

## Real Analog: Chapters 1-12

1300 Henley Court  
Pullman, WA 99163  
509.334.6306  
www.store.digilent.com

---

## Table of Contents

Table of Contents .....	1
Chapter 1: Circuit Analysis Fundamentals .....	39
1 Introduction and Chapter Objectives .....	39
After Completing this Chapter, You Should be Able to: .....	39
1.1 Basic Circuit Parameters and Sign Conventions .....	40
1.1.1 Electrical Charge .....	40
1.1.2 Voltage .....	40
1.1.3 Current .....	41
1.1.4 Power .....	41
1.1.5 International System of Units and Prefixes .....	41
1.1.6 Passive Sign Convention .....	42
Example 1.1: .....	42
Example 1.2: .....	43
Example 1.3: .....	43
1.1.7 Voltage Subscript and Sign Conventions .....	44
1.1.8 Reference Voltages and Ground .....	44
Example 1.4: .....	44
1.1.9 Power and Sign Conventions .....	45
Example 1.5: .....	45
Section Summary .....	45
1.1 Exercises .....	46

1.2	Power Sources.....	47
1.2.1	Independent Voltage Sources.....	47
1.2.2	Independent Current Sources.....	48
1.2.3	Dependent Sources.....	48
	Section Summary .....	50
1.2	Exercises: .....	50
1.3	Resistors and Ohm's Law .....	50
1.3.1	Conductance .....	52
1.3.2	Power Dissipation .....	52
1.3.3	Practical Resistors .....	52
	Example 1.6:.....	54
	Section Summary .....	54
1.3	Exercises .....	54
1.4	Kirchhoff's Laws.....	55
1.4.1	Nodes .....	55
	Example 1.7.....	55
1.4.2	Loops.....	56
	Example 1.8.....	56
1.4.3	Kirchhoff's Current Law .....	56
	Example 1.9:.....	57
	Example 1.10:.....	57
1.4.4	Kirchhoff's Voltage Law .....	58
	Example 1.11:.....	59
	Application Examples: Solving for Circuit Element Variables.....	60
	Example 1.12:.....	60
	Example 1.13:.....	61
	Section Summary .....	62
1.4	Exercises: .....	62
	Chapter Summary: .....	62
	<b>Real Analog Chapter 1: Lab Projects.....</b>	<b>64</b>

1.1: Solderless Breadboards, Open-circuits, and Short-circuits .....	64
Symbol Key.....	64
General Discussion:.....	64
Pre-lab: None .....	64
Lab Procedures: .....	65
<b>Real Analog Chapter 1: Lab Worksheets .....</b>	<b>67</b>
1.1: Breadboards and Ohmmeters (20 points total) .....	67
<b>Real Analog Chapter 1: Lab Projects .....</b>	<b>68</b>
1.2.1: Independent Power Supplies, Ammeters, and Voltmeters .....	68
Symbol Key:.....	68
General Discussion:.....	68
Notes: .....	69
Note: .....	69
Pre-lab: None .....	69
Lab Procedures: .....	70
<b>Real Analog Chapter 1: Lab Worksheets .....</b>	<b>71</b>
1.2.1: Power Supplies, Voltmeters, Ammeters (25 points total).....	71
<b>Real Analog Chapter 1: Lab Projects .....</b>	<b>73</b>
1.2.2: Dependent Sources and MOSFETs.....	73
Overview: .....	73
Symbol Key:.....	73
General Discussion:.....	73
Pre-lab: None .....	74
Lab Procedures: .....	74
Appendix A: Waveform Generator to Apply Constant Voltages .....	76
Important notes relative to the Enabled/Disabled button:.....	77
<b>Real Analog Chapter 1: Lab Worksheets .....</b>	<b>78</b>

1.2.2: Dependent Sources and MOSFETs (55 points total).....	78
<b>Real Analog Chapter 1: Lab Projects.....</b>	<b>79</b>
1.3.1: Resistors and Ohms Law – Resistance Variations.....	79
Symbol Key:.....	79
General Discussion:.....	79
Important Note: .....	79
Related information:.....	80
Pre-lab: None .....	80
Lab Procedures: .....	80
Appendix A: Basic Statistics .....	80
Mean value: .....	80
Median value: .....	80
Standard deviation:.....	80
MATLAB syntax: .....	81
<b>Real Analog Chapter 1: Lab Worksheets.....</b>	<b>82</b>
1.3.1: Resistance Variations (20 points total).....	82
<b>Real Analog Chapter 1: Lab Projects.....</b>	<b>83</b>
1.3.2: Resistors and Ohms Law – Voltage-Current Characteristics .....	83
Symbol Key:.....	83
General Discussion:.....	83
Pre-lab: None .....	84
Lab Procedures: .....	84
Related information:.....	85
Post-lab Exercises:.....	85
Appendix A: Linear Regression: .....	85
Least-squares curve fitting:.....	85
Using MATLAB for least squares curve fitting: .....	86
<b>Real Analog Chapter 1: Lab Worksheets.....</b>	<b>88</b>

1.3.2: Resistance from Measured Current-voltage Data (25 points total) .....	88
<b>Real Analog Chapter 1: Lab Projects .....</b>	<b>89</b>
1.4.1: Dusk-to-Dawn Light .....	89
Symbol Key:.....	89
General Discussion:.....	89
Diodes and LEDs:.....	90
Bipolar Junction Transistors (BJTs): .....	91
Photocell: .....	91
Pre-lab:.....	92
Lab Procedures: .....	92
Real Analog Chapter 1: Lab Worksheets.....	93
1.4.1: Dusk-to-dawn Light (20 points total).....	93
<b>Real Analog Chapter 1: Lab Projects .....</b>	<b>94</b>
1.4.2: Resistive Network Power Dissipation .....	94
Symbol Key:.....	94
General Discussion:.....	94
Pre-lab:.....	95
Lab Procedures: .....	95
<b>Real Analog Chapter 1: Lab Worksheets .....</b>	<b>96</b>
1.4.2: Resistive Network Power Dissipation (20 points total) .....	96
<b>Real Analog Chapter 1: Lab Projects .....</b>	<b>97</b>
1.4.3: Input Resistance.....	97
Symbol Key:.....	97
General Discussion:.....	97
Pre-lab:.....	98
Lab Procedures: .....	98
<b>Real Analog Chapter 1: Lab Worksheets .....</b>	<b>99</b>

1.4.3: Circuit Input Resistance (25 points total) .....	99
<b>Real Analog Chapter 1: Lab Projects .....</b>	<b>100</b>
1.4.4: Temperature Measurement System.....	100
Symbol Key:.....	100
General Discussion:.....	100
Related Material: .....	101
Pre-lab:.....	102
Note on the design process: .....	102
Lab Procedures: .....	102
Post-Lab Exercises:.....	103
1.4.4 Background Information: Resistive Temperature Sensors.....	103
Resistive Temperature Detectors: .....	104
RTD Signal Conditioning:.....	105
Thermistors: .....	105
Thermistor Signal Conditioning: .....	106
Issues associated with resistive temperature measurement:.....	106
<b>Real Analog Chapter 1: Lab Worksheets .....</b>	<b>107</b>
1.4.4: Temperature Measurement System (40 points total).....	107
<b>Real Analog Chapter 1: Homework .....</b>	<b>109</b>
<b>Chapter 2: Circuit Reduction .....</b>	<b>115</b>
<b>2 Introduction and Chapter Objectives .....</b>	<b>115</b>
After Completing this Chapter, You Should be Able to: .....	115
2.1 Series Circuit Elements and Voltage Division.....	115
2.1.1 Series Connections.....	116
2.1.2 Voltage Division .....	117
Example 2.1.....	118
Section Summary: .....	118
2.1 Exercises .....	119

2.2	Parallel Circuit Elements and Current Division .....	119
2.2.1	Current Division .....	120
	Section Summary .....	121
2.2	Exercises .....	122
2.3	Circuit Reduction and Analysis.....	122
	Example 2.2.....	123
	Example 2.3.....	123
	Example 2.4.....	124
	Example 2.5: Wheatstone Bridge .....	125
	Section Summary .....	126
2.3	Exercises .....	126
2.4	Non-ideal Power Supplies .....	126
2.4.1	Non-ideal Voltage Sources.....	126
	Example 2.6.....	127
	Example 2.7: Charging a Battery.....	127
2.4.2	Non-ideal Current Sources.....	128
	Example 2.8.....	129
	Section Summary .....	130
2.4	Exercises .....	130
2.5	Practical Voltage and Current Measurement .....	130
2.5.1	Voltmeter and Ammeter Models.....	131
2.5.2	Voltage Measurement .....	131
2.5.3	Current Measurement .....	132
	Caution .....	132
	Section Summary .....	133
2.5	Exercises .....	133
<b>Real Analog Chapter 2: Lab Projects.....</b>		<b>134</b>
2.1.1:	Temperature Measurement System.....	134
	Symbol Key:.....	134
	General Discussion:.....	134

Pre-lab:.....	135
Lab Procedures: .....	136
<b>Real Analog Chapter 2: Lab Worksheets.....</b>	<b>137</b>
2.1.1: Temperature Measurement System (50 points total).....	137
<b>Real Analog Chapter 2: Lab Projects.....</b>	<b>138</b>
2.3.1: Series and Parallel Resistances and Circuit Reduction .....	138
Symbol Key:.....	138
General Discussion:.....	138
Pre-lab:.....	139
Lab Procedures: .....	139
<b>Real Analog Chapter 2: Lab Worksheets.....</b>	<b>140</b>
2.3.1: Series and Parallel Resistors and Equivalent Resistance (20 points total) .....	140
<b>Real Analog Chapter 2: Lab Projects.....</b>	<b>141</b>
2.3.2: Series and Parallel Resistances and Circuit Reduction .....	141
Symbol Key:.....	141
General Discussion:.....	141
Pre-lab:.....	142
Lab Procedures: .....	142
Note: .....	142
<b>Real Analog Chapter 2: Lab Worksheets.....</b>	<b>143</b>
2.3.2: Series and Parallel Resistors and Equivalent Resistance (25 points total) .....	143
<b>Real Analog Chapter 2: Lab Projects.....</b>	<b>144</b>
2.4: Non-ideal Power Sources.....	144
Symbol Key:.....	144
General Discussion:.....	144
Pre-lab:.....	145

Hint – non-ideal voltage sources: .....	145
Lab Procedures: .....	145
Note: .....	146
<b>Real Analog Chapter 2: Lab Worksheets .....</b>	<b>147</b>
2.4: Non-ideal Power Sources (20 points total).....	147
<b>Real Analog Chapter 2: Lab Projects .....</b>	<b>148</b>
2.5: Practical Voltage and Current Measurement .....	148
Symbol Key:.....	148
General Discussion:.....	148
Pre-lab:.....	149
Hint – non-ideal voltmeters:.....	149
Lab Procedures: .....	150
Note: .....	150
<b>Real Analog Chapter 2: Lab Worksheets .....</b>	<b>151</b>
2.5: Practical Voltage and Current Measurement (25 points total) .....	151
<b>Real Analog Chapter 2: Homework .....</b>	<b>152</b>
<b>Chapter 3: Nodal and Mesh Analysis .....</b>	<b>155</b>
<b>3 Introduction and Chapter Objectives .....</b>	<b>155</b>
After Completing this Chapter, You Should be Able to: .....	155
3.1 Introduction and Terminology .....	155
Basic Idea .....	156
Basic Idea .....	156
Section Summary .....	157
3.1 Exercises .....	157
3.2 Nodal Analysis .....	158
Example 3.1.....	161
Note .....	163

3.2.1	Supernodes .....	163
3.2.2	Alternate Approach: Constraint Equations.....	165
	Example 3.2.....	167
3.2.3	Dependent Sources.....	168
	Example 3.3.....	168
	Example 3.4.....	169
	Section Summary .....	170
3.2	Exercises .....	170
3.3	Mesh Analysis.....	170
	Example 3.5.....	173
3.3.1	Alternate Approach to Constraint Loops: Constraint Equations .....	174
3.3.2	Clarification: Constraint Loops.....	175
	Example 3.6: Version 1 .....	176
	Example 3.6: Version 2 .....	177
	Example 3.6: Version 3 .....	177
3.3.3	Dependent Sources.....	178
	Example 3.7.....	178
	Example 3.8.....	179
	Section Summary .....	180
3.3	Exercises .....	180
<b>Real Analog Chapter 3: Lab Projects.....</b>		<b>182</b>
3.2.1:	Nodal Analysis.....	182
	Symbol Key:.....	182
	General Discussion:.....	182
	Pre-lab:.....	183
	Lab Procedures: .....	183
<b>Real Analog Chapter 3: Lab Worksheets.....</b>		<b>184</b>
3.2.1:	Nodal Analysis (25 points total).....	184
<b>Real Analog Chapter 3: Lab Projects.....</b>		<b>185</b>

3.2.2: Nodal Analysis.....	185
Symbol Key:.....	185
General Discussion:.....	185
Pre-lab:.....	186
Lab Procedures: .....	186
<b>Real Analog Chapter 3: Lab Worksheets .....</b>	<b>187</b>
3.2.2: Nodal Analysis (25 points total).....	187
<b>Real Analog Chapter 3: Lab Projects.....</b>	<b>188</b>
3.2.3: Nodal Analysis.....	188
Symbol Key:.....	188
General Discussion:.....	188
Pre-lab:.....	189
Lab Procedures: .....	189
<b>Real Analog Chapter 3: Lab Worksheets.....</b>	<b>190</b>
3.2.3: Nodal Analysis (25 points total).....	190
<b>Real Analog Chapter 3: Lab Projects.....</b>	<b>191</b>
3.3.1: Mesh Analysis .....	191
Symbol Key:.....	191
General Discussion:.....	191
Pre-lab:.....	192
Lab Procedures: .....	192
<b>Real Analog Chapter 3: Lab Worksheets.....</b>	<b>193</b>
3.3.1: Mesh Analysis (25 points total) .....	193
<b>Real Analog Chapter 3: Lab Projects.....</b>	<b>194</b>
3.3.2: Mesh Analysis .....	194
Symbol Key:.....	194

General Discussion:.....	194
Pre-lab:.....	195
Lab Procedures: .....	195
<b>Real Analog Chapter 3: Lab Worksheets .....</b>	<b>196</b>
3.3.2: Mesh Analysis (25 points total) .....	196
<b>Real Analog Chapter 3: Lab Projects .....</b>	<b>197</b>
3.3.3: Mesh Analysis .....	197
Symbol Key:.....	197
General Discussion:.....	197
Pre-lab:.....	198
Lab Procedures: .....	198
<b>Real Analog Chapter 3: Lab Worksheets .....</b>	<b>199</b>
3.3.3: Mesh Analysis (25 points total) .....	199
<b>Real Analog Chapter 3: Homework .....</b>	<b>200</b>
<b>Chapter 4: System and Network Theorems .....</b>	<b>204</b>
<b>4 Introduction and Chapter Objectives .....</b>	<b>204</b>
After Completing this Chapter, You Should be Able to: .....	204
4.1 Signals and Systems .....	205
Example 4.1: Mass Subjected to an External Force .....	206
Example 4.2: Electrical Circuit.....	206
Example 4.3: Temperature Control System.....	207
Section Summary .....	208
4.1 Exercises .....	209
4.2 Linear Systems.....	209
4.2.1 Linearity: .....	211
4.2.2 Dependent Variables and Linearity: .....	211
4.2.3 Linearity: .....	212

Section Summary .....	212
4.2 Exercises .....	212
4.3 Superposition .....	213
Example 4.4.....	214
Example 4.5.....	214
Section Summary .....	215
4.3 Exercises .....	216
4.4 Two-terminal Networks .....	216
4.4.1 Resistive Networks.....	217
Example 4.6.....	217
4.4.2 Two-terminal Networks with Sources .....	218
Example 4.7.....	219
Example 4.8.....	219
Section Summary .....	221
4.4 Exercises .....	221
4.5 Thévenin's and Norton's Theorems.....	222
4.5.1 Thévenin's Theorem .....	222
Example 4.9.....	224
4.5.2 Norton's Theorem.....	225
Example 4.10.....	226
4.5.3 Source Transformations.....	227
Example 4.11.....	228
4.5.4 Voltage-current Characteristics of Thévenin and Norton Circuits .....	229
Section Summary .....	230
4.5 Exercises .....	230
4.6 Maximum Power Transfer.....	231
4.6.1 Practical Power Supplies.....	233
Section Summary .....	234
4.6 Exercises .....	234
<b>Real Analog Chapter 4: Lab Projects.....</b>	<b>235</b>

4.3.1: Superposition .....	235
Symbol Key:.....	235
General Discussion:.....	235
Pre-lab:.....	236
Lab Procedures: .....	236
<b>Real Analog Chapter 4: Lab Worksheets .....</b>	<b>237</b>
4.3.1: Superposition (25 points total).....	237
<b>Real Analog Chapter 4: Lab Projects .....</b>	<b>238</b>
4.3.2: Superposition .....	238
Symbol Key:.....	238
General Discussion:.....	238
Pre-lab:.....	239
Lab Procedures: .....	239
<b>Real Analog Chapter 4: Lab Worksheets .....</b>	<b>240</b>
4.3.2: Superposition (25 points total).....	240
<b>Real Analog Chapter 4: Lab Projects .....</b>	<b>241</b>
4.4.1: Two-terminal Characteristics.....	241
Symbol Key:.....	241
General Discussion:.....	241
Pre-lab:.....	242
Lab Procedures: .....	242
<b>Real Analog Chapter 4: Lab Worksheets .....</b>	<b>243</b>
4.4.1: Two-terminal Characteristics (25 points total).....	243
<b>Real Analog Chapter 4: Lab Projects .....</b>	<b>244</b>
4.5.1: Thevenin's Theorem .....	244
Symbol Key:.....	244

General Discussion:.....	244
Pre-lab:.....	245
Lab Procedures: .....	245
<b>Real Analog Chapter 4: Lab Worksheets .....</b>	<b>246</b>
4.5.1: Thevenin's Theorem (35 points total) .....	246
<b>Real Analog Chapter 4: Lab Projects .....</b>	<b>248</b>
4.6.1: Maximum Power Transfer .....	248
Symbol Key:.....	248
General Discussion:.....	248
Pre-lab:.....	248
Lab Procedures: .....	249
<b>Real Analog Chapter 4: Lab Worksheets .....</b>	<b>251</b>
4.6.1: Maximum Power Transfer (30 points total) .....	251
<b>Real Analog Chapter 4: Homework .....</b>	<b>252</b>
<b>Chapter 5: Operational Amplifiers .....</b>	<b>254</b>
<b>5 Introduction and Chapter Objectives .....</b>	<b>254</b>
After Completing this Chapter, You Should be Able to: .....	255
5.1 Ideal Operational Amplifier Model .....	255
Section Summary .....	257
5.1 Exercises .....	258
5.2 Operational Amplifier Model Background .....	258
Section Summary .....	261
5.2 Exercises .....	261
5.3 Commercially Available Operational Amplifiers .....	261
5.3 Exercises: .....	263
5.4 Analysis of Op-amp Circuits .....	263
Example 5.1.....	264

Example 5.2.....	266
Example 5.3.....	267
Example 5.4.....	268
Example 5.5.....	269
Section Summary .....	270
5.4 Exercises .....	271
5.5 Comparators.....	271
5.5 Exercises .....	272
5.6 A Few Non-ideal Effects .....	272
5.6.1 Input Resistance Effects.....	273
5.6.2 Output Resistance Effects.....	274
5.6.3 Finite Gain Effects .....	274
Section Summary .....	275
<b>Real Analog Chapter 5: Lab Projects.....</b>	<b>276</b>
5.4.1: Inverting Voltage Amplifier.....	276
Symbol Key:.....	276
General Discussion:.....	276
Pre-lab:.....	277
Lab Procedures: .....	277
<b>Real Analog Chapter 5: Lab Worksheets.....</b>	<b>278</b>
5.4.1: Inverting Voltage Amplifier (30 points total).....	278
<b>Real Analog Chapter 5: Lab Projects.....</b>	<b>279</b>
5.4.2: Summing Amplifier .....	279
Symbol Key:.....	279
General Discussion:.....	279
Pre-lab:.....	280
Lab Procedures: .....	280
<b>Real Analog Chapter 5: Lab Worksheets.....</b>	<b>281</b>

5.4.2: Summing Amplifier (30 points total) .....	281
<b>Real Analog Chapter 5: Lab Projects .....</b>	<b>282</b>
5.4.3: Non-inverting Voltage Amplifier .....	282
Symbol Key:.....	282
General Discussion:.....	282
Pre-lab: .....	283
Lab Procedures: .....	283
<b>Real Analog Chapter 5: Lab Worksheets .....</b>	<b>284</b>
5.4.3: Non-inverting Voltage Amplifier (30 points total).....	284
<b>Real Analog Chapter 5: Lab Projects .....</b>	<b>285</b>
5.4.3: Difference Amplifier.....	285
Symbol Key:.....	285
General Discussion:.....	285
Pre-lab:.....	286
Lab Procedures: .....	286
<b>Real Analog Chapter 5: Lab Worksheets .....</b>	<b>287</b>
5.4.4: Difference Amplifier (30 points total).....	287
<b>Real Analog Chapter 5: Lab Projects .....</b>	<b>288</b>
5.4.5: Temperature Measurement System Design.....	288
Symbol Key:.....	288
General Discussion:.....	288
Pre-lab:.....	289
Lab Procedures: .....	289
Design Hint:.....	290
Design Hint:.....	290
Appendix A: Wheatstone Bridge Circuits.....	291
Wheatstone bridge sensitivity to resistance variations: .....	291

Balancing the Wheatstone Bridge Circuit:.....	292
Governing equations for balanced circuit: .....	292
Practical Note:.....	293
Appendix B: Difference Amplifier .....	293
<b>Real Analog Chapter 5: Lab Worksheets .....</b>	<b>294</b>
5.4.5: Temperature Measurement System (100 points total).....	294
<b>Real Analog Chapter 5: Homework .....</b>	<b>296</b>
<b>Chapter 6: Energy Storage Elements .....</b>	<b>302</b>
<b>6 Introduction and Chapter Objectives .....</b>	<b>302</b>
After completing this chapter, you should be able to:.....	302
6.2 Fundamental Concepts .....	303
6.2.1 Systems with no Energy Storage.....	303
6.2.2 Systems with Energy Storage.....	304
Example 6.1: Mass-damper system .....	305
Example 6.2.....	306
Section Summary .....	308
6.1 Exercises .....	308
6.2 Basic Time-varying Signals.....	309
6.2.1 Step Function .....	309
6.2.2 Exponential Functions.....	312
Section Summary .....	313
6.2 Exercises .....	314
6.3 Capacitors.....	314
6.3.1 Capacitors .....	314
Example 6.3.....	316
Example 6.4.....	317
6.3.2 Energy Storage .....	318
Example 6.5.....	319

6.3.3	Capacitors in Series .....	320
6.3.4	Capacitors in Parallel.....	321
6.3.5	Practical Capacitors.....	322
	Section Summary .....	323
6.3	Exercises .....	324
6.4	Inductors .....	324
6.4.1	Inductors .....	324
	Example 6.6.....	326
6.4.2	Energy Storage .....	328
6.4.3	Inductors in Series.....	328
6.4.4	Inductors in Parallel .....	329
	Summary: Series and Parallel Inductors .....	330
6.5	Practical Inductors.....	331
	Section Summary .....	331
6.4	Exercises .....	332
<b>Real Analog Chapter 6: Lab Projects.....</b>		<b>333</b>
6.2.1:	Time-varying Signals .....	333
	Symbol Key:.....	333
	General Discussion:.....	333
	Pre-lab:.....	335
	Lab Procedures: .....	336
	Note on selecting parameters: .....	336
<b>Real Analog Chapter 6: Lab Worksheets.....</b>		<b>338</b>
6.2.1:	Time-varying Signals (40 points total) .....	338
<b>Real Analog Chapter 6: Lab Projects.....</b>		<b>339</b>
6.3.1:	Capacitor Voltage-current Relations .....	339
	Symbol Key:.....	339
	General Discussion:.....	339
	Pre-lab:.....	340

Lab Procedures: .....	340
Post-lab Exercises:.....	341
<b>Real Analog Chapter 6: Lab Worksheets .....</b>	<b>342</b>
6.3.1: Capacitor Voltage-current Relations (35 points total) .....	342
<b>Real Analog Chapter 6: Lab Projects .....</b>	<b>343</b>
6.3.2: Leakage Currents and Electrolytic Capacitors .....	343
Symbol Key:.....	343
General Discussion:.....	343
Caution:.....	344
Pre-lab:.....	345
Lab Procedures: .....	345
Post-lab Exercises:.....	346
<b>Real Analog Chapter 6: Lab Worksheets .....</b>	<b>348</b>
6.3.2: Electrolytic Capacitor Leakage Currents (35 points total) .....	348
<b>Real Analog Chapter 6: Lab Projects .....</b>	<b>349</b>
6.4.1: Inductor Voltage-current Relations .....	349
Symbol Key:.....	349
General Discussion:.....	349
Pre-lab:.....	350
Lab Procedures: .....	350
Post-lab Exercises:.....	351
Appendix A: Math channel to calculate current from resistor's voltage .....	351
<b>Real Analog Chapter 6: Lab Worksheets .....</b>	<b>353</b>
6.4.1: Inductor Voltage-current Relations (40 points total) .....	353
<b>Real Analog Chapter 6: Lab Projects .....</b>	<b>354</b>
6.4.2: Non-Ideal Inductor Effects.....	354

Symbol Key:.....	354
General Discussion:.....	354
Pre-lab:.....	355
Lab Procedures: .....	356
<b>Real Analog Chapter 6: Lab Worksheets .....</b>	<b>358</b>
6.4.2: Non-ideal Inductor Effects (45 points total) .....	358
<b>Real Analog Chapter 6: Homework .....</b>	<b>359</b>
<b>Chapter 7: First Order Circuits.....</b>	<b>360</b>
<b>7 Introduction and Chapter Objectives .....</b>	<b>360</b>
After completing this chapter, you should be able to:.....	360
7.1 Introduction to First Order Systems.....	361
7.1.1 Mass-damper System Example.....	362
Summary .....	366
Section Summary .....	368
7.1 Exercises .....	369
7.2 Natural Response of RC Circuits.....	369
7.2.1 Generalization to Multiple Resistors .....	372
Example 7.1.....	372
7.2.2 Determining Initial Conditions .....	374
Example 7.2: Switched Circuit Natural Response .....	374
Section Summary .....	375
7.2 Exercises .....	376
7.3 Natural Response of RL Circuits .....	376
7.3.1 Generalization to Multiple Resistors .....	378
7.3.2 Determining Initial Conditions .....	378
Example 7.3: Switched RL Circuit Natural Response .....	379
Section Summary .....	380
7.3 Exercises .....	381

7.4	Forces Response of First Order Circuits .....	381
7.4.1	Generalization to Multiple Resistors .....	383
	Example 7.4.....	384
	Example 7.5.....	385
	Example 7.6.....	386
7.4.2	Cross-checking Results.....	386
	Example 7.7.....	387
	Section Summary .....	387
7.4	Exercises .....	388
7.5	Step Response of First Order Circuits .....	389
	Example 7.8.....	390
	Example 7.9.....	392
	Example 7.10.....	394
7.5.1	Cross-checking Results.....	395
	Example 7.11.....	396
	Example 7.12.....	396
7.5.2	DC Gain.....	396
	Example 7.13.....	396
	Section Summary .....	397
7.5	Exercises .....	398
<b>Real Analog Chapter 7: Lab Projects.....</b>		<b>400</b>
7.2.1:	Passive RC Circuit Natural Response .....	400
	Symbol Key:.....	400
	General Discussion:.....	400
	Pre-lab:.....	401
	Lab Procedures: .....	401
Appendix A: Triggering and Single Acquisition .....		402
Appendix B: Continuous Acquisition.....		404
<b>Real Analog Chapter 7: Lab Worksheets.....</b>		<b>405</b>
7.2.1:	Passive RC Circuit Natural Response (35 points total) .....	405

<b>Real Analog Chapter 7: Lab Projects</b> .....	<b>407</b>
7.3.1: Passive RL Circuit Natural Response.....	407
Symbol Key:.....	407
General Discussion:.....	407
Pre-lab:.....	408
Lab Procedures: .....	408
Appendix A: Triggering and Single Acquisition .....	409
Appendix B: Continuous Acquisition.....	411
<b>Real Analog Chapter 7: Lab Worksheets</b> .....	<b>412</b>
7.3.1: Passive RL Circuit Natural Response (35 points total) .....	412
<b>Real Analog Chapter 7: Lab Projects</b> .....	<b>414</b>
7.4.1: Inverting Differentiator.....	414
Symbol Key:.....	414
General Discussion:.....	414
Pre-lab:.....	415
Note: .....	415
Lab Procedures: .....	415
Post-lab Exercises:.....	415
<b>Real Analog Chapter 7: Lab Worksheets</b> .....	<b>417</b>
7.4.1: Passive RL Circuit Natural Response (30 points total) .....	417
<b>Real Analog Chapter 7: Lab Projects</b> .....	<b>418</b>
7.5.1: Passive RC Circuit Step Response .....	418
Symbol Key:.....	418
General Discussion:.....	418
Pre-lab:.....	419
Lab Procedures: .....	419
Post-lab Exercises:.....	420

<b>Real Analog Chapter 7: Lab Worksheets</b> .....	<b>421</b>
7.5.1: Passive RC Circuit Step Response (40 points total) .....	421
<b>Real Analog Chapter 7: Lab Projects</b> .....	<b>423</b>
7.5.2: Passive RL Circuit Step Response .....	423
Symbol Key: .....	423
General Discussion: .....	423
Pre-lab: .....	424
Lab Procedures: .....	424
Post-lab Exercises: .....	425
<b>Real Analog Chapter 7: Lab Worksheets</b> .....	<b>426</b>
7.5.2: Passive RL Circuit Step Response (40 points total) .....	426
<b>Real Analog Chapter 7: Lab Projects</b> .....	<b>428</b>
7.5.3: Active RC Circuit Step Response .....	428
Symbol Key: .....	428
General Discussion: .....	428
Pre-lab: .....	429
Lab Procedures: .....	429
<b>Real Analog Chapter 7: Lab Worksheets</b> .....	<b>431</b>
7.5.3: Active RC Circuit Step Response (50 points total) .....	431
<b>Real Analog Chapter 7: Homework</b> .....	<b>434</b>
<b>Chapter 8: Second Order Circuits</b> .....	<b>438</b>
<b>8 Introduction and Chapter Objectives</b> .....	<b>438</b>
After completing this chapter, you should be able to: .....	438
8.1 Introduction to Second Order Systems .....	439
Example 8.1: Series RLC Circuit .....	439

Example 8.2: Parallel RLC Circuit .....	441
Section Summary .....	442
8.1 Exercises .....	442
8.2 Second Order System Natural Response, Part 1 .....	443
Section Summary .....	445
8.3 Sinusoidal Signals and Complex Exponentials.....	446
8.3.1 Sinusoidal Signals .....	446
8.3.2 Complex Exponentials.....	447
Section Summary .....	449
8.2 Exercises .....	449
8.4 Second Order System Natural Response, Part 2 .....	449
1. Overdamped System.....	450
2. Underdamped System .....	451
3. Critically Damped System .....	452
Example 8.3.....	453
Section Summary .....	458
8.3 Exercises .....	458
8.5 Second Order System Step Response .....	459
Section Summary .....	461
8.4 Exercises .....	462
<b>Real Analog Chapter 8: Lab Projects.....</b>	<b>463</b>
8.5.1: Series RLC Circuit Step Response .....	463
Symbol Key:.....	463
Second Order Series RLC Circuit: .....	463
Pre-lab:.....	464
Lab Procedures: .....	464
Critically Damped Series RLC Circuit: .....	464
Pre-lab:.....	465
Lab Procedures: .....	465
<b>Real Analog Chapter 8: Lab Worksheets.....</b>	<b>466</b>

8.5.1 Part 1: Second Order Series RLC Circuit (35 points total).....	466
<b>Real Analog Chapter 8: Lab Worksheets .....</b>	<b>467</b>
8.5.1 Part 2: Critically Damped Series RLC Circuit (25 points total) .....	467
<b>Real Analog Chapter 8: Lab Projects .....</b>	<b>468</b>
8.5.2: Parallel RLC Circuit Response.....	468
Symbol Key:.....	468
General Discussion:.....	468
Pre-lab:.....	469
Lab Procedures: .....	469
<b>Real Analog Chapter 8: Lab Worksheets .....</b>	<b>470</b>
8.5.2: Parallel RLC Circuit Response (40 points total).....	470
<b>Real Analog Chapter 8: Lab Projects .....</b>	<b>471</b>
8.5.3: RLC Circuit Response.....	471
Symbol Key:.....	471
General Discussion:.....	471
Pre-lab:.....	472
Lab Procedures: .....	472
<b>Real Analog Chapter 8: Lab Worksheets .....</b>	<b>473</b>
8.5.3: RLC Circuit Response (40 points total).....	473
<b>Real Analog Chapter 8: Homework .....</b>	<b>474</b>
<b>Chapter 9: Introduction to State Variable Models .....</b>	<b>477</b>
<b>9 Introduction and Chapter Objectives .....</b>	<b>477</b>
After completing this chapter, you should be able to:.....	477
9.1 Introduction to State Variable Models.....	478
9.1.1 Background and Introduction .....	478

9.1.2	Form of State Variable Models .....	478
	Example 9.1: State Variable Model of Series RLC Circuit .....	479
	Example 9.2: State Variable Model of Third Order Circuit .....	480
	Section Summary .....	481
9.1	Exercises .....	482
9.2	Numerical Simulation of System Responses Using MATLAB .....	482
9.2.1	Basic Commands for Simulation of Linear, Time-invariant Systems .....	483
	Example 9.3: Step Response of Series RLC Circuit .....	483
	Section Summary: .....	485
9.2	Exercises .....	486
9.3	Numerical Simulation of System Responses Using Octave.....	486
9.3.1	Basic Commands for Simulation of Linear, Time-invariant Systems .....	486
	Example 9.4: Step Response of Series RLC Circuit .....	487
	Section Summary .....	489
9.3	Exercises .....	489
<b>Real Analog Chapter 9: Lab Projects .....</b>		<b>491</b>
9.3.1:	State Variable Model of Series RLC Circuits.....	491
	Symbol Key:.....	491
	General Discussion:.....	491
	Pre-lab:.....	492
	Lab Procedures: .....	492
	Post-lab Exercises:.....	493
<b>Real Analog Chapter 9: Lab Worksheets .....</b>		<b>494</b>
9.3.1:	State Variable Model of Series RLC Circuits (50 points total).....	494
<b>Real Analog Chapter 9: Lab Projects .....</b>		<b>496</b>
9.3.2:	Second Order Circuit Response .....	496
	Symbol Key:.....	496
	General Discussion:.....	496
	Pre-lab:.....	497

Lab Procedures: .....	497
Post-lab Exercises:.....	498
<b>Real Analog Chapter 9: Lab Worksheets .....</b>	<b>499</b>
9.3.2: Second Order Circuit Response (50 points total) .....	499
<b>Real Analog Chapter 9: Homework .....</b>	<b>501</b>
<b>Chapter 10: Steady-state Sinusoidal Analysis .....</b>	<b>503</b>
<b>10 Introduction and Chapter Objectives .....</b>	<b>503</b>
After completing this chapter, you should be able to:.....	503
10.1 Introduction to Steady-state Sinusoidal Analysis.....	504
Example 10.1: Series RLC Circuit Response .....	505
Section Summary .....	506
10.1 Exercises.....	507
10.2 Sinusoidal Signals, Complex Exponentials, and Phasors .....	507
10.2.1 Sinusoidal Signals .....	507
10.2.2 Complex Exponentials and Phasors .....	510
10.2.3 Complex Arithmetic Review.....	511
Section Summary .....	514
10.2 Exercises.....	514
10.3 Sinusoidal Steady-state System Response .....	515
Example 10.2: RC Circuit Sinusoidal Steady-state Response via Time-domain Analysis....	516
Example 10.3: RC Circuit Sinusoidal Steady-state Response via Frequency-domain Analysis	517
Example 10.4: Numerical Example and Phasor Diagrams .....	518
Example 10.5: RL Circuit Sinusoidal Steady-state Response .....	519
Example 10.6: Series RLC Circuit Sinusoidal Steady-state Response.....	520
Section Summary .....	522
10.3 Exercises.....	522
10.4 Phasor Representations of Circuit Elements .....	522
10.4.1 Phasor Domain Voltage-current Relationships.....	523

10.4.2	Resistors .....	523
10.4.3	Inductors .....	525
10.3.4	Capacitors .....	526
10.3.5	Impedance and Admittance.....	528
	Example 10.7: Provide the Phasor-domain Representation of the Circuit Below.....	530
	Section Summary .....	530
10.4	Exercises.....	531
10.5	Direct Frequency Domain Circuit Analysis .....	531
10.5.1	Kirchhoff's Voltage Law .....	532
10.5.2	Kirchhoff's Current Law.....	532
	Example 10.8: RC Circuit Steady-state Sinusoidal Response.....	533
10.5.3	Parallel and Series Impedances & Circuit Reduction.....	534
	Example 10.9.....	535
	Example 10.10.....	536
10.5.4	Nodal and Mesh Analysis.....	537
	Example 10.11.....	537
	Example 10.12.....	538
10.5.5	Superposition .....	538
	Example 10.13.....	539
	Example 10.14.....	540
10.5.6	Thévenin's & Norton's Theorems, Source Transformations, and Maximum Power Transfer	542
	Example 10.15.....	543
	Example 10.16.....	544
	Example 10.17.....	545
	Section Summary .....	546
10.5	Exercises:.....	546
10.6	Frequency Domain System Characterization .....	547
	Example 10.18.....	548
	Example 10.19.....	549
	Example 10.20.....	550
	Section Summary .....	551

10.5 Exercises.....	551
<b>Real Analog Chapter 10: Lab Projects .....</b>	<b>553</b>
10.4.1: Impedance .....	553
Symbol Key:.....	553
General Discussion:.....	553
Pre-lab: .....	554
Lab Procedures: .....	554
Appendix A: Measuring Gain and Phase:.....	556
<b>Real Analog Chapter 10: Lab Worksheets .....</b>	<b>557</b>
10.4.1: Impedance (75 points total) .....	557
a. Resistor (25 points) .....	557
b. Inductor (25 points) .....	558
c. Capacitor (25 points).....	559
<b>Real Analog Chapter 10: Lab Projects .....</b>	<b>560</b>
10.6.1: Passive RL Circuit Response .....	560
Symbol Key:.....	560
General Discussion:.....	561
Pre-lab: .....	561
Notes:.....	561
Lab Procedures: .....	562
Appendix A: Measuring Gain and Phase.....	562
<b>Real Analog Chapter 10: Lab Worksheets .....</b>	<b>564</b>
10.6.1: Passive RL Circuit Response (40 points total) .....	564
<b>Real Analog Chapter 10: Lab Projects .....</b>	<b>566</b>
10.6.2: Passive RC Circuit Response .....	566
Symbol Key:.....	567
General Discussion:.....	567

Pre-lab:.....	567
Lab Procedures: .....	567
Hint:.....	568
Appendix A: Measuring Gain and Phase:.....	568
<b>Real Analog Chapter 10: Lab Worksheets .....</b>	<b>570</b>
10.6.2: Passive RC Circuit Response (30 points total).....	570
<b>Real Analog Chapter 10: Lab Projects .....</b>	<b>572</b>
10.6.3: Inverting Voltage Amplifier.....	572
Symbol Key:.....	572
General Discussion:.....	573
Pre-lab:.....	573
Lab Procedures: .....	573
Hint:.....	574
Appendix A: Measuring Gain and Phase:.....	574
<b>Real Analog Chapter 10: Lab Worksheets .....</b>	<b>576</b>
10.6.3: Inverting Voltage Amplifier (45 points total).....	576
<b>Real Analog Chapter 10: Lab Projects .....</b>	<b>578</b>
10.6.4: Non-inverting Voltage Amplifier.....	578
Symbol Key:.....	578
General Discussion:.....	579
Pre-lab:.....	579
Lab Procedures: .....	579
Hint:.....	580
Appendix A: Measuring Gain and Phase.....	580
<b>Real Analog Chapter 10: Lab Worksheets .....</b>	<b>582</b>
10.6.4: Non-inverting Voltage Amplifier (40 points total).....	582
<b>Real Analog Chapter 10: Homework .....</b>	<b>584</b>

<b>Chapter 11: Frequency Response and Filtering .....</b>	<b>587</b>
<b>11 Introduction and Chapter Objectives .....</b>	<b>587</b>
After completing this chapter, you should be able to: .....	588
11.1 Introduction to Steady-state Sinusoidal Analysis .....	588
Example 11.1 .....	589
Example 11.2 .....	590
Example 11.3 .....	590
Section Summary .....	591
11.1 Exercises .....	592
11.2 Signal Spectra and Frequency Response Plots .....	592
11.2.1 Signal Spectra .....	593
Example 11.4 .....	594
Example 11.5 .....	594
11.2.2 Frequency Response Plots .....	595
Example 11.6 .....	595
Example 11.7 .....	596
11.2.3 Checking the Frequency Response .....	597
Example 11.8 .....	597
Example 11.9 .....	598
Example 11.10 .....	598
11.2.4 Graphical Representation of System Response .....	599
Example 11.11 .....	599
Section Summary .....	601
11.2 Exercises .....	602
11.3 Frequency Selective Circuits and Filters .....	602
11.3.2 Ideal Low-pass and High-pass Filters .....	603
11.3.3 First Order Low-pass Filters .....	604
11.3.4 First Order High-pass Filters .....	605
Example 11.12: First Order Low-pass Filter .....	606
Example 11.13: First Order High-pass Filter .....	606

Section Summary .....	607
11.3 Exercises.....	608
11.4 Introduction to Bode Plots .....	608
11.4.1 Properties of Logarithms .....	609
11.4.2 Decibel Scales.....	610
11.4.3 Bode Plots .....	610
11.4.4 Bode Plots for First Order Low-pass Filters .....	611
Example 11.14.....	612
Section Summary .....	613
11.4 Exercises.....	614
<b>Real Analog Chapter 11: Lab Projects .....</b>	<b>615</b>
11.2.1: Signals with Multiple Frequency Components .....	615
Symbol Key:.....	615
General Discussion:.....	615
Pre-lab:.....	616
Lab Procedures: .....	616
Appendix A: Creating a Custom Mathematical Waveform .....	617
Appendix B: Creating a Sinusoidal Sweep.....	618
<b>Real Analog Chapter 11: Lab Worksheets .....</b>	<b>620</b>
11.2.1: Signals with Multiple Frequency Components (45 points total) .....	620
<b>Real Analog Chapter 11: Lab Projects .....</b>	<b>622</b>
11.3.1: Passive RL Filter.....	622
Symbol Key:.....	622
General Discussion:.....	622
Pre-lab:.....	623
Lab Procedures: .....	623
Post-lab Exercises:.....	623
<b>Real Analog Chapter 11: Lab Worksheets .....</b>	<b>625</b>

11.3.1: Passive RL Filter (55 points total).....	625
<b>Real Analog Chapter 11: Lab Projects .....</b>	<b>627</b>
11.3.2: Passive RC Filter .....	627
Symbol Key:.....	627
General Discussion:.....	627
Pre-lab: .....	628
Lab Procedures: .....	628
Post-lab Exercises:.....	628
<b>Real Analog Chapter 11: Lab Worksheets .....</b>	<b>630</b>
11.3.2: Passive RC Filter (50 points total) .....	630
<b>Real Analog Chapter 11: Lab Projects .....</b>	<b>632</b>
11.3.3: Active Low-pass Filter .....	632
Symbol Key:.....	632
General Discussion:.....	632
Pre-lab: .....	633
Lab Procedures: .....	633
<b>Real Analog Chapter 11: Lab Worksheets .....</b>	<b>634</b>
11.3.3: Active Low-pass Filter (40 points total) .....	634
<b>Real Analog Chapter 11: Lab Projects .....</b>	<b>635</b>
11.3.4: Signal Conditioning – Audio Application.....	635
Symbol Key:.....	635
1    Microphone.....	635
Pre-lab: .....	636
Lab Procedures: .....	636
2    Signal Conditioning Circuit.....	637
Pre-lab: .....	637
Lab Procedures: .....	638

3 Overall System Integration .....	638
Pre-lab: .....	638
Lab Procedures: .....	638
<b>Real Analog Chapter 11: Lab Worksheets .....</b>	<b>639</b>
11.3.4: Signal Conditioning – Audio Application (100 points total).....	639
1. Microphone (20 points total).....	639
2. Signal Conditioning Circuit (50 points total).....	639
3. Overall System Integration (30 points total) .....	641
<b>Real Analog Chapter 11: Lab Projects .....</b>	<b>642</b>
11.3.5: Signal Conditioning – Vibration Measurement .....	642
Symbol Key:.....	642
1 Piezoelectric Sensor Vibration .....	642
Pre-lab:.....	643
Lab Procedures: .....	643
2 Signal Conditioning Circuit.....	644
Pre-lab:.....	645
Lab Procedures: .....	645
3 Overall System Integration .....	646
Pre-lab:.....	646
Lab Procedures: .....	646
Appendix A: Piezoelectric sensors .....	646
Appendix B: Measuring Gain and Phase.....	647
<b>Real Analog Chapter 11: Lab Worksheets .....</b>	<b>649</b>
11.3.5: Signal Conditioning – Vibration Measurement (100 points total) .....	649
1. Piezoelectric Sensor Vibration (20 points total) .....	649
2. Signal Conditioning Circuit (50 points total).....	649
3. Overall System Integration (30 points total) .....	651
<b>Real Analog Chapter 11: Lab Projects .....</b>	<b>652</b>

11.4.1: Passive Low-pass Filter .....	652
Symbol Key:.....	652
General Discussion:.....	652
Pre-lab:.....	653
Lab Procedures: .....	653
Post-lab Exercises:.....	653
<b>Real Analog Chapter 11: Lab Worksheets .....</b>	<b>655</b>
11.4.1: Passive Low-pass Filter (45 points total) .....	655
<b>Real Analog Chapter 11: Lab Projects .....</b>	<b>657</b>
11.4.2: Non-inverting Low-pass Filter.....	657
Symbol Key:.....	657
General Discussion:.....	657
Pre-lab:.....	658
Lab Procedures: .....	658
Post-lab Exercises:.....	659
<b>Real Analog Chapter 11: Lab Worksheets .....</b>	<b>660</b>
11.4.2: Non-inverting Low-pass Filter (50 points total).....	660
<b>Real Analog Chapter 11: Lab Projects .....</b>	<b>662</b>
11.4.3: Non-inverting High-pass Filter .....	662
Symbol Key:.....	662
General Discussion:.....	662
Pre-lab:.....	663
Lab Procedures: .....	663
Post-lab Exercises:.....	664
<b>Real Analog Chapter 11: Lab Worksheets .....</b>	<b>665</b>
11.4.3: Non-inverting High-pass Filter (50 points total).....	665
<b>Real Analog Chapter 11: Homework .....</b>	<b>667</b>

<b>Chapter 12: Steady-state Sinusoidal Power .....</b>	<b>670</b>
<b>12 Introduction and Chapter Objectives .....</b>	<b>670</b>
After completing this chapter, you should be able to: .....	670
12.1 Instantaneous Power .....	671
Section Summary: .....	672
12.1 Exercises .....	672
12.2 Average and Reactive Power .....	672
Section Summary: .....	673
12.2 Exercises: .....	674
12.3 RMS Values .....	674
Section Summary: .....	675
12.3 Exercises .....	675
12.4 Apparent Power and Power Factor .....	675
Section Summary .....	676
12.4 Exercises .....	676
12.5 Complex Power .....	676
Example 12.1 .....	677
Section Summary: .....	678
12.5 Exercises .....	679
12.6 Power Factor Correction .....	680
Example 12.2 .....	680
Section Summary: .....	681
<b>Real Analog Chapter 12: Lab Projects .....</b>	<b>682</b>
12.4.1: Apparent Power and Power Factor .....	682
Symbol Key: .....	682
General Discussion: .....	682
Pre-lab: .....	683
Lab Procedures: .....	684
Post-lab Exercises: .....	685

Real Analog Chapter 12: Homework ..... 686

# Chapter 1: Circuit Analysis Fundamentals

## 1 Introduction and Chapter Objectives

In this chapter, we introduce all fundamental concepts associated with circuit analysis. Electrical circuits are constructed in order to direct the flow of electrons to perform a specific task. In other words, in circuit analysis and design, we are concerned with transferring electrical energy in order to accomplish a desired objective. For example, we may wish to use electrical energy to pump water into a reservoir; we can adjust the amount of electrical energy applied to the pump to vary the rate at which water is added to the reservoir. The electrical circuit, then, might be designed to provide the necessary electrical energy to the pump to create the desired water flow rate.

This chapter begins with introduction to the basic parameters which describe the energy in an electrical circuit: *charge*, *voltage*, and *current*. Movement of charge is associated with electrical energy transfer. The energy associated with charge motion is reflected by two parameters: voltage and current. Voltage is indicative of an electrical energy change resulting from moving a charge from one point to another in an electric field. Current indicates the rate at which charge is moving, which is associated with the energy of a magnetic field. We will not be directly concerned with charge, electrical fields, or magnetic fields in this course, we will work almost exclusively with voltages and currents. Since *power* quantifies the rate of energy transfer, we will also introduce power in this chapter.

Electrical circuits are composed of interconnected *components*. In this chapter, we will introduce two basic types of components: *power supplies* and *resistors*. Power supplies are used to provide power to our electrical circuits, and resistors dissipate electrical power by converting it to heat. These two types of components will allow us to introduce and exercise virtually all available circuit analysis techniques. Electrical components are described in terms of the relationships between the voltages and currents at their terminals, these relationships are called the *voltage-current characteristics* of the device. In this chapter, we will introduce voltage-current characteristics for power supplies and resistors. In later chapters, we will introduce additional circuit components, but our circuit analysis approaches will not change – we will simply substitute voltage-current characteristics for these components as appropriate to model future circuits.

Finally, we introduce the two fundamental rules of circuit analysis: *Kirchhoff's Current Law* and *Kirchhoff's Voltage Law*. These rules form the basis of all circuit analysis techniques used throughout this textbook.

Please pay special attention to the *passive sign convention* introduced in this chapter. Voltages and currents have signs – they can be positive or negative – and these signs are crucial to understanding the effect of these parameters on the energy transferred by the circuit. No useful circuit analysis can be performed without following the passive sign convention.

In summary, this chapter introduces virtually all the basic concepts which will be used throughout this textbook. After this chapter, little information specific to electrical circuit analysis remains to be learned – the remainder of the textbook is devoted to developing analysis methods used to increase the efficiency of our circuit analysis and introducing additional circuit components such as capacitors, inductors, and operational amplifiers. The student should be aware, however, that all of our circuit analysis is based on energy transfer among circuit components; this energy transfer is governed by Kirchhoff's Current Law and Kirchhoff's Voltage Law and the circuit components are modeled by their voltage-current relationships.

### After Completing this Chapter, You Should be Able to:

- Define voltage and current in terms of electrical charge

- State common prefixes and the symbols used in scientific notation
- State the passive sign convention from memory
- Determine the power absorbed or generated by a circuit element, based on the current and voltage provided to it
- Write symbols for independent voltage and current sources
- State from memory the function of independent voltage and current sources
- Write symbols for dependent voltage and current sources
- State governing equations for the four types of dependent sources
- State Ohm's Law from memory
- Use Ohm's Law to perform voltage and current calculations for resistive circuit elements
- Identify nodes in an electrical circuit
- Identify loops in an electrical circuit
- State Kirchhoff's current law from memory, both in words and as a mathematical expression
- State Kirchhoff's voltage law from memory, both in words and as a mathematical expression
- Apply Kirchhoff's voltage and current laws to electrical circuits

## 1.1 Basic Circuit Parameters and Sign Conventions

This section introduces the basic engineering parameters for electric circuits: *voltage*, *current*, and *power*. The international system of units is commonly used to describe the units of these parameters; this system as it relates to electrical circuit analysis is briefly discussed in this section.

This section also introduces the *passive sign convention*. It is extremely important when analyzing electrical circuits to use the correct sign convention between the voltage across a circuit element and the current going through the element. Some of the most common errors of beginning students are associated with applying incorrect sign conventions when analyzing circuits.

### 1.1.1 Electrical Charge

Electron flow is fundamental to operation of electric circuits; the concept of *charge* can be used to describe the distribution of electrons in the circuit. Charge can be represented as either positive or negative – generally relative to some reference level. Charge is represented by the variable  $q$  and is measured in *coulombs*, abbreviated as  $C$ . The charge of one electron corresponds to  $-1.6022 \times 10^{-19} C$ . Charge can only exist in integer multiples of the charge of a single electron. Charge, however, is not widely used in electrical circuit analysis; voltage and current are more convenient ways to represent the electric charge in a system.

### 1.1.2 Voltage

*Voltage* is energy per unit charge. Energy is specified relative to some reference level; thus, voltages are more accurately specified as voltage *differences* between two points in a circuit. The voltage difference between two points can be thought of as a difference in potential energy between charges placed at those two points. Units of voltage are volts, abbreviated  $V$ . The voltage difference between two points indicates the energy necessary to move a unit charge from one of the points to the other. Voltage differences can be either positive or negative.

Mathematically, voltage is expressed in differential form as:

$$v = \frac{dw}{dq} \quad \text{Eq. 1.1}$$

Where  $v$  is the voltage difference (in volts),  $w$  is the energy (in joules), and  $q$  is the charge (in coulombs). The differences in equation (1.1) are all defined relative to different spatial positions; thus, the

differentials  $dw$  and  $dq$  are between two different points in space, and the voltage is defined as being between these same two spatial points.

### 1.1.3 Current

Current is the rate at which charge is passing a given point. Current is specified at a particular point in the circuit, and is not relative to a reference. Since current is caused by charge in motion, it can be thought of as indicating *kinetic* energy.

Mathematically, current is represented as:

$$i = \frac{dq}{dt} \quad \text{Eq. 1.2}$$

Where  $i$  is the current in amperes,  $q$  is the charge in coulombs, and  $t$  is the time in seconds. Thus, current is the time rate of change of charge and units of charge are coulombs per second, or *amperes* (abbreviated as A).

### 1.1.4 Power

An electrical system is often used to drive a non-electrical system (in an electric stove burner, for example, electric energy is converted to heat). Interactions between electrical and non-electrical systems are generally described in terms of *power*. Electrical power associated with a particular circuit element is the product of the current passing through the element and the voltage difference across the element. This is often written as:

$$p(t) = v(t) \cdot i(t) \quad \text{Eq. 1.3}$$

Where  $p(t)$  is the *instantaneous* power at time  $t$ ,  $v(t)$  is the voltage difference at time  $t$ , and  $i(t)$  is the current at time  $t$ . Power can either be absorbed by a circuit element or generated by a circuit element; the determination as to whether the element is absorbing or generating power can be made by the relative signs of the values of voltage and current. These sign conventions are an important issue, and will be addressed separately in the next chapter. Units of power are watts, abbreviated  $W$ .

### 1.1.5 International System of Units and Prefixes

We will use the international system of units (SI). The scales of parameters that are of interest to engineers can vary over many orders of magnitudes. For example, voltages experienced during lightning strikes can be on the order of  $10^7V$ , while voltages measured from an electroencephalograph (EEG) can be on the order of  $10^{-4}V$ . For this reason, numbers represented in SI units are often associated with a prefix, which helps account for the order-of-magnitude variations in numbers. Table 1 below provides a list of common prefixes and the symbols used to represent them.

Multiple	Prefix	Symbol
$10^9$	giga	G
$10^6$	mega	M
$10^3$	kilo	k
$10^{-3}$	milli	m
$10^{-6}$	micro	$\mu$
$10^{-9}$	nano	n
$10^{-12}$	pico	p

Table 1.1. SI prefixes.

## 1.1.6 Passive Sign Convention

A general two-terminal electrical circuit element is shown in Fig. 1.1. In general, there will be some current,  $i$ , flowing through the element and some voltage difference,  $v$ , across its terminals. Note that we are currently representing both voltage and current as constants, but none of the assertions made in this chapter change if they are functions of time.

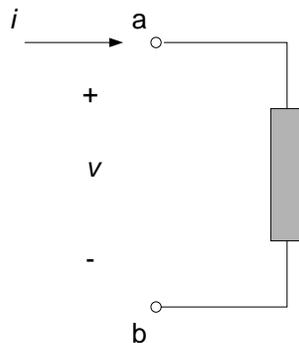
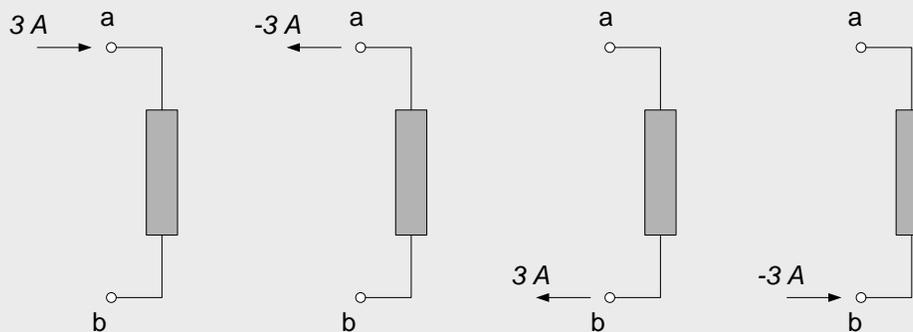


Figure 1.1. General circuit element and passive sign convention.

The assumed direction of the current,  $i$ , passing through the element is shown by the arrow on Fig. 1.1. In Fig. 1.1,  $i$  is assumed to be positive if it is going into node a. A negative value of  $i$  simply indicates a change in direction of the current – if  $i$  is negative, the current is going into node b (or, equivalently, out of node a). We will assume that our circuit elements do not accumulate charge, so any current entering node a must leave node b.

### Example 1.1:

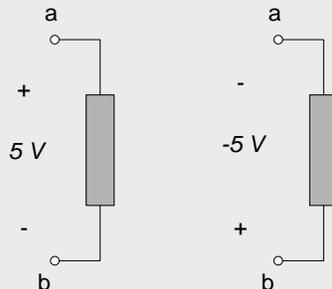
Three amperes (3 A) of current is passing through a circuit element connecting nodes a and b. The current is flowing from node a to node b. The physical situation can be represented schematically by any of the figures shown below – all four figures represent the same current flow and direction.



The assumed polarity of the voltage difference  $v$  across the element is shown by the + and - signs on Fig. 1.1. The polarity shown in Fig. 1.1 indicates that a positive value for  $v$  indicates that the voltage at the terminal marked with a + sign is higher than the voltage at the terminal marked with a - sign (that is, the voltage at node a is higher than the voltage at node b). A negative value for  $v$  simply reverses this polarity (negative voltage means that the voltage at node b is higher than the voltage at node a).

### Example 1.2:

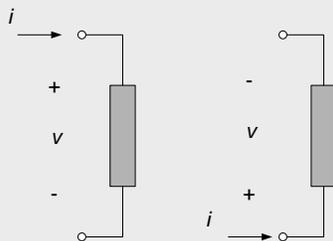
A 5 volt (5 V) voltage potential difference is applied across a circuit element connecting nodes a and b. The voltage at node a is positive relative to the voltage at node b. The physical situation can be represented schematically by either of the figures shown below – both figures represent the same voltage potential difference.



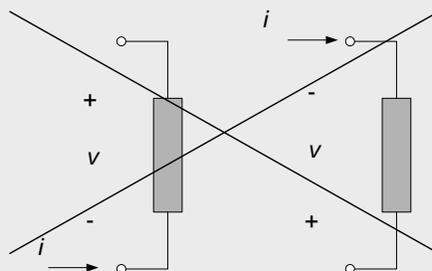
The assumed voltage polarity and current direction are not individually significant - the assumed direction of voltage polarity relative to current direction is important. To satisfy our sign convention, we will assume that positive current enters the node at which the positive voltage polarity is defined. This sign convention is called the *passive sign convention*. In the passive sign convention, the relative assumed sign convention between voltage and current is as shown in Fig. 1.1.

### Example 1.3:

The passive sign convention is satisfied for either of the two voltage-current definitions shown below - the current is assumed to enter the positive voltage node.



The passive sign convention is not satisfied for either of the two voltage current definitions shown below – the current is assumed to enter the negative voltage node.



**Note:** Many students attempt to choose current directions and voltage polarities so that their calculations result in positive values for voltages and currents. In general, this is a waste of time - it is best to arbitrarily assume either a voltage polarity of current direction for each circuit element.

Choice of a positive direction for current dictates the choice of positive voltage polarity, per Fig. 1.1. Choice of a positive voltage polarity dictates the choice of positive current direction, per Fig. 1.1.

Analysis of the circuit is performed using the above assumed signs for voltage and current. The sign of the results indicates whether the assumed choice of voltage polarity and current direction was correct. A positive magnitude of a calculated voltage indicates that the assumed sign convention is correct; a negative magnitude indicates that the actual voltage polarity is opposite to the assumed polarity. Likewise, a positive magnitude of a calculated current indicates that the assumed current direction is correct; a negative magnitude indicates that the current direction is opposite to that assumed.

### 1.1.7 Voltage Subscript and Sign Conventions

The assumed sign convention for voltage potentials is sometimes expressed by using subscripts. The first subscript denotes the node at which the positive voltage polarity is assumed and the second subscript is the negative voltage polarity. For example,  $v_{ab}$  denotes the voltage difference between nodes  $a$  and  $b$ , with node  $a$  assumed as having positive voltage relative to node  $b$ . Switching the order of the subscripts changes the assumed polarity of the voltage difference and thus the sign of the voltage, so  $v_{ab} = -v_{ba}$ . Since our passive sign convention dictates the direction of current relative to voltage polarity, a circuit element whose voltage difference is denoted as  $v_{ab}$  will have positive current entering node  $a$ .

### 1.1.8 Reference Voltages and Ground

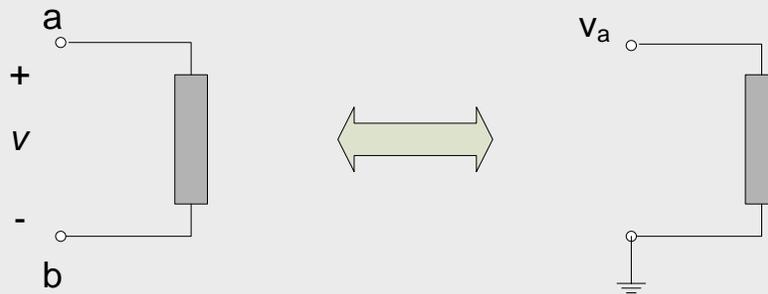
For convenience, voltage differences are often not explicitly stated as being differences between two potential levels - a node will simply be referred to as having some "voltage". This voltage must still be interpreted as a voltage difference, however. The difference in this case is assumed to be relative to some reference voltage, with the reference generally assumed to be 0V. The reference voltage is often referred to as *ground*. The symbol, displayed below, is used to denote the ground or reference voltage from which all other voltages are measured. When this convention is used, voltages at a node are often identified with a single subscript. For example,  $v_a$  would be the voltage at node  $a$ , relative to ground. It is assumed that positive voltages are positive relative to ground and negative voltages are negative relative to ground.



#### Example 1.4:

The two figures below show identical ways of specifying the voltage across a circuit element. In the circuit to the left, the voltage  $v$  is the voltage potential between nodes  $a$  and  $b$ , with the voltage at node  $a$  being assumed positive relative to voltage at node  $b$ . This can be equivalently specified as  $v_{ab}$ . In the figure to the right below, node  $b$  has been specified as our ground with the use of the ground symbol. In this figure, the voltage at node  $a$  can be specified simply as  $v_a$ , with the polarity being assumed positive relative to ground which is implied to be 0V. Thus, for the figures below:

$$v = v_{ab} = v_a \quad \text{Eq. 1.3}$$



### 1.1.9 Power and Sign Conventions

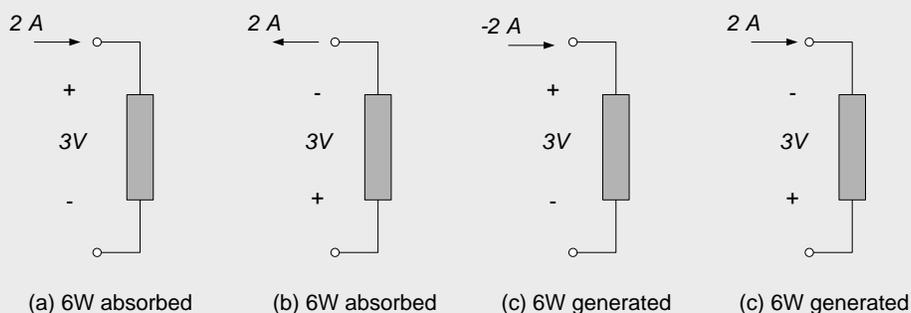
The sign of the voltage across an element relative to the sign of the current through the element governs the sign of the power. Equation (1.3) above defines power as the product of the voltage times current:

$$P = vi \quad \text{Eq. 1.4}$$

The power is *positive* if the signs of voltage and current agree with the passive sign convention - that is, if a positive current enters the positive voltage polarity node. If the power is positive, the element is said to be absorbing power. The power is *negative* if the signs of voltage and current disagree with the passive sign convention - that is, if positive current enters the negative voltage polarity node. If the power is negative, the element is said to be *generating* power.

#### Example 1.5:

In Fig. (a) below, the element agrees with the passive sign convention since a positive current is entering the positive voltage node. Thus, the element of Fig. (a) is absorbing energy. In Fig. (b), the element is absorbing power - positive current is leaving the negative voltage node, which implies that positive current enters the positive voltage node. The element of Fig. (c) generates power; negative current enters the positive voltage node, which disagrees with the passive sign convention. Fig. (d) also illustrates an element which is generating power, since positive current is entering a negative voltage node.



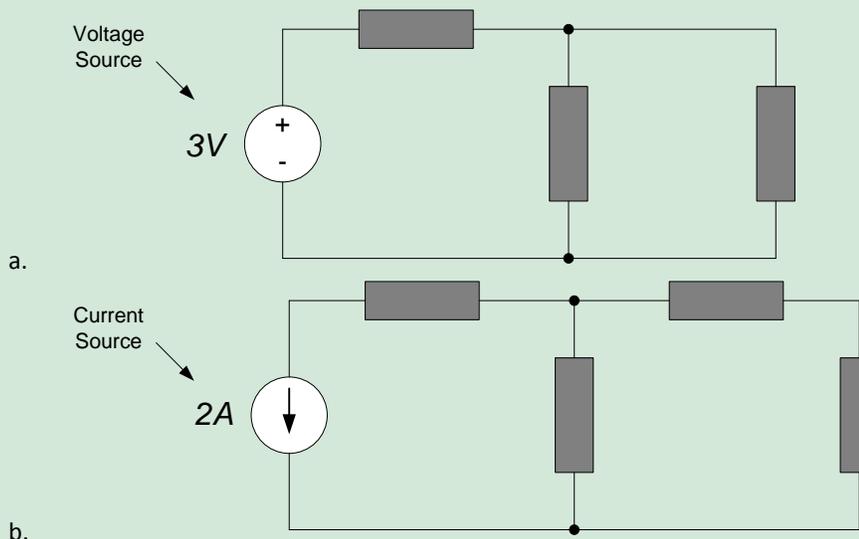
### Section Summary

- In this text, we will be primarily concerned with the movement of electrical charge. Electrical charge motion is represented by voltage and current. Voltage indicates the energy change associated with the movement of a charge from one location to another, while current is indicative of the rate of current motion past a particular point.
  - Voltage is an energy difference between two physically separated points. The polarity of a voltage is used to indicate which point is to be assumed to be at the higher energy level. The positive terminal (+) is assumed to be at a higher voltage than the negative terminal (-). A

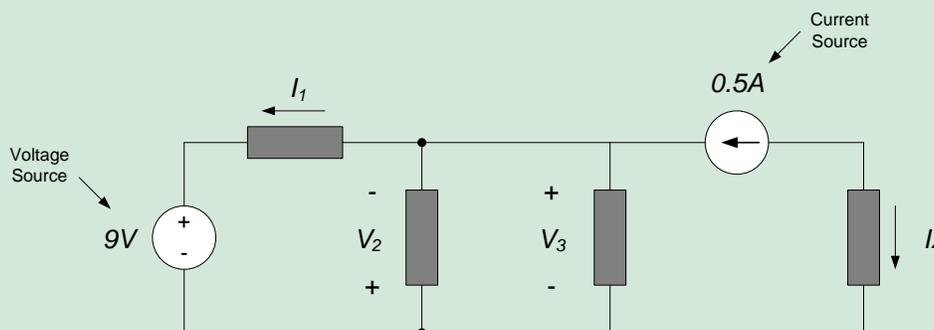
- negative voltage value simply indicates that the actual voltage polarity is opposite to the assumed polarity.
- The sign of the current indicates the assumed direction of charge motion past a point. A change in the sign of the current value indicates that the current direction is opposite to the assumed direction.
  - The assumed polarity of the voltage across a passive circuit element must be consistent with the assumed current direction through the element. The assumed positive direction for current must be such that positive current enters the positive voltage terminal of the element. Since this sign convention is applied only to passive elements, it is known as the *passive sign convention*.
    - The assumed current direction or the assumed voltage polarity can be chosen arbitrarily, but once one parameter is chosen, the other must be chosen to agree with the passive sign convention.
  - The power absorbed or generated by an electrical circuit component is the product of the voltage difference across the element and the current through the element:  $p=iv$ . The relative sign of the voltage and current are set according to the passive sign convention. Positive power implies that the voltage and current are consistent with the passive sign convention (the element absorbs or dissipates energy) while negative power indicates that the relative signs between voltage and current are opposite to the passive sign convention (the element generates or supplies energy to the circuit).

### 1.1 Exercises

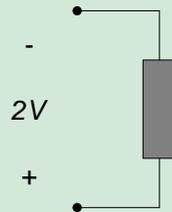
1. Assign reference voltage and current directions to the circuit elements represented by the shaded boxes in the circuits below.



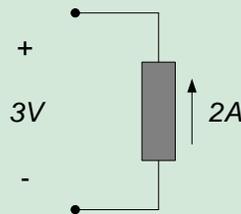
2. Either the reference voltage polarity or the reference current direction is provided for the circuit elements below. Provide the appropriate sign convention for the missing parameters.



3. Determine the magnitude and direction of the current in the circuit element below if the element absorbs 10W.



4. Determine the power absorbed or supplied by the circuit element below. State whether the power is absorbed or supplied.



## 1.2 Power Sources

Circuit elements are commonly categorized as either *passive* or *active*. A circuit element is passive if the total amount of energy it delivers to the rest of the circuit (over all time) is non-positive (passive elements can temporarily deliver energy to a circuit, but only if the energy was previously stored in the passive element by the circuit). An active circuit element has the ability to create and provide power to a circuit from mechanisms external to the circuit. Examples of active circuit elements are batteries (which create electrical energy from chemical processes) and generators (which create electrical energy from mechanical processes, such as spinning a turbine).

In this section we consider some very important active circuit elements: voltage and current sources. We will discuss two basic types of sources: *independent sources* and *dependent sources*. Independent sources provide a specified voltage or current, regardless of what is happening elsewhere in the circuit to which they are connected - batteries and generators are generally considered to be independent sources. Dependent sources provide a voltage or current based on a voltage or current elsewhere in the circuit (the source voltage or current is dependent upon some other voltage or current). Dependent sources are often used in the mathematical modeling of common devices such as metal-oxide semiconductor field-effect transistors (MOSFETs) and bipolar junction transistors (BJTs).

### 1.2.1 Independent Voltage Sources

An independent voltage source maintains a specified voltage across its terminals. The symbol used to indicate a voltage source delivering a voltage  $v_s(t)$  is shown in Fig. 1.2. As indicated in Fig. 1.2, the voltage supplied by the source can be time-varying or constant (a constant voltage is a special case of a time-varying voltage). An alternate symbol that is often used to denote a constant voltage source is shown in Fig. 1.3; we, however, will generally use the symbol of Fig. 1.2 for both time-varying and constant voltages.

Note that the sign of the voltage being applied by the source is provided on the source symbol - there is no need to assume a voltage polarity for voltage sources. The current direction, however, is unknown and must be determined (if necessary) from an analysis of the overall circuit.

*Ideal* voltage sources provide a specified voltage regardless of the current flowing through the device. Ideal sources can, obviously, provide infinite power; all real sources will provide only limited power to the circuit. We will discuss approaches for modeling non-ideal sources in later chapters.

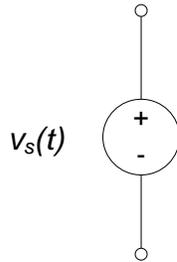


Figure 1.2. Independent voltage source.

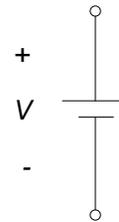


Figure 1.3. Constant voltage source.

## 1.2.2 Independent Current Sources

An independent current source maintains a specified current. This current is maintained regardless of the voltage differences across the terminals. The symbol used to indicate a current source delivering a current  $i_s(t)$  is shown in Fig. 1.4. The current supplied by the source can be time-varying or constant.

Note that the sign of the current being applied by the source is provided on the source symbol - there is no need to assume a current direction. The voltage polarity, however, is unknown and must be determined (if necessary) from an analysis of the overall circuit.

*Ideal* current sources provide a specified current regardless of the voltage difference across the device. Ideal current sources can, like ideal voltage sources, provide infinite power; all real sources will provide only limited power to the circuit. We will discuss approaches for modeling non-ideal current sources in later chapters.

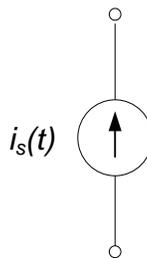


Figure 1.4. Independent current source.

## 1.2.3 Dependent Sources

Dependent sources can be either voltage or current sources; Fig. 1.5(a) shows the symbol for a dependent voltage source and Fig. 1.5(b) shows the symbol for a dependent current source. Since each type of source can be controlled by either a voltage or current, there are four types of dependent current sources:

- Voltage-controlled voltage source (VCVS)
- Current-controlled voltage source (CCVS)

- Voltage-controlled current source (VCCS)
- Current-controlled current source (CCCS)



Figure 1.5. Symbols for dependent sources.

Figure 1.6 illustrates the voltage-controlled dependent sources, and Fig. 1.7 illustrates the current-controlled dependent sources. In all cases, some electrical circuit exists which has some voltage and current combination at its terminals. Either the voltage or current at these terminals is used to set the voltage or current of the dependent source. The parameters  $\mu$  and  $\beta$  in Figs. 1.6 and 1.7 are dimensionless constants.  $\mu$  is the *voltage gain* of a VCVS and  $\beta$  is the *current gain* of a CCCS. The parameter  $r$  is the voltage-to-current ratio of a CCVS and has units of volts/ampere, or *ohms*. The parameter  $g$  is the current-to-voltage ratio of a VCCS and has units of amperes/volt, or *siemens*. The units of ohms and siemens will be discussed in more depth in section 1.3.

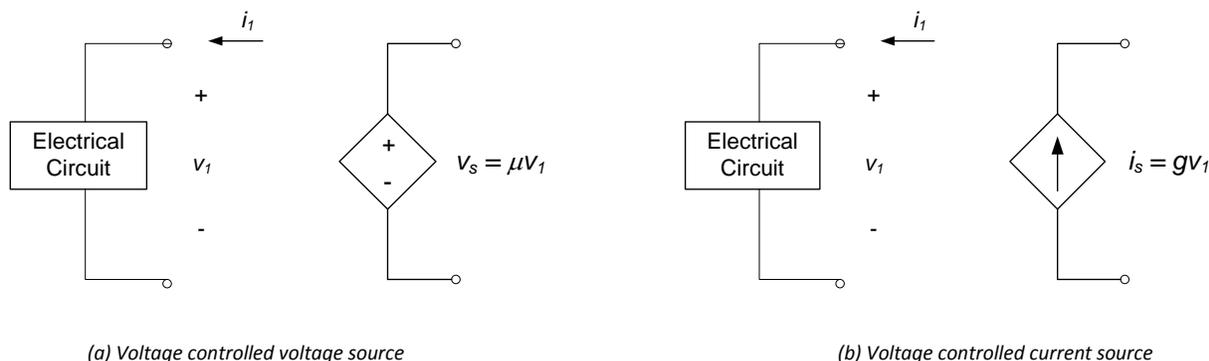


Figure 1.6. Voltage-controlled dependent sources

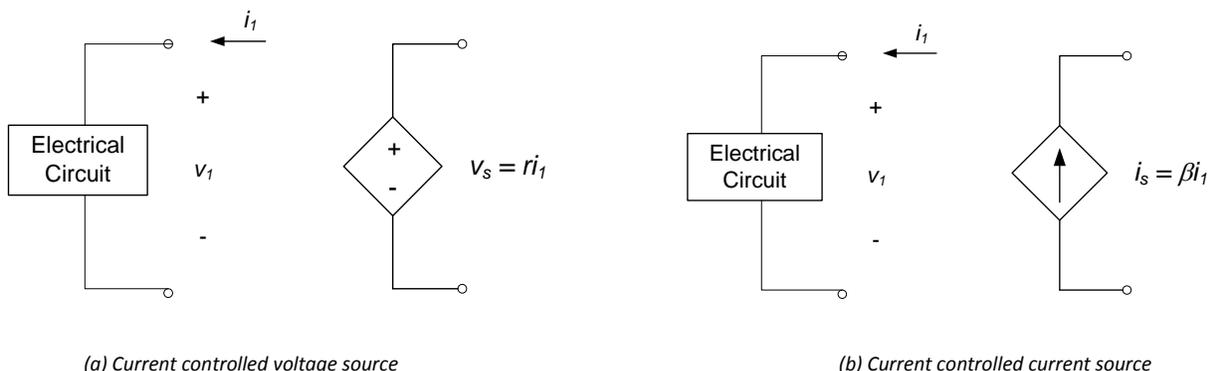


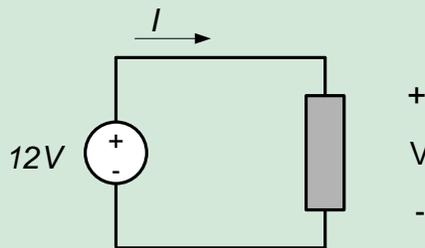
Figure 1.7. Current-controlled dependent sources

## Section Summary

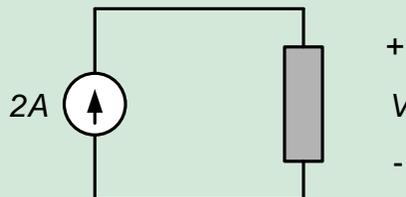
- Circuit elements can be either active or passive. Active elements provide electrical energy from a circuit from sources outside the circuit; active elements can be considered to create energy (from the standpoint of the circuit, anyway). Passive elements will be discussed in section 1.3, when we introduce resistors. Active circuit elements introduced in this section are ideal independent and dependent voltage and current sources.
  - Ideal independent sources presented in this section are voltage and current sources. Independent voltage sources deliver the specified voltage, regardless of the current demanded of them. Independent current sources provide the specified current, regardless of the voltage levels required to provide this current. Devices such as batteries are often modeled as independent sources.
  - Dependent sources provide a voltage or current which is controlled by a voltage or current elsewhere in the circuit. Devices such as operational amplifiers and transistors are often modeled as dependent sources. We will revisit the subject of dependent sources in chapter 5 of this text, when we discuss operational amplifier circuits.

### 1.2 Exercises:

1. The ideal voltage source shown in the circuit below delivers 12V to the circuit element shown. What is the current  $I$  through the circuit element?



2. The ideal current source shown in the circuit below delivers 2A to the circuit element shown. What is the voltage difference  $V$  across the circuit element?



## 1.3 Resistors and Ohm's Law

Resistance is a property of all materials - this property characterizes the loss of energy associated with passing an electrical current through some conductive element. Resistors are circuit elements whose characteristics are dominated by this energy loss. Since energy is always lost when current is passed through an electrical circuit element, all electrical elements exhibit resistive properties which are characteristic of resistors. Resistors are probably the simplest and most commonly used circuit elements.

All materials impede the flow of current through them to some extent. Essentially, this corresponds to a statement that energy is always lost when transferring charge from one point in a circuit to another - this energy loss is generally due to heat generation and dissipation. The amount of energy required to transfer current in a particular

element is characterized by the *resistance* of the element. When modeling a circuit, this resistance is represented by *resistors*. The circuit symbol for a resistor is shown in Fig. 1.8. The value of resistance is labeled in Fig. 1.8 as  $R$ .  $i$  in Fig. 1.8 is the current flowing through the resistor and  $v$  is the voltage drop across the resistor, caused by the energy dissipation induced by the resistor. The units of resistance are ohms, abbreviated  $\Omega$ .

The relationship between voltage and current for a resistor is given by *Ohm's Law*:

$$v(t) = Ri(t) \quad \text{Eq. 1.5}$$

Where voltage and current are explicitly denoted as functions of time. Note that in Fig. 1.8, the current is flowing from a higher voltage potential to a lower potential, as indicated by the polarity (+ and -) of the voltage and the arrow indicating direction of the current flow. The relative polarity between voltage and current for a resistor must be as shown in Fig. 1.8; the current enters the node at which the voltage potential is highest. Values of resistance,  $R$  are always positive, and resistors always absorb power.

**Note:** The voltage-current relationship for resistors always agrees with the passive sign convention. Resistors always absorb power.

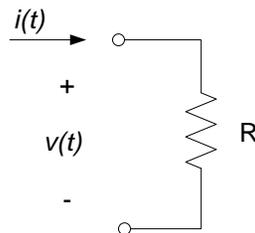


Figure 1.8. Circuit symbol for resistor.

Figure 1.9 shows a graph of  $v$  vs.  $i$  according to equation (1.5); the resulting plot is a straight line with slope  $R$ . Equation (1.5) thus describes the voltage-current relationship for a *linear* resistor. Linear resistors do not exist in reality - all resistors are *nonlinear*, to some extent. That is, the voltage-current relationship is not exactly a straight line for all values of current (for example, all electrical devices will fail if enough current is passed through them). Fig. 1.10 shows a typical nonlinear voltage-current relationship; however, many nonlinear resistors exhibit an approximately linear voltage-current characteristic over some range of voltages and currents; this is also illustrated in Fig. 1.10. We will assume for now that any resistor we use is operating within a range of voltages and currents over which its voltage-current characteristic is linear and can be approximated by equation (1.5).

**Note:** For the most part, we will consider only linear resistors in this text. These resistors obey the linear voltage-current relationship shown in equation (1.5). All real resistors are nonlinear to some extent, but can often be assumed to operate as linear resistors over some range of voltages and currents.

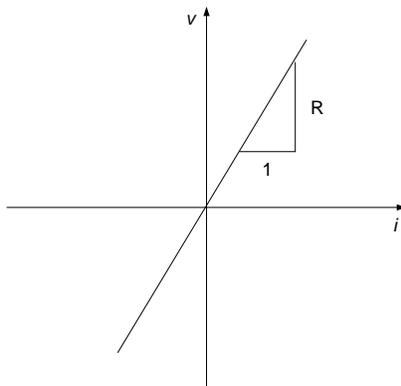


Figure 1.9. Linear resistor voltage vs. current characteristic.

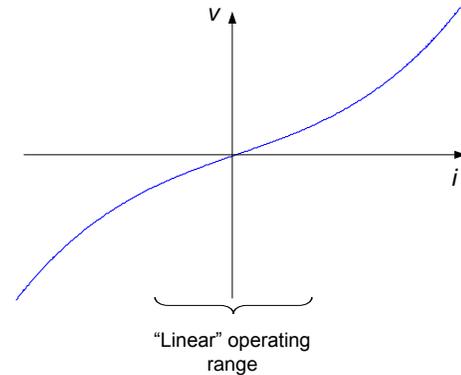


Figure 1.10. Typical nonlinear resistor voltage vs. current characteristic.

### 1.3.1 Conductance

*Conductance* is an important quantity in circuit design and analysis. Conductance is simply the reciprocal of resistance, defined as:

$$G = \frac{1}{R} \quad \text{Eq. 1.6}$$

The unit for conductance is siemens, abbreviated S. Ohm's law, written in terms of conductance, is:

$$i(t) = Gv(t) \quad \text{Eq. 1.7}$$

Some circuit analyses can be performed more easily and interpreted more readily if the elements' resistance is characterized in terms of conductance.

**Note:** In section 1.2, we characterized a current-controlled voltage source in terms of a parameter with units of ohms, since it had units of volts/amp. We characterized a voltage-controlled current source in terms of a parameter with units of siemens, since it had units of amps/volts.

### 1.3.2 Power Dissipation

Instantaneous power was defined by equation (1.3) in section 1.1 as:

$$P(t) = v(t) \cdot i(t)$$

For the special case of a resistor, we can re-write this (by substituting equation (1.5) into the above) as:

$$P(t) = Ri^2(t) = \frac{v^2(t)}{R} \quad \text{Eq. 1.8}$$

Likewise, we can write the power dissipation in terms of the conductance of a resistor as:

$$P(t) = \frac{i^2(t)}{G} = Gv^2(t) \quad \text{Eq. 1.9}$$

**Note:** We can write the power dissipation from a resistor in terms of the resistance or conductance of the resistor and either the current through the resistor or the voltage drop across the resistor.

### 1.3.3 Practical Resistors

All materials have some resistance, so all electrical components have non-zero resistance. However, circuit design often relies on implementing a specific, desired resistance at certain locations in a circuit; resistors are often placed in the circuit at these points to provide the necessary resistance. Resistors can be purchased in certain standard values. Resistors are manufactured in a variety of ways, though most commonly available commercial resistors are carbon composition or wire-wound. Resistors can have either a fixed or variable resistance.

*Fixed* resistors provide a single specified resistance value and have two terminals, as shown in Fig. 1.5 above. *Variable* resistors or *potentiometers* (commonly called “pots”) have three terminals, two are “fixed” and one is “movable”. The symbol for a variable resistor is shown in Fig. 1.11. The resistance between two of the terminals -  $R_{23}$  in Fig. 1.11 - of a variable resistor can be set as some fraction of the overall resistance of the device -  $R_{13}$  in Fig. 1.11. The ratio of  $R_{23}$  to  $R_{13}$  is generally set by a dial or screw on the side of the device.

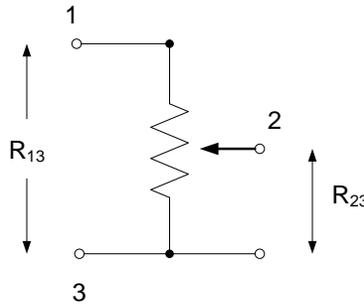
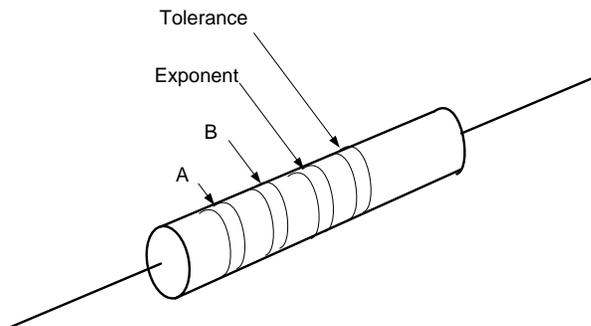


Figure 1.11. Schematic for variable resistor.

Resistors, which are physically large enough, will generally have their resistance value printed directly on them. Smaller resistors generally will use a color code to identify their resistance value. The color coding scheme is provided in Fig. 1.12. The resistance values indicated on the resistor will provide a *nominal* resistance value for the component; the actual resistance value for the component will vary from this by some amount. The expected tolerance between the allowable actual resistance values and the nominal resistance is also provided on the resistor, either printed directly on the resistor or provided as an additional color band. The color-coding scheme for resistor tolerances is also provided in Fig. 1.12.



Fourth Band	Tolerance
None	±20%
Silver	±10%
Gold	±5%

Color	Digit
Black	0
Brown	1
Red	2
Orange	3
Yellow	4
Green	5
Blue	6
Violet	7
Grey	8
White	9

$$\text{Resistance} = (10 \times A + B) \times 10^{\text{Exponent}} \pm \text{Tolerance}$$

Figure 1.12. Resistor color code.

**Example 1.6:**

A resistor has the following color bands below. Determine the resistance value and tolerance.

- First band (A): Red
- Second band (B): Black
- Exponent: Orange
- Tolerance: Gold

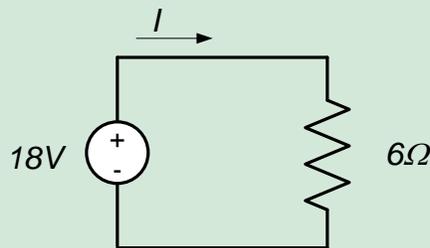
$$\text{Resistance} = (20+0) \times 10^3 \pm 5\% = 20 \text{ k}\Omega \pm 1 \text{ k}\Omega$$

**Section Summary**

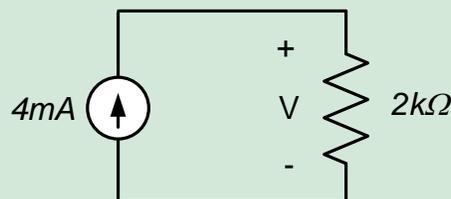
- The relationship between voltage and current for a resistor is Ohm's Law:  $v=iR$ . Since a resistor only dissipates energy, the voltage and current for a resistor must always agree with the passive sign convention.
- As noted in section 1.2, circuit elements can be either active or passive. Resistors are passive circuit elements. Passive elements can store or dissipate electrical energy provided to them by the circuit; they can subsequently return energy to the circuit which they have previously stored, but they cannot create energy. Resistors cannot store electrical energy; they can only dissipate energy by converting it to heat.

**1.3 Exercises**

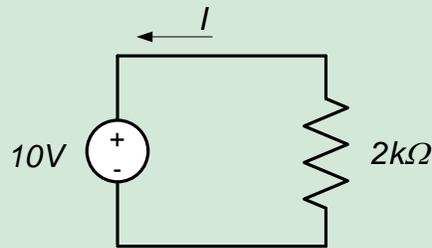
1. The ideal voltage source shown in the circuit below delivers 18V to the resistor shown. What is the current  $I$  through the resistor?



2. The ideal current source shown in the circuit below delivers 4mA to the resistor shown. What is the voltage difference  $V$  across the resistor?



3. The ideal voltage source shown in the circuit below delivers 10V to the resistor shown. What is the current  $I$  in the direction shown?



## 1.4 Kirchhoff's Laws

This section provides some basic definitions and background information for two important circuit analysis tools: Kirchhoff's Current Law and Kirchhoff's Voltage Law. These laws, together with the voltage-current characteristics of the circuit elements in the system, provide us with the ability to perform a systematic analysis of any electrical network.

We will use a *lumped-parameters* approach to a circuit analysis. This means that the circuit will consist of a number of discrete elements, connected by *perfect conductors*. Perfect conductors have no resistance, thus there is no voltage drop across a perfect conductor regardless of how much current flows through it. There is no energy stored or dissipated by a perfect conductor. All energy dissipation and energy storage is thus assumed to reside (or is *lumped*) in the circuit elements connected by the perfect conductors.

The lumped parameters approach toward modeling circuits is appropriate if the voltage and currents in the circuit change slowly relative to the rate with which information can be transmitted through the circuit. Since information propagates in an electrical circuit at a rate comparable to the speed of light and circuit dimensions are relatively small, this modeling approach is often appropriate.

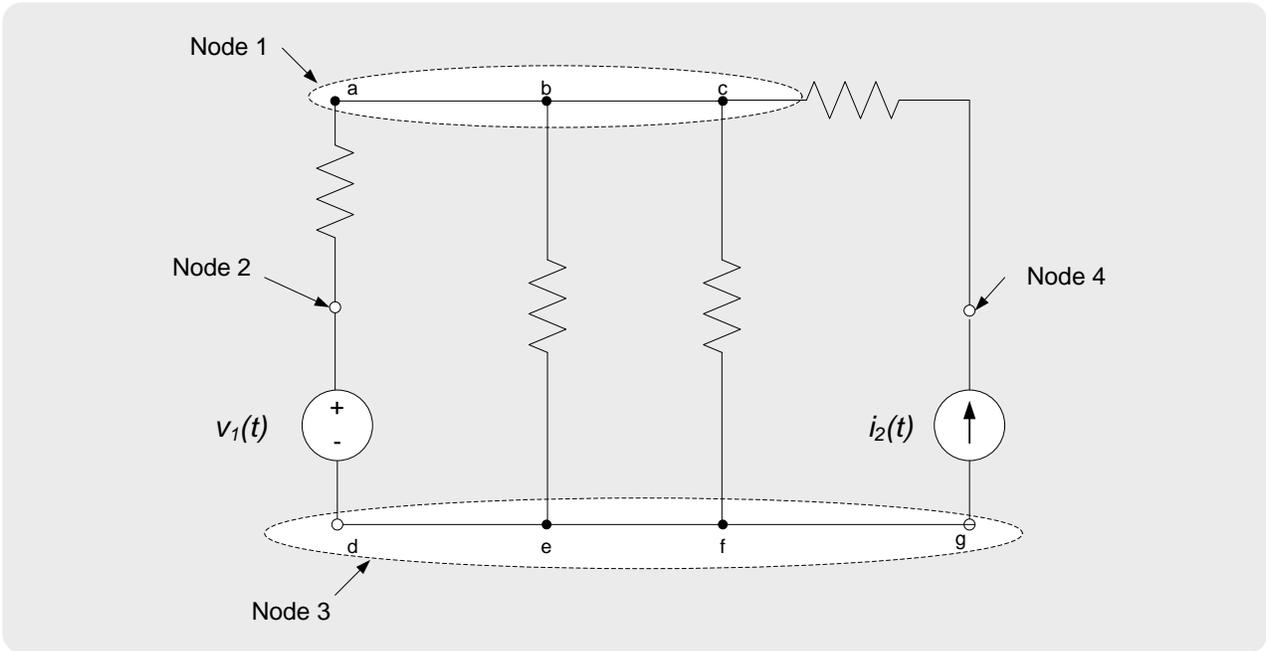
An alternate approach to circuit analysis is a *distributed-parameters* approach. This approach is considerably more mathematically complicated than the lumped parameters approach, but is necessary when dimensions become very large (as in cross-country power transmission) or when signals are varying extremely rapidly (such as the rate of bit transmission in modern computers).

### 1.4.1 Nodes

Identification of circuit nodes will be extremely important to the application of Kirchhoff's Laws. A *node* is a point of connection of two or more circuit elements. A node has a single, unique voltage. Since there is no voltage drop across a perfect conductor, any points in a circuit which are connected by perfect conductors will be at the same voltage and will thus be part of the same node.

#### Example 1.7

The circuit below has four nodes, as shown. A common error for beginning students is to identify points a, b, and c as being separate nodes, since they appear as separate points on the circuit diagram. However, these points are connected by perfect connectors (no circuit elements are between points a, b, and c) and thus the points are at the same voltage and are considered electrically to be at the same point. Likewise, points d, e, f, and g are at the same voltage potential and are considered to be the same node. Node 2 interconnects two circuit elements (a resistor and a source) and must be considered as a separate node. Likewise, node 4 interconnects two circuit elements and qualifies as a node.

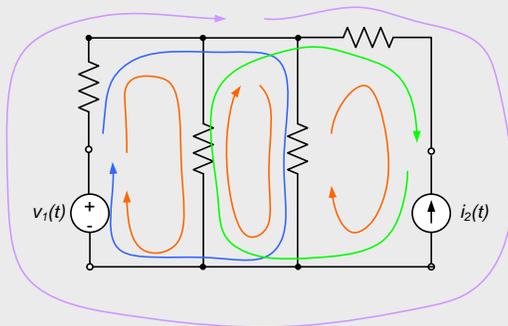


### 1.4.2 Loops

A *loop* is any closed path through the circuit which encounters no node more than once.

#### Example 1.8

There are six possible ways to loop through the circuit of the previous example. These loops are shown below.



### 1.4.3 Kirchhoff's Current Law

Kirchhoff's Current Law is one of the two principle approaches we will use for generating the governing equations for an electrical circuit. Kirchhoff's Current Law is based upon our assumption that charges cannot accumulate at a node.

Kirchhoff's Current Law (commonly abbreviated in these chapters as KCL) states:

***The algebraic sum of all currents entering (or leaving) a node is zero.***

A common alternate statement for KCL is:

***The sum of the currents entering any node equals the sum of the currents leaving the node.***

A general mathematical statement for Kirchhoff's Current Law is:

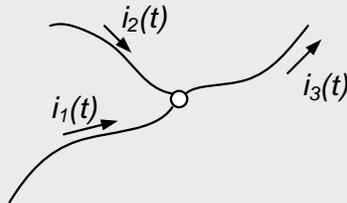
$$\sum_{k=1}^N i_k(t) = 0$$

Eq. 1.10

**Note:** Current directions (entering or leaving the node) are based on assumed directions of currents in the circuit. As long as the assumed directions of the currents are consistent from node to node, the final result of the analysis will reflect the actual current directions in the circuit.

### Example 1.9:

In the figure below, the assumed directions of  $i_1(t)$ ,  $i_2(t)$  and  $i_3(t)$  are as shown.



If we (arbitrarily) choose a sign convention such that currents entering the node are positive then currents leaving the node are negative and KCL applied at this node results in:

$$i_1(t) + i_2(t) - i_3(t) = 0$$

If, on the other hand, we choose a sign convention that currents entering the node are negative, then currents leaving the node are positive and KCL applied at this node results in:

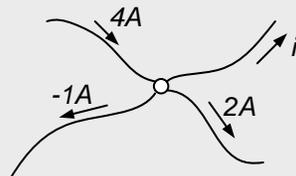
$$-i_1(t) - i_2(t) + i_3(t) = 0$$

These two equations are the same; the second equation is simply the negative of the first equation. Both of the above equations are equivalent to the statement:

$$i_1(t) + i_2(t) = i_3(t)$$

### Example 1.10:

Use KCL to determine the value of the current  $i$  in the figure below.

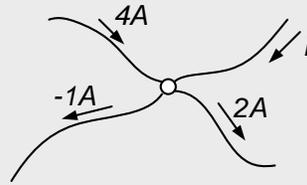


Summing the currents entering the node results in:

$$4A - (-1A) - 2A - i = 0 \Rightarrow i = 4A + 1A - 2A = 3A$$

and  $i = 3A$ , leaving the node.

In the figure below, we have reversed our assumed direction of  $i$  in the above circuit:



Now, if we sum currents entering the node:

$$4A - (-1A) - 2A + i = 0 \Rightarrow i = -4A - 1A + 2A = -3A$$

So now  $i = -3A$ , entering the node. The negative sign corresponds to a change in direction, so we can interpret this result to a  $+3A$  current leaving the node, which is consistent with our previous result. Thus, the assumed current direction has not affected our results.

We can generalize Kirchhoff's Current Law to include any enclosed portion of a circuit. To illustrate this concept, consider the portion of a larger circuit enclosed by a surface as shown in Fig. 1.13 below. Since none of the circuit elements within the surface store charge, the total charge which can be stored within any enclosed surface is zero. Thus, the net charge entering an enclosed surface must be zero. This leads to a generalization of our previous statement of KCL:

***The algebraic sum of all currents entering (or leaving) any enclosed surface is zero.***

Applying this statement to the circuit of Fig. 1 results in:

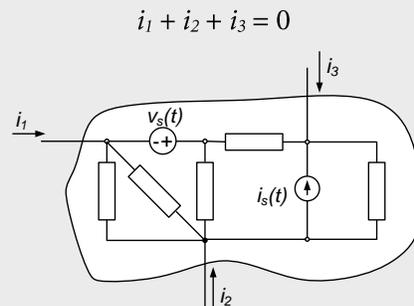


Figure 1.13 KCL applied to closed surface.

## 1.4.4 Kirchhoff's Voltage Law

Kirchhoff's Voltage Law is the second of two principle approaches we will use for generating the governing equations for an electrical circuit. Kirchhoff's Voltage Law is based upon the observation that the voltage at a node is unique.

Kirchhoff's Voltage Law (commonly abbreviated in these chapters as KVL) states:

***The algebraic sum of all voltage differences around any closed loop is zero.***

An alternate statement of this law is:

***The sum of the voltage rises around a closed loop must equal the sum of the voltage drops around the loop.***

A general mathematical statement for Kirchhoff's Voltage Law is:

$$\sum_{k=1}^N v_k(t) = 0$$

Eq. 1.11

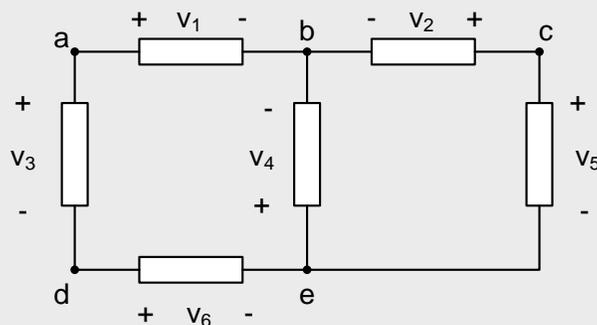
Where  $V_k$  is the  $k^{\text{th}}$  voltage difference in the loop and  $N$  is the total number of voltage differences in the loop.

**Note:** Voltage polarities are based on assumed polarities of the voltage differences in the loop. As long as the assumed directions of the voltages are consistent from loop to loop, the final result of the analysis will reflect the actual voltage polarities in the circuit.

### Example 1.11:

In the figure below, the assumed (or previously known) polarities of the voltages  $v_1$ ,  $v_2$ ,  $v_3$ ,  $v_4$ ,  $v_5$ , and  $v_6$  are as shown. There are three possible loops in the circuit: a-b-e-d-a, a-b-c-e-d-a, and b-c-e-b. We will apply KVL to each of these loops.

Our sign convention for applying signs to the voltage polarities in our KVL equations will be as follows: when traversing the loop, if the positive terminal of a voltage difference is encountered before the negative terminal, the voltage difference will be interpreted as positive in the KVL equation. If the negative terminal is encountered first, the voltage difference will be interpreted as negative in the KVL equation. We use this sign convention for convenience; it is not required for proper application of KVL, as long as the signs on the voltage differences are treated consistently.



Applying KVL to the loop a-b-e-d-a, and using our sign convention as above results in:

$$v_1 - v_4 - v_6 - v_3 = 0$$

The starting point of the loop and the direction that we loop in is arbitrary; we could equivalently write the same loop equation as loop d-e-b-a-d, in which case our equation would become:

$$v_6 + v_4 - v_1 + v_3 = 0$$

This equation is identical to the previous equation, the only difference is that the signs of all variables has changed and the variables appear in a different order in the equation.

We now apply KVL to the loop b-c-e-b, which results in:

$$-v_2 + v_5 + v_4 = 0$$

Finally, application of KVL to the loop a-b-c-e-d-a provides:

$$v_1 - v_2 + v_5 - v_6 - v_3 = 0$$

## Application Examples: Solving for Circuit Element Variables

Typically, when analyzing a circuit, we will need to determine voltages and/or currents in one or more elements in the circuit. In this chapter, we discuss use of the tools presented in previous chapters for circuit analysis.

The complete solution of a circuit consists of determining the voltages and currents for every elements in the circuit. A complete solution of a circuit can be obtained by:

4. Writing a voltage-current relationship for each element in the circuit (e.g. write Ohm's Law for the resistors).
5. Applying KCL at all but one of the nodes in the circuit.
6. Applying KVL for all but one of the loops in the circuit.

This approach will typically result in a set of  $N$  equations in  $N$  unknowns, the unknowns consisting of the voltages and currents for each element in the circuit. Methods exist for defining a reduced set of equations or a complete analysis of a circuit; these approaches will be presented in later chapters.

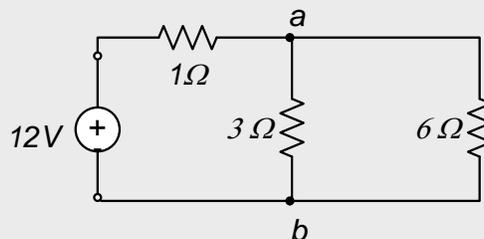
If KCL is written for every node in the circuit and KVL written for every loop in the circuit, the resulting set of equations will typically be over-determined and the resulting equations will, in general, not be independent. That is, there will be more than  $N$  equations in  $N$  unknowns and some of the equations will carry redundant information.

Generally, we do not need to determine all the variables in a circuit. This often means that we can write fewer equations than those listed above. The equations to be written will, in these cases, be problem dependent and are often at the discretion of the person doing the analysis.

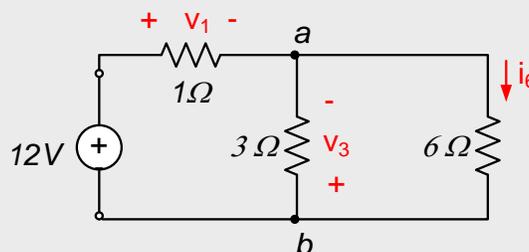
Examples of using Ohm's Law, KVL, and KCL for circuit analysis are provided below.

### Example 1.12:

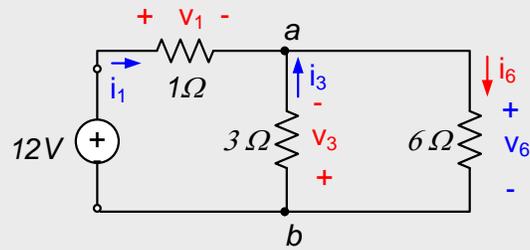
For the circuit below, determine  $v_{ab}$ .



We are free to arbitrarily choose either the voltage polarity or the current direction in each element. Our choices are shown below:



Once the above voltage polarities and current directions are chosen, we must choose all other parameters in a way that satisfies the passive sign convention. (Current must enter the positive voltage polarity node.) Our complete definition of all circuit parameters is shown below:



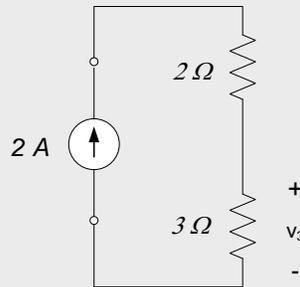
We now apply the steps outlined above for an exhaustive circuit analysis.

1. Ohm's law, applied for each resistor, results in:  
 $v_1 = (1\Omega)i_1;$       $v_3 = (3\Omega)i_3;$       $v_6 = (6\Omega)i_6$
2. KCL, applied at node a:  
 $i_1 + i_3 - i_6 = 0$
3. KVL, applied over any two of the three loops in the circuit:  
 $-12V + v_1 - v_3 = 0$   
 $v_3 + v_6 = 0$

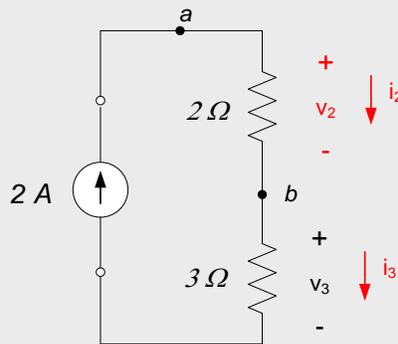
The above provide six equations in six unknowns. Solving these for  $v_3$  results in  $v_3 = -8V$ . Since  $v_3 = -v_{ab}$ ,  $v_{ab} = 8V$

### Example 1.13:

Determine  $v_3$  in the circuit shown below.



We choose voltages and currents as shown below. Since  $v_3$  is defined in the problem statement, we define it to be consistent with the problem statement.



KVL around the single loop in the circuit does not help us – the voltage across the current source is unknown, so inclusion of this parameter in a KVL equation simply introduces an additional unknown to go with the equation we write. KVL would, however, be useful if we wished to determine the voltage across the current source.

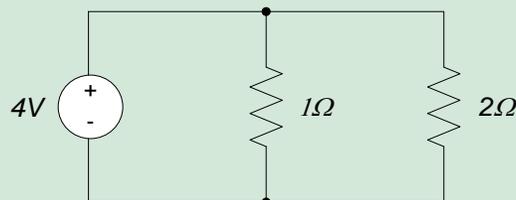
KCL at node a tells us that  $i_2 = 2A$ . Likewise, KCL at node b tells us that  $i_2 - i_3 = 0$ , so  $i_3 = i_2 = 2A$ . Ohms law tells us that  $v_3 = (3\Omega)(i_3) = (3\Omega)(2A) = 6V$ .

## Section Summary

- Kirchhoff's Current Law (KCL) and Kirchhoff's Voltage Law (KVL) govern the interactions between circuit elements. Governing equations for a circuit are created by applying KVL and KCL and applying the circuit element governing equations, such as Ohm's Law.
- Kirchhoff's current law states that the sum of the currents entering or leaving a node must be zero. A node in a circuit is a point which has a unique voltage.
  - A node is a point of interconnection between two or more circuit elements. A circuit node has a particular voltage. Nodes can be "spread out" with perfect conductors.
- Kirchhoff's voltage law states that the sum of the voltage differences around any closed loop in a circuit must sum to zero. A loop in a circuit is any path which ends at the same point at which it starts.
  - A loop is a closed path through a circuit. Loops end at the same node at which they start, and typically are chosen so that no node is encountered more than once.

## 1.4 Exercises:

1. For the circuit below, determine:
  - a) The current through the  $2\Omega$  resistor
  - b) The current through the  $1\Omega$  resistor
  - c) The power (absorbed or generated) by the 4V power source



## Chapter Summary:

- In this text, we will be primarily concerned with the movement of electrical charge. Electrical charge motion is represented by voltage and current. Voltage indicates the energy change associated with the movement of a charge from one location to another, while current is indicative of the rate of current motion past a particular point.
  - Voltage is an energy difference between two physically separated points. The polarity of a voltage is used to indicate which point is to be assumed to be at the higher energy level. The positive terminal (+) is assumed to be at a higher voltage than the negative terminal (-). A negative voltage value simply indicates that the actual voltage polarity is opposite to the assumed polarity.
  - The sign of the current indicates the assumed direction of charge motion past a point. A change in the sign of the current value indicates that the current direction is opposite to the assumed direction.

- Circuit elements can be either active or passive. Active elements provide electrical energy from a circuit, from sources outside the circuit; active elements can be considered to create energy (from the standpoint of the circuit, anyway). Passive elements can store or dissipate electrical energy provided to them by the circuit; they can subsequently return energy to the circuit which they have previously stored, but they cannot create energy.
  - Active circuit elements introduced in this section are independent voltage and current sources, and dependent voltage and current sources.
  - Resistors are the only passive circuit elements introduced in this chapter. Resistors cannot store electrical energy, they can only dissipate energy by converting it to heat.
- The assumed polarity of the voltage across a passive circuit element must be consistent with the assumed current direction through the element. The assumed positive direction for current must be such that positive current enters the positive voltage terminal of the element. Since this sign convention is applied only to passive elements, it is known as the *passive sign convention*.
  - The assumed current direction or the assumed voltage polarity can be chosen arbitrarily, but once one parameter is chosen, the other must be chosen to agree with the passive sign convention.
- The power absorbed or generated by an electrical circuit component is the product of the voltage difference across the element and the current through the element:  $p = iv$ . The relative sign of the voltage and current are set according to the passive sign convention. Positive power implies that the voltage and current are consistent with the passive sign convention (the element absorbs or dissipates energy) while negative power indicates that the relative signs between voltage and current are opposite to the passive sign convention (the element generates or supplies energy to the circuit).
- The relationship between voltage and current for a resistor is Ohm's Law:  $v = iR$ . Since a resistor only dissipates energy, the voltage and current for a resistor must always agree with the passive sign convention.
- Kirchoff's voltage law states that the sum of the voltage differences around any close loop in a circuit must sum to zero. A loop in a circuit is any path which ends at the same point at which it starts.
- Kirchoff's current law states that the sum of the currents entering or leaving a node must be zero. A node in a circuit is an point which has a unique voltage.

# Real Analog Chapter 1: Lab Projects

## 1.1: Solderless Breadboards, Open-circuits, and Short-circuits

The purpose of this lab assignment is to provide users with an introduction to some of the equipment which will be used in subsequent lab assignments. The following topics are discussed:

- breadboards
- use of digital multimeters (DMMs) to measure resistance.

The concept of electrical *resistance* is introduced qualitatively in this lab assignment. A more rigorous discussion of resistance will be presented in later modules and explored more thoroughly in later lab assignments.

### Before beginning this lab, you should be able to:

- Define the connectivity between holes in a solderless breadboard (see related background material relative to breadboards)
- State how to use a DMM to measure resistance (see related material relative to DMMs)
- State rules governing proper usage of a lab notebook (see related materials relative to lab notebooks)

### After completing this lab, you should be able to:

- Use a breadboard to connect electrical components
- Create open circuits and short circuits on a breadboard
- Identify nodes on a breadboard circuit
- Use a digital multimeter to measure resistance

### This lab exercise requires:

- Hand-held DMM
- Solderless breadboard

## Symbol Key

- DEMO** Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
- ANALYSIS** Analysis; include principle results of analysis in laboratory report.
- SIM** Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
- DATA** Record data in your lab notebook.

## General Discussion:

In this portion of the lab assignment, we will examine the connections between different holes in your breadboard using a DMM as an ohmmeter. Specifically, we will see which connections correspond to open circuits and short circuits. We will use jumper wires to change the connectivity between holes on the breadboard.

## Pre-lab: None

## Lab Procedures:

In this portion of the lab assignment, we will examine the connections between different holes in your breadboard using a DMM as an ohmmeter. Specifically, we will see which connections correspond to open circuits and short circuits. We will use jumper wires to change the connectivity between holes on the breadboard.

### DATA

- Using your DMM as an ohmmeter, connect the leads of the DMM to two holes in the same row on your breadboard, as shown in Figure 1(a) below. The DMM should indicate a very low resistance between the holes – generally fractions of an ohm. Record your results in your lab notebook.

### DATA

- Check the resistance between two rows of holes on opposite sides of the central channel of the breadboard, as shown in Figure 1(b) below. Your DMM should indicate a very high – or infinite – resistance between the holes. Record your results in your lab notebook.

### Note:

Different ohmmeters have different ways of indicating what they consider to be an infinite resistance. For example, some models may read out an overload (sometimes displayed as OL or a “flashing” number) condition.

### DATA

- Check the resistance between two arbitrary holes (not in the same row) of your breadboard (e.g. between nodes a and c in Figure 1(b)). Your DMM should again indicate a very high or infinite resistance. Record your results in your lab notebook.

### DATA

- Finally, use a jumper wire to connect two different rows on the breadboard. Add a jumper wire between nodes a and c as shown in Figure 1(c). The DMM should indicate that the resistance between these nodes is now very low. Record your results in your lab notebook.

### DATA

- In your lab notebook, comment on whether the connections in lab procedures 1 – 4 correspond to “open circuit” or “short circuit” conditions.

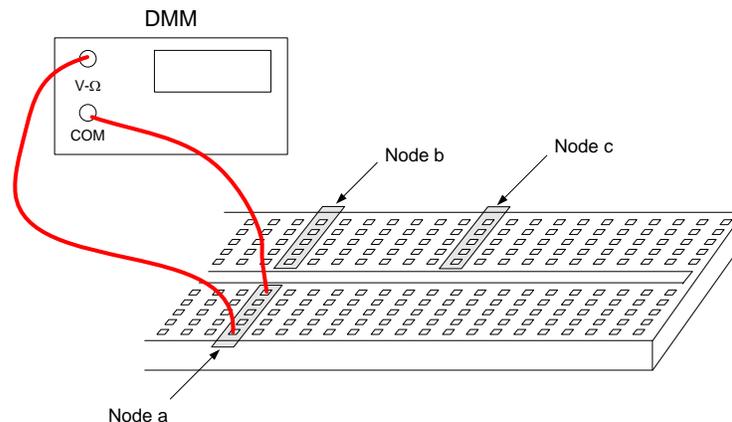


Figure 1(a).

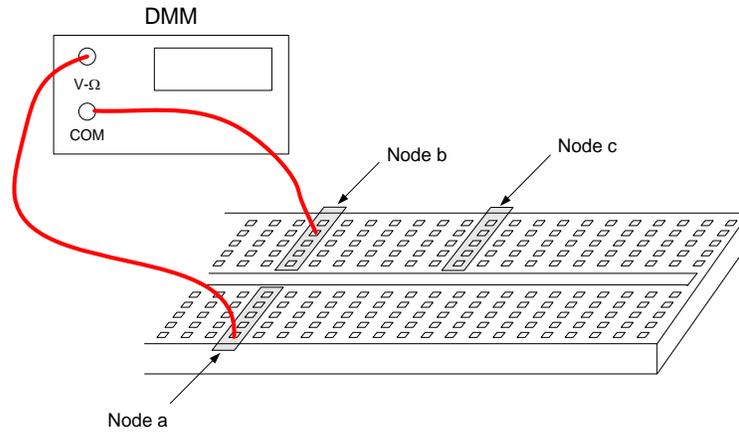


Figure 1(b).

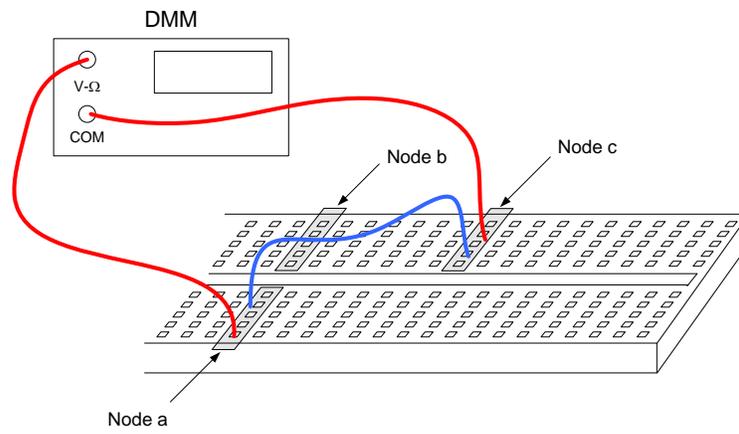


Figure 1(c).

## Real Analog Chapter 1: Lab Worksheets

### 1.1: Breadboards and Ohmmeters (20 points total)

1. Provide the resistance you measured between two holes in same row. State whether these operate as open or closed circuit. (5 pts)
2. Provide the resistance you measured between two holes opposite sides of channel. Characterize as open or closed circuit. (5 pts)
3. Provide the resistance you measured between two arbitrary holes in the breadboard. Characterize as open or closed circuit. (5 pts)
4. Provide the resistance you measured between holes connected by a jumper wire. Characterize as open or closed circuit. (5 pts)

# Real Analog Chapter 1: Lab Projects

## 1.2.1: Independent Power Supplies, Ammeters, and Voltmeters

The purpose of this lab assignment is to provide users with an introduction to some of the equipment which will be used in the next few lab assignments. The following topics are discussed:

- Power supplies
- Use a digital multimeter (DMM) to measure voltage and current

### Before beginning this lab, you should be able to:

- State how to use a DMM to measure voltage and current (see related material relative to DMMs)
- Write symbols for independent voltage and current sources
- State the purpose of independent voltage and current sources

### After completing this lab, you should be able to:

- Use a digital multimeter to measure voltage and current
- Use the Analog Discovery 2 voltmeter to measure a constant voltage
- Use the Analog Discovery 2 power supplies to provide a constant voltage to a circuit

### This lab exercise requires:

- Analog Discovery 2
- Digilent Analog Parts Kit
- Digital multimeter

### Symbol Key:

**DEMO**

Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.

**ANALYSIS**

Analysis; include principle results of analysis in laboratory report.

**SIM**

Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.

**DATA**

Record data in your lab notebook.

### General Discussion:

Consider the circuit shown schematically in Fig. 1(a). A DC voltage supply of 5V is connected between nodes a and c to provide power to the circuit. We want to know the current,  $i$ , out of the power source, and the voltage across the open circuit,  $v_{bc}$ . Since there is an open circuit between nodes b and c, we will expect no current to flow through the circuit ( $i = 0$  A) and the voltage difference between nodes b and c should be the same as the voltage applied to the circuit ( $v_{bc} = 5$  V).

The schematic of Fig. 1(b) shows the same circuit, with the measurements which will be made explicitly indicated. A DMM is connected between nodes a and b to measure the current  $i$ . We will use one oscilloscope channel, connected across nodes b and c to measure the voltage  $v_{bc}$ . (Since the voltage  $v_{bc}$  will be a constant value, it is not absolutely necessary to measure this value using an oscilloscope. It would be more common to use a DMM to measure the voltage  $v_{bc}$ , if one were available.) You may need to use a combination of jumper wires and

connectors with alligator clips to make the connections between the breadboard and the DMMs and power supplies.

### Notes:

- A DMM, used as an ammeter, behaves approximately as a short circuit.
- A DMM or oscilloscope, used to measure voltage, behaves approximately as an open circuit.

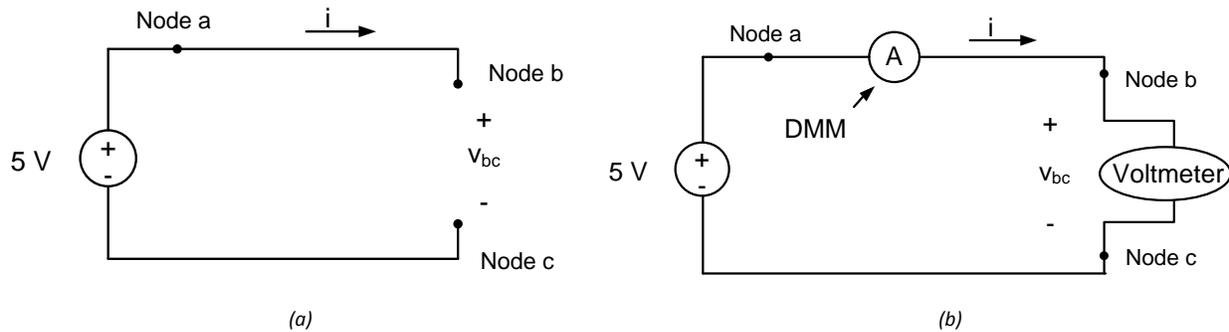


Figure 1. Circuit schematic.

### Note:

Constant voltages are commonly measured using DMMs. However, it is easy to use the Analog Discovery 2 Voltmeter instrument to perform this task. The Analog Discovery 2 Voltmeter allows measurement of up to two voltages simultaneously on two different channels. Probes used by the Voltmeter are labeled as **1+** and **1-** (positive and negative terminals for the channel 1 voltage) and **2+** and **2-** (positive and negative terminals for the channel 2 voltage) on the Analog Discovery 2. Usage of the Voltmeter is briefly discussed below:

- Connect the probes of the desired voltmeter channel to the appropriate nodes of your circuit.
- Click on Voltmeter in the WaveForms main window. A Voltmeter window should open.
- Make sure that the Voltmeter is running –To run the Voltmeter click on the Run button. If it is running the run button will be replaced with the stop button<sup>1</sup>.
- The measured constant voltages for both channels are displayed in the “DC” row on the Voltmeter window<sup>2</sup>. If the probes for a channel are not connected to a circuit, the Voltmeter reading should be approximately zero for that channel.

The above process can be applied rather automatically, as long as the voltage being measured is constant. We will examine the measurement of voltages, which vary with time in later projects.

### Pre-lab: None

<sup>1</sup> The Voltmeter instrument and Scope instruments share the same set of probes. If the Voltmeter instrument is enabled, the Scope instrument is disabled. Disabling the Voltmeter instrument enables the Scope instrument.

<sup>2</sup> The rows labeled **True RMS** and **AC RMS** are used for measuring voltages which change as a function of time. We’ll discuss these measurements in later lab assignments.

**Lab Procedures:****DATA**

1. Connect the circuit shown in Fig. 1(b) except do not connect the power supply to the circuit. Use your DMM to perform the current measurement and channel 1 of your voltmeter to measure the voltage  $v_{bc}$ . Note in your lab notebook the voltage and current with no power supplied to the circuit.
2. Use VP+ on the Analog Discovery 2 to apply the 5V source to the circuit. Turn the power supply on.

**DATA**

3. Measure the voltage  $v_{bc}$  and the current  $i$ . Note your results in your lab notebook. Do your results agree with your expectations? What is the approximate power generated or absorbed by the power source?

**DATA**

4. Turn off the power supply, reverse the leads on the voltmeter, turn the power supply back on, and note the resulting voltage and current measurements in your lab notebook. Do the results agree with your expectations? Why or why not? Comment on the agreement (or lack thereof) between your voltage and current measurements in your lab notebook.

**DEMO**

5. Demonstrate operation of your circuit to the Teaching Assistant. Have the TA initial the appropriate page(s) of your lab notebook and the lab checklist.

## Real Analog Chapter 1: Lab Worksheets

### 1.2.1: Power Supplies, Voltmeters, Ammeters (25 points total)

1. Sketch the circuit below, indicate the voltages  $v_{ac}$ ,  $v_{bc}$  and the current  $i$  on your sketch. Indicate on your sketch how the current and voltage measurements are being made. (2 pts)
2. What are the voltage and current measurements  $v_{bc}$  and  $i$  with no power applied to the circuit? (3 pts)
3. What are the measured voltages  $v_{ac}$  and  $v_{bc}$  and the measured current  $i$  with 5V applied to the circuit? (3 pts)
4. Comment on the following relative to your results.
  - a) What is the percent difference between the applied voltage  $v_{ab}$  indicated by the voltage source and the measured voltage  $v_{ab}$ ? (2 pts)
  - b) What is the voltage difference across the ammeter (e.g the difference between  $v_{ac}$  and the voltage  $v_{bc}$ )? Is the ammeter behaving like an open circuit or a short circuit? (2 pts)
  - c) What is the power generated by the source? (4 pts)

5. What are the measured voltages  $v_{ac}$  and  $v_{bc}$  and the measured current  $i$  with the voltmeter leads reversed? Comment on your results vs. expectations, especially relative to the signs of the voltages and current. (5 pts)

6. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuits' operation. (4 pts)

TA Initials: \_\_\_\_\_

# Real Analog Chapter 1: Lab Projects

## 1.2.2: Dependent Sources and MOSFETs

### Overview:

In this lab assignment, a qualitative discussion of dependent sources is presented in the context of MOSFETs (Metal Oxide Semiconductor Field Effect Transistors). A simple voltage controlled current source is constructed and tested.

#### Before beginning this lab, you should be able to:

- Use a DMM to measure voltage and current (Lab 1.2.1)
- Use the Analog Discovery 2 power supplies to apply a fixed 5V voltage (Lab 1.2.1)
- Use the Analog Discovery 2 voltmeter to measure a constant voltage (Lab 1.2.1)
- Write symbols for dependent voltage and current sources
- State governing equations for the four types of dependent sources

#### After completing this lab, you should be able to:

- Use MOSFETs as dependent sources
- Use the Analog Discovery 2 waveform generators to apply constant voltages

#### This lab exercise requires:

- Analog Discovery 2
- Digilent Analog Parts Kit
- Digital multimeter

### Symbol Key:

- DEMO** Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
- ANALYSIS** Analysis; include principle results of analysis in laboratory report.
- SIM** Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
- DATA** Record data in your lab notebook.

### General Discussion:

Many common circuit elements are *modeled* as dependent sources – that is, the mathematics describing the operation of the element is conveniently described by the equations governing a dependent source. In this portion of the lab assignment, we will build and test a circuit which acts as a Voltage Controlled Current Source (VCCS).

The primary circuit element used in this assignment is a Metal Oxide Semiconductor Field Effect Transistor (MOSFET). There are two basic types of MOSFETs: *n-channel* and *p-channel*; the discussion presented here is for n-channel MOSFETs, though similar concepts apply to p-channel MOSFETs. A MOSFET is a three-terminal device; the symbol commonly used to represent a MOSFET in circuit diagrams is shown in Fig. 1(a). The three terminals of the device are called the source (S), the drain (D) and the gate (G). Our circuit will employ a ZVN2210A MOSFET; the

physical appearance of this MOSFET is shown in Fig. 1(b), along with the relative locations of the drain, gate and source for that model MOSFET.

An extremely simplified discussion of a MOSFET's operation is as follows: A "channel" is opened in the MOSFET by application of a voltage at the gate of the MOSFET. This channel allows current to flow from the drain to the source of the MOSFET ( $i_D$  in Fig. 1(a)). Thus, if a power supply is connected to the drain of the MOSFET, the MOSFET can be used to control the power supply's current: increasing the gate voltage increases the current out of the power supply. A rough analogy to this process is a valve placed at the base of a water tank – opening the valve allows water to flow out of the tank. Likewise, increasing the gate voltage allows current to "flow" out of the power supply. A MOSFET, therefore, in conjunction with a power supply, can act as a voltage controlled current source in which the drain current is controlled by the gate voltage.

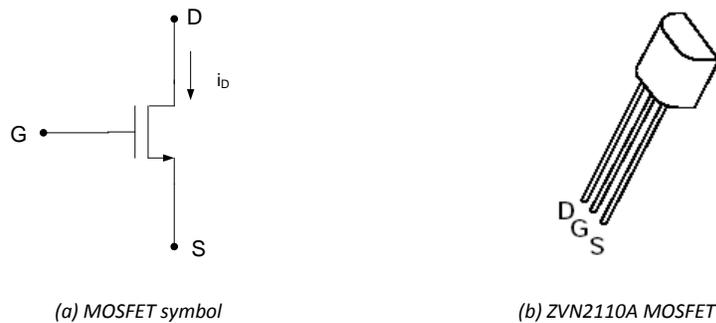


Figure 1. MOSFET symbol and typical physical appearance.

## Pre-lab: None

## Lab Procedures:

1. Connect the circuit shown in Fig. 2. Two power supplies are used in the circuit. Use channel 1 of your Arbitrary Waveform Generator (W1) to apply the (variable) gate voltage,  $V_G$ . Use of the waveform generator to apply constant voltages is presented in Appendix A of this assignment. Use the positive power supply (VP+) to provide a constant 5V to the MOSFET drain; this power supply provides the drain current  $I_D$ .

The  $100\Omega$  resistor in Fig. 2 is used to limit the amount of current flowing through the MOSFET. If no resistor is used between the power supply and the MOSFET, an excessive amount of current can flow through the MOSFET resulting in damage to the MOSFET and/or the rest of the circuit. The  $100\Omega$  resistors in your analog parts kit can be identified by the color bands on the side of the resistor – they will be as shown in Fig. 3. We will discuss resistors in detail in later modules. Use an ohmmeter to measure the resistance of the resistor and record this value in your lab notebook (the actual resistance will most likely be slightly different from  $100\Omega$ ).

### DATA

Connect your DMM as shown in Fig. 2 to measure the current  $I_D$ .

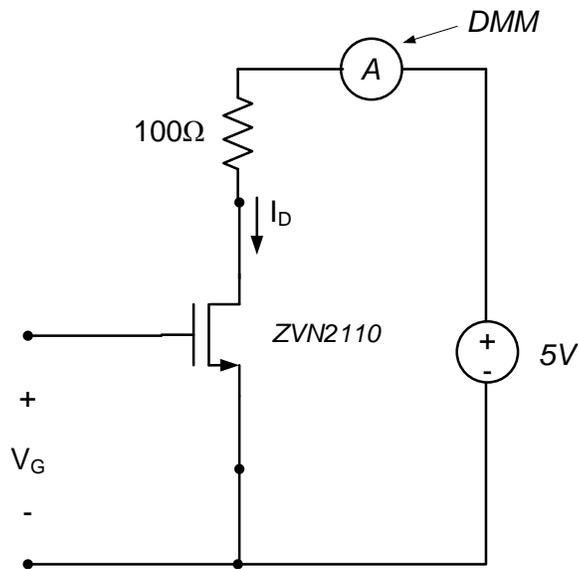


Figure 2. VCCS circuit schematic.

**DATA**

- MOSFETs have a *threshold voltage*, below which essentially no current passes through the MOSFET. To determine the threshold voltage for our MOSFET, begin with zero voltage applied at the gate by the variable voltage source  $V_G$  ( $V_G = 0V$ ). The drain current, with no voltage applied at the gate, should be essentially zero. Gradually increase the MOSFET gate voltage while monitoring the MOSFET drain current  $I_D$ . Record in your lab notebook the voltage at which the drain current begins to increase significantly. This is the MOSFET's threshold voltage.

**DATA**

- Now characterize the MOSFET's relationship between gate voltage and drain current. Starting at the threshold voltage, continue to increase the gate voltage at increments of about 0.3V up to a maximum of about 5V. Record the gate voltages and their corresponding drain currents in your lab notebook. Plot the gate voltage vs. drain current data in your lab notebook. Comment on your observations relative to the data, especially relative to how the circuit behaves like a dependent source.

**ANALYSIS**

- The parameter  $g$  of a VCCS provides a relationship between the rate of change between the applied voltage and the resulting current. This is essentially the slope of the data you plotted in part 3 above. Use the curve of part 3 to estimate the value of  $g$  for the circuit you built. Note: Your curve will most likely not be a straight line. Do your best to fit a straight line to the data you acquired in part 3 for your estimate of  $g$ .

**DEMO**

- Demonstrate operation of your circuit to the Teaching Assistant. Have the TA initial the appropriate page(s) of your lab notebook and the lab checklist.

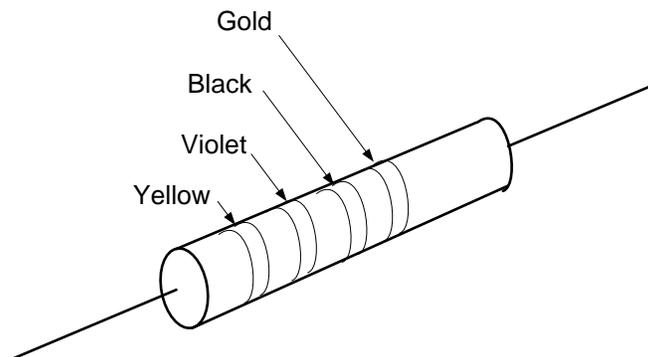
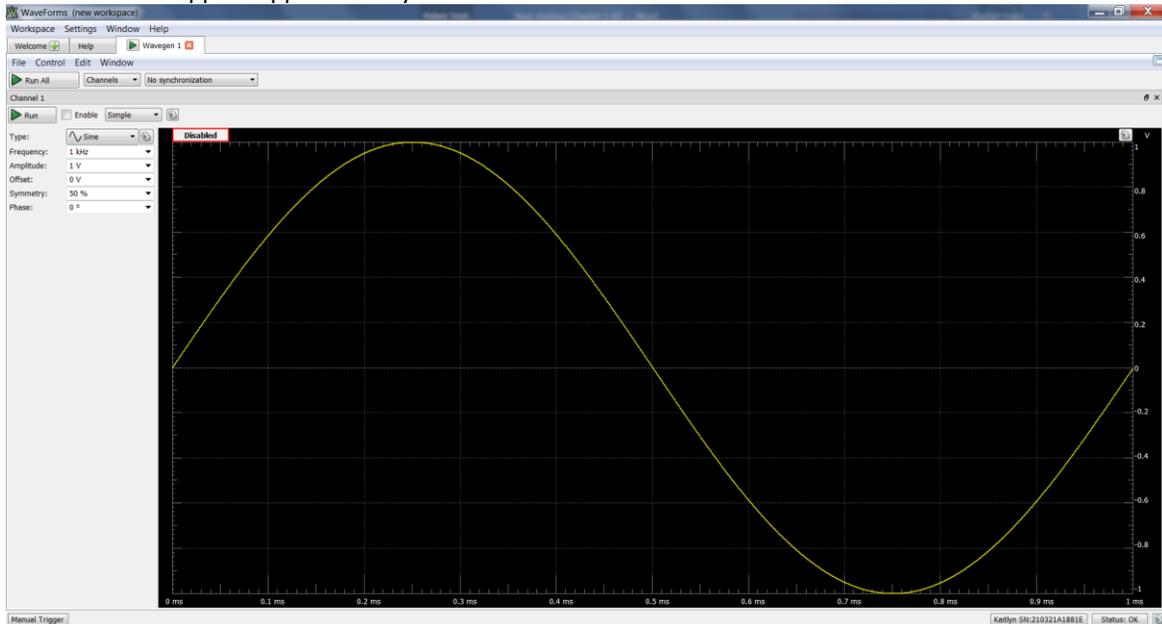


Figure 3. Color bands for 47Ω resistor.

## Appendix A: Waveform Generator to Apply Constant Voltages

The Voltage instrument on the Analog Discovery 2 provides the capability to apply between  $\pm 5V$  voltages to a circuit. You can also apply arbitrary voltages to a circuit using the Waveform Generator in the case that you need more than one positive or negative supply. The waveform generator on the Analog Discovery 2 provides this capability. To do this, follow the steps below:

1. Click on the **WaveGen** icon in the **WaveForms** main window to open the waveform generator window. This window should appear approximately as shown below:



2. There are a series of options in a drop-down in the upper-left corner of the screen. These icons allow you to select the “shape” of the waveform which will be applied to your circuit. In order to apply a constant voltage, click on the DC option. The waveform generator window should alter its appearance so that it looks as shown below. The button to the immediate right of the “Run” button should appear as “**Disabled**”. If it displays **Enabled**, click on it to disable the waveform generator.



3. The value of the constant voltage to be applied to the circuit is specified in the **Offset** drop-down (the value is limited to between +5V and -5V). The desired value can be set by typing the desired value into the drop-down.
4. Click on “Run” to apply power to channel 1 (W1) of the waveform generator. (Each waveform generator channel has its own interface; you can select the desired channel with the button.) Clicking on the “Run” button automatically enables the function generator; the button to the immediate right of the “Run AWG 1” button should now appear as “Enabled”.

#### Important notes relative to the Enabled/Disabled button:

- The button to the right of the “Run AWG 1” button allows you to “enable” or “disable” the AWG. This button is used to turn off all power provided by the waveform generator channel. Thus, if you are working on your circuit with the waveform generator probes are connected, this button should always be set to “Disabled”.
  - Clicking on “Stop”, when the waveform generator is running, only removes the time-varying component of the signal, not any constant components. If we are applying only a constant voltage to the circuit (as in this lab assignment), the “Stop” button has no effect on the voltages being applied to the circuit.
  - Clicking on “Run” automatically enables the waveform generator, so that both time-varying and constant components are applied to the circuit.
  - Closing the waveform generator window also turns off all power applied by the waveform generator.
5. To turn off power to your circuit, click on the “Enabled” button so that it displays “Disabled”.

## Real Analog Chapter 1: Lab Worksheets

### 1.2.2: Dependent Sources and MOSFETs (55 points total)

1. Diagram of circuit, including measured resistance value. (5 pts)
2. What MOSFET threshold voltage. (10 pts)
3. Attach to this worksheet a table providing your measured gate-to-source voltage vs. drain current values and a plot of data. (15 pts)
4. What type of dependent source is the transistor behaving like? Why? (5 pts)
5. Estimated value of  $g$  for circuit. Annotate the plot attached to this worksheet, indicating how the value of  $g$  was determined. (10 pts)
6. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuits' operation. (10 pts)

TA Initials: \_\_\_\_\_

## Real Analog Chapter 1: Lab Projects

### 1.3.1: Resistors and Ohms Law – Resistance Variations

In this lab, we will experimentally explore the characteristics of resistors. Resistance is measured using an ohmmeter; this measured resistance is compared to a voltage-current characteristic curve measured for the resistor. Some basic statistical data analysis methods are introduced briefly in this lab assignment. MATLAB commands used to perform basic statistical analysis of data are provided in the background material related to this lab assignment.

#### Before beginning this lab, you should be able to:

- State Ohm's law from memory
- Use a digital multimeter to measure resistance and voltage (Labs 1.1, 1.2.1)
- Use color codes on resistors to determine the resistor's nominal resistance

#### After completing this lab, you should be able to:

- Calculate the mean, median, and standard deviation values of a set of data

#### This lab exercise requires:

- Digilent Analog Parts Kit
- Digital multimeter (DMM)

#### Symbol Key:

- DEMO** Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
- ANALYSIS** Analysis; include principle results of analysis in laboratory report.
- SIM** Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
- DATA** Record data in your lab notebook.

#### General Discussion:

Fixed resistors are fabricated with a nominal resistance value. Individual resistors will, in general, not have exactly this resistance – their actual resistance will vary from this nominal value to some extent. In this portion of the lab assignment, we will measure the resistance of several resistors which all have the same nominal resistance, in order to determine the actual resistor-to-resistor resistance variations. These variations will be compared with the manufacturer's specified tolerances.

#### Important Note:

Since the resistance of individual resistors will vary from the value specified by the manufacturer, it is generally important, when incorporating a resistor in a circuit, to measure the resistor's actual resistance and record that value in your laboratory notebook when you are constructing a circuit. It is possible that the variations in resistance can affect your experimental results.

We will use *statistical analyses* to in this section to examine variations in resistors' resistances. Statistics are a way to quantify variations in results which are due to random effects such as variations in manufacturing, uncertainty in measured quantities, and measurement noise.

### Related information:

Some basic information relative to common statistical quantities used to characterize data is provided in Appendix A of this experiment. Basic MATLAB commands which can be used to perform these statistical analyses are also provided in this Appendix

**Pre-lab: None**

### Lab Procedures:

**DATA**

1. Pick five (5) 2.2KΩ resistors from your analog parts kit. Using an ohmmeter, measure the resistance of each of the resistors and record the values in your lab notebook. Calculate the mean, median, and standard deviation of the resistance values and record these quantities in your lab notebook. Convert these values to percentages of the nominal resistance, and record these results in your lab notebook. Determine the manufacturer's tolerance on the resistance values of the resistance and comment on whether your resistors lie within the specified tolerance.

**DATA**

2. Repeat procedure 1 above with five (5) 47KΩ resistors.

## Appendix A: Basic Statistics

In this appendix, we briefly present some common statistical values used to characterize data which have some random component. We will also present MATLAB commands which can be used to determine these values.

### Mean value:

If we have N sample values,  $\{y_1, y_2, \dots, y_N\}$ , we can determine the mean of the values by the following expression:

$$\bar{y} = \frac{1}{N} (y_1 + y_2 + y_3 + \dots + y_N) = \frac{1}{N} \sum_{i=1}^N y_i \quad \text{Eq. 1}$$

Thus, the mean can be determined by summing all the sample values and dividing by the total number values. It is essentially what we do when we determine an *average* of a series of values.

### Median value:

The median value of a set of N sample values  $\{y_1, y_2, \dots, y_N\}$  is the value which has the same number of samples above the value as there are below the value. This definition, however, does not necessarily uniquely determine the median value of a set of numbers. We will, therefore, refine our definition as follows:

- If N is an odd number, sort the data so that they are ordered from smallest to largest. The median value is then the middle data value.
- If N is an even number, again sort the data so that they are ordered from smallest to largest. There is no value exactly at the middle of the resulting sequence, so we choose the median value as the mean of the middle two numbers.

### Standard deviation:

The standard deviation of a set of values  $\{y_1, y_2, \dots, y_N\}$  can be thought of as the amount of "spread" that the numbers have from their mean value. Thus, the standard deviation provides an indication as to how closely grouped the values are around their mean. The definition of standard deviation that we will use is:

$$s = \frac{1}{N-1} \sqrt{\sum_{i=1}^N (y_i - \bar{y})^2} \quad \text{Eq. 2}$$

where  $s$  is the standard deviation and  $\bar{y}$  is the mean of the values, as defined above. The positive square root is used in the calculation, so that  $s$  is always represented as a positive number. Some textbooks use an alternate version of the standard deviation; we will exclusively use equation (2). When using a software package to perform statistical analyses, always check the definition of standard deviation that is used.

### MATLAB syntax:

If the variable  $\mathbf{y}$  has been defined as a vector of numbers in the workspace, the following MATLAB syntax will calculate the mean, median, and standard deviation of the numbers.

- Mean: **mean (y)**
- Median: **median (y)**
- Standard Deviation: **std (y)**

## Real Analog Chapter 1: Lab Worksheets

### 1.3.1: Resistance Variations (20 points total)

1. 2.2K $\Omega$  resistors:
  - a. Attach to this worksheet a table providing your measured resistances (3 pts)
  
  - b. Provide below the mean and median values, and the standard deviation of your resistance values. (2 pts)
  
  - c. Provide below the mean, median, and standard deviation as percentages of the nominal resistance. (2 pts)
  
  - d. Are the estimated variations within the manufacturer's tolerances? Explain. (3 pts)
  
2. 47K $\Omega$  resistors:
  - a. Attach to this worksheet a table providing your measured resistances (3 pts)
  
  - b. Provide below the mean and median values, and the standard deviation of your resistance values. (2 pts)
  
  - c. Provide below the mean, median, and standard deviation as percentages of the nominal resistance. (2 pts)
  
  - d. Are the estimated variations within the manufacturer's tolerances? Explain. (3 pts)

## Real Analog Chapter 1: Lab Projects

### 1.3.2: Resistors and Ohms Law – Voltage-Current Characteristics

In this lab, we will continue to explore the characteristics of resistors. In this lab, we measure several combinations of voltage and current for a resistor and plot the resulting voltage-current characteristic curve measured for the resistor. The resistance of the resistor will be estimated from the slope of the voltage-current characteristic. The slope of the curve will be estimated using linear regression techniques; MATLAB commands used to perform linear regression are provided in the background material associated with this lab assignment.

#### Before beginning this lab, you should be able to:

- State Ohm's law from memory
- Use a digital multimeter to measure current and voltage (Lab 1.2.1)
- Use color codes on resistors to determine the resistor's nominal resistance
- Use the Analog Discovery 2's arbitrary waveform generator (AWG) to apply constant voltages to a circuit (Lab 1.2.2)

#### After completing this lab, you should be able to:

- Determine the least-squares best fit straight line approximating a set of data
- Calculate the correlation coefficient between a set of data and a line approximating the data
- Estimate resistance from measured voltage-current data

#### This lab exercise requires:

- Analog Discovery 2
- Digilent Analog Parts Kit
- Digital multimeter

#### Symbol Key:

	Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
	Analysis; include principle results of analysis in laboratory report.
	Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
	Record data in your lab notebook.

#### General Discussion:

We have previously noted that the resistance of a component is the slope of the current vs. voltage curve for the component. In this part of the lab assignment, we will measure a current-voltage characteristic curve for a resistor and estimate a resistance from this data. We will compare this resistance from the resistance measured by an ohmmeter.

In order to experimentally determine the current-voltage characteristic for our resistor, we will use the circuit shown schematically in Fig. 1. The arbitrary waveform generator will be used to apply the voltage  $v_s$ . We will measure the voltage across the resistor,  $v_R$ , and the current through the resistor,  $i_R$ , using our DMM. By varying  $v_s$ , we can measure a set of values for  $v_R$  and  $i_R$  and plot  $v_R$  vs  $i_R$ , as shown in Fig. 2(a). The slope of the line that "best fits" the measured data can then be used to estimate the component's resistance, as shown in Fig. 2(b).

**Note:**

Do not use the values displayed by the power supply as the resistor voltage and current,  $v_R$  and  $i_R$ . The values displayed by the power supply may differ from the resistor's voltage and current due to non-ideal power supply effects, such as the power supply internal resistance.

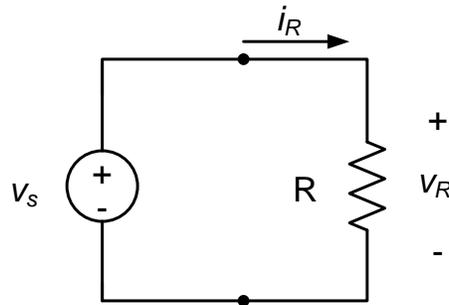


Figure 1. Circuit schematic.

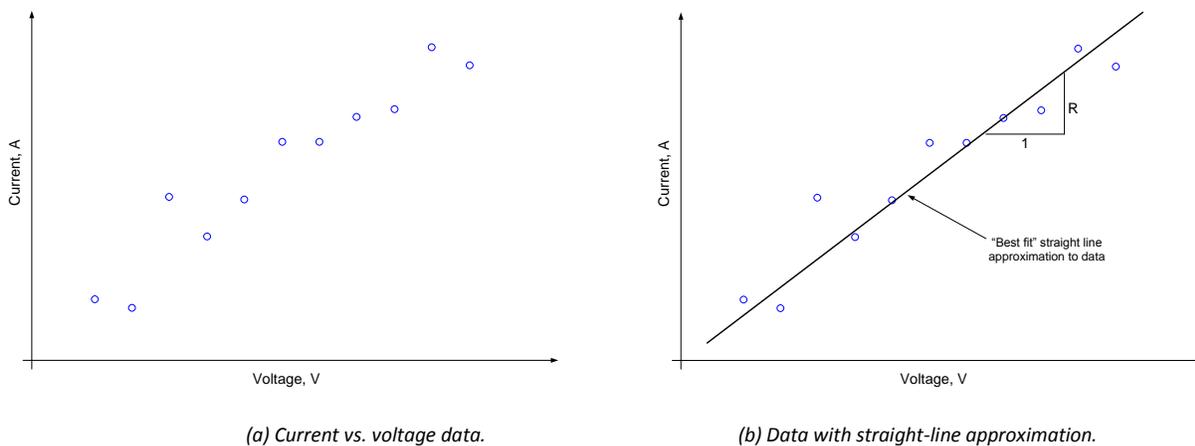


Figure 2. Measured data with “best fit” straight-line approximation.

**Pre-lab: None****Lab Procedures:****DATA**

1. Connect the circuit shown in Fig. 1. Use a  $100\Omega$  resistor and one of the AWG channels for the variable supply. Use an ohmmeter to measure the actual resistance of your resistor and record the value in your lab notebook.

**DATA**

2. Vary the supply voltage  $v_s$  from 0V to approximately 2V. Measure  $v_R$  and  $i_R$  for at least 10 different values of  $v_s$  over this range of applied voltage (e.g. measure  $v_R$  and  $i_R$  at approximately 0.2V increments in  $v_s$ ). Tabulate the measured values of  $v_R$  and  $i_R$  in your lab notebook. *Note: since we only have one DMM, the voltage and current measurements will have to be performed separately. If you have access to two DMMs, the two measurements can be made simultaneously.*

**DEMO**

3. Demonstrate operation of your circuit to the Teaching Assistant. Have the TA initial the appropriate page(s) of your lab notebook and the lab checklist.

### Related information:

Resistance is estimated in the post-lab exercises using linear regression of these data. Linear regression is discussed in Appendix A of this lab.

### Post-lab Exercises:

Determine a least-squares curve fit of the  $v_R$  vs.  $i_R$ . Plot the resulting line and the measured  $v_R$  vs.  $i_R$  data on the same graph. Comment on your results. Calculate a correlation coefficient for the data. Comment on your correlation coefficient relative to the qualitative agreement between the line and the data as shown on your plot.

## Appendix A: Linear Regression:

### Least-squares curve fitting:

Experimental data will always contain some uncertainty, so measured current-voltage data for a resistor will never lie exactly on a straight line – see, for example, Fig. 2. Thus, the notion of a “best” straight-line approximation to the data is rather nebulous. It is simplest to draw by eye a straight line through the plotted data. This approach, while used fairly often, has the drawback that no two engineers are likely to draw the same straight line. Thus, we look for a more objective and readily quantifiable approach toward fitting a line to a set of measured data. One common approach toward determining a line which provides a “best fit” to the available data is *least squares curve fitting*. The basic idea behind the least-squares approach toward fitting a curve to data is as follows:

- We have a set of  $x, y$  data, where the  $x$  data points are  $\{x_1, x_2, \dots, x_N\}$  and the  $y$  data points are  $\{y_1, y_2, \dots, y_N\}$
- Assume that a straight line will approximate the data. The equation for the straight line is

$$y = mx + b \quad \text{Eq.1}$$

where  $m$  and  $b$  – the slope of the line and its  $y$ -intercept – are unknowns to be determined.

- We define the error between the estimated line and the measured data to be the square of the distance between the line and the data at the  $x$  data points. Thus, our error is:

$$E = \sum_{i=1}^N [y_i - (mx_i + b)]^2 \quad \text{Eq. 2}$$

- If we minimize the error of equation (2) with respect to  $m$  and  $b$ , we obtain the least-squares straight line fit to the data. We will not discuss the mathematical details of this step here – they are rather tedious.
- We can determine how well our straight line agrees with the data by calculating a *correlation coefficient*. The correlation coefficient,  $r$ , is calculated by:

$$r = \frac{1}{N-1} \sum_{i=1}^N \left[ \left( \frac{x_i - \bar{x}}{s_x} \right) \cdot \left( \frac{y_i - \bar{y}}{s_y} \right) \right] \quad \text{Eq. 3}$$

where  $\bar{x}$  and  $\bar{y}$  are the means of the  $x$  and  $y$  data, respectively, and  $s_x$  and  $s_y$  are the standard deviations of the  $x$  and  $y$  data. The correlation coefficient is a number between -1 and 1 ( $-1 \leq r \leq 1$ ); it essentially tells us how well our data agrees with the straight line curve fit. If all the data lie exactly on a straight line with positive slope, the correlation coefficient will be identically one ( $r = 1$ ). If the data have noise or

follow a nonlinear relationship, the correlation coefficient will be reduced. Data which are entirely uncorrelated have a correlation coefficient of zero ( $r = 0$ ). A correlation coefficient of -1 simply means that there is a perfect negative relation between  $x$  and  $y$  – the data will lie on a straight line with negative slope. Fig. 3 provides several examples of data with various degrees of correlation.

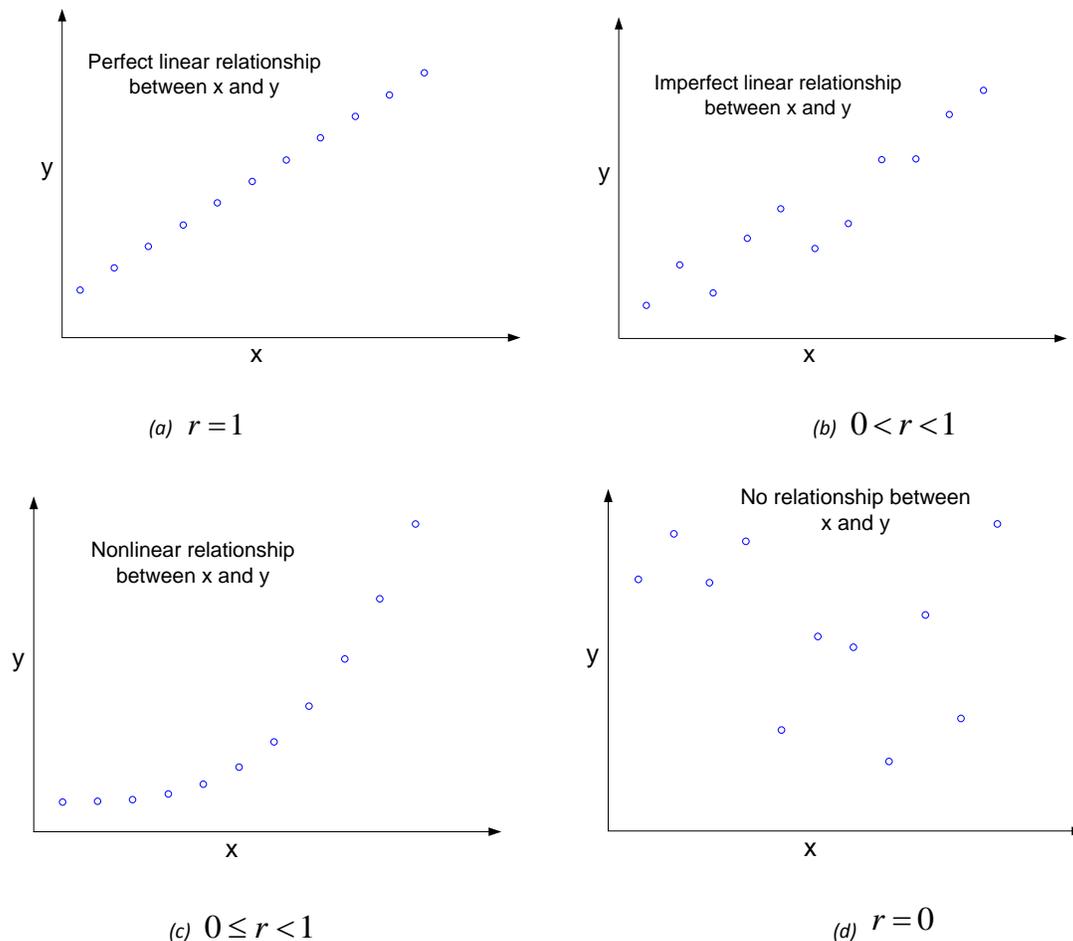


Figure 3. Example data and representative correlation coefficients.

### Using MATLAB for least squares curve fitting:

MATLAB's `polyfit` function performs least-squares curve fitting. Polyfit will fit an arbitrary-order polynomial to a set of data. Syntax for the function is

$$\mathbf{p} = \text{polyfit}(\mathbf{x}, \mathbf{y}, \mathbf{n})$$

where  $\mathbf{x}$  and  $\mathbf{y}$  are vectors containing the data to be fit,  $\mathbf{n}$  is the order of polynomial to be fit to the data (a straight line is a first order polynomial, so we will always set  $n = 1$ ). The function returns a vector containing the coefficients of the polynomial which provides a least-squares fit to the data. For  $n = 1$  a two-element vector will be returned; the first element of the vector will be the slope of the line ( $m$ , in equation (1)) and the second element will be the y-intercept of the line ( $b$ , in equation (1)).

MATLAB's `corrcoef` function provides the correlation coefficient of two data sets. Possible syntax for using this function is:

$$\mathbf{r} = \text{corrcoef}(\mathbf{x}, \mathbf{y})$$

where  $\mathbf{x}$  and  $\mathbf{y}$  are vectors containing the data. This use of the function will return a  $2 \times 2$  matrix; it will have the following form:

$$r = \begin{bmatrix} r_{xx} & r_{xy} \\ r_{yx} & r_{yy} \end{bmatrix} \quad \text{Eq. 4}$$

This matrix provides correlations between all possible combinations of the data provided to the function.  $r_{xx}$  is the correlation between the  $x$  data and itself. Likewise,  $r_{yy}$  is the correlation between the  $y$  data and itself. Since data is always perfectly correlated with itself,  $r_{xx} = r_{yy} = 1$  always.  $r_{xy}$  is the correlation between the  $x$  data and the  $y$  data, and  $r_{yx}$  is the correlation between the  $y$  data and the  $x$  data. For us,  $r_{xy} = r_{yx}$ . Thus, either the  $r_{xy}$  or  $r_{yx}$  terms will give us the correlation coefficient as defined in equation (3).

## Real Analog Chapter 1: Lab Worksheets

### 1.3.2: Resistance from Measured Current-voltage Data (25 points total)

1. Circuit diagram; measured resistance value of resistor. (5 pts)
2. Attach a table to this worksheet, providing your measured  $v_R$  and  $i_R$  over range of  $v_s$  from 0V – 2V (minimum 10 data points) (8 pts)
3. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuits' operation. (5 pts)

**TA Initials:** \_\_\_\_\_

4. Provide below an equation for least-squares best fit line to the data and the correlation coefficient between the data and the curve fit. Attach to this worksheet a plot of data vs. best fit line. (7 pts)

# Real Analog Chapter 1: Lab Projects

## 1.4.1: Dusk-to-Dawn Light

In this lab, we will create our first circuit which appears to “do” something which is readily perceivable without instrumentation. The circuit operates as a “dusk-to-dawn” light; the circuit turns a light on when the ambient light level goes below a certain level.

### Before beginning this lab, you should be able to:

- State Ohm’s law from memory
- Use Ohm’s law to perform voltage and current calculations for resistive circuit elements
- Use a digital multimeter to measure resistance, current, and voltage (Labs 1.1, 1.2.1)
- Use transistors as dependent sources (Lab 1.2.2)
- Use color codes on resistors to determine the resistor’s nominal resistance
- State KVL and KCL from memory

### After completing this lab, you should be able to:

- Use a bipolar junction transistor (BJT) as a dependent source
- Use a photoresistor to measure ambient light levels
- Use an LED as a light source
- Integrate a number of components into an overall system

### This lab exercise requires:

- Analog Discovery 2 module
- Digilent Analog Parts Kit
- Digital multimeter (optional)

### Symbol Key:

-  **DEMO** Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
-  **ANALYSIS** Analysis; include principle results of analysis in laboratory report.
-  **SIM** Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
-  **DATA** Record data in your lab notebook.

### General Discussion:

In this part of the lab assignment, we will create a light-sensitive lighting system. A photocell – a light-sensitive resistor – will be used to sense the ambient light level. A Bipolar Junction Transistor (BJT) will be used as a switch to turn on a light-emitting diode when the ambient light level becomes low. The circuit we will use is shown in Fig. 1.

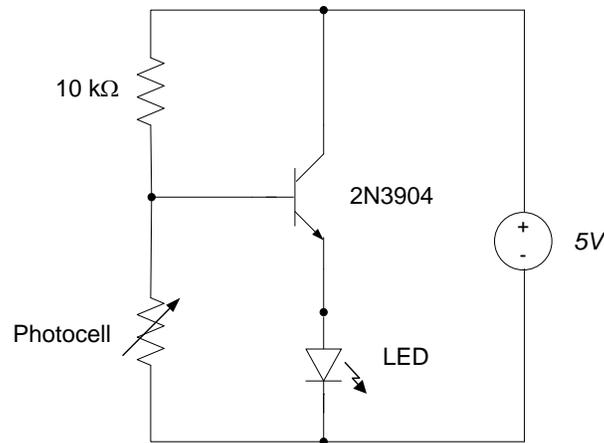


Figure 1. Dusk-to-dawn lighting circuit.

There are several unfamiliar components in the circuit of Fig. 1: a photocell, an LED, and a BJT. A detailed understanding of the operation of these components is beyond the scope of this lab, but a brief overview of their operating characteristics is provided below. Later courses in a typical electrical engineering curriculum will provide background information necessary to fully understand these components.

### Diodes and LEDs:

Diodes are two-terminal semiconductor devices that conduct current in only one direction. The terminals of a diode are called the *anode* and the *cathode*; diodes are intended to conduct current from the anode to the cathode. Diodes have a minimum *threshold voltage* (or  $V_{th}$ , usually around 0.7V) that must be present between the anode and cathode in order for current to flow. If the anode voltage is not at least  $V_{th}$  greater than the cathode voltage, no current will flow. Likewise, if the cathode voltage is greater than the anode voltage, the diode is said to be *reverse-biased* and no current will flow. In an ideal diode, if the diode voltage equals the threshold voltage (plus a small amount), then unlimited current can flow without causing the voltage across the diode to increase. And, if the diode is reverse-biased, no current will flow regardless of reverse-voltage magnitude

As with diodes, LED's are two-terminal semiconductor devices that conduct current in only one direction (from the anode to the cathode). The small LED chips are secured inside a plastic housing, and they emit light at a given frequency when a small electric current (typically 10mA to 25mA) flows through them. When the voltage difference across the LED exceeds the threshold voltage of the LED, current flows through the LED and light is emitted. If the LED voltage is less than the threshold voltage, no current flows and no light is emitted. LEDs are available in a number of colors; the Digilent analog parts kit contains red, yellow, and green LEDs.

Since LEDs are *polarized* devices, they must be placed in the circuit with the correct orientation; the anode must be at higher voltage potential than the cathode in order for the diode to emit light. An LED schematic symbol is shown in Fig. 2 below, together with a sketch of a physical LED. The anode and cathode on a physical LED can be identified because the anode pin is longer than the cathode pin and the cathode side of the plastic diffusion lens is typically slightly flattened.

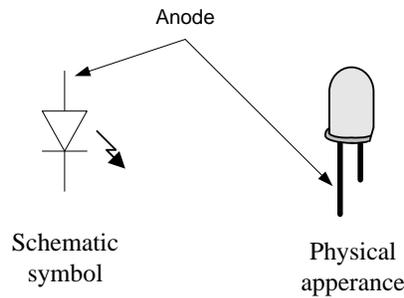


Figure 2. LED schematic symbol and physical appearance.

## Bipolar Junction Transistors (BJTs):

In Lab 1.2.1, we used a MOSFET as a voltage controlled current source. Bipolar Junction Transistors, BJTs, are also conveniently modeled as dependent sources. Like MOSFETs, BJTs are three-terminal devices; the terminals of a BJT are called the base (B), the collector (C), and the emitter (E). The symbol commonly used to represent the type of BJT we will be using is shown in Fig. 3(a). Our circuit employs a 2N3904 BJT; the physical appearance of this BJT is shown in Fig. 3(b), along with the relative locations of the base, collector, and emitter for that BJT.

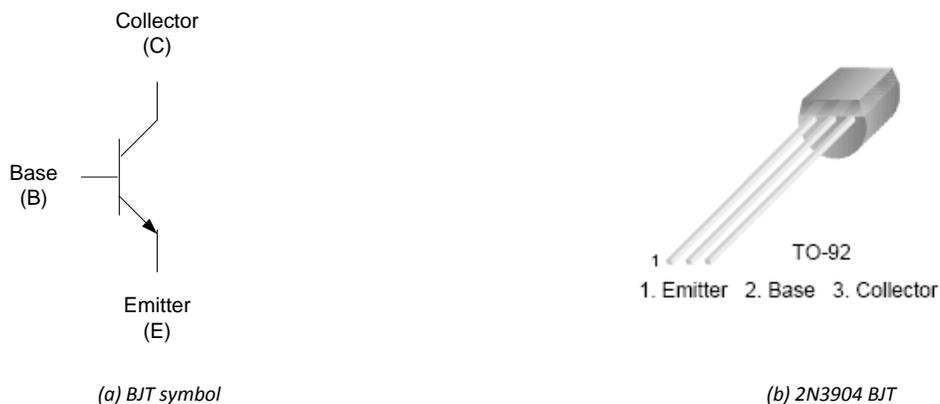


Figure 3. BJT symbol and physical appearance of 2N3904 BJT.

An extremely simplified discussion of a BJT's operation is as follows: application of voltage to the base of the BJT allows current to flow from the collector to the emitter of the BJT. Typically the current flowing into the base of the BJT is much smaller than the collector and emitter currents. Thus, the BJT can be conceptualized as a current controlled current source. Thus, if a power supply is connected to the collector of the BJT, the base voltage of the BJT can be used to control BJT's emitter current: increasing the base voltage typically increases the emitter current. In the case of two relatively discrete values of base voltage, the BJT can act as a switch; low base voltages turn off the switch (the emitter current is zero) while high base voltages turn the switch on (the emitter current is non-zero).

## Photocell:

*Photocells* (sometimes called *photoresistors* or *photoconductors*) are devices whose resistance changes according to the light intensity applied to the sensor. The photocells in the Digilent analog parts kits have resistances which vary from about  $5\text{K}\Omega$  at relatively high light levels to about  $20\text{K}\Omega$  at relatively low light levels. A variable resistor is commonly indicated on a circuit schematic by a resistor symbol with an arrow through it, as shown in Fig. 4.

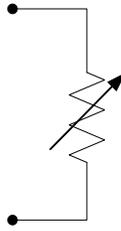


Figure 4. Variable resistor circuit symbol.

## Pre-lab:

## ANALYSIS

Apply KVL around the outer loop of the circuit of Fig. 1 (as shown in Fig. 5) to determine the voltage  $V_B$  for photocell resistances of  $5\text{k}\Omega$  and  $20\text{k}\Omega$ . You may assume that the current into the base of the BJT is negligible.

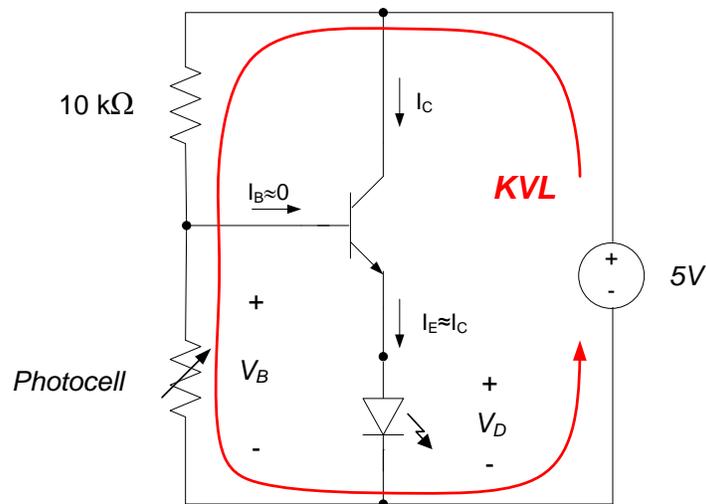


Figure 5. Circuit analysis to determine BJT base voltage.

## Lab Procedures:

## DATA

1. Construct the circuit shown in Fig. 1. The LED should not light under normal light levels. Using the scope instrument on your Analog Discovery 2 (or a DMM), measure the base voltage of the BJT ( $V_B$  in Fig. 5) and the voltage difference across the diode ( $V_D$  in Fig. 5). Record these voltages in your lab notebook and compare them with your calculated values from the prelab.

## DATA

2. Cover the photocell; the LED should light up. Using the scope instrument on your Analog Discovery 2 (or a DMM), measure the base voltage of the BJT ( $V_B$  in Fig. 5) and the voltage difference across the diode ( $V_D$  in Fig. 5). Record these voltages in your lab notebook and compare them with your calculated values from the prelab.

## DEMO

3. Demonstrate operation of your circuit to the Teaching Assistant. Have the TA initial the appropriate page(s) of your lab notebook and the lab checklist.



# Real Analog Chapter 1: Lab Projects

## 1.4.2: Resistive Network Power Dissipation

In this lab assignment, we will use KVL and KCL to analyze a simple circuit. The circuit will be constructed and tested and the actual circuit response compared to the expected response from the analysis.

### Before beginning this lab, you should be able to:

- State Ohm's law from memory
- Use a digital multimeter to measure resistance, voltage, and current (Lab 1.1, 1.2.1)
- Use the Analog Discovery 2's arbitrary waveform generator (AWG) to apply constant voltages to a circuit (Lab 1.2.2)
- Use the Analog Discovery 2 voltmeter to measure a constant voltage (Lab 1.2.1)
- Use color codes on resistors to determine the resistor's nominal resistance
- Use KVL and KCL for circuit analysis

### After completing this lab, you should be able to:

- Measure power dissipation of a resistor in a resistive network

### This lab exercise requires:

- Analog Discovery 2
- Digilent Analog Parts Kit
- Digital multimeter (optional)

### Symbol Key:

-  Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
-  Analysis; include principle results of analysis in laboratory report.
-  Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
-  Record data in your lab notebook.

### General Discussion:

This portion of the lab assignment concerns the circuit shown in Fig. 1 below. A power supply is used to apply the 5V voltage difference. We wish to determine the power dissipated by the 4.7K $\Omega$  resistor.

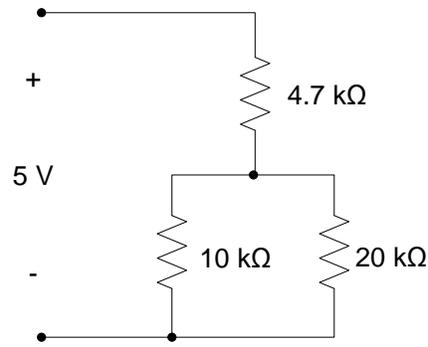


Figure 1. Circuit schematic.

### Pre-lab:

**ANALYSIS**

Analyze the circuit of Fig. 1 to estimate the power dissipated by the 4.7kΩ resistor.

### Lab Procedures:

**DATA**

1. Construct the circuit of Fig. 1. Measure and record the actual resistance values used in your circuit. Measure the parameters (voltages and/or currents) necessary to determine the power dissipated by the 4.7kΩ resistor. Determine the power dissipated by the 4.7kΩ resistor.

**ANALYSIS**

Compare the measured power with your estimate from the pre-lab. Comment on any differences between the estimated and measured values.

**DEMO**

2. Demonstrate operation of your circuit to the Teaching Assistant. Have the TA initial the appropriate page(s) of your lab notebook and the lab checklist.

## Real Analog Chapter 1: Lab Worksheets

### 1.4.2: Resistive Network Power Dissipation (20 points total)

1. Expected power dissipated by 4.7K $\Omega$  resistor (pre-lab analysis). (5 pts)
2. Measured power dissipated by 4.7K $\Omega$  resistor. Provide a circuit schematic and include all measurements taken: actual resistance values, voltages/currents and power calculation. (10 pts)
3. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuits' operation. (5 pts)

TA Initials: \_\_\_\_\_

## Real Analog Chapter 1: Lab Projects

### 1.4.3: Input Resistance

An extremely important circuit characteristic – the circuit’s input resistance – will be determined using Kirchoff’s laws. The input resistance of a circuit can be a significant design parameter; if the circuit is later incorporated in part of a larger overall system, the circuit’s input resistance can have a significant effect on the behavior of the overall system.

#### Before beginning this lab, you should be able to:

- State Ohm’s law from memory
- Use a digital multimeter to measure resistance, voltage, and current (Lab 1.1, 1.2.1)
- Use the Analog Discovery 2’s arbitrary waveform generator (AWG) to apply constant voltages to a circuit (Lab 1.2.2)
- Use the Analog Discovery 2 voltmeter to measure a constant voltage (Lab 1.2.1)
- Use color codes on resistors to determine the resistor’s nominal resistance
- Use KVL and KCL for circuit analysis

#### After completing this lab, you should be able to:

- Estimate the input resistance of a resistor network from measured voltage-current characteristics

#### This lab exercise requires:

- Analog Discovery 2
- Digilent Analog Parts Kit
- Digital multimeter

#### Symbol Key:

-  Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
-  Analysis; include principle results of analysis in laboratory report.
-  Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
-  Record data in your lab notebook.

#### General Discussion:

The *input resistance* of a circuit is the effective resistance that a source (or another stage of an overall system) will see when it is connected to the circuit. This value can be an extremely important design parameter for a circuit. In this part of the lab assignment, we will determine the input resistance of the circuit shown in Fig. 1.

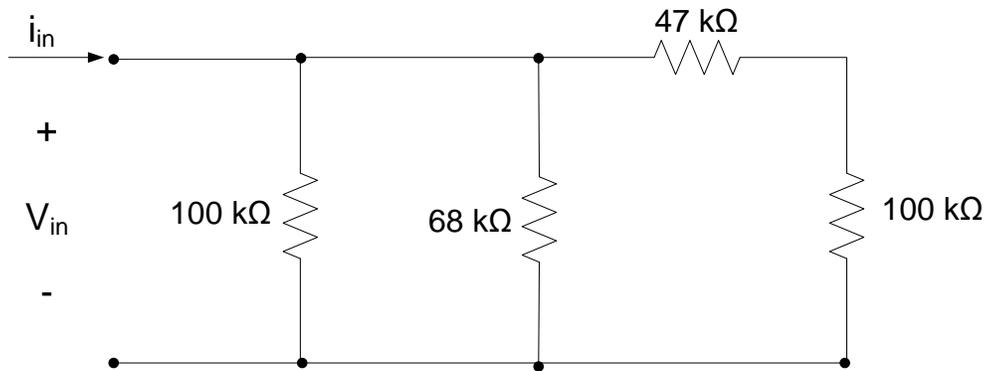


Figure 1. Circuit schematic.

**Pre-lab:****ANALYSIS**

Determine the relationship between  $V_{in}$  and  $i_{in}$  for the circuit shown in Fig. 1. Defining the input resistance according to Ohm's law as  $R_{in} = \frac{V_{in}}{i_{in}}$ , determine the input resistance of the circuit of Fig. 1.

**Lab Procedures:****DATA**

1. Construct the circuit of Fig. 1. Measure and record the actual resistance of all resistors used in your circuit. Use a variable voltage supply (the arbitrary waveform generator of the Analog Discovery 2 will work) to apply  $V_{in}$  and  $i_{in}$  to the circuit. For at least five different values of  $V_{in}$  between 0V and 5V, measure  $V_{in}$  and  $i_{in}$ .

**ANALYSIS**

2. Plot the voltage-current characteristic for the five points determined in part 1. Estimate the input resistance of the circuit from the slope of the voltage-current characteristic. (Feel free to draw your own best-fit line to the data points, without performing a least-squares best fit.)

**ANALYSIS**

3. Calculate the input resistance for each combination of  $V_{in}$  and  $i_{in}$  measured in part 1. Determine the average input resistance over the five measurements and the standard deviation of the measurements. Compare this value of input resistance to the one determined in part 2.

**DEMO**

4. Demonstrate operation of your circuits to the Teaching Assistant. Have the TA initial the appropriate page(s) of your lab notebook and the lab checklist.

## Real Analog Chapter 1: Lab Worksheets

### 1.4.3: Circuit Input Resistance (25 points total)

1. Estimated relationship between  $V_{in}$  and  $i_{in}$  from prelab. Estimated input resistance for circuit (4 pts)
2. Circuit schematic with measured resistance values implemented in circuit (2 pts)
3. Attach to this worksheet a plot of the measured voltage-current data. Your plot should include the five measured points and a straight line approximating the data. In the space below, provide your estimate of the input resistance of the circuit, based on these data. (8 pts)
4. Attach to this worksheet a table including (at least) four combinations of  $V_{in}$  and  $i_{in}$  applied to circuit, calculated input resistances for each combination, and the average input resistance over all measured values. In the space below, comment on the agreement between the input resistance determined in parts 3 and 4. (6 pts)
5. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuits' operation. (5 pts)

TA Initials: \_\_\_\_\_

## Real Analog Chapter 1: Lab Projects

### 1.4.4: Temperature Measurement System

This lab assignment also includes our first design-related task: we will design a circuit whose output voltage provides a crude temperature measurement. A thermistor – a device whose resistance changes with temperature – is used to sense the temperature. We will create an electrical circuit which uses this resistance change to output a voltage which indicates the temperature of the thermistor.

#### Before beginning this lab, you should be able to:

- State Ohm’s law from memory
- Use a digital multimeter to measure resistance, voltage, and current (Lab 1.1, 1.2.1)
- Use the Analog Discovery 2’s arbitrary waveform generator (AWG) to apply constant voltages to a circuit (Lab 1.2.2)
- Use the Analog Discovery 2 voltmeter to measure a constant voltage (Lab 1.2.1)
- Use color codes on resistors to determine the resistor’s nominal resistance
- Use KVL and KCL for circuit analysis

#### After completing this lab, you should be able to:

- Design a thermistor-based circuit to measure temperature

#### This lab exercise requires:

- Analog Discovery 2 module
- Digilent Analog Parts Kit
- Digital multimeter (optional)

#### Symbol Key:

- DEMO** Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
- ANALYSIS** Analysis; include principle results of analysis in laboratory report.
- SIM** Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
- DATA** Record data in your lab notebook.

#### General Discussion:

In this portion of the lab assignment, we will design and construct a temperature measurement system. The system will use a *thermistor* to detect temperature changes. A thermistor is a device whose electrical resistance changes as a function of the temperature of the thermistor. The thermistor we will use, for example, has a temperature-resistance curve approximately as shown in Fig. 1. It is worthwhile noting that the relationship between temperature and resistance is not linear. However, during the design process it is common to approximate the data as a straight line – at least over some range of temperatures.

### Related Material:

Thermistors are classified as NTC (Negative Temperature Coefficient) or PTC (Positive Temperature Coefficient) depending on whether their resistance decreases or increases with temperature. Thermistor specifications also include their nominal resistance at some temperature. The thermocouple with the temperature-resistance curve shown in Figure 2 is specified as an “NTC 10K $\Omega$  @ 25°C” thermistor. Additional information about thermistors can be found in the backup information provided as a companion document on to this lab.

It can be seen from Fig. 1 that the temperature can be inferred from the resistance of the thermistor. However, resistance is not a common quantity to use to represent a physical parameter – it is much more usual to use voltage to represent the parameter. (Voltages are generally easier to use than resistance to represent information. For example, digital logic circuits generally operate based on voltages applied to them.) We will design a circuit which outputs a voltage, from which the temperature of the thermistor can be inferred.

Our design criteria are as follows: Design a circuit like that shown in Fig. 2, containing a thermistor as one of the resistances, which satisfies the following specifications:

1. +5V input voltage to the system
2. Output voltage varies by a minimum of 0.5V over a temperature range of 25°C to 37°C.
3. Output voltage must increase as temperature increases

The selected temperature range provided in the design requirements corresponds (approximately) to the change between room temperature and human body temperature. We can thus check our temperature measurement system by measuring the output voltage when the thermistor is at room temperature and then changing the temperature by firmly holding the thermistor between two fingers. The voltage should increase by at least 0.5V as a result of this temperature change.

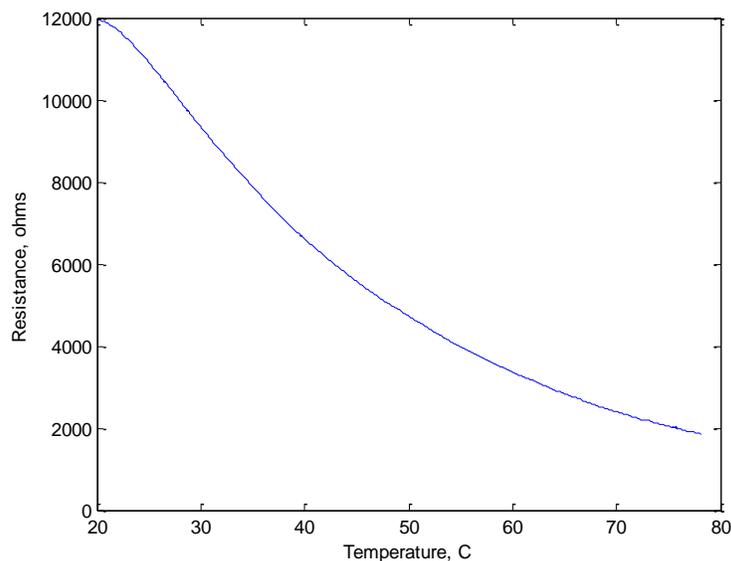


Figure 1. Thermistor temperature-resistance characteristic.

## Pre-lab:

In the circuit of Fig. 2, the resistance  $R_{TH}$  is the variable resistance of the thermistor. (The arrow through the resistor symbol typically means that the resistance is not necessarily constant.) The voltage  $v_{out}$  is the voltage that we will use to indicate temperature. The 5V input voltage is applied across the two resistors as shown. The design problem is to choose a value for  $R$  so that  $v_{out}$  increases by a minimum of 0.5V over a temperature range of 25°C to 37°C. To do this,

### ANALYSIS

1. Analyze the circuit of Fig. 2 to determine  $v_{out}$  as a function of  $R_{TH}$  and  $R$ .
2. Verify that  $v_{out}$  increases as temperature increases (or, equivalently, as  $R_{TH}$  decreases)
3. Using the temperature-resistance curve provided in Fig. 1, choose a value for  $R$  such that  $v_{out}$  changes by at least 0.5V over the specified temperature difference.

In your lab notebook, be sure to include the results of your analyses, including the equation(s) governing the system, your desired value for  $R$ , your expected output voltage change over the specified temperature range, and your expected output voltage at room temperature.

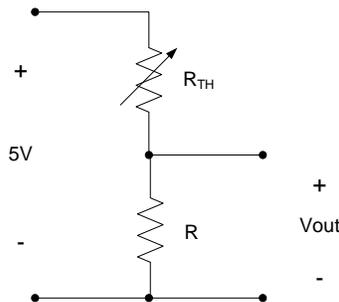


Figure 2. Temperature measurement circuit schematic.

### Note on the design process:

In general, design is an *iterative* process. As new information becomes available, the design is revised to ensure that the design requirements are met or the design requirements are revised to ensure that they are realistic. Since this is our first experience with design, we will not iterate on the preliminary design from the pre-lab. Our pre-lab analysis, however, still provides valuable information which can be used to discuss our final design performance relative to system requirements. For example, we may claim that the design requirements were not met because our thermistor did not have the sensitivity provided in Fig. 1 or that our fingertips did not apply the desired 37°C temperature. Either way, we have a reason why we did not meet the design requirements – this can be important when discussing your system performance relative to the design requirements with your customer!

### Lab Procedures:

Implement and test your design from the pre-lab. It is suggested that you perform at least the following steps when implementing your design:

### DATA

1. Check the resistance variation of your particular thermistor, over the specified temperature variation. To do this, measure and record the resistance of the thermistor at room temperature. Then hold the thermistor firmly between two fingers and record the resulting resistance value. This provides the resistance variation over the desired temperature range.

DATA

DATA

ANALYSIS

DEMO

2. Check your preliminary design generated in the pre-lab with the measured resistance variation determined in step 1 above. It is entirely possible that this step may indicate that your design does not meet the design requirements, however, do not modify your design to meet the design requirements.
3. Implement your design. Be sure to record actual resistance values for any fixed resistors used in your design.
4. Measure the voltage response to the specified temperature change. Record the output voltage at the high and low temperature conditions. Discuss your circuit's performance relative to the design specifications. (e.g. Were requirements met? If not, why?)
5. Calculate the percent error between the expected performance (based on your pre-lab analysis) and the measured performance.
6. Demonstrate operation of your circuit to the Teaching Assistant. Have the TA initial the appropriate page(s) of your lab notebook and the lab checklist.

### Post-Lab Exercises:

Using the thermistor provided in this lab assignment, design a temperature measurement system which meets the following requirements:

1. The output voltage must increase as temperature increases.
2. The output sensitivity of the device must be at least 0.1 V/°C (or 100 mV per degree Centigrade of temperature change).

Analyze the design to show that the requirements are met. You do not need to implement your design.

## 1.4.4 Background Information: Resistive Temperature Sensors

Most people have an intuitive concept of temperature, heat or cold is sensed directly via the sense of touch. Temperature is also an extremely important quantity from the standpoint of engineering and physics, in these disciplines we require a more technical definition of temperature. Temperature, engineering terms, provides a measure of the amount of internal kinetic energy that the molecules in an object have. This property governs a number of important physical phenomena, including:

- *Heat transfer.* Heat tends to flow from regions of high temperature to low temperature. The field of heat transfer, therefore, relies heavily upon determining the temperature distribution in a material.
- *Thermodynamics.* Temperature is a primary parameter which determines the thermodynamic state of liquids and gasses. For example, the amount of propane which can safely be stored in a given tank is a strong function of the temperatures that the gas will be subjected to; if the temperature exceeds specified levels, the tank may rupture – with potentially disastrous consequences.

Due to the extreme importance of temperature in many engineering analysis, temperature is one of the most commonly measured physical parameters. Many approaches have been developed to measure temperature. In this module, we will discuss temperature measurement processes which rely upon temperature-induced changes in electrical resistance. Measuring electrical resistance under these circumstances, allows us to infer temperature. Electrical resistance approaches to temperature measurement are extremely popular, since the signals resulting from these types of measurements are readily recorded, transmitted and analyzed.

Two relatively common approaches toward electrical resistance-based temperature measurements are resistive temperature detectors (RTDs) and thermistors. Thermistors are somewhat more widely used than RTDs, primarily because thermistors are considerably less expensive than RTDs. This module is by no means intended to be a comprehensive discussion of how to use either RTDs or thermocouples. We restrict ourselves to a presentation of the principle concepts relative to the operation of these types of temperature measurement device, along with some general information about how these instruments are used in practice.

**Before beginning this module, you should be able to:**

- Perform basic algebra
- Analyze resistive electrical circuits

**After completing this module, you should be able to:**

- Write an approximate linear relationship governing the temperature-resistance characteristic of RTDs
- Sketch an electrical circuit to convert resistance change to voltage change
- Write a nonlinear temperature-resistance relationship for thermistors
- List two sources of error commonly arising in resistive temperature measurements

**References:**

1. Holman, J.P., Experimental Methods for Engineers, 3<sup>rd</sup> edition, McGraw-Hill Book Company, 1978.

**Resistive Temperature Detectors:**

Many materials, especially metals, have resistivities which increase as temperature increases. Resistive Temperature Detectors (RTDs) utilize this property to provide a temperature measurement. RTDs can be made by winding a thin wire into a coil or depositing a thin film of metal on a plastic or ceramic substrate. This wire or film is then exposed to the temperature being measured; as the temperature increases, the resistivity of the wire or film increases, and the overall electrical resistance of the device increases. Platinum is commonly used in RTDs, since its resistivity varies linearly with temperature over a wide temperature range. RTDs have a number of advantages over other types of temperature measurements: their response is extremely linear over a wide temperature range and their readings are extremely stable. Among their disadvantages is their high cost: the high cost of platinum makes RTDs relatively expensive compared to other temperature measurement devices.

As noted above, the resistance vs. temperature relation is approximately linear. This relationship is commonly defined in terms of a linear temperature coefficient of resistance,  $\alpha$ , as follows:

$$R = R_{nom} + \alpha R_{nom} (T - T_{nom}) \quad \text{Eq. 1}$$

where  $R$  is the resistance at the temperature  $T$  and  $R_{nom}$  is the resistance at some nominal (reference) temperature  $T_{nom}$  (often taken to be some convenient temperature such as 0°C or 0°F). This can be rearranged to determine the temperature from the measured resistance:

$$T = T_{nom} + \left( \frac{R - R_{nom}}{\alpha \cdot R_{nom}} \right) \quad \text{Eq. 2}$$

The temperature coefficient of resistance for platinum is approximately 0.00392 (reference [1]), though this value depends upon the grade of platinum used. It is always important to determine  $\alpha$  accurately for the particular RTD being used, especially if large ranges in temperature are to be measured.

Over very large temperature ranges, the linear relationship of equation (1) between resistance and temperature may not be appropriate. In these cases, a quadratic relationship is generally employed:

$$R = R_{nom} [1 + a(T - T_{nom}) + b(T - T_{nom})^2] \quad \text{Eq. 3}$$

where  $R$ ,  $R_{nom}$ ,  $T$ , and  $T_{nom}$  are as defined above.  $a$  and  $b$  are experimentally determined constants.

## RTD Signal Conditioning:

The resistance changes of RTDs are usually converted to a voltage change by electrical circuitry. Voltage levels have the advantage over resistances in that voltages can be readily transmitted, recorded, and used for control purposes. The resistance-to-voltage conversion is normally performed by a Wheatstone bridge circuit; a very simple Wheatstone bridge circuit is shown in Fig. 1. In the bridge circuit of Fig. 1, the RTD resistance is represented by a resistor with resistance  $R+\Delta R$ . The resistance  $R$  is the nominal resistance of the RTD, the resistance  $\Delta R$  represents the change in resistance of the strain gage due to some temperature change. There are three fixed resistors in the bridge circuit of Fig. 1; these resistors all have the same resistance as the nominal resistance of the RTD. A fixed voltage,  $V_s$ , is applied to the circuit as shown. The output voltage,  $v_{ab}$ , is the voltage which is used to represent the change in resistance of the RTD. Without proof, we claim that the output voltage is:

$$v_{ab} = \frac{\Delta R}{2(2R + \Delta R)} \cdot V_s \quad \text{Eq. 4}$$

if  $\Delta R \ll R$ , which is the usual case for RTDs, this simplifies to:

$$v_{ab} \approx \frac{V_s}{4R} \Delta R \quad \text{Eq. 5}$$

and the output voltage is linearly related to the change in resistance of the RTD. Since the RTD resistance is linearly related to the temperature, the output voltage is thus a linear function of temperature.

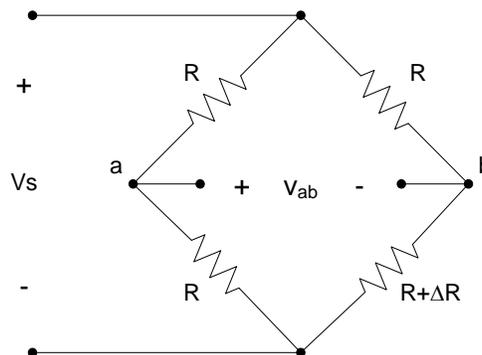


Figure 1. Wheatstone bridge circuit.

## Thermistors:

Thermistors are semiconductor devices whose resistance, like RTDs, varies with temperature. In fact, thermistors can be considered to be a special case of an RTD. Thermistors, however, have a number of properties which distinguish them from RTDs as presented in the previous section. These differences are outlined below:

- The resistance of thermistors commonly decreases as temperature increases. Thermistors with this type of resistance-temperature characteristic are called Negative Temperature Coefficient (NTC) thermistors. Thermistors whose resistance increases with temperature are available; these are called Positive Temperature Coefficient (PTC) thermistors.
- Thermistors generally do not exhibit a linear relationship between temperature and resistance. The relationship between temperature and resistance for a thermistor is commonly modeled by an exponential relation. Reference [1], for example, provides the following resistance-temperature relationship:

$$R = R_{nom} e^{\left[ \beta \left( \frac{1}{T} - \frac{1}{T_{nom}} \right) \right]}$$

Eq. 6

where  $R$  is the resistance at the temperature  $T$  and  $R_{nom}$  is the resistance at some reference temperature,  $T_{nom}$ . This nonlinear resistance-temperature relation is a significant drawback to the use of thermistors; however, the thermistors' advantages often outweigh this disadvantage, making thermistors one of the most common choices for temperature measurement devices.

- Because of the thermistors' nonlinear resistance-temperature relation, thermistors are useful over smaller temperature ranges than RTDs.
- Thermistors generally have a much higher sensitivity than RTDs – the resistance change resulting from a given temperature variation is considerably larger for a thermistor than for an RTD.
- Thermistors generally have a faster temperature response than RTDs. This makes them a better choice for time-varying (transient) temperature changes than RTDs.
- Thermistors are less expensive than RTDs.

### Thermistor Signal Conditioning:

A bridge circuit like that of Fig. 1, with the thermistor used for the  $R+\Delta R$  resistance, is often used to convert the thermistor's change in resistance to a voltage change. Unlike the RTD, however, the output voltage vs. temperature relation is generally nonlinear. Also, if an NTC thermistor is used, the output voltage of the circuit of Fig. 1 will decrease as temperature increases. Modifications to the bridge circuit of Fig. 1 can alleviate both of these drawbacks – bridge circuits using a second thermistor can help provide a more linear response, while switching the polarity of the output voltage will result in an output voltage which increases with increasing temperature.

### Issues associated with resistive temperature measurement:

Several issues must be considered when using resistive temperature measurement devices. When using a bridge circuit like that shown in Fig. 1, the externally supplied voltage  $V_s$  will result in some current flowing through the resistive temperature measurement device. This current will induce heating of the resistive temperature measurement device, due to the resistive power dissipation in the device. Care must be taken that this so-called *self heating* of the device does not introduce excessive heating in the device itself, which can invalidate the temperature measurement.

Another common source of error in electrical resistance temperature measurements is the resistance of the leads connecting the device to the bridge circuit used to convert the resistance change to a voltage change. Alternate bridge circuits can compensate for this effect.



4. In the space below, provide your measured circuit voltage response over desired temperature range. Also discuss your results relative to design requirements, including the percent difference between expected and measured values. (8 pts)

5. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuits' operation. (5 pts)

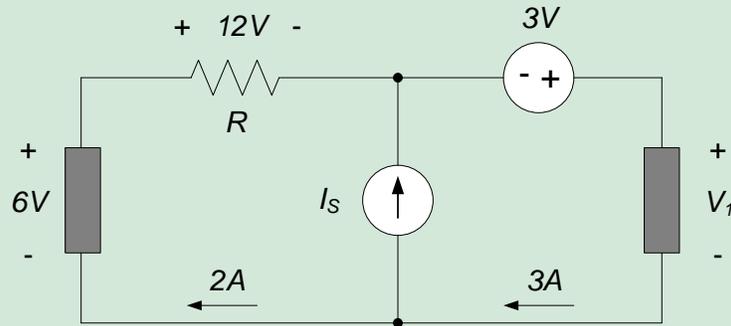
**TA Initials:** \_\_\_\_\_

6. Post-lab Exercise: Temperature Measurement System Re-Design (10 pts total)

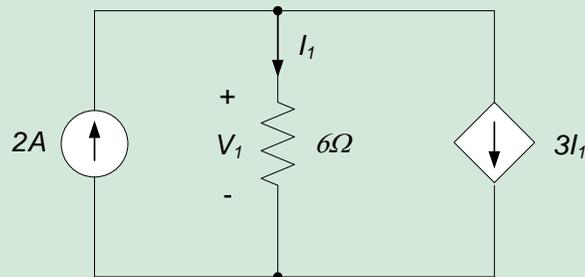
Provide below a schematic of your proposed design, including any voltage supply requirements and component values. Provide a brief analysis of your design's performance, along with a comparison of the performance vs. the design specification (e.g. does your design meet the design requirements?)

# Real Analog Chapter 1: Homework

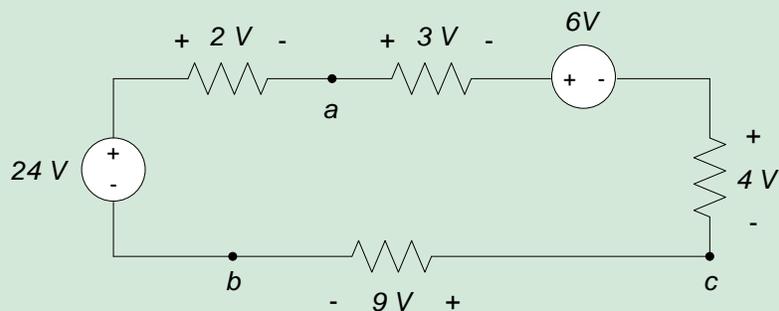
- 1.1 For the circuit below, determine
- the voltage,  $V_1$
  - the resistance,  $R$
  - the current,  $I_s$
  - the power (absorbed or supplied) by the current source
  - the power (absorbed or supplied) by the 3V voltage source



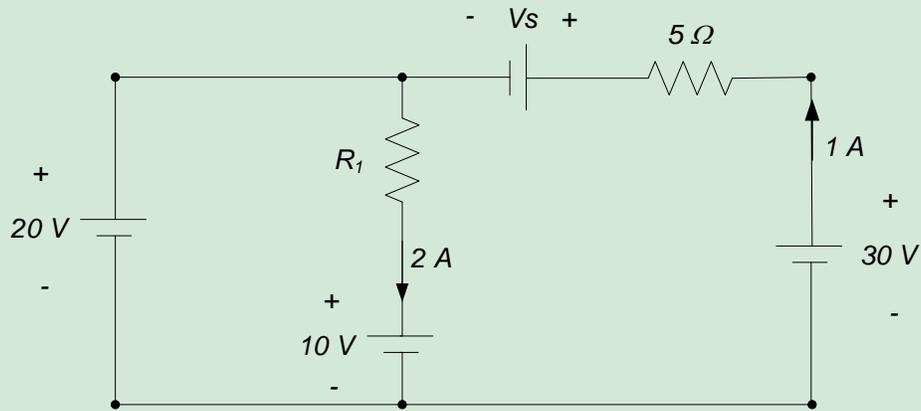
- 1.2 Find  $I_1$  and  $V_1$  in the circuit below.



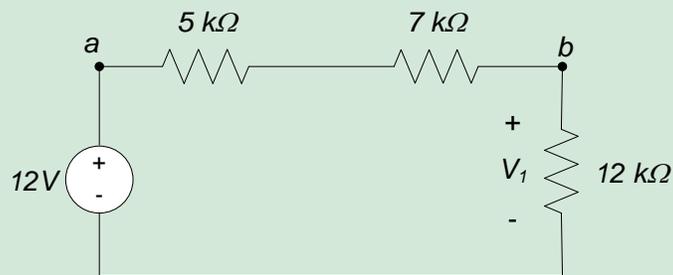
- 1.3 In the circuit below, determine  $v_{ab}$ ,  $v_{ac}$ , and  $v_{cb}$ .



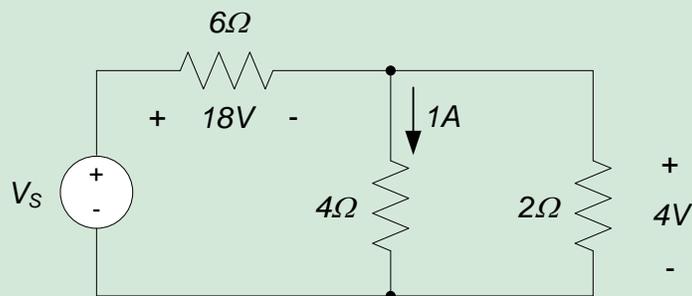
- 1.4 For the circuit shown, find:
- $V_s$
  - $R_1$
  - Power absorbed or generated by the 10V source. State whether the power is absorbed or generated.



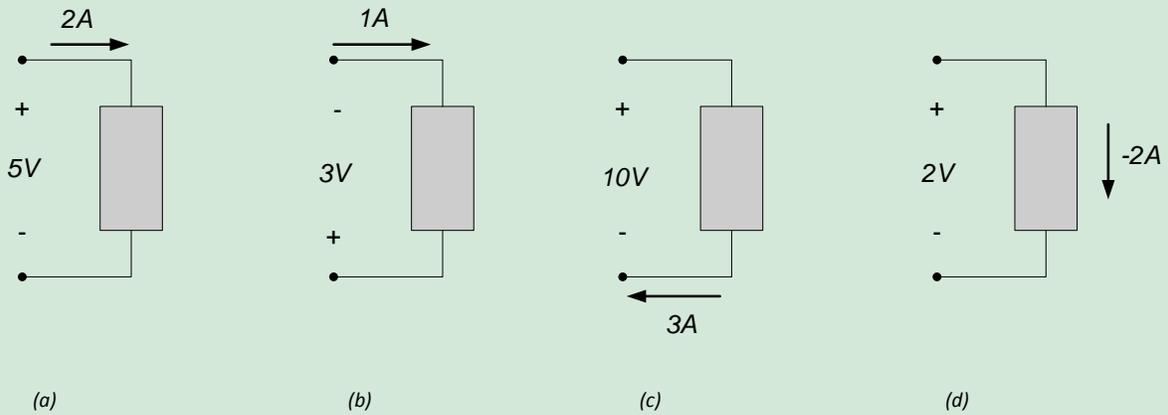
1.5 Find  $V_{ab}$  and  $V_1$  in the circuit below.



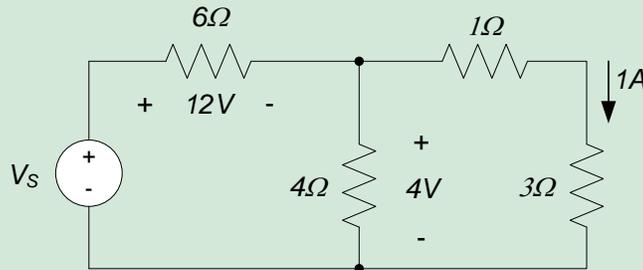
- 1.6 For the circuit below, determine
- the power dissipated by the  $6\Omega$  resistor
  - the voltage across the  $4\Omega$  resistor
  - the current through the  $2\Omega$  resistor



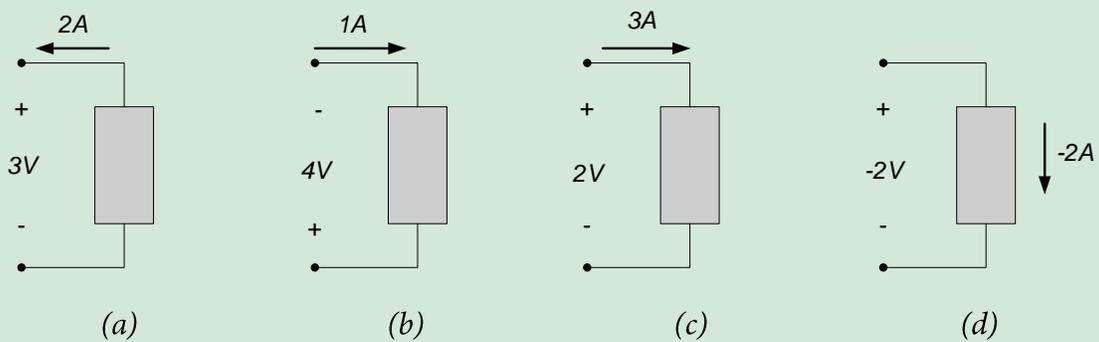
1.7 For the circuit elements below, determine the power generated or absorbed by the element. State whether the power is generated or absorbed.



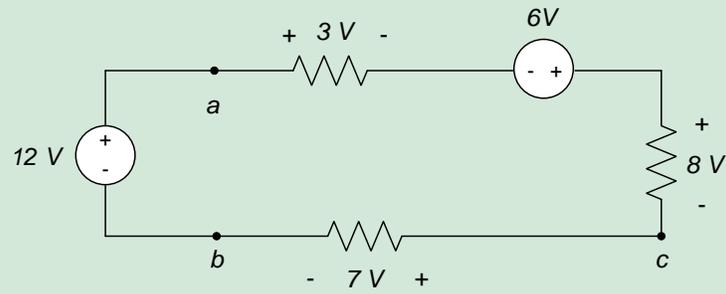
- 1.8 For the circuit below, determine
- the power dissipated by the  $6\Omega$  resistor
  - the voltage across the  $3\Omega$  resistor
  - the current through the  $4\Omega$  resistor



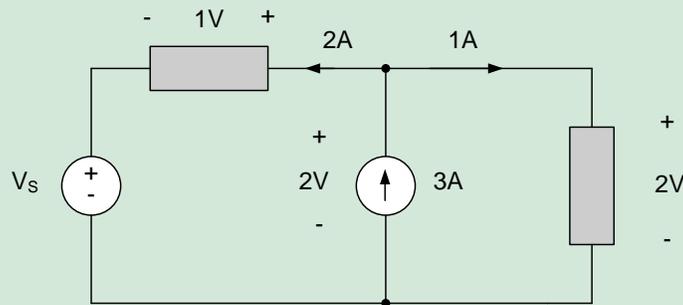
- 1.9 For the circuit elements below, determine the power generated or absorbed by the element. State whether the power is generated or absorbed.



- 1.10 In the circuit below, determine  $v_{ab}$ ,  $v_{ac}$ , and  $v_{cb}$ .

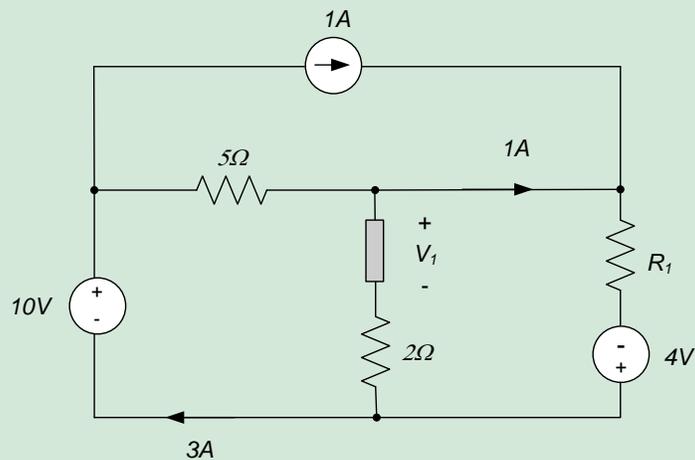


1.11 In the circuit below, determine the power (absorbed or generated) by the voltage source  $V_s$ .



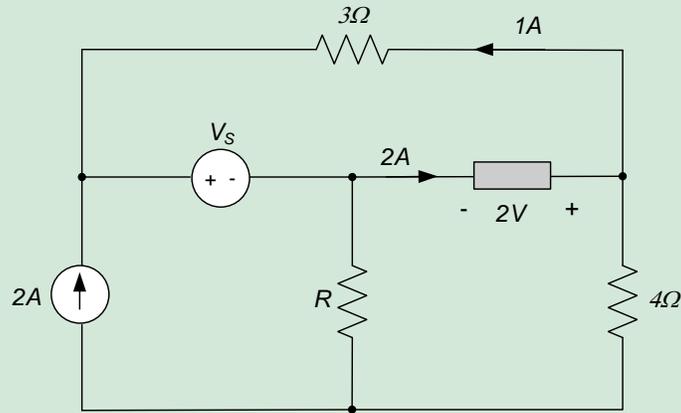
1.12 For the circuit shown, find:

- a)  $V_1$
- b)  $R_1$
- c) The power (generated or absorbed) by the 1A source

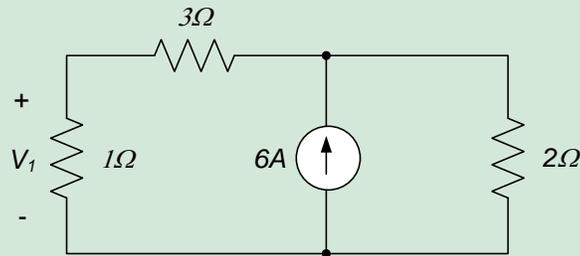


1.13 For the circuit shown, find:

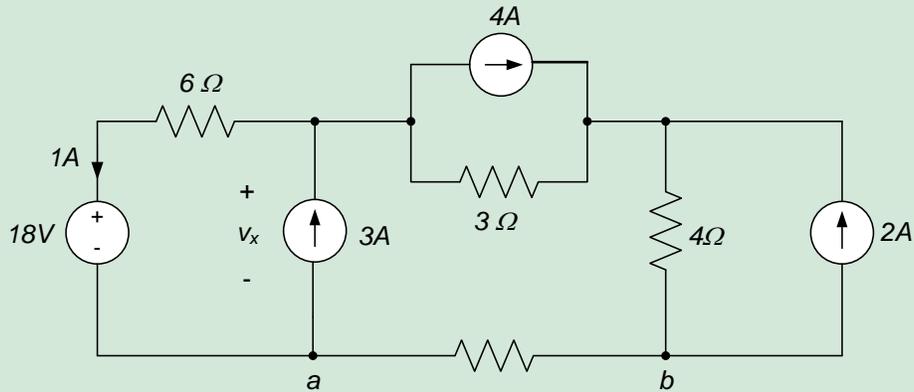
- a)  $V_s$
- b)  $R$
- c) The power (generated or absorbed) by the 2A source



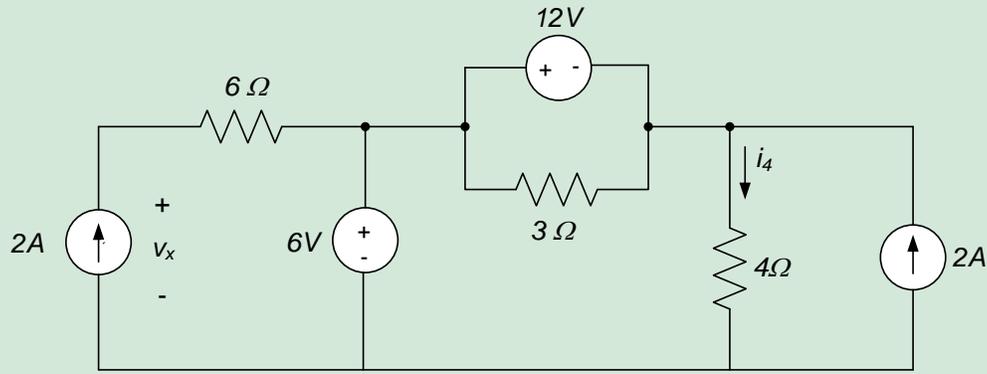
1.14 Write a set of equations from which you can determine the current  $V_1$  in the circuit below. You do not need to solve the equations.



1.15 For the circuit below, determine the voltage  $v_x$ , the voltage  $v_{ab}$ , and the power (generated or absorbed) by the 3A source.



1.16 For the circuit shown, find the voltage  $v_x$  and the current  $i_4$ .



## Chapter 2: Circuit Reduction

### 2 Introduction and Chapter Objectives

In Chapter 1, we presented Kirchhoff's laws (which govern the interaction between circuit elements) and Ohm's law (which governs the voltage-current relationships for resistors). These analytical tools provide us with the ability to analyze any circuit containing only resistors and ideal power supplies. However, we also saw in Chapter 1 that a circuit analysis, which relies strictly on a brute-force application of these tools can become complex rapidly - we essentially must use as our unknowns the voltage differences across all resistors and the currents through all resistors. This generally results in a large number of unknowns and a correspondingly large number of equations, which must be written and solved in order to analyze any but the simplest circuit.

In the next few chapters, we will still apply Kirchhoff's laws and Ohm's law in our circuit analysis, but we will focus on improving the efficiency of our analyses. Typically, this improvement in efficiency is achieved by reducing the number of unknowns in the circuit, which reduces the number of equations, which must be written to describe the circuit's operation.

In this chapter, we introduce analysis methods based on *circuit reduction*. Circuit reduction consists of combining resistances in a circuit to a smaller number of resistors, which are (in some sense) equivalent to the original resistive network. Reducing the number of resistors, of course, reduces the number of unknowns in a circuit.

We begin our discussion of circuit reduction techniques by presenting two specific, but very useful, concepts: *Series* and *parallel* resistors. These concepts will lead us to *voltage* and *current divider* formulas. We then consider reduction of more general circuits, which typically corresponds to identifying multiple sets of series and parallel resistances in a complex resistive network. This chapter then concludes with two important examples of the application of circuit reduction techniques: the analysis of *non-ideal power sources* and *non-ideal measurement* devices; without an understanding of these devices, it is impossible to build practical circuits or understand the consequences of a voltage or current measurement.

#### After Completing this Chapter, You Should be Able to:

- Identify series and parallel combinations of circuit elements
- Determine the equivalent resistance of series resistor combinations
- Determine the equivalent resistance of parallel resistor combinations
- State voltage and current divider relationships from memory
- Determine the equivalent resistance of electrical circuits consisting of series and parallel combinations of resistors
- Sketch equivalent circuits for non-ideal voltage and current meters
- Analyze circuits containing non-ideal voltage or current sources
- Determine the effect of non-ideal meters on the parameter being measured

### 2.1 Series Circuit Elements and Voltage Division

There are a number of common circuit element combinations that are quite easily analyzed. These “special cases” are worth noting since many complicated circuits contain these circuit combinations as sub-circuits. Recognizing these sub-circuits and analyzing them appropriately can significantly simplify the analysis of a circuit.

This chapter emphasizes two important circuit element combinations: elements in series and elements in parallel. Also discussed is the use of these circuit element combinations to reduce the complexity of a circuit's analysis.

## 2.1.1 Series Connections

Circuit elements are said to be connected in *series* if all of the elements carry the same current. An example of two circuit elements connected in series is shown in Fig. 2.1. Applying KCL at node a and taking currents out of the node as positive we see that:

$$-i_1 + i_2 = 0$$

Or

$$i_1 = i_2 \tag{Eq. 2.1}$$

Equation (2.1) is a direct outcome of the fact that the (single) node a in Fig. 2.1 interconnects only two elements - there are no other elements connected to this node through which current can be diverted. This observation is so apparent (in many cases<sup>3</sup>) that equation (2.1) is generally written by inspection for series elements such as those shown in Fig. 2.1 without explicitly writing KCL.

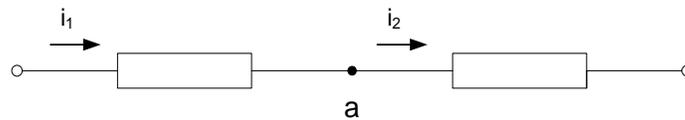


Figure 2.1. Circuit elements connected in series.

When resistors are connected in series, a simplification of the circuit is possible. Consider the resistive circuit shown in Fig. 2.2(a). Since the resistors are in series, they both carry the same current. Ohm's law gives:

$$v_1 = R_1 i$$

$$v_2 = R_2 i \tag{Eq. 2.2}$$

Applying KVL around the loop:

$$-v + v_1 + v_2 = 0 \Rightarrow v = v_1 + v_2 \tag{Eq. 2.3}$$

Substituting equations (2.2) into equation (2.3) and solving for the current  $i$  results in:

$$i = \frac{v}{R_1 + R_2} \tag{Eq. 2.4}$$

Now consider the circuit of Fig. 2.2(b). Application of Ohm's law to this circuit and solution for the current  $i$  gives:

$$i = \frac{v}{R_{eq}} \tag{Eq. 2.5}$$

<sup>3</sup> If there is any doubt whether the elements are in series, apply KCL! Assuming elements are in series which are not in series can have disastrous consequences.

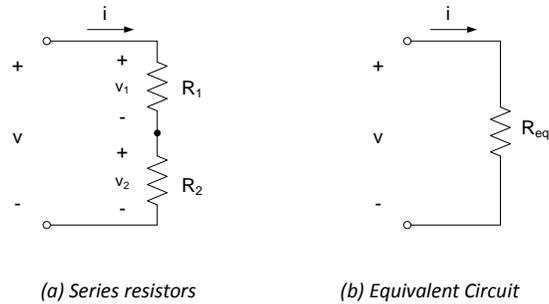


Figure 2.2. Series resistors and equivalent circuit.

Comparing equation (2.4) with equation (2.5), we can see that the circuits of Figs. 2.2(a) and 2.2(b) are indistinguishable if we select:

$$R_{eq} = R_1 + R_2 \quad \text{Eq. 2.6}$$

Figs. 2.2(a) and 2.2(b) are called *equivalent circuits* if the equivalent resistance of Fig. 2.2(b) is chosen as shown in equation (2.6).  $R_{eq}$  of equation (2.6) is called the equivalent resistance of the series combination of resistors  $R_1$  and  $R_2$ .

This result can be generalized to a series combination of  $N$  resistances as follows:

A series combination of  $N$  resistors  $R_1, R_2, \dots, R_N$  can be replaced with a single equivalent resistance  $R_{eq} = R_1 + R_2 + \dots + R_N$ . The equivalent circuit can be analyzed to determine the current through the series combination of resistors.

## 2.1.2 Voltage Division

Combining equations (2.2) with equation (2.4) results in the following expressions for  $V_i$  and  $v_2$ :

$$v_1 = \frac{R_1}{R_1 + R_2} v \quad \text{Eq. 2.7}$$

$$v_2 = \frac{R_2}{R_1 + R_2} v \quad \text{Eq. 2.8}$$

These results are commonly called *voltage divider* relationships, because they state that the total voltage drop across a series combination of resistors is divided among the individual resistors in the combination. The ratio of each individual resistor's voltage drop to the overall voltage drop is the same as the ratio of the individual resistance to the total resistance.

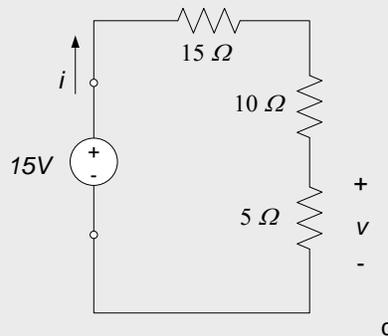
The above results can be generalized for a series combination of  $N$  resistance as follows:

The voltage drop across any resistor in a series combination of  $N$  resistances is proportional to the total voltage drop across the combination of resistors. The constant of proportionality is the same as the ratio of the individual resistor value to the total resistance of the series combination. For example, the voltage drop of the  $k^{\text{th}}$  resistance in a series combination of resistors given by:

$$v_k = \frac{R_k}{R_1 + R_2 + \dots + R_N} v \quad \text{Eq. 2.9}$$

### Example 2.1

For the circuit below, determine the voltage across the  $5\Omega$  resistor,  $v$ , the current supplied by the source,  $i$ , and the power supplied by the source.



The voltage across the  $5\Omega$  resistor can be determined from our voltage divider relationship:

$$v = \left[ \frac{5\Omega}{5\Omega + 15\Omega + 10\Omega} \right] \cdot 15V = \frac{5}{30} \cdot 15V = 2.5V$$

The current supplied by the source can be determined by dividing the total voltage by the equivalent resistance:

$$i = \frac{15V}{R_{eq}} = \frac{15V}{5\Omega + 15\Omega + 10\Omega} = \frac{15V}{30\Omega} = 0.5A$$

The power supplied by the source is the product of the source voltage and the source current:

$$P = iv = (0.5A)(15V) = 7.5W$$

We can double-check the consistency between the voltage  $v$  and the current  $i$  with Ohm's law. Applying Ohm's law to the  $5\Omega$  resistor, with a  $0.5A$  current, results in  $v = (5\Omega)(0.5A) = 2.5V$ , which agrees with the result obtained using the voltage divider relationship.

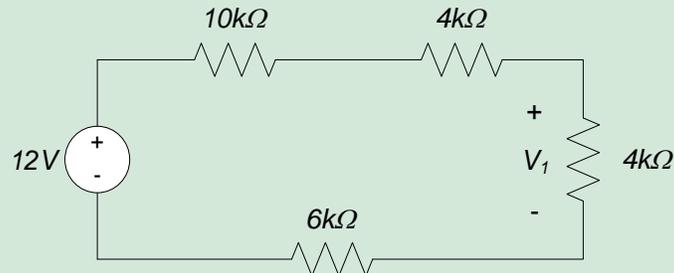
### Section Summary:

- If only two elements connect at a single node, the two elements are in *series*. A more general definition, however, is that circuit elements in series all share the same current - this definition allows us to determine series combinations that contain more than two elements. Identification of series circuit elements allows us to simplify our analysis, since there is a reduction in the number of unknowns: there is only a single unknown current for all series elements.
- A series combination of resistors can be replaced by a single *equivalent resistance*, if desired. The equivalent resistance is simply the sum of the individual resistances in the series combination. Therefore, a series combination of  $N$  resistors  $R_1, R_2, \dots, R_N$  can be replaced with a single equivalent resistance  $R_{eq} = R_1 + R_2 + \dots + R_N$ .
- If the total voltage difference across a set of series is known, the voltage differences across any individual resistor can be determined by the concept of *voltage division*. The term voltage division comes from the fact that the voltage drop across a series combination of resistors is divide among the individual resistors. The ratio between the voltage difference across a particular resistor and the total voltage difference is the same as the ratio between the resistance of that resistor and the total resistance of the combination. If  $v_k$  is the voltage across the  $k_{th}$  resistor, and  $R_{TOT}$  is the total resistance of the series combination, the mathematical statement of this concept is:

$$\frac{v_k}{v_{TOT}} = \frac{R_k}{R_{TOT}}$$

## 2.1 Exercises

- Determine the voltage  $V_1$  in the circuit below.



## 2.2 Parallel Circuit Elements and Current Division

Circuit elements are said to be connected in *parallel* if all of the elements share the same pair of nodes. An example of two circuit elements connected in parallel is shown in Fig. 2.3. Applying KVL around the loop of Fig. 2.3 results in:

$$v_1 = v_2 \quad \text{Eq. 2.10}$$

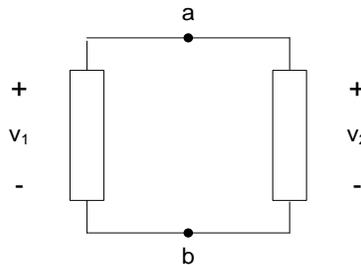


Figure 2.3. Parallel connection of circuit elements.

We can simplify circuits, which consist of resistors connected in parallel. Consider the resistive circuit shown in Fig. 2.4(a). The resistors are connected in parallel, so both resistors have a voltage difference of  $v$ . Ohm's law applied to each resistor results in:

$$i_1 = \frac{v}{R_1}$$

$$i_2 = \frac{v}{R_2} \quad \text{Eq. 2.11}$$

Applying KCL at node a:

$$i = i_1 + i_2 \quad \text{Eq. 2.12}$$

Substituting equations (2.11) into equation (2.12):

$$i = \left[ \frac{1}{R_1} + \frac{1}{R_2} \right] v \quad \text{Eq. 2.13}$$

Or

$$v = \frac{1}{\frac{1}{R_1} + \frac{1}{R_2}} \cdot i \quad \text{Eq. 2.14}$$

If we set  $R_{eq} = \frac{1}{\frac{1}{R_1} + \frac{1}{R_2}}$ , we can draw Fig. 2.4(b) as being equivalent to Fig. 2.4(a).

We can generalize this result for  $N$  parallel resistances:

A parallel combination of  $N$  resistors  $R_1, R_2, \dots, R_N$  can be replaced with a single equivalent resistance:

$$R_{eq} = \frac{1}{\frac{1}{R_1} + \frac{1}{R_2} + \dots + \frac{1}{R_N}} \quad \text{Eq. 2.15}$$

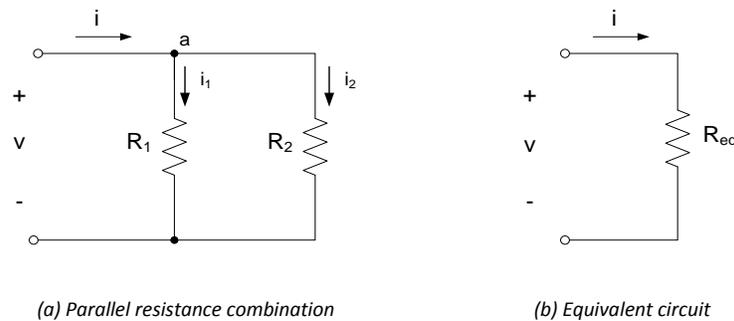


Figure 2.4. Parallel resistances and equivalent circuit.

For the special case of two parallel resistances,  $R_1$  and  $R_2$ , the equivalent resistance is commonly written as:

$$R_{eq} = \frac{R_1 R_2}{R_1 + R_2} \quad \text{Eq. 2.16}$$

This alternative way to calculate  $R_{eq}$  can be also used to calculate  $R_{eq}$  for larger numbers of parallel resistors since any number of resistors could be combined two at a time.

## 2.2.1 Current Division

Substituting equation (2.14) into equations (2.11) results in:

$$i_1 = \frac{1}{R_1} \cdot \frac{i}{\frac{1}{R_1} + \frac{1}{R_2}} \quad \text{Eq. 2.17}$$

Simplifying:

$$i_1 = \frac{R_2}{R_1 + R_2} i \quad \text{Eq. 2.18}$$

Likewise, for the current  $i_2$ :

$$i_2 = \frac{R_1}{R_1 + R_2} i \quad \text{Eq. 2.19}$$

Equations (2.18) and (2.19) are the current *divider relationships* for two parallel resistances, so called because the current into the parallel resistance combination is divided between the two resistors. The ratio of one resistor's current to the overall current in the same as the ratio of the other resistance to the total resistance.

The above results can be generalized for a series combination of  $N$  resistances. By Ohm's law,  $v = R_{eq} i$ .

Substituting our previous result for the equivalent resistance for a parallel combination of  $N$  resistors results in:

$$v = \frac{1}{\frac{1}{R_1} + \frac{1}{R_2} + \dots + \frac{1}{R_N}} \cdot i \quad \text{Eq. 2.20}$$

Since the voltage difference across all resistors is the same, the current through the  $k^{\text{th}}$  resistor is, by Ohm's law:

$$i_k = \frac{v}{R_k} \quad \text{Eq. 2.21}$$

Where  $R_k$  is the resistance of the  $k^{\text{th}}$  resistor. Combining equations (2.20) and (2.21) gives:

$$i_k = \frac{\frac{1}{R_k}}{\frac{1}{R_1} + \frac{1}{R_2} + \dots + \frac{1}{R_N}} \cdot i \quad \text{Eq. 2.22}$$

It is often more convenient to provide the generalized result of equation (2.20) in terms of the conductance of the individual resistors. Recall that the conductance is the reciprocal of the resistance,  $G = \frac{1}{R}$ . Thus, equation (2.22) can be re-expressed as follows:

The Current through any resistor in a parallel combination of  $N$  resistances is proportional to the total current into the combination of resistors. The constant of proportionality is the same as the ratio of the conductance of the individual resistor value to the total conductance of the parallel combination. For example, the current through the  $k^{\text{th}}$  resistance in a parallel combination of resistors is given by:

$$i_k = \frac{G_k}{G_1 + G_2 + \dots + G_N} i \quad \text{Eq. 2.23}$$

Where  $i$  is the total current through the parallel combination of resistors.

One final comment about notation: two parallel bars are commonly used as shorthand notation to indicate that two circuit elements are in parallel. For example, the notation  $R_1 \parallel R_2$  indicates that the resistors  $R_1$  and  $R_2$  are in parallel. The notation  $R_1 \parallel R_2$  is often used as shorthand notation for the equivalent resistance of the parallel resistance combination, in lieu of equation (2.16).

Double-checking results for parallel resistances:

- The equivalent resistance for a parallel combination of  $N$  resistors will always be less than the smallest resistance in the combination. In fact, the equivalent resistance will always obey the following inequalities:

$$\frac{R_{min}}{N} \leq R_{eq} \leq R_{min}$$

- Where  $R_{min}$  is the smallest resistance value in the parallel combination.
- In a parallel combination of resistances, the resistor with the smallest resistance will have the largest current and the resistor with the largest resistance will have the smallest current.

## Section Summary

- If several elements interconnect the same two nodes, the two elements are in *parallel*. A more general definition, however, is that circuit elements in parallel all share the same voltage difference. As with series circuit elements, identification of parallel circuit elements allows us to simplify our analysis, since there is a reduction in the number of unknowns: there is only a single unknown voltage difference for all of the parallel elements.
- A parallel combination of resistors can be replaced by a single *equivalent resistance*, if desired. The conductance of the parallel combination is simply the sum of the individual conductance of the parallel

resistors. Therefore, a parallel combination of  $N$  resistors  $R_1, R_2, \dots, R_N$  can be replaced with a single equivalent resistance:

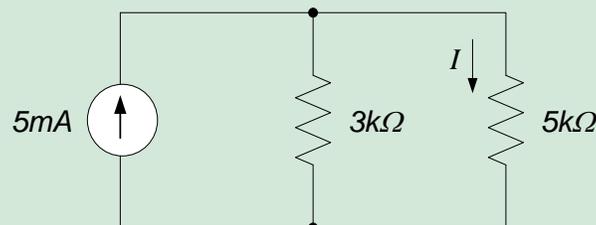
$$R_{eq} = \frac{1}{\frac{1}{R_1} + \frac{1}{R_2} + \dots + \frac{1}{R_N}}$$

- If the total current through a set of parallel resistors is known, the current through any individual resistor can be determined by the concept of *current division*. The term current division comes from the fact that the current through a parallel combination of resistors is divided among the individual resistors. The ratio between the current through a particular resistor and the total current is the same as the ratio between the conductance of that resistor and the total conductance of the combination. If  $i_k$  is the current through the  $k^{\text{th}}$  resistor,  $i_{TOT}$  is the total current through the parallel combination,  $G_k$  is the conductance of the  $k^{\text{th}}$  resistor, and  $G_{TOT}$  is the total conductance of the parallel combination, the mathematical statement of this concept is:

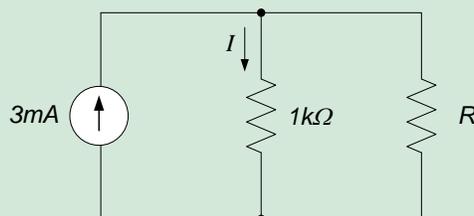
$$\frac{i_k}{i_{TOT}} = \frac{G_k}{G_{TOT}}$$

## 2.2 Exercises

1. Determine the value of  $I$  in the circuit below.



2. Determine the value of  $R$  in the circuit below which makes  $I=2\text{mA}$ .



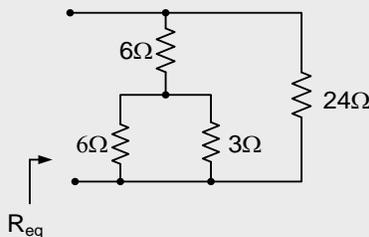
## 2.3 Circuit Reduction and Analysis

The previous results give us an ability to potentially simplify the analysis of some circuits. This simplification results if we can use *circuit reduction* techniques to convert a complicated circuit to a simpler, but equivalent, circuit which we can use to perform the necessary analysis. Circuit reduction is not always possible, but when it is applicable it can significantly simplify the analysis of a circuit.

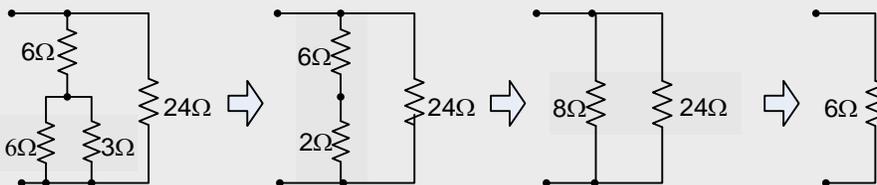
Circuit reduction relies upon identification of parallel and series combinations of circuit elements. The parallel and series elements are then combined into equivalent elements and the resulting *reduced* circuit is analyzed. The principles of circuit reduction are illustrated below in a series of examples.

### Example 2.2

Determine the equivalent resistance seen by the terminals of the resistive network shown below.

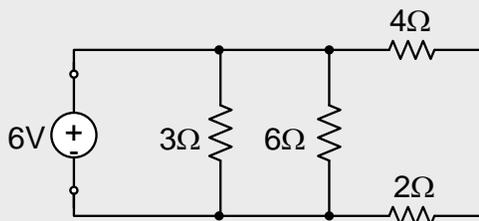


The sequence of operations performed is illustrated below. The 6Ω and 3Ω resistances are combined in parallel to obtain an equivalent 2Ω resistance. This 2Ω resistance and the remaining 6Ω resistance are in series, these are combined into an equivalent 8Ω resistance. Finally, this 8Ω resistor and the 24Ω resistor are combined in parallel to obtain an equivalent 6Ω resistance. Thus, the equivalent resistance of the overall network is 6Ω.



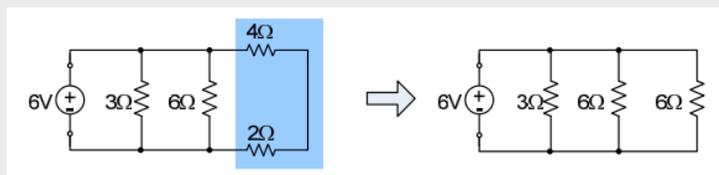
### Example 2.3

In the circuit below, determine the power delivered by the source.

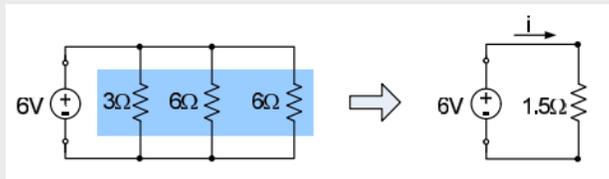


In order to determine power delivery, we need to determine the total current provided by the source to the rest of the circuit. We can determine current easily if we convert the resistor network to a single, equivalent resistance. A set of steps for doing this are outlined below.

**Step 1:** The 4-ohm and 2-ohm resistors, highlighted in the figure to the left in blue, are in series. Series resistances add directly, so these can be replaced with a single 6-ohm resistor, as shown on the figure to the right below.



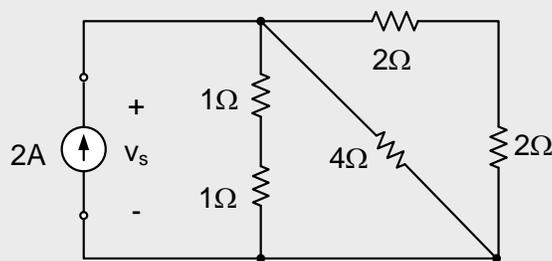
**Step 2:** The 3-ohm resistor and the two 6-ohm resistors are now all in parallel, as indicated on the figure to the left below. These resistances can be combined into a single equivalent resistor  $R_{eq} = \frac{1}{\frac{1}{3} + \frac{1}{6} + \frac{1}{6}} = 1.5\Omega$ . The resulting equivalent circuit is shown to the right below.



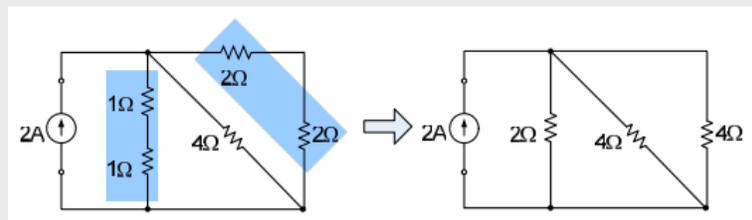
The current out of the source can now be readily determined from the figure to the right above. The voltage drop across the 1.5Ω resistor is 6V, so Ohm's law gives  $i = \frac{6V}{1.5\Omega} = 4A$ . Thus, the power delivered by the source is  $P = (4A)(6V) = 24W$ . Since the sign of the current relative to the current does not agree with the passive sign convention, the power is generated by the source.

### Example 2.4

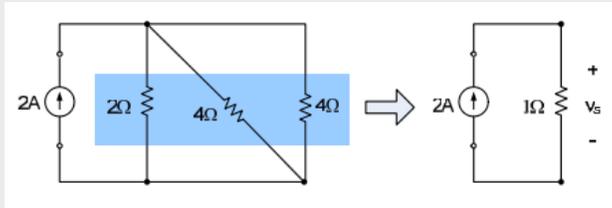
For the circuit shown below, determine the voltage,  $v_s$ , across the 2A source.



The two 1Ω resistors and the two 2Ω resistors are in series with one another, as indicated on the figure to the left below. These can be combined by simply adding the series resistances, leading to the equivalent circuit shown to the right below.



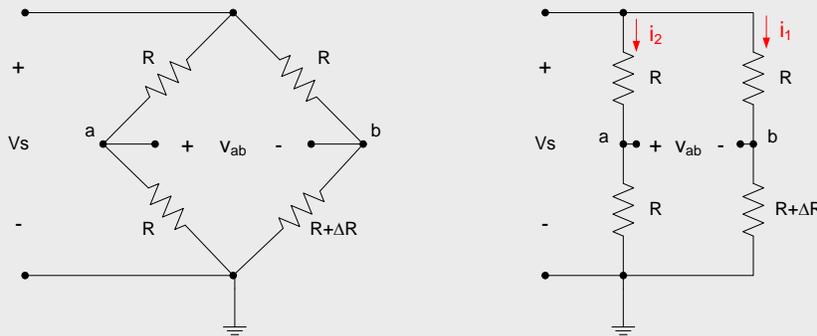
The three remaining resistors are all in parallel (they all share the same nodes) so they can be combined using the relation  $R_{eq} = \frac{1}{\frac{1}{2} + \frac{1}{4} + \frac{1}{4}}$ . Note that it is not necessary to combine all three simultaneously, the same result is obtained by successive combinations of two resistances. For example, the two 4Ω resistors can be combined using equation (2.16) to obtain:  $R_{eq1} = \frac{4 \cdot 4}{4 + 4} = 2\Omega$ . The total equivalent resistance can then be determined by a parallel combination of  $R_{eq1}$  and the 2Ω resistor:  $R_{eq} = \frac{2 \cdot 2}{2 + 2} = 1\Omega$ .



The voltage across the source can now be determined from Ohm's law:  $v_s = (1\Omega)(2A) = 2V$ . The assumed polarity of the source voltage is correct.

### Example 2.5: Wheatstone Bridge

A Wheatstone bridge circuit is shown below. The bridge is generally presented as shown in the figure to the left; we will generally use the equivalent circuit shown to the right. A Wheatstone bridge is commonly used to convert a variation in resistance to a variation in voltage. A constant supply voltage  $V_s$  is applied to the circuit. The resistors in the circuit all have a nominal resistance of  $R$ ; the variable resistor has a variation  $\Delta R$  from this nominal value. The output voltage  $v_{ab}$  indicates the variation  $\Delta R$  in the variable resistor. The variable resistor in the network is often a transducer whose resistance varies dependent upon some external variable such as temperature.



By voltage division, the voltages  $v_b$  and  $v_a$  (relative to ground) are:

$$v_b = \frac{(R + \Delta R)}{2R + \Delta R} V_s$$

And

$$v_a = Ri_2 = \frac{V_s \cdot R}{2R} = \frac{V_s}{2}$$

The voltage  $v_{ab}$  is then:

$$v_{ab} = v_a - v_b = \left(\frac{1}{2} - \frac{R + \Delta R}{2R + \Delta R}\right) V_s = \left(\frac{(2R + \Delta R) - 2(R + \Delta R)}{2(2R + \Delta R)}\right) V_s = -\frac{\Delta R}{2(2R + \Delta R)} \cdot V_s$$

For the case in which  $\Delta R \ll 2R$ , this simplifies to:

$$v_{ab} \approx -\frac{V_s}{4R} \Delta R$$

And the output voltage is proportional to the change in resistance of the variable resistor.

#### Practical applications:

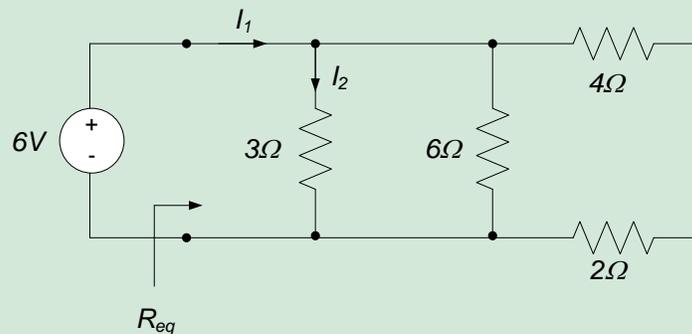
A number of common sensors result in a resistance variation resulting from some external influence. *Thermistors* change resistance as a result of temperature changes; *strain gages* change resistance as a result of deformation, generally due to application of a load to the part to which the gage is bonded; *photoconductive transducers*, or *photoresistors*, change resistance as a result of changes in light intensity. Wheatstone bridges are commonly used in conjunction with these types of sensors.

## Section Summary

- In a circuit, which contains obvious series and/or parallel combinations of resistors, analysis can be simplified by combining these resistances into equivalent resistances. The reduction in the overall number of resistances reduces the number of unknowns in the circuit, with a corresponding reduction in the number of governing equations. Reducing the number of equations and unknowns typically simplifies the analysis of the circuit.
- Not all circuits are reducible.

## 2.3 Exercises

1. For the circuit shown, determine:
  - a.  $R_{eq}$  (the equivalent resistance seen by the source)
  - b. The currents  $I_1$  and  $I_2$



## 2.4 Non-ideal Power Supplies

In section 1.2, we discussed ideal power sources. In that section, an ideal voltage supply was characterized as providing a specified voltage regardless of the current requirements made upon the device. Likewise, an ideal current source was defined as providing a specified current regardless of the voltage potential difference across the source. These models are not realistic - since an ideal voltage source can provide infinite current with non-zero voltage difference and an ideal current source can provide infinite voltage difference with non-zero current, either device is capable of delivering infinite power. In many cases, the ideal voltage and current source models will be adequate, but in cases where we need to more accurately replicate the operation of realistic power supplies, we will need to modify our models of these devices.

In this section, we present simple models for voltage and current sources which incorporate more realistic assumptions as to the behavior of these devices.

### 2.4.1 Non-ideal Voltage Sources

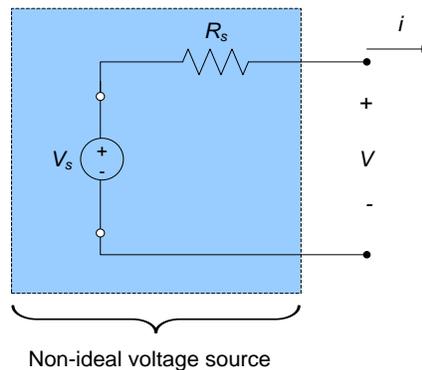
An ideal voltage source was defined in section 1.2 as providing a specified voltage, regardless of the current flow out of the device. For example, an ideal 12V battery will provide 12V across its terminals, regardless of the load connected to the terminals. A real 12V battery, however, provides 12V across its terminals only when its terminals

are open-circuited. As we draw current from the terminals, the battery will provide less than 12V - the voltage will decrease as more and more current is drawn from the battery. The real battery thus appears to have an internal voltage drop which increases with increased current.

We will model a real or *practical* voltage source as a series connection of an ideal voltage source and an *internal resistance*. This model is depicted schematically in Fig. 2.5, in which the non-ideal voltage source contains an ideal voltage source providing voltage  $V_s$  and an internal resistance  $R_s$ . The non-ideal voltage source delivers a voltage  $V$  and a current  $i$ , where:

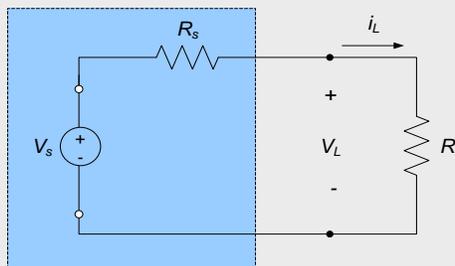
$$V = V_s - i \cdot R_s \quad \text{Eq. 2.24}$$

Equation (2.24) indicates that the voltage delivered by our non-ideal voltage source model decreases as the current out of the voltage source increases, which agrees with expectations.



### Example 2.6

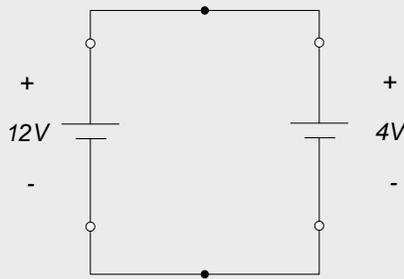
Consider the case in which we connect a resistive load to the non-ideal voltage source. The figure below provides a schematic of the overall system;  $R_L$  is the load resistance,  $V_L$  is the voltage delivered to the load, and  $i_L$  is the current delivered to the load.



In the case above, the current delivered to the load is  $i = \frac{V_s}{R_s + R_L}$  and the load voltage is  $V_L = V_s \frac{R_L}{R_s + R_L}$ . Thus, if the load resistance is infinite (the load is an open circuit),  $V_L = V_s$ , but the power supply delivers no current and hence no power to the load. If the load resistance is zero (the load is a short circuit),  $V_L = 0$  and the power supply delivers current  $i_L = \frac{V_s}{R_s}$  to the load; the power delivered to the load, however, is still zero.

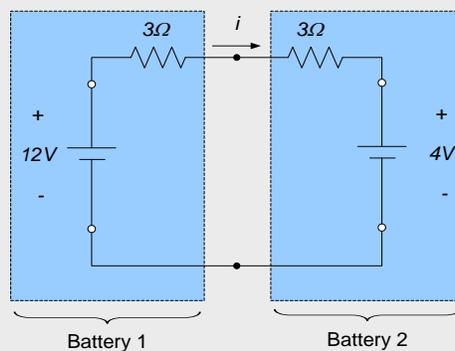
### Example 2.7: Charging a Battery

We have a “dead” car battery which is providing only 4V across its terminals. We want to charge the battery using a spare battery which is providing 12V across its terminals. To do this, we connect the two batteries as shown below:



If we attempt to analyze this circuit by applying KVL around the loop, we obtain  $12V=4V$ . This is obviously incorrect and we cannot proceed with our analysis - our model disagrees with reality!

To resolve this issue, we will include the internal resistance of the batteries. Assuming a  $3\Omega$  internal resistance in each battery, we obtain the following model for the system:



Applying KVL around the loop, and using Ohm's law to write the voltages across the battery internal resistances in terms of the current between the batteries results in:

$$-12V + (3\Omega)i + (3\Omega)i + 4V = 0$$

Which can be solved for the current  $i$  to obtain:

$$i = \frac{12V - 4V}{6\Omega} = 1.33A$$

Notice that as the voltage of the “dead” battery increases during the charging process, the current delivered to the “dead” battery decreases.

### 2.4.2 Non-ideal Current Sources

An ideal current source was defined in section 1.2 as providing a specified current, regardless of the voltage difference across the device. This model suffers from the same basic drawback as our ideal voltage source model - the model can deliver infinite power, which is inconsistent with the capabilities of a real current source.

We will use the circuit shown schematically in Fig. 2.6 to model a non-ideal current source. The non-ideal model consists of an ideal current source,  $i_s$ , placed in parallel with an internal resistance,  $R_s$ . The source delivers a voltage  $V$  and current  $i$ . The output current is given by:

$$i = i_s - \frac{V}{R_s} \tag{Eq. 2.25}$$

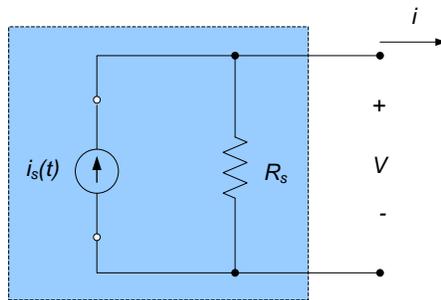
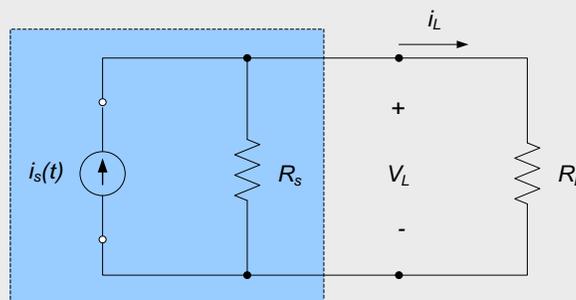


Figure 2.6. Non-ideal current source model.

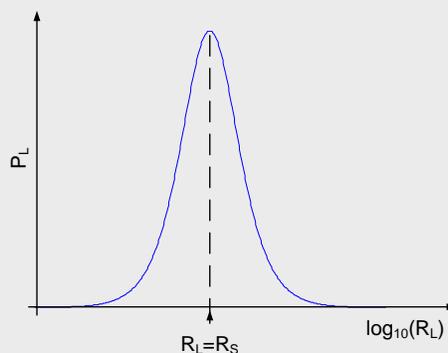
### Example 2.8

Consider the case in which we connect a resistive load to the non-ideal current source. The figure below provides a schematic of the overall system;  $R_L$  is the load resistance,  $V_L$  is the voltage delivered to the load, and  $i_L$  is the current delivered to the load.



In the case above, the current delivered to the load can be determined from a current divider relation as  $i_L = i_s \cdot \frac{R_s}{R_s + R_L}$  and the load voltage, by Ohm's law, is  $V_L = i_L R_L = i_s \frac{R_s R_L}{R_s + R_L}$ . If the load resistance is zero (the load is a short circuit),  $i_L = i_s$ , but the power supply delivers no voltage and hence no power to the load. In the case of infinite load resistance (the load is open circuit),  $i_L = 0$ . In this case, we can neglect  $R_s$  in the denominator of the load voltage equation to obtain  $V_L \approx i_s \frac{R_s R_L}{R_L}$  so that  $V_L \approx i_s R_s$ . Since the current is zero, however, the power delivered to the load is still zero.

If we explicitly calculate the power delivered to the load, we obtain  $V_L = i_s^2 \frac{R_s R_L}{R_s + R_L} \cdot \frac{R_s}{R_s + R_L}$ . A plot of the power delivered to the load as a function of the load resistance is shown below; a logarithmic scale is used on the horizontal axis to make the plot more readable. As expected, the power is zero for high and low load resistances. The peak of the curve occurs when the load resistance is equal to the source resistance,  $R_L = R_s$ .

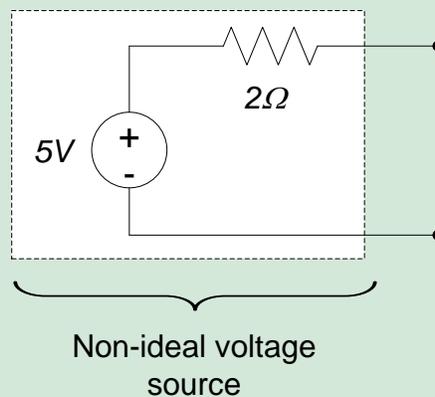


## Section Summary

- In many cases, power supplies can be modeled as ideal power supplies, as presented in section 1.2. However, in some cases representation as a power supply as ideal results in unacceptable errors. For example, ideal power supplies can deliver infinite power, which is obviously unrealistic.
- In this section, we present a simple model for a non-ideal power supply.
  - Our non-ideal voltage source consists of an ideal voltage source in series with a resistance which is internal to the power supply.
  - Our non-ideal current supply consists of an ideal current source in parallel with a resistance which is internal to the power supply.
- Voltage and current divider formulas allow us to easily quantify the effects of the internal resistances of the non-ideal power supplies. Our analysis indicates that the non-ideal effects are negligible, as long as the resistance of the load is large relative to the internal resistance of the power supply.

## 2.4 Exercises

1. A voltage source with an internal resistance of  $2\Omega$  as shown below is used to apply power to a  $3\Omega$  resistor. What voltage would you measure across the  $3\Omega$  resistor?



2. The voltage source of exercise 1 above is used to apply power to a  $2k\Omega$  resistor. What voltage would you measure across the  $2k\Omega$  resistor?

## 2.5 Practical Voltage and Current Measurement

The process of measuring a physical parameter will almost invariably change the parameter being measured. This effect is both undesirable and, in general, unavoidable. One goal of any measurement is to affect the parameter being measured as little as possible.

The above statement is true of voltage and current measurements. An ideal voltmeter, connected in parallel with some circuit element, will measure the voltage across the element without affecting the current flowing through the element. Unfortunately, any real or practical voltmeter will draw some current from the circuit it is connected to; this *loading effect* will change the circuit's operating conditions, causing some difference between the measured voltage and the corresponding voltage without the voltmeter present in the circuit. Likewise, an ideal ammeter, connected in series with some circuit element, will measure current without affecting the voltage in the circuit. A practical ammeter, like a practical voltmeter, will introduce loading effects which change the operation of the circuit on which the measurement is being made.

In this section, we introduce some effects of measuring voltages and currents with practical meters.

## 2.5.1 Voltmeter and Ammeter Models

We will model both voltmeters and ammeters as having some internal resistance and a method for displaying the measured voltage difference or current. Fig. 2.7 shows schematic representations of voltmeters and ammeters.

The ammeter in Fig. 2.7(a) has an internal resistance  $R_M$ ; the current through the ammeter is  $i_A$  and the voltage difference across the ammeter is  $V_M$ . The ammeter's voltage difference should be as small as possible - an ammeter, therefore, should have an extremely small internal resistance.

The voltmeter in Fig. 2.7(b) is also represented as having an internal resistance  $R_M$ ; the current through the meter is  $i_A$  and the voltage difference across the meter is  $R_M$ . The current through the voltmeter should be as small as possible - the voltmeter should have an extremely high internal resistance.

The effects of non-zero ammeter voltages and non-zero voltmeter currents are explored in more detail in the following subsections.

## 2.5.2 Voltage Measurement

Consider the circuit shown in Fig. 2.8(a). A current source,  $i_s$ , provides current to a circuit element with resistance,  $R$ . We want to measure the voltage drop,  $V$ , across the circuit element. We do this by attaching a voltmeter across the circuit element as shown in Fig. 2.8(b).

In Fig. 2.8(b) the voltmeter resistance is in parallel to the circuit element we wish to measure the voltage across and the combination of the circuit element and the voltmeter becomes a current divider. The current through the resistor  $R$  then becomes:

$$i = i_s \frac{R_M}{R + R_M} \quad \text{Eq. 2.26}$$

The voltage across the resistor  $R$  is then, by Ohm's law:

$$V = i_s \frac{R \cdot R_M}{R + R_M} \quad \text{Eq. 2.27}$$

If  $R_M \gg R$ , this expression simplifies to:

$$V \approx i_s \frac{R \cdot R_M}{R_M} = R \cdot i_s \quad \text{Eq. 2.28}$$

And negligible error is introduced into the measurement - the measured voltage is approximately the same as the voltage without the voltmeter. If, however, the voltmeter resistance is comparable to the resistance  $R$ , the simplification of equation (2.28) is not appropriate and significant changes are made to the system by the presence of the voltmeter.

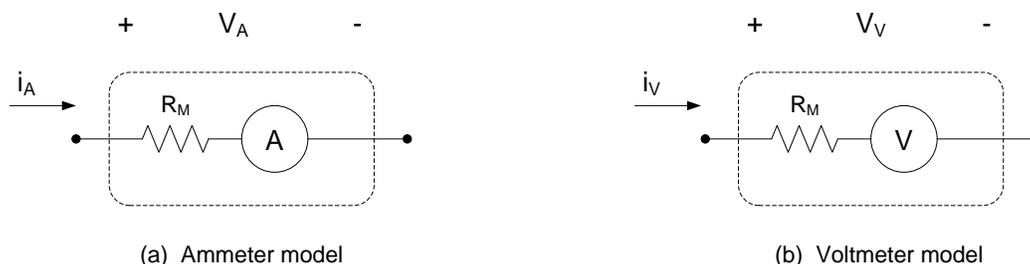


Figure 2.7. Ammeter and voltmeter models.

### 2.5.3 Current Measurement

Consider the circuit shown in Fig. 2.9(a). A voltage source,  $V_s$ , provides power to a circuit element with resistance,  $R$ . We want to measure the current,  $i$ , through the circuit element. We do this by attaching an ammeter in series with the circuit element as shown in Fig. 2.9(b).

In Fig. 2.9(b) the series combination of the ammeter resistance and the circuit element whose current we wish to measure creates a voltage divider. KVL around the single circuit loop provides:

$$V_s = i(R_M + R) \tag{Eq. 2.29}$$

Solving for the current results in:

$$i = \frac{V_s}{R_M + R} \tag{Eq. 2.30}$$

If  $R_M \ll R$ , this simplifies to:

$$i \approx \frac{V_s}{R} \tag{Eq. 2.31}$$

And the measured current is a good approximation to current in the circuit of Fig. 2.9(a). However, if the ammeter resistance is not small compared to the resistance  $R$ , the approximation of equation (2.31) is not appropriate and the measured current is no longer representative of the circuit's operation without the ammeter.

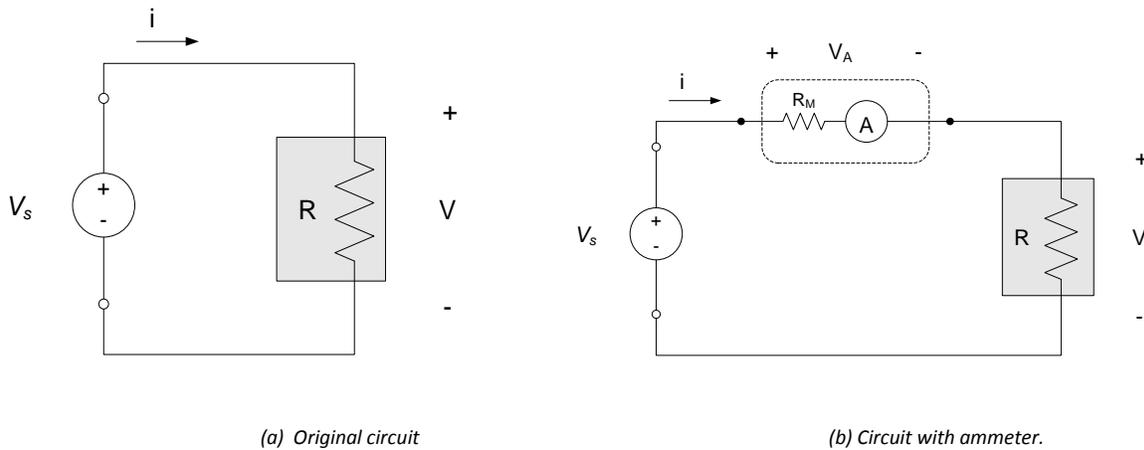
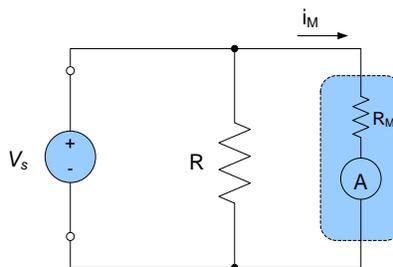


Figure 2.9. Current measurement.

#### Caution

Incorrect connections of ammeters or voltmeters can cause damage to the meter. For example, consider the connection of an ammeter in parallel with a relatively large resistance, as shown below.



In this configuration, the ammeter current,  $i_M = \frac{V_S}{R_M}$ . Since the ammeter resistance is typically very small, this can result in high currents being provided to the ammeter. This, in turn, may result in excessive power being provided to the ammeter and resulting damage to the device.

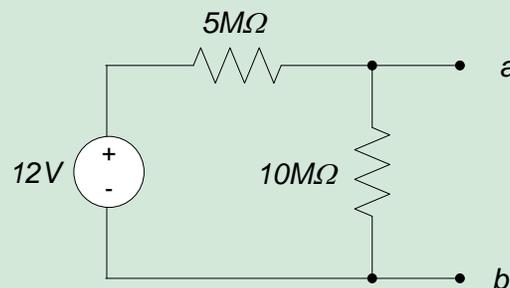
Ammeters are generally intended to be connected in series with the circuit element(s) whose current is being measured. Voltmeters are generally intended to be connected in parallel with the circuit element(s) whose voltage is being measured. Alternate connections can result in damage to the meter.

## Section Summary

- Measurement of voltage and/or current in a circuit will always result in some effect on the circuit's behavior - that is, our measurement will always change the parameter being measured. One goal when measuring a voltage or current is to ensure that the measurement effects are negligible.
- In this section, we present simple models for voltmeters and ammeters (voltage and current measurement devices, respectively).
  - Our non-ideal voltmeter consists of an ideal voltmeter (which had infinite resistance, and thus draws no current from the circuit) in parallel with a resistance which is internal to the voltmeter. This model replicates the finite current which is necessarily drawn from the circuit by a real voltmeter.
  - Our non-ideal ammeter consists of an ideal ammeter (which has zero resistance, and thus introduces no voltage drop in the circuit) in series with a resistance which is internal to the ammeter. This resistance allows us to model the finite voltage drop which is introduced into the circuit by a real current measurement.
- Voltage and current divider formulas allow us to easily quantify the effects of the internal resistances of voltage and current meters. Our analysis indicates that the non-ideal effects are negligible, as long as:
  - The resistance of the voltmeter is large relative to the resistance across which the voltage measurement is being made.
  - The resistance of the ammeter is small compared to the overall circuit resistance.

## 2.5 Exercises

A voltmeter with an internal resistance of  $10M\Omega$  is used to measure the voltage  $v_{ab}$  in the circuit below. What is the measured voltage? What voltage measurement would you expect from an ideal voltmeter?



## Real Analog Chapter 2: Lab Projects

### 2.1.1: Temperature Measurement System

In this lab assignment, students will design another temperature-measuring circuit. Unlike our previous temperature measuring circuit, the output voltage of this circuit is to be relative to the output voltage at room temperature. The output voltage is to be positive if the temperature is above room temperature, and negative if the temperature is below room temperature. As with our previous temperature measuring circuit, this circuit will use a thermistor to sense temperature changes.

#### Before beginning this lab, you should be able to:

- State Ohm's law
- Determine the equivalent resistance of series and parallel resistive networks
- State the voltage divider and current divider formulae
- Use a digital multimeter to measure resistance, voltage, and current (Labs 1.1 and 1.2.1)
- Use the Analog Discovery 2's waveform generator to apply constant voltages to a circuit (Lab 1.2.2)
- Use the Analog Discovery 2 voltmeter to measure a constant voltage (Lab 1.2.1)
- Use color codes on resistors to determine the resistor's nominal resistance

#### After completing this lab, you should be able to:

- Design a thermistor-based circuit to measure temperature
- Use a potentiometer to provide a desired resistance value
- Use multiple power supplies in an electrical circuit.

#### This lab exercise requires:

- Analog Discovery 2
- Digilent Analog Parts Kit
- Digital multimeter (optional)

#### Symbol Key:

- DEMO** Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
- ANALYSIS** Analysis; include principle results of analysis in laboratory report.
- SIM** Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
- DATA** Record data in your lab notebook.

#### General Discussion:

In this portion of the lab assignment, we will refine the temperature measurement system we designed in Lab 1.4.4. The system will still use a thermistor to detect temperature changes. (Recall that a thermistor is a device whose electrical resistance changes as a function of the temperature of the thermistor. The thermistor we will use has a temperature-resistance curve approximately as shown in Fig. 1. Thermistor operation is discussed in more detail in the companion document to Lab 1.4.4.)

Our design requirements for this assignment are as follows:

1.  $\pm 5V$  input voltage to the system
2. Output voltage is  $0V \pm 10mV$  at room temperature (approximately  $25^\circ C$ )
3. Output voltage is positive for temperatures above room temperature, negative for temperatures below room temperature
4. Output voltage increases by a minimum of  $1V$  over a temperature range of  $25^\circ C$  to  $37^\circ C$ . (These temperatures correspond approximately to room temperature and body temperature, respectively.)

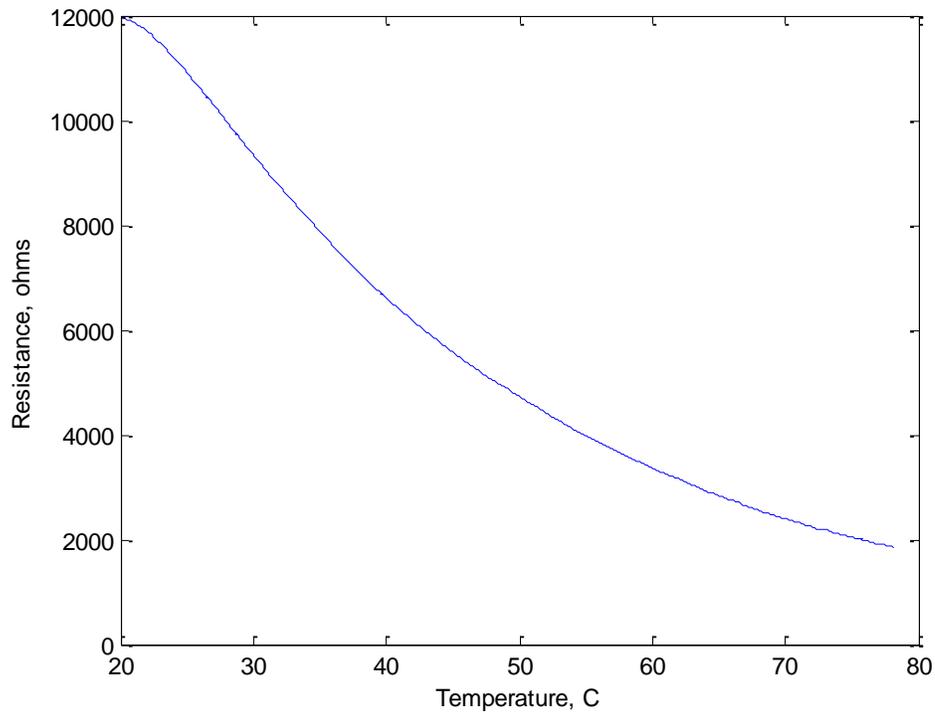


Figure 1. Thermistor temperature-resistance characteristic.

### Pre-lab:

#### ANALYSIS

In the circuit of Fig. 2, the resistance  $R_{TH}$  is the variable resistance of the thermistor. The voltage  $v_{out}$  is the voltage that we will use to indicate temperature. Two 5V voltage supplies are used to apply power to the circuit as shown – note that  $V_{ba} = +5V$  and  $V_{ca} = -5V$ .  $v_{out}$  is measured between nodes d and a with the polarity shown. The design problem is to choose a value for R so that  $v_{out}$  satisfies the given design requirements. It is recommended that you choose R based on requirement 2, and then check to see that this resistance satisfies the remaining design requirements.

Be sure to document your analyses (preferably in a lab notebook), including the equation(s) governing the system, your desired value for R, your expected output voltage change over the specified temperature range, and your expected output voltage at room temperature.

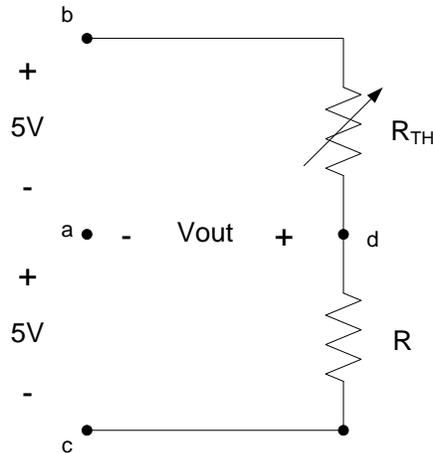


Figure 2. Temperature measurement circuit schematic.

### Lab Procedures:

Implement and test the design you created in the pre-lab. It is suggested that you perform at least the following steps when doing this:

#### DATA

1. Measure the room-temperature resistance of your particular thermistor. Compare this value to the assumed value used in your pre-lab and modify your desired value of  $R$  accordingly.

#### DATA

2. Implement your design. Be sure to record actual resistance values for any fixed resistors used in your design. In order to meet requirement 2, it may be necessary for you to implement a very specific resistance. A potentiometer (variable resistor) can be used to provide an arbitrary resistance value. You can monitor the output voltage while adjusting the potentiometer to ensure that requirement 2 is met. If desired, the potentiometer can be placed in parallel or series with a fixed resistor.

#### DATA

3. Measure and record the voltage response at room temperature. Measure and record the output voltage at the high temperature condition by firmly holding the thermistor between two fingers. Verify that the output voltage becomes negative when the thermistor is below room temperature by holding a cold can (or bottle) of your favorite beverage against the thermistor. Discuss your circuit's performance relative to the design specifications. (e.g. Were requirements met? If not, why?)

#### DEMO

4. Demonstrate operation of your circuit to the Teaching Assistant. Have the TA initial the appropriate page(s) of your lab notebook and the lab checklist.

## Real Analog Chapter 2: Lab Worksheets

### 2.1.1: Temperature Measurement System (50 points total)

1. In the space below, provide your preliminary design from pre-lab. Include your estimate of output voltage at room temperature and output voltage variation resulting from specified temperature variation. Compare the expected results vs. specified performance. (15 pts)
2. Provide the measured thermistor resistance at room temperature; compare this value with data used in pre-lab to design circuit. Design changes resulting from measured thermistor response. (7 pts)
3. Actual resistance values used in implementation of circuit. (3 pts)
4. Measured circuit voltage response. Discuss your results, especially the measured performance vs. the design requirements. (15 pts)
5. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuits' operation. (10 pts)

TA Initials: \_\_\_\_\_

## Real Analog Chapter 2: Lab Projects

### 2.3.1: Series and Parallel Resistances and Circuit Reduction

In this lab, we will examine resistance networks consisting of resistors in series and parallel. We will measure the equivalent resistance of the resistance network and comparing the measured results to analytical expectations. DMMs will be used to measure the voltage and current across individual resistors within series and parallel combinations of resistors; the experimental measurements will be compared to analytical expectations based on the governing equations for voltage and current dividers.

#### Before beginning this lab, you should be able to:

- State Ohm's law
- Determine the equivalent resistance of series and parallel resistive networks
- State the voltage divider and current divider formulae
- Use a digital multimeter to measure resistance, voltage, and current
- Use the Analog Discovery 2's waveform generator to apply constant voltages
- Use the Analog Discovery 2 voltmeter to measure a constant voltage
- Use color codes on resistors to determine the resistor's nominal resistance

#### After completing this lab, you should be able to:

- Measure the equivalent resistance of a resistive network
- Measure the voltage and/or current in a resistor in a series or parallel resistance combination

#### This lab exercise requires:

- Analog Discovery 2 module
- Digilent Analog Parts Kit
- Digital multimeter (optional)

#### Symbol Key:

-  Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
-  Analysis; include principle results of analysis in laboratory report.
-  Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
-  Record data in your lab notebook.

#### General Discussion:

This portion of the lab assignment concerns the circuit shown in Fig. 1 below. A power supply is used to apply the 5V voltage difference. We wish to determine the power dissipated by the 1K $\Omega$  resistor.

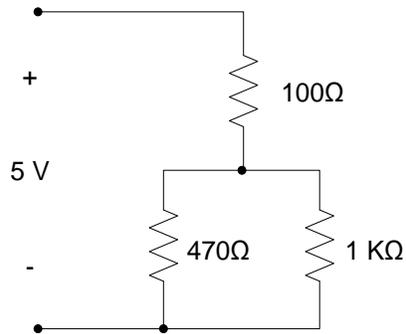


Figure 1. Circuit schematic.

### Pre-lab:

**ANALYSIS**

Analyze the circuit of Fig. 1 to estimate the power dissipated by the 1KΩ resistor.

### Lab Procedures:

**DATA**

1. Construct the circuit of Fig. 1; measure and record all actual resistance values. Measure the parameters necessary to determine the power dissipated by the 1KΩ resistor. Determine the power dissipated by the 1KΩ resistor. Compare the measured power with your estimate from the pre-lab. Comment on any differences between the estimated and measured values.

**DEMO**

2. Demonstrate operation of your circuit to the Teaching Assistant. Have the TA initial the appropriate page(s) of your lab notebook and the lab checklist.

## Real Analog Chapter 2: Lab Worksheets

### 2.3.1: Series and Parallel Resistors and Equivalent Resistance (20 points total)

1. Expected power dissipated by  $1\text{K}\Omega$  resistor (pre-lab analysis). (5 pts)
2. Provide a schematic of the circuit below, including measured resistance values. (3 pts)
3. Measured power dissipated by  $1\text{K}\Omega$  resistor (provide all measurements taken: actual resistance values, voltages, currents, power calculation). Comment on the agreement between measured and expected power dissipation – calculating a percent difference is always good! (8 pts)
4. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuit's operation. (4 pts)

TA Initials: \_\_\_\_\_

## Real Analog Chapter 2: Lab Projects

### 2.3.2: Series and Parallel Resistances and Circuit Reduction

In this lab assignment, we will perform some simple design-related exercises. Specifically, we will design resistive networks, composed of the available fixed resistors, to provide specified resistances.

#### Before beginning this lab, you should be able to:

- State Ohm's law
- Determine the equivalent resistance of series and parallel resistive networks
- State the voltage divider and current divider formulae
- Use a digital multimeter to measure resistance, voltage, and current
- Use the Analog Discovery 2's waveform generator to apply constant voltages
- Use the Analog Discovery 2 oscilloscope to measure a constant voltage
- Use color codes on resistors to determine the resistor's nominal resistance

#### After completing this lab, you should be able to:

- Measure the equivalent resistance of a resistive network
- Measure the voltage and/or current in a resistor in a series or parallel resistance combination

#### This lab exercise requires:

- Digilent Analog Parts Kit
- Digital multimeter

#### Symbol Key:

-  Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
-  Analysis; include principle results of analysis in laboratory report.
-  Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
-  Record data in your lab notebook.

#### General Discussion:

We need resistors with the following resistance values and tolerances:

1.  $9\text{K}\Omega \pm 5\%$
2.  $800\Omega \pm 5\%$
3.  $35\text{K}\Omega \pm 5\%$

Resistors with these resistances are not included in the analog parts kit; we will use available fixed resistors to construct circuits with the required equivalent resistance.

**Pre-lab:****ANALYSIS**

Using only fixed-value resistors available in your analog parts kit, design circuits which have the equivalent resistances listed above.

**Lab Procedures:****DATA**

1. Construct the three circuits you designed in the pre-lab. Use an ohmmeter to measure the equivalent resistance of each of the circuits. Comment on your results – specifically, whether the design requirements were met.

**DEMO**

2. Demonstrate operation of your circuit to the Teaching Assistant. Have the TA initial the appropriate page(s) of your lab notebook and the lab checklist.

**Note:**

As always, measure and record the resistance of the individual resistors used in your circuits.



## Real Analog Chapter 2: Lab Projects

### 2.4: Non-ideal Power Sources

Though many theoretical models of electrical circuits assume that power supplies are ideal, actual circuit implementations can depend upon non-ideal limitations of the power supplies. In this lab assignment, we will experimentally explore the behavior of non-ideal power sources. The experiments in this assignment illustrate some of the effects of non-ideal power supplies.

#### Before beginning this lab, you should be able to:

- State Ohm's law
- Determine the equivalent resistance of series and parallel resistive networks
- State the voltage divider and current divider formulae
- Use a digital multimeter to measure resistance, voltage, and current
- Use the Analog Discovery 2's waveform generator to apply constant voltages
- Use the Analog Discovery 2 voltmeter to measure a constant voltage
- Model non-ideal sources

#### After completing this lab, you should be able to:

- Estimate the internal resistance of a non-ideal power supply
- Describe qualitatively, the effects of power supply internal resistances

#### This lab exercise requires:

- Analog Discovery 2 module
- Diligent Analog Parts Kit
- Digital multimeter

#### Symbol Key:

-  Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
-  Analysis; include principle results of analysis in laboratory report.
-  Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
-  Record data in your lab notebook.

#### General Discussion:

Consider the circuit of Fig. 1. Our goal is to reduce the circuit resistance to the point where the internal resistance of the power source begins to have an effect on the circuit's behavior.

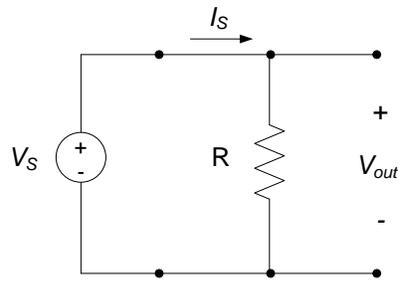


Figure 1. Circuit schematic.

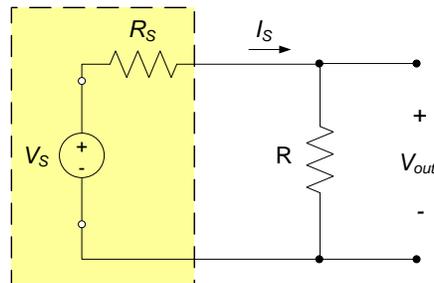
**Pre-lab:****ANALYSIS**

If  $R = 25\Omega$  and  $V_S = 1V$ , analyze the circuit of Fig. 1 to determine expected values for the measured voltage  $V_{out}$ , source current  $I_S$ , and power dissipated by the resistor, for the cases in which

- The voltage source is ideal, and
- The voltage source is non-ideal, and has an internal resistance  $R_S$ . (Note: your answer here will be a function of  $R_S$ .)

**Hint – non-ideal voltage sources:**

Per module 1.5.1, a non-ideal voltage source can be modeled as an ideal voltage source in series with an internal resistance as shown in the figure below.



Applying KVL around the loop of the above circuit results in:

$$V_S - R_S I_S = V_{out}$$

Note that we do not have the ability to measure  $V_S$  or  $R_S$  (they are inside the power supply), but we can potentially measure  $V_{out}$  and  $I_S$ .

**Lab Procedures:**

- Construct the circuit of Fig. 1, using  $R \approx 25\Omega$  and  $V_S \approx 1V$ , and estimate the source resistance of your voltage source. The following steps are suggested in order to do this:

**DATA**

- Measure and record the actual value of  $R$  for your circuit.

**DATA**

- Use the waveform generator to apply the voltage  $V_S = 1V$ . Measure and record the actual value of  $V_S$  by open-circuiting the waveform generator terminals and using a voltmeter or your Analog Discovery 2 module to measure the voltage across its terminals. Fine tune the waveform generator voltage until you measure exactly 1V across the terminals. (Note

that under these conditions, there is essentially no current supplied by the voltage source so that the internal source resistance is not affecting your voltage measurement.)

**DATA**

3. Connect the resistor to your waveform generator. Measure the voltage  $V_{out}$  using your Analog Discovery 2 module and the source current  $I_s$  using your digital multimeter. Compare the measured voltage with your estimate from part (a) of the pre-lab. Use your measurement of  $V_{out}$  and the  $V_s$  from part (2) above to estimate the internal resistance of the voltage source. (Note that we are assuming that the ideal voltage source portion of the non-ideal voltage source is not affected by the application of the load resistance  $R$ . This is a fairly large assumption!)

**DEMO**

4. Demonstrate operation of your circuit to the Teaching Assistant. Have the TA initial the appropriate page(s) of your lab notebook and the lab checklist.
- b) Repeat the source resistance estimate of part (a) above for two other values of  $R$ :  $R \approx 30\Omega$  and  $R \approx 20\Omega$ . Comment on the consistency between the results for the three different values of  $R$ .

**Note:**

This is not a good way to get an accurate estimate of the internal resistance of the power source, but it should give you an idea of the overall concepts involved.

## Real Analog Chapter 2: Lab Worksheets

### 2.4: Non-ideal Power Sources (20 points total)

1. Estimated  $V_{out}$ , for both ideal and non-ideal voltage sources, from your pre-lab analysis. (3 pts)
2. Circuit schematic, including measured resistance values. (2 pts)
3. Estimated power source internal resistance, for  $R \approx 25\Omega$ . (5 pts)
4. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuit's operation. (5 pts)  
  
**TA Initials:** \_\_\_\_\_
5. Estimated power source internal resistance for  $R \approx 30\Omega$  and  $R \approx 20\Omega$ . Discussion of consistency of results for various values of R. (5 pts)

## Real Analog Chapter 2: Lab Projects

### 2.5: Practical Voltage and Current Measurement

Theoretical models of electrical circuits often assume that we can determine voltages and currents within the circuit without affecting the circuit's operation. In reality, any time we measure a voltage or current, we alter the circuit's behavior to some extent – sometimes the effects of the measurement process can be very significant. In this lab assignment, we will experimentally explore the behavior of non-ideal meters. The experiments in this assignment illustrate the effects of non-ideal voltage measurements.

#### Before beginning this lab, you should be able to:

- State Ohm's law
- Determine the equivalent resistance of series and parallel resistive networks
- State the voltage divider and current divider formulae
- Use a digital multimeter to measure resistance, voltage, and current
- Use the Analog Discovery 2's waveform generator to apply constant voltages
- Use the Analog Discovery 2 voltmeter to measure a constant voltage
- Model non-ideal sources

#### After completing this lab, you should be able to:

- Estimate a voltmeter's internal resistance
- Describe qualitatively the effects of voltmeter internal resistances on voltage measurements

#### This lab exercise requires:

- Analog Discovery 2
- Digiilent Analog Parts Kit
- Digital multimeter

#### Symbol Key:

-  Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
-  Analysis; include principle results of analysis in laboratory report.
-  Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
-  Record data in your lab notebook.

#### General Discussion:

Very large resistors are used in the voltage divider circuit of Fig. 1. Due to these large resistances, measurement of the voltage  $V_{out}$  will likely result in measurement errors due to non-ideal voltmeter effects.

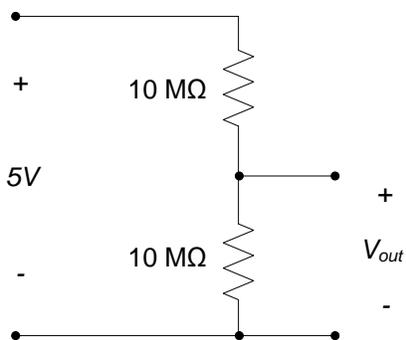


Figure 1. Circuit schematic.

**Pre-lab:**

**ANALYSIS**

Analyze the circuit of Fig. 1 to determine an expected value for the measured voltage  $V_{out}$  for the cases in which

- a) The measurement of  $V_{out}$  is determined using an ideal voltmeter (a voltmeter with infinite internal resistance), and
- b) The measurement of  $V_{out}$  is determined using a voltmeter with internal resistance  $R_M$ . (In this case, your result will be a formula which depends upon  $R_M$ .)

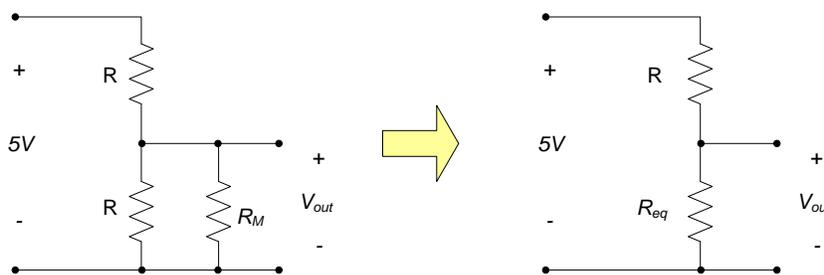
**Hint – non-ideal voltmeters:**

Per section 2.5 of the text, a voltmeter can be modeled as an equivalent resistance  $R_M$  in parallel with the voltage being measured. Thus, the circuit of Fig. 1, with the voltmeter resistance included, becomes as shown in the figure to the left below. The parallel combination of the voltmeter and the  $10M\Omega$  resistor can then be represented as a single equivalent resistance  $R_{eq}$  as shown in the figure to the right below, where

$$R_{eq} = \frac{(R)(R_M)}{R+R_M}$$

Therefore,

$$V_{OUT} = 5V \left( \frac{R_{eq}}{R+R_{eq}} \right)$$



From the above, it can be seen that if  $R \gg R_M$ ,  $R_{eq} \approx R$  and the measured  $V_{out}$  will be essentially the same as the  $V_{out}$  indicated in Fig. 1. If, however, this condition is not true, the voltmeter’s internal resistance can have a significant (and generally undesirable) effect on the voltage being measured.

**Lab Procedures:****DATA**

a) Construct the circuit of Fig. 1. Measure the voltage  $V_{out}$  using your DMM. Using your pre-lab results, estimate the internal resistance of the voltmeter.

**DEMO**

b) Demonstrate operation of your circuit to the Teaching Assistant. Have the TA initial the appropriate page(s) of your lab notebook and the lab checklist.

**DATA**

c) Repeat the test of part (a), except use the voltmeter on your Analog Discovery 2 module to measure  $V_{out}$ . Using your pre-lab results, estimate the internal resistance of the scope instrument.

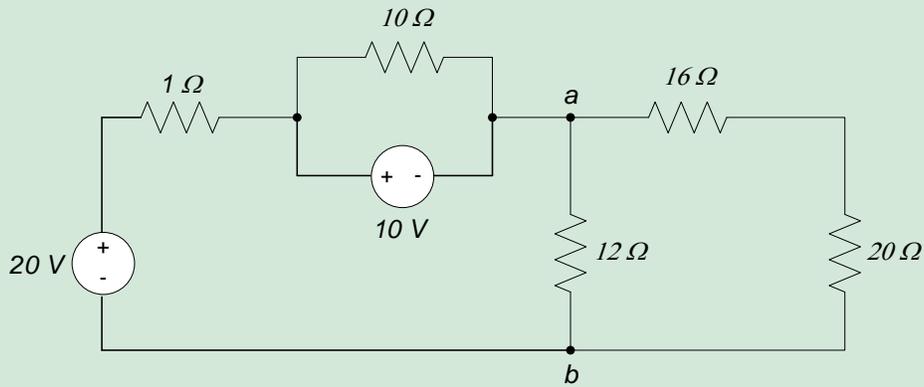
**Note:**

- This is not a good way to get an accurate estimate of the internal resistance of the voltmeter, but it should give you an idea of the overall concepts involved.
- It is likely that the Analog Discovery 2 internal resistance will be significantly lower than the internal resistance of most commercially available DMMs. This is at least partly due to the fact that the Analog Discovery 2 is primarily intended for making time-varying measurements, while DMMs are intended to measure constant values.

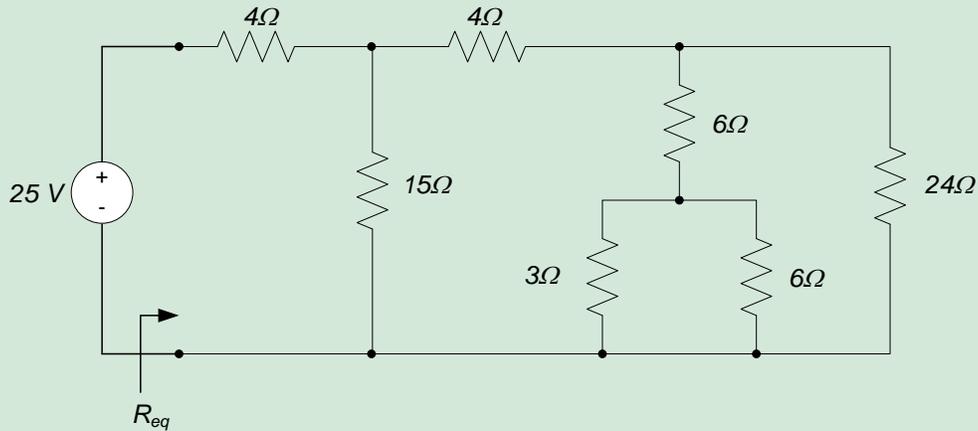


## Real Analog Chapter 2: Homework

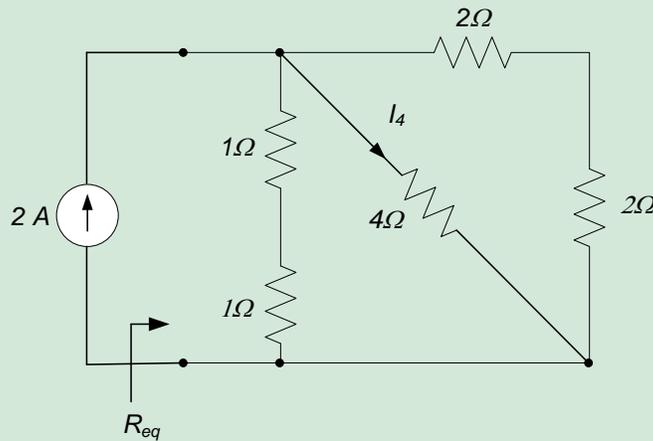
2.1 For the circuit shown, find the voltage  $v_{ab}$  and the power (generated or absorbed) by the 20V source.



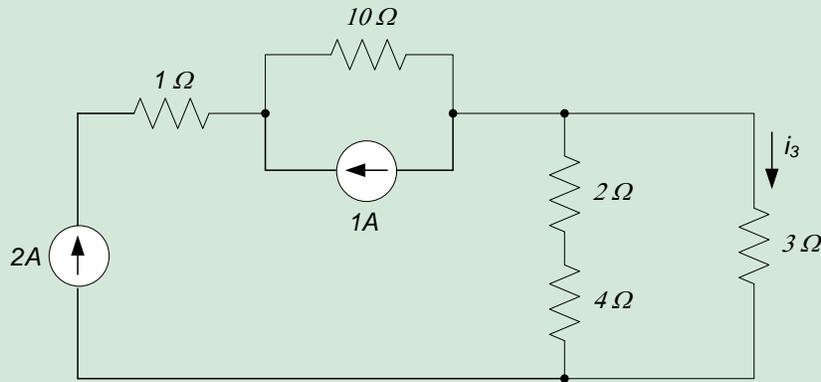
2.2 For the circuit shown, find  $R_{eq}$  (the equivalent resistance “seen” by the source) and the current out of the source.



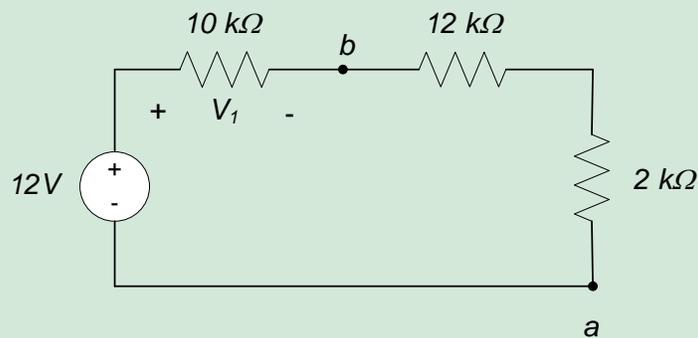
2.3 For the circuit shown, find  $R_{eq}$  (the equivalent resistance “seen” by the source) and the current  $I_4$ .



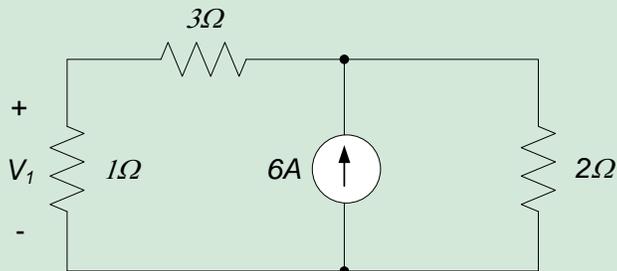
2.4 For the circuit below, determine the current  $i_3$ .



2.5 Find  $V_{ab}$  and  $V_1$  in the circuit below.

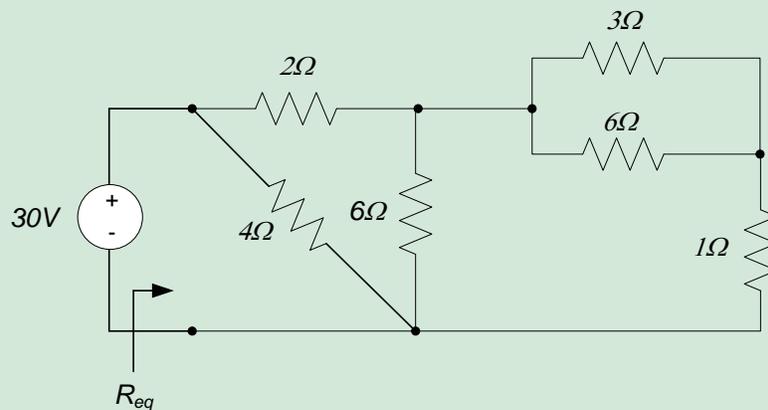


2.6 Determine the voltage  $V_1$  in the circuit below.

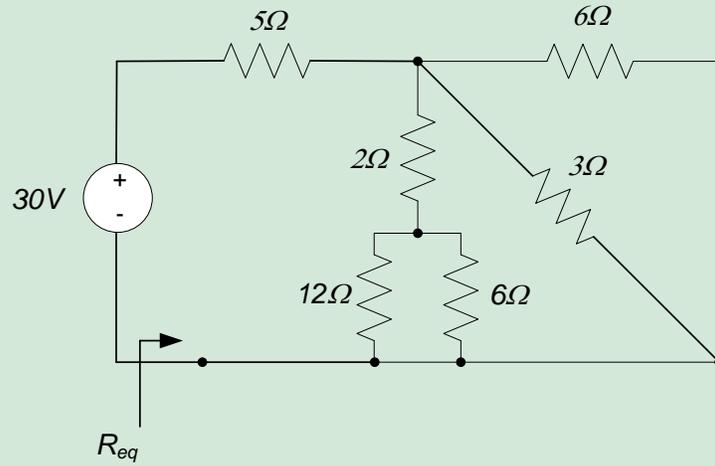


2.7 For the circuit shown, find:

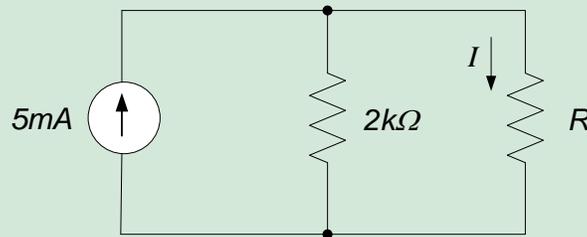
- a)  $R_{eq}$  (the equivalent resistance "seen" by the source)
- b) the power generated by the source
- c) The current through the  $4\Omega$  resistor



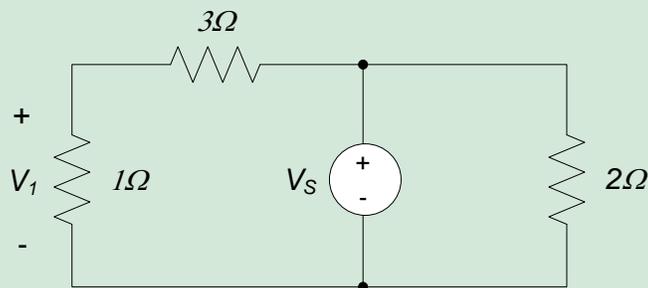
- 2.8 For the circuit shown, find:
- $R_{eq}$  (the equivalent resistance "seen" by the source)
  - the current delivered by the source
  - The voltage difference across the  $3\Omega$  resistor.
  - The voltage difference across the  $2\Omega$  resistor.



- 2.9 Determine the value of the resistance  $R$  which makes the current  $I = 2\text{mA}$ .



- 2.10 Determine the value of the source voltage  $V_S$  which makes the voltage  $V_1 = 2\text{V}$ .



## Chapter 3: Nodal and Mesh Analysis

### 3 Introduction and Chapter Objectives

In Chapters 1 & 2, we introduced several tools in circuit analysis:

- Ohm's Law
- Kirchhoff's laws
- Circuit reduction

Circuit reduction, it should be noted, is not fundamentally different from direct application of Ohm's and Kirchhoff's laws - it is simply a convenient re-statement of these laws for specific combinations of circuit elements.

In Chapter 1, we saw that direct application of Ohm's law and Kirchhoff's laws to a specific circuit using the *exhaustive method* often results in a large number of unknowns - even if the circuit is relatively simple. A correspondingly large number of equations must be solved to determine these unknowns. Circuit reduction allows us, in some cases, to simplify the circuit to reduce the number of unknowns in the system. Unfortunately, not all circuits are reducible and even analysis of circuits that are reducible depends upon the engineer "noticing" certain resistance combinations and combining them appropriately.

In cases where circuit reduction is not feasible, approaches are still available to reduce the total number of unknowns in the system. *Nodal analysis* and *mesh analysis* are two of these. Nodal and mesh analysis approaches still rely upon application of Ohm's law and Kirchhoff's laws - we are just applying these laws in a very specific way in order to simplify the analysis of the circuit. One attractive aspect of nodal and mesh analysis is that the approaches are relatively rigorous - we are assured of identifying a reduced set of variables, if we apply the analysis rules correctly. Nodal and mesh analysis are also more general than circuit reduction methods - virtually any circuit can be analyzed using nodal or mesh analysis.

Since nodal and mesh analysis approaches are fairly closely related, section 3.1 introduces the basic ideas and terminology associated with both approaches. Section 3.2 provides details of nodal analysis, and mesh analysis is presented in section 3.3.

#### After Completing this Chapter, You Should be Able to:

- Use nodal analysis techniques to analyze electrical circuits
- Use mesh analysis techniques to analyze electrical circuits

### 3.1 Introduction and Terminology

As noted in the introduction, both nodal and mesh analysis involve identification of a "minimum" number of unknowns, which completely describe the circuit behavior. That is, the unknowns themselves may not directly provide the parameter of interest, but any voltage or current in the circuit can be determined from these unknowns. In nodal analysis, the unknowns are the *node voltages*. In mesh analysis, the unknowns are the *mesh currents*. We introduce the concept of these unknowns via an example below.

Consider the circuit shown in Fig. 3.1(a). The circuit nodes are labeled in Fig. 3.1(a), for later convenience. The circuit is not readily analyzed by circuit reduction methods. If the exhaustive approach toward applying KCL and KVL is taken, the circuit has 10 unknowns (the voltages and currents of each of the five resistors), as shown in Fig. 3.1(b). Ten circuit equations must be written to solve for the ten unknowns. Nodal analysis and mesh analysis

provide approaches for defining a reduced number of unknowns and solving for these unknowns. If desired, any other desired circuit parameters can subsequently be determined from the reduced set of unknowns.

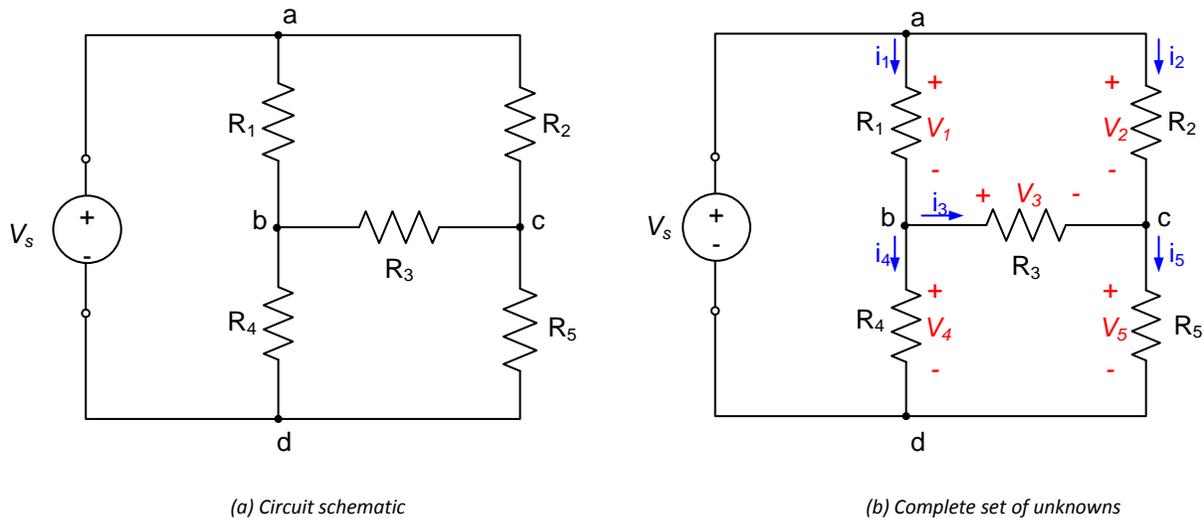


Figure 3.1. Non-reducible circuit.

In nodal analysis, the unknowns will be *node voltages*. Node voltages, in this context, are the *independent voltages* in the circuit. It will be seen later that the circuit of Fig. 3.1 contains only two independent voltages - the voltages at nodes b and c<sup>4</sup>. Only two equations need be written and solved to determine these voltages! Any other circuit parameters can be determined from these two voltages.

### Basic Idea

In nodal analysis, Kirchhoff's current law is written at each independent voltage node; Ohm's law is used to write the currents in terms of the node voltages in the circuit.

In mesh analysis, the unknowns will be *mesh currents*. Mesh currents are defined only for *planar circuits*; planar circuits are circuits which can be drawn in a single plane such that no elements overlap one another. When a circuit is drawn in a single plane, the circuit will be divided into a number of distinct areas; the boundary of each area is a *mesh* of the circuit. A *mesh current* is the current flowing around a mesh of the circuit. The circuit of Fig. 3.1 has three meshes:

- The mesh bounded by  $V_s$ , node a, and node d
- The mesh bounded by node a, node c, and node b
- The mesh bounded by node b, node c, and node d

These three meshes are illustrated schematically in Fig. 3.2. Thus, in a mesh analysis of the circuit of Fig. 3.1, three equations must be solved in three unknowns (the mesh currents). Any other desired circuit parameters can be determined from the mesh currents.

### Basic Idea

In mesh analysis, Kirchhoff's voltage law is written around each mesh loop; Ohm's law is used to write the voltage in terms of the mesh currents in the circuit. Since KVL is written around closed loops in the circuit, mesh analysis is

<sup>4</sup> The voltages at nodes a and d are not independent; the voltage source  $v_s$  constrains the voltage at node a relative to the voltage at node d (KVL around the leftmost loop indicates the  $v_{ab} = V_s$ )

sometimes known as *loop analysis*.

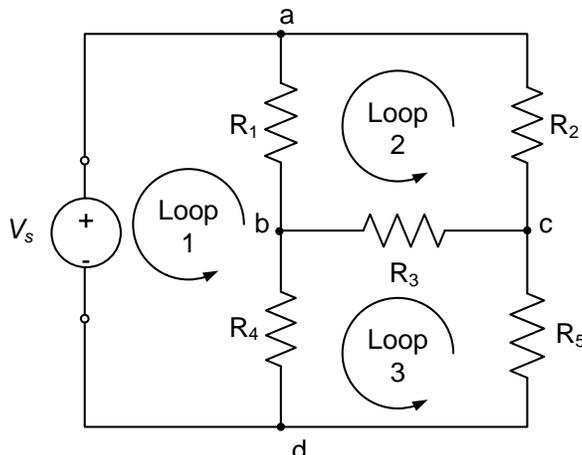


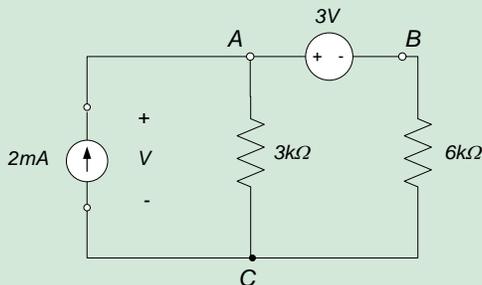
Figure 3.2. Meshes for circuit of Figure 3.1.

### Section Summary

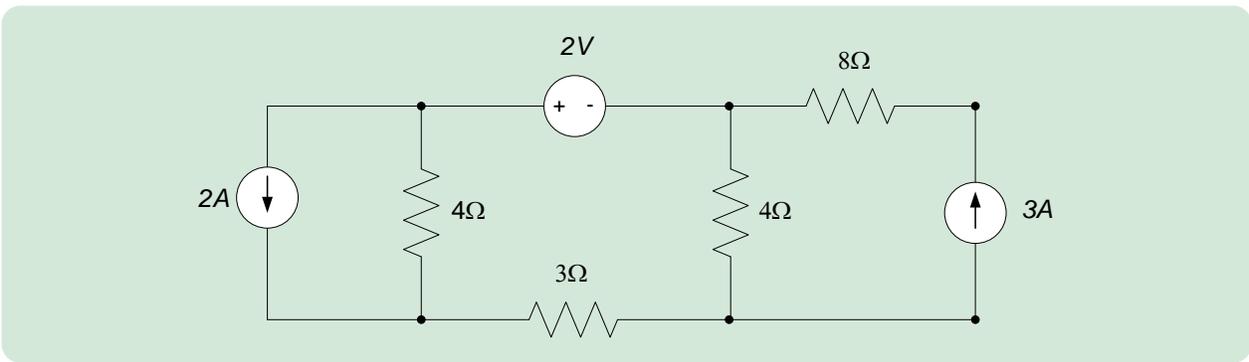
1. In nodal analysis:
  - o Unknowns in the analysis are called the *node voltages*
  - o Node voltages are the voltages at the independent nodes in the circuit
  - o Two nodes connected by a voltage source are not independent. The voltage source constrains the voltages at the nodes relative to one another. A node which is not independent is also called dependent.
2. In mesh analysis:
  - o Unknowns in the analysis are called *mesh currents*.
  - o Mesh currents are defined as flowing through the circuit elements which form the perimeter of the circuit meshes. A mesh is any enclosed, non-overlapping region in the circuit (when the circuit schematic is drawn on a piece of paper).

### 3.1 Exercises

1. The circuit below has three nodes, A, B, and C. Which two nodes are dependent? Why?



2. Identify meshes in the circuit below.



### 3.2 Nodal Analysis

As noted in section 3.1, in nodal analysis we will define a set of node voltages and use Ohm's law to write Kirchhoff's current law in terms of these voltages. The resulting set of equations can be solved to determine the node voltages; and other circuit parameters (e.g. currents) can be determined from these voltages.

The steps used in nodal analysis are provided below. The steps are illustrated in terms of the circuit of Fig. 3.3.

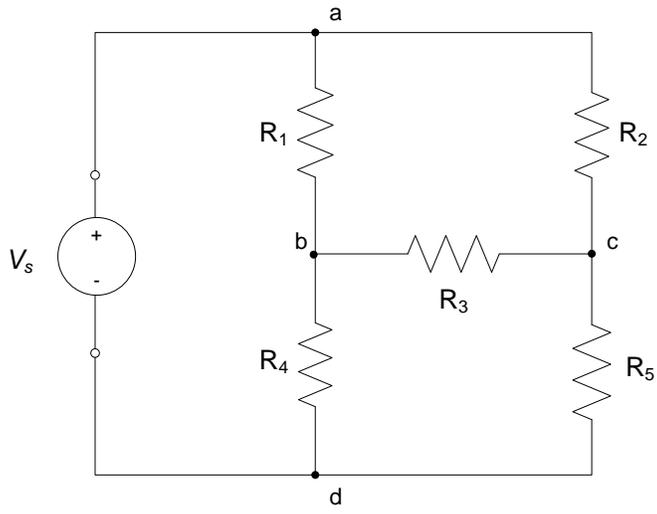


Figure 3.3. Example circuit.

#### Step 1: Define Reference Voltage

One node will be arbitrarily as a *reference node* or *datum node*. The voltages of all other nodes in the circuit will be defined to be relative to the voltage of this node. Thus, for convenience, it will be assumed that the reference node voltage is zero volts. It should be emphasized that this definition is arbitrary - since voltages are actually potential differences, choosing the reference voltage as zero is primarily a convenience.

For our example circuit, we will choose node d as our reference node and define the voltage at this node to be 0V, as shown in Fig. 3.4.

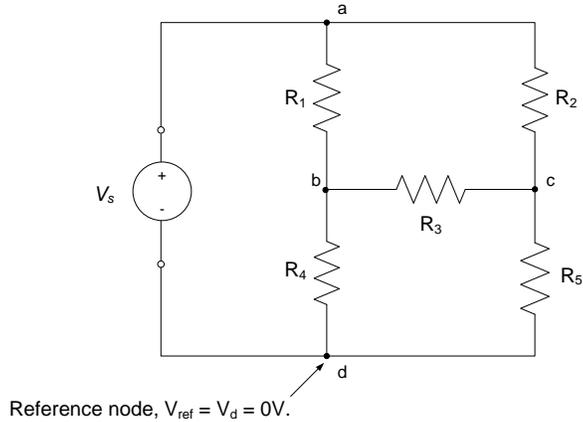


Figure 3.4. Definition of reference node and reference voltage.

**Step 2: Determine Independent Nodes**

We now define the voltages at the *independent* nodes. These voltages will be the unknowns in our circuit equations. In order to define independent nodes:

- “Short-circuit” all voltage sources
- “Open-circuit” all current sources

After removal of the sources, the remaining nodes (with the exception of the reference node) are defined as *independent* nodes (the nodes which were removed in this process are *dependent* nodes. The voltages at these nodes are sometimes said to be *constrained*). Label the voltages at these nodes - they are the unknowns for which we will solve.

For our example circuit of Fig. 1, removal of the voltage source (replacing it with a short circuit) results in nodes remaining only at nodes b and c. This is illustrated in Fig. 3.

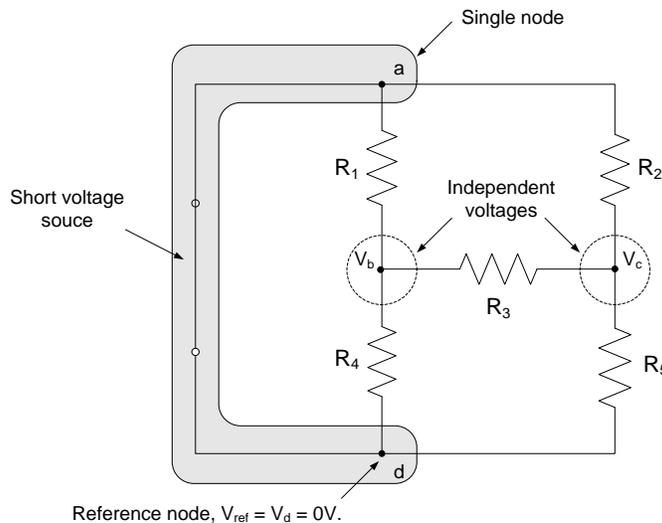


Figure 3.5. Independent voltages  $V_b$  and  $V_c$ .

**Step 3: Replace Sources in the Circuit and Identify Constrained Voltages**

With the independent voltages defined as in Step 2, replace the sources and define the voltages at the dependent nodes in terms of the independent voltages and the known voltage difference.

For our example, the voltage at node a can be written as a known voltage  $V_s$  above the reference voltage, as shown in Fig. 3.6.

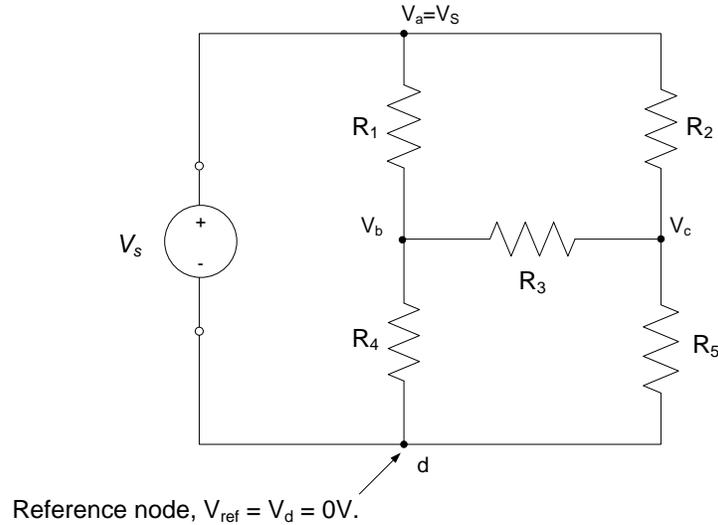


Figure 3.6. Dependent voltages defined.

**Step 4: Applying KCL at Independent Nodes**

Define currents and write Kirchhoff's current law at all independent nodes. Currents for our example are shown in Fig. 3.7 below. The defined currents include the assumed direction of positive current - this defines the sign convention for our currents. To avoid confusion, these currents are defined consistently with those shown in Fig. 3.1(a). The resulting equations are (assuming that currents leaving the node are defined as positive):

Node b:

$$-i_1 + i_3 + i_4 = 0 \tag{Eq. 3.1}$$

Node c:

$$-i_2 + i_3 + i_5 = 0 \tag{Eq. 3.2}$$

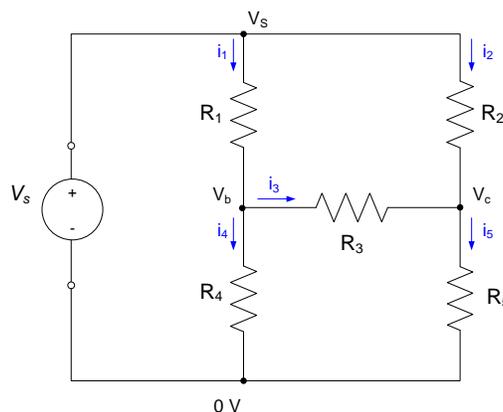


Figure 3.7. Current definitions and sign conventions.

### Step 5: Use Ohm's Law to Write the Equations From Step 4 in Terms of Voltages

The currents defined in Step 4 can be written in terms of the node voltages defined previously. For example, from Fig. 3.7:  $i_1 = \frac{V_S - V_b}{R_1}$ ,  $i_3 = \frac{V_b - V_c}{R_3}$ , and  $i_4 = \frac{V_b - 0}{R_4}$ , so equation (3.1) can be written as:

So the KCL equation for node b becomes:

$$\left(\frac{1}{R_1} + \frac{1}{R_3} + \frac{1}{R_4}\right)V_b - \frac{1}{R_3}V_c = \frac{1}{R_1}V_S \quad \text{Eq. 3.3}$$

Likewise, the KCL equation for node c can be written as:

$$-\frac{1}{R_3}V_b + \left(\frac{1}{R_3} + \frac{1}{R_2} + \frac{1}{R_5}\right)V_c = \frac{1}{R_2}V_S \quad \text{Eq. 3.3}$$

### Double-checking Results

If the circuit being analyzed contains only independent sources, and the sign convention used in KCL equations is the same as used above (currents leaving nodes are assumed positive), the equations written at each node will have the following form:

- The term multiplying the voltage at that node will be the sum of the conductances connected to that node. For the example above, the term multiplying  $V_b$  in the equation for node b is  $\frac{1}{R_1} + \frac{1}{R_3} + \frac{1}{R_4}$  while the term multiplying  $V_c$  in the equation for node c is  $\frac{1}{R_3} + \frac{1}{R_2} + \frac{1}{R_5}$ .
- The term multiplying the voltages adjacent to the node will be the negative of the conductance connecting the two nodes. For the example above, the term multiplying  $V_c$  in the equation for node b is  $-\frac{1}{R_3}$ , and the term multiplying  $V_b$  in the equation for node c is  $-\frac{1}{R_3}$ .

If the circuit contains dependent sources, or a different sign convention is used when writing the KCL equations, the resulting equations will not necessarily have the above form.

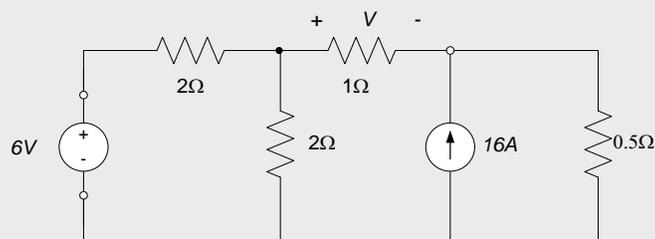
### Step 6: Solve the System of Equations Resulting from Step 5

Step 5 will always result in N equations in N unknowns, where N is the number of independent nodes identified in Step 2. These equations can be solved for the independent voltages. Any other desired circuit parameters can be determined from these voltages.

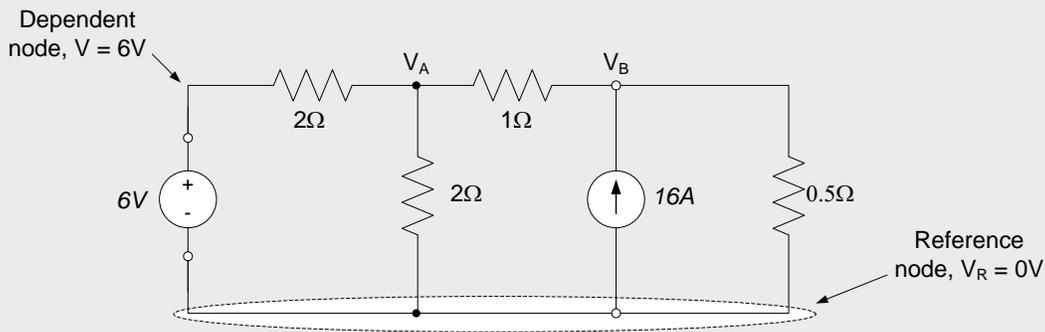
The example below illustrates the above approach.

### Example 3.1

Find the voltage V for the circuit shown below:



Steps 1, 2, and 3: Choosing the reference voltage as shown below, identifying voltages at dependent nodes, and defining voltages  $V_A$  and  $V_B$  at the independent nodes results in the circuit schematic shown below:



Steps 4 and 5: Writing KCL at nodes A and B and converting currents to voltages using Ohm's law results in the following two equations:

Node A:

$$\frac{V_A - 6}{2} + \frac{V_A - V_B}{2} + \frac{V_A - V_B}{1} = 0 \Rightarrow \left(\frac{1}{2} + \frac{1}{2} + \frac{1}{1}\right) V_A - V_B = 3 \Rightarrow 2V_A - V_B = 3$$

Node B:

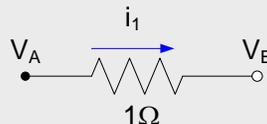
$$\frac{V_B - V_A}{1} + \frac{V_B - 0}{0.5} + 16 = 0 \Rightarrow \left(\frac{1}{1} + \frac{1}{0.5}\right) V_B - V_A = -16 \Rightarrow 3V_B = 16$$

Step 6: Solving the above equations results in  $V_A = 5V$  and  $V_B = 7V$ . The voltage  $V$  is:

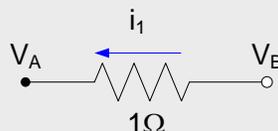
$$V = V_A - V_B = -2V$$

Several comments should be made relative to the above example:

- Steps 4 and 5 (applying KCL at each independent node and using Ohm's law to write these equations in terms of voltages) have been combined into a single step. This approach is fairly common, and can provide a significant savings in time.
- There may be a perceived inconsistency between the two node equations, in the assumption of positive current direction in the  $1\Omega$  resistor. In the equation for node A, the current is apparently assumed to be positive from node A to node B, as shown below:



This leads to the corresponding term in the equation for node A becoming:  $\frac{V_A - V_B}{1}$ . In the equation for node B, however, the positive current direction appears to be from node B to node A, as shown below:



This definition leads to the corresponding term in the equation for node B becoming:  $\frac{V_B - V_A}{1}$ .

The above inconsistency in sign is, however, insignificant. Suppose that we had assumed (consistently with the equation for node A) that the direction of positive current for the node B equation is from Node A to B. Then, the corresponding term in the equation for node B would have been:  $-\frac{V_A - V_B}{1}$  (note that a negative sign has been applied to this term to accommodate our assumption that currents flowing into nodes are negative). This is equal to  $\frac{V_A - V_B}{1}$ , which is exactly what our original result was.

- The current source appears directly in the nodal equations.

### Note

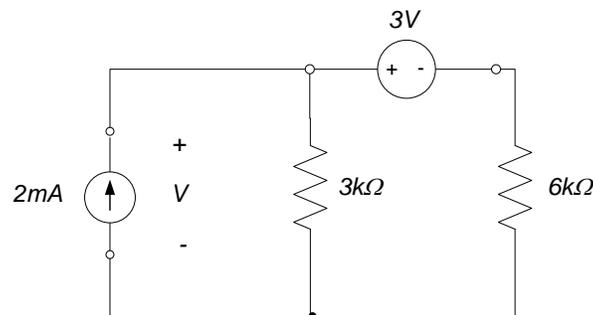
When we write nodal equations in these chapters, we will generally assume that any unknown currents are flowing away from the node for which we are writing the equation, regardless of any previous assumptions we have made for the direction of that current. The signs will work out, as long as we are consistent in our sign convention between assumed voltage polarity and current direction and our sign convention relative to positive currents flowing out of nodes.

The sign applied to currents induced by current sources must be consistent with the current direction assigned by the source.

## 3.2.1 Supernodes

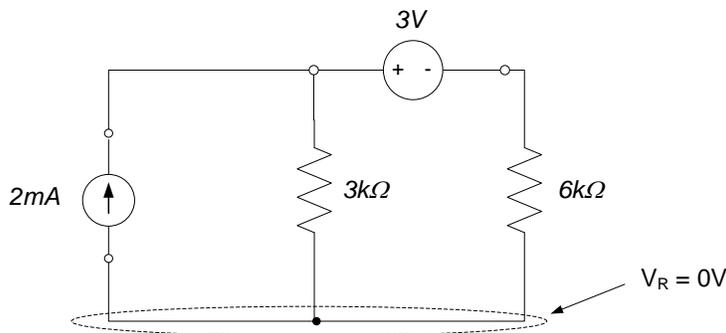
In the previous examples, we identified dependent nodes and determined constrained voltages. Kirchhoff's current law was then only written at independent nodes. Many readers find this somewhat confusing, especially if the dependent voltages are not relative to the reference voltage. We will thus discuss these steps in more detail here in the context of an example, introducing the concept of a *supernode* in the process.

Example: for the circuit below, determine the voltage difference,  $V$ , across the 2mA source.



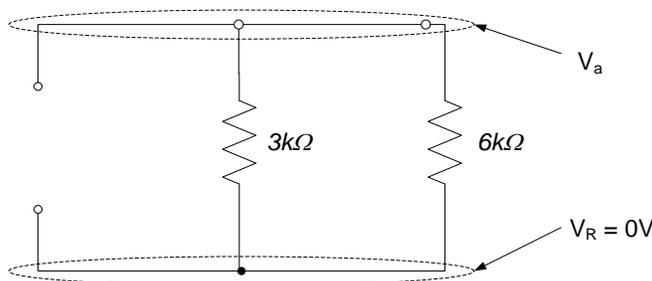
### Step 1: define Reference Node

Choose reference node (somewhat arbitrarily) as shown below; label the reference node voltage,  $V_R$ , as zero volts.



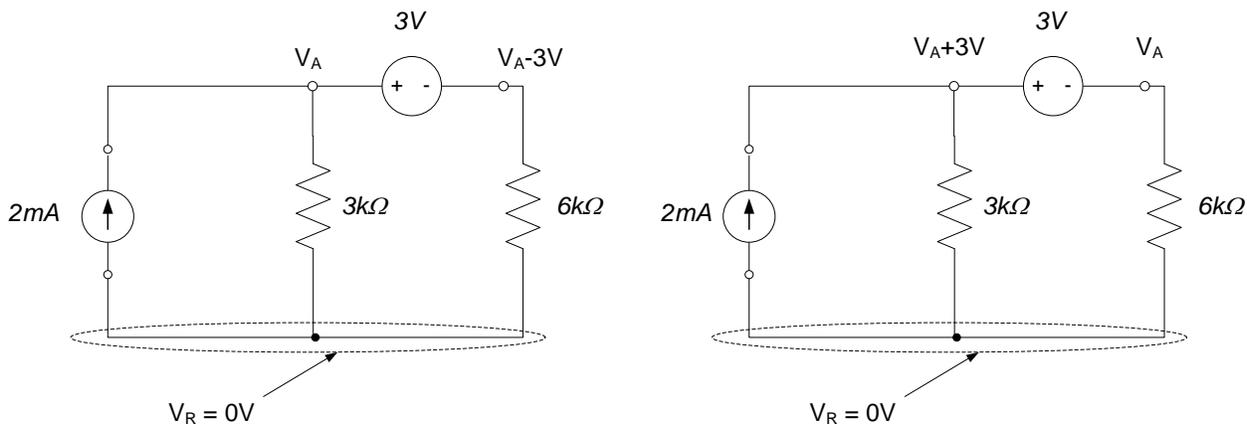
**Step 2: Define Independent Nodes**

Short circuit voltage sources, open circuit current sources as shown below and identify independent nodes/voltages. For our example, this result in only one independent voltage, labeled as  $V_A$  below.



**Step 3: Replace Sources and Label any Known Voltages**

The known voltages are written in terms of node voltages identified above. There is some ambiguity in this step. For example, either of the representations below will work equally well - either side of the voltage source can be chosen as the node voltage, and the voltage on the other side of the source written in terms of this node voltage. Make sure, however, that the correct polarity of the voltage source is preserved. In our example, the left side of the source has a potential that is three volts higher than the potential of the right side of the source. This fact is represented correctly by both of the choices below.



**Step 4: Apply KCL at the Independent Nodes**

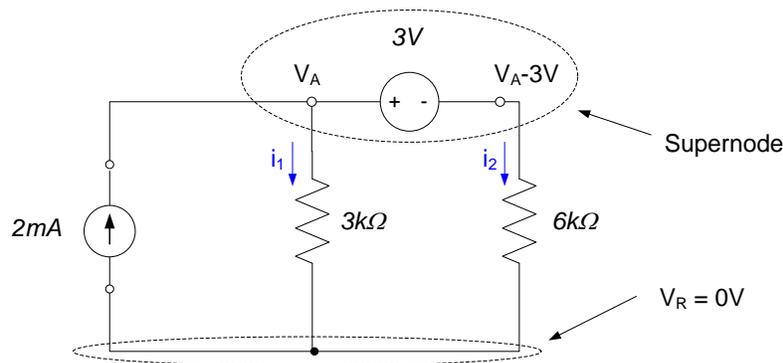
It is this step that sometimes causes confusion among readers, particularly when voltage sources are present in the circuit. Conceptually, it is possible to think of two nodes connected by an ideal voltage source as forming a single *supernode* (some authors use the term *generalized node* rather than supernode). A node is rigorously

defined as having a single, unique voltage. However, although the two nodes connected by a voltage source do not share the same voltage, they are not entirely independent - the two voltages are *constrained* by one another. This allows us to simplify the analysis somewhat.

For our example, we will arbitrarily choose the circuit to the left above to illustrate this approach. The supernode is chosen to include the voltage source and both nodes to which it is connected, as shown below. We define two currents leaving the supernode,  $i_1$  and  $i_2$ , as shown. KCL, applied at the supernode, results in:

$$-2mA + i_1 + i_2 = 0$$

As before, currents leaving the node are assumed to be positive. This approach allows us to account for the current flowing through the voltage source without ever explicitly solving for it.



### Step 5: Use Ohm's Law to Write the KCL Equations in Terms of Voltages

For the single KCL equation written above, this results in:

$$-2mA + \frac{V_A - 0}{3k\Omega} + \frac{(V_A - 3) - 0}{6k\Omega} = 0$$

### Step 6: Solve the System of Equations to Determine the Nodal Voltages

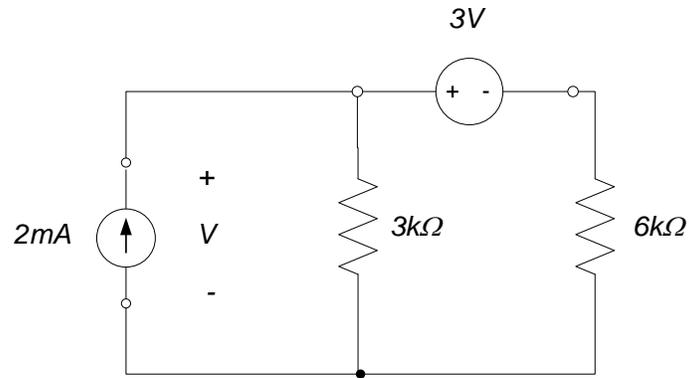
Solution of the equation above results in  $V_A = 5V$ . Thus, the voltage difference across the current source is  $V = 5V$ .

## 3.2.2 Alternate Approach: Constraint Equations

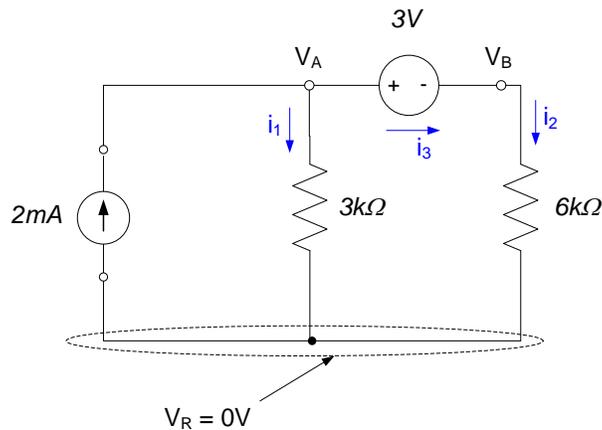
The use of supernodes can be convenient, but is not a necessity. An alternate approach, for those who do not wish to identify supernodes, is to restrain separate nodes on either side of the voltage source and then write a constraint equation relating these voltages. Thus, in cases where the reader does not recognize a supernode, the analysis can proceed correctly. We now revisit the previous example, but use constraint equations rather than the previous supernode technique.

In this approach, Steps 2 and 3 (identification of independent nodes) are not necessary. One simply writes Kirchhoff's current law at all nodes and then writes constraint equations for the voltage sources. A disadvantage of this approach is that currents through voltage sources must be accounted for explicitly; this results in a greater number of unknowns (and equations to be solved) than the supernode technique.

Example (revisited): for the circuit below, determine the voltage difference,  $V$ , across the 2mA source.



Choice of a reference voltage proceeds as previously. However, now we will not concern ourselves too much with identification of independent nodes. Instead, we will just make sure we account for voltages and currents everywhere in the circuit. For our circuit, this results in the node voltages and currents shown below. Notice that we have now identified two unknown voltages ( $V_A$  and  $V_B$ ) and three unknown currents, one of which ( $i_3$ ) is the current through the voltage source.



Now we write KCL at each of the identified nodes, making sure to account for the current through the voltage source. This results in the following equations (assuming currents leaving the node are positive):

$$\text{Node A: } -2mA + i_1 + i_3 = 0$$

$$\text{Node B: } -i_3 + i_2 = 0$$

Using Ohm's law to convert the currents  $i_1$  and  $i_2$  to voltages results in:

$$\text{Node A: } -2mA + \frac{V_A - 0}{3k\Omega} + i_3 = 0$$

$$\text{Node B: } -i_3 + \frac{V_B - 0}{6k\Omega} = 0$$

Notice that we cannot, by inspection, determine anything about the current  $i_3$  from the voltages; the voltage-current relationship for an ideal source is not known.

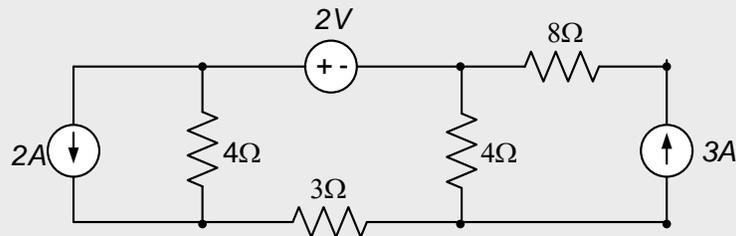
The two equations above have three unknowns - we cannot solve for the node voltages from them without a third equation. This third equation is the constraint equation due to the presence of the voltage source. For our circuit, the voltage source causes a direct relationship between  $V_A$  and  $V_B$ :

$$V_B = V_A - 3$$

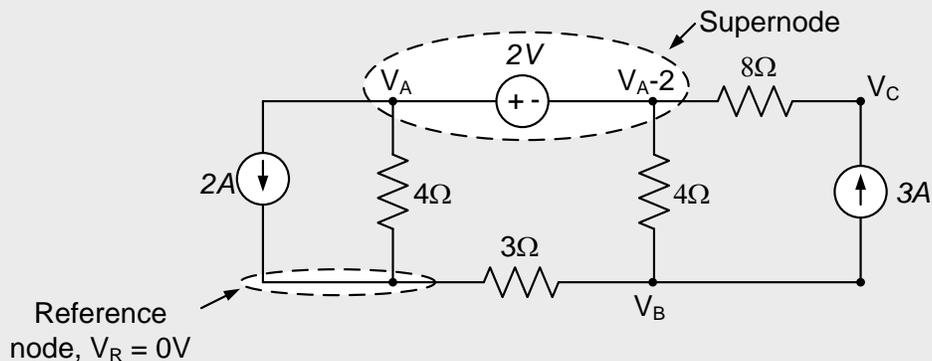
These three equations (the two KCL equations, written in terms of the node voltages and the constraint equation) constitute three equations in three unknowns. Solving these for the node voltage  $V_A$  results in  $V_A = 5V$ , so the voltage across the current source is  $V = 5V$ .

### Example 3.2

For the circuit below, find the power generated or absorbed by the 2V source and the power generated or absorbed by the 2A source.



*Steps 1, 2, and 3:* we choose our reference node (arbitrarily) as shown below. Shorting voltage sources and open-circuiting current sources identifies three independent node voltages (labeled below as  $V_A$ ,  $V_B$ , and  $V_C$ ) and one dependent node, with voltage labeled below as  $V_A - 2$ .



*Steps 4 and 5:* Writing KCL at nodes A, B, and C and converting the currents to voltages using Ohm's law results in the equations below. Note that we have (essentially) assumed that all unknown currents at a node are flowing out of the node, consistent with our node 2 for example 1 above.

Node A:

$$2A + \frac{V_A - 0V}{4\Omega} + \frac{(V_A - 2V) - V_B}{4\Omega} + \frac{(V_A - 2V) - V_C}{8\Omega} = 0 \Rightarrow 5V_A - 2V_B - V_C = -10$$

Node B:

$$\frac{V_B - 0V}{3\Omega} + \frac{(V_A - 2V)}{4\Omega} + 3A = 0 \Rightarrow 7V_B - 3V_A = 42$$

Node C:

$$\frac{V_C - (V_A - 2)}{8\Omega} - 3A = 0 \Rightarrow V_C - V_A = 22$$

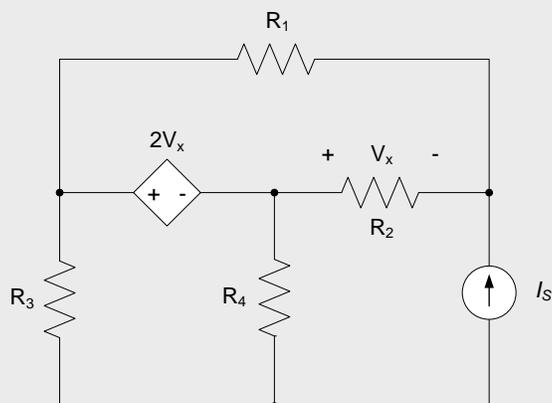
Step 6: Solving the above results in  $V_A = 0V$ ,  $V_B = -6V$ , and  $V_C = 22V$ . Thus, the voltage difference across the 2A source is zero volts, and the 2A source delivers no power. KCL at node A indicates that the current through the 2V source is 2A, and the 2V source generates 4W.

### 3.2.3 Dependent Sources

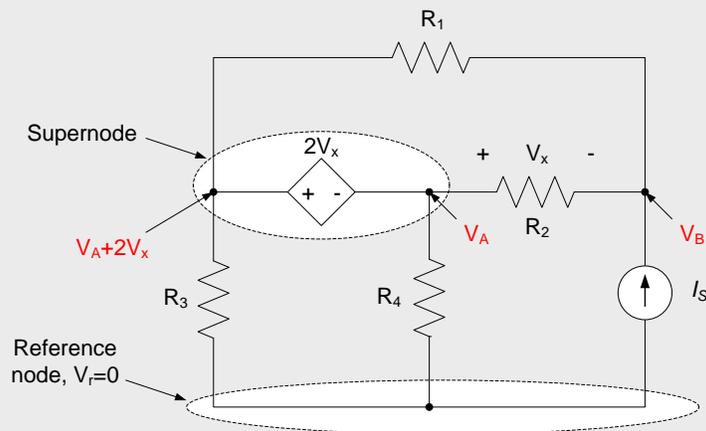
In the presence of dependent sources, nodal analysis proceeds approximately as outlined above. The main difference is the presence of additional equations describing the dependent source. As before, we will discuss the treatment of dependent sources in the context of examples.

#### Example 3.3

Write the nodal equations for the circuit below. The dependent source is a voltage controlled voltage source.  $I_S$  is an independent current source.



As always, the choice of reference node is arbitrary. To determine independent voltages, dependent voltage sources are short-circuited in the same way as independent voltage sources. Thus, the circuit below has two independent nodes; the dependent voltage source and the nodes on either side of it form a supernode. The reference voltage, independent voltages, supernode, and resulting dependent voltage are shown below.



We now, as previously, write KCL for each independent node, taking into account the dependent voltage resulting from the presence of the supernode:

$$\frac{(V_A - 2V_x) - 0}{R_3} + \frac{V_A - 0}{R_4} + \frac{V_A - V_B}{R_2} = 0$$

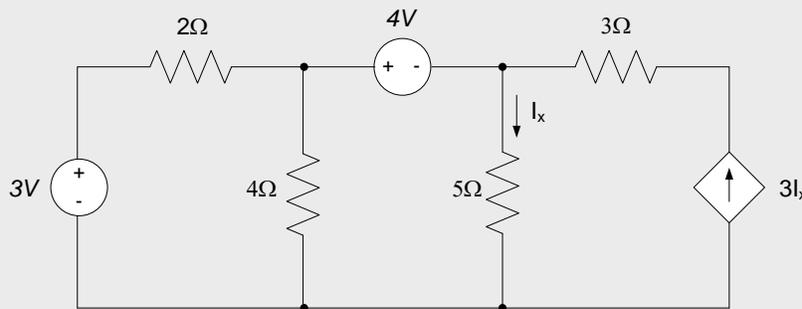
$$\frac{V_B - V_A}{R_2} + \frac{V_B - (V_A + 2V_X)}{R_1} - I_S = 0$$

The above equations result in a system with two equations and three unknowns:  $V_A$ ,  $V_B$ , and  $V_X$  ( $I_S$  is a known current). We now write any equations governing the dependent sources. Writing the controlling voltage in terms of the independent voltages results in:

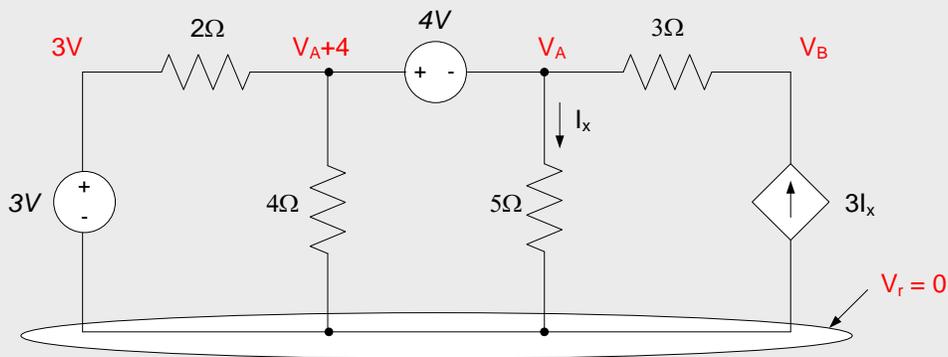
$$V_X = V_A - V_B$$

### Example 3.4

Write the nodal equations for the circuit below.



The reference node, independent voltages and dependent voltages are shown on the figure below. A supernode, consisting of the 4V source and the nodes on either side of it, exists but is not shown explicitly on the figure.



Applying KCL for each independent node results in:

$$\frac{(V_A + 4V) - 3V}{2\Omega} + \frac{(V_A + 4V) - 0}{4\Omega} + \frac{V_A - 0}{5\Omega} + \frac{V_A - V_B}{3\Omega} = 0$$

$$\frac{V_B - V_A}{3\Omega} - 3I_x = 0$$

This consists of two equations with three unknowns. The equation governing the dependent current source provides the third equation. Writing the controlling current in terms of independent voltages results in:

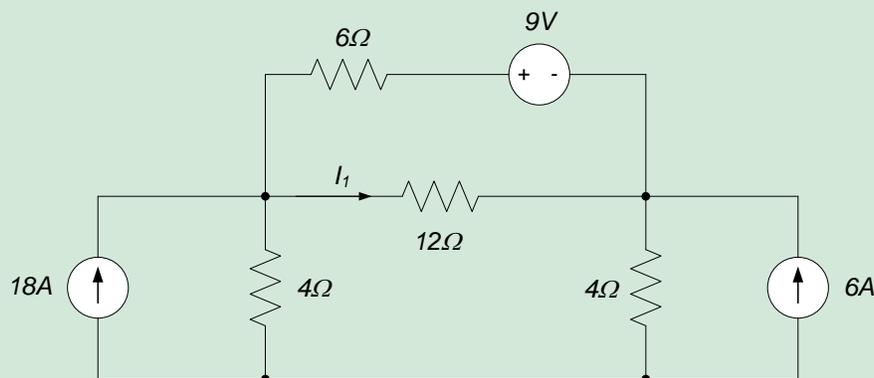
$$I_x = \frac{V_A - 0}{5\Omega}$$

## Section Summary

- Basic steps in nodal analysis are:
  - Define a reference node. All node voltages will be relative to this reference voltage.
  - Identify independent nodes. This can be done by short-circuiting voltage sources, open-circuiting current sources, and identifying the remaining nodes in the circuit. The voltages at these nodes are the node voltages.
  - Determine dependent voltages. This can be done by replacing the sources in the circuit schematic, and writing voltage constraints introduced by voltage sources.
  - Use Ohm's law to write KCL at each independent node, in terms of the node voltages. This will result in  $N$  equations in  $N$  unknowns, where  $N$  is the number of node voltages. Independent "nodes" can be *supernodes*; supernodes typically contain a voltage source; this minimizes the number of equations being written by taking advantage of voltage constraints introduced in step 3.
  - Solve the equations of step 4 to determine the node voltages.
  - Use the node voltages to determine any other desired voltages/currents in the circuit.
- Modifications to the above approach are allowed. For example, it is not necessary to define supernodes in step 4 above. One can define unknown voltages at either terminal of a voltage source and write KCL at each of these nodes. However, the unknown current through the voltage source must be accounted for when writing KCL - this introduces an additional unknown into the governing equations. This added unknown requires an additional equation. This equation is obtained by explicitly writing a constraint equation relating the voltages at the two terminals of the voltage source.

### 3.2 Exercises

1. Use nodal analysis to write a set of equations from which you can find  $I_1$ , the current through the  $12\Omega$  resistor. Do not solve the equations.



2. Use nodal analysis to find the current  $I$  flowing through the  $10\Omega$  resistor in the circuit below.

### 3.3 Mesh Analysis

In mesh analysis, we will define a set of mesh currents and use Ohm's law to write Kirchhoff's voltage law in terms of these voltages. The resulting set of equations can be solved to determine the mesh currents; any other circuit parameters (e.g. voltages) can be determined from these currents.

Mesh analysis is appropriate for *planar circuits*. Planar circuits can be drawn in a single plane<sup>5</sup> such that no elements overlap one another. Such circuits, when drawn in a single plane will be divided into a number of distinct areas; the boundary of each area is a *mesh* of the circuit. A *mesh current* is the current flowing around a mesh of the circuit.

The steps used in mesh analysis are provided below. The steps are illustrated in terms of the circuit of Fig. 3.8.

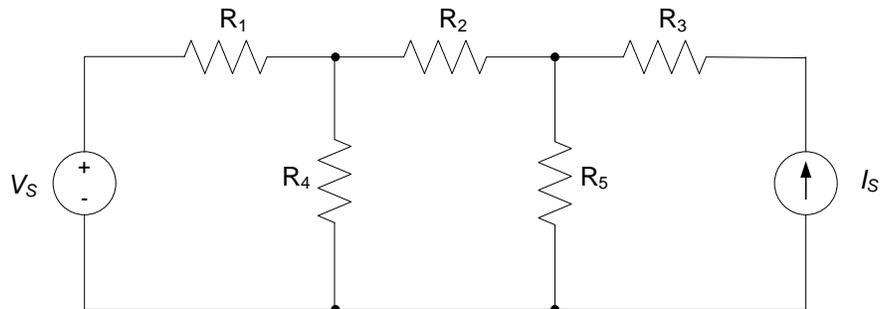


Figure 3.8. Example circuit.

### Step 1: Define Mesh Currents

In order to identify our mesh loops, we will turn off all sources, much like what we did in nodal analysis. To do this, we:

- Short-circuit all voltage sources.
- Open-circuit all current sources.

Once the sources have been turned off, the circuit can be divided into a number of non-overlapping areas, each of which is completely enclosed by circuit elements. The circuit elements bounding each of these areas form the meshes of our circuit. The mesh currents flow around these meshes. Our example circuit has two meshes after removal of the sources, the resulting mesh currents are as shown in Fig. 3.9.

### Note

We will always choose our mesh currents as flowing clockwise around the meshes. This assumption is not fundamental to the application of mesh analysis, but it will result in a special form for the resulting equations which will later allow us to do some checking of our results.

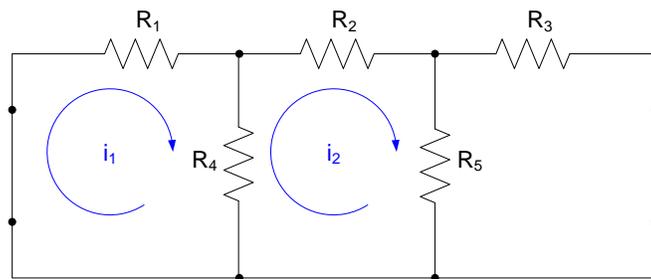


Figure 3.9. Example circuit meshes.

<sup>5</sup> Essentially, you can draw the schematic on a piece of paper without ambiguity.

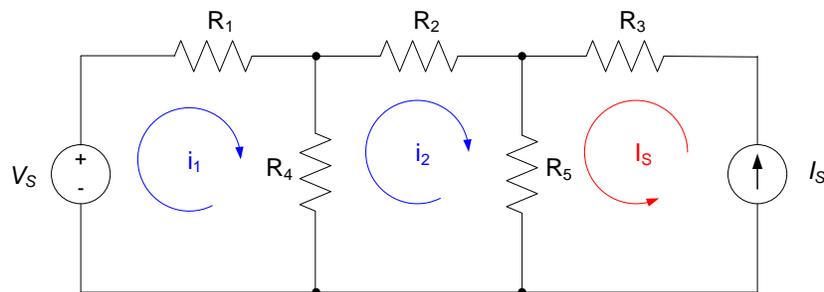
### Step 2: Replace Sources and Identify Constrained Loops

The presence of current sources in our circuit will result in the removal of some meshes during Step 1. We must now account for these meshes in our analysis by returning the sources to the circuit and identifying *constrained* loops.

We have two rules for constrained loops:

3. Each current must have one and only one constrained loop passing through it.
4. The direction and magnitude of the constrained loop current must agree with the direction and magnitude of the source current.

For our example circuit, we choose our constrained loop as shown below. It should be noted that constrained loops can, if desired, cross our mesh loops - we have, however, chosen the constrained loop so that it does not overlap any of our mesh loops.



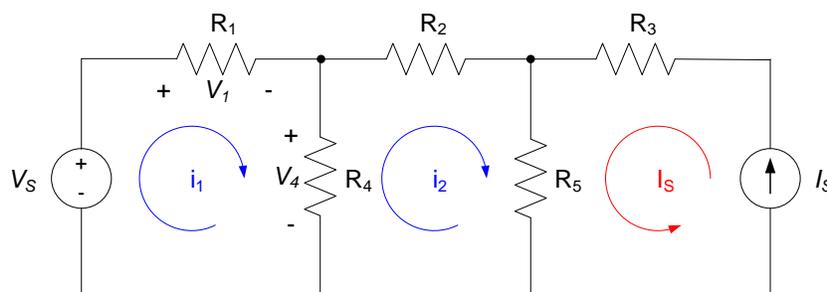
### Step 3: Write KVL Around the Mesh Loops

We will apply Kirchhoff's voltage law around each mesh loop in order to determine the equations to be solved. Ohm's law will be used to write KVL in terms of the mesh currents and constrained loop currents as identified in Steps 1 & 2 above.

Note that more than one mesh current may pass through a circuit element. When determining voltage drops across individual elements, the contributions from all mesh currents passing through that element must be included in the voltage drop.

When we write KVL for a given mesh loop, we will base our sign convention for the voltage drops on the direction of the mesh current for that loop.

For example, when we write KVL for the mesh current  $i_1$  in our example, we choose voltage polarities for resistors  $R_1$  and  $R_4$  as shown in the figure below - these polarities agree with the passive sign convention for voltages relative to the direction of the mesh current  $i_1$ .



From the above figure, the voltage drops across the resistor  $R_1$  can then be determined as:

$$V_1 = R_1 i_1$$

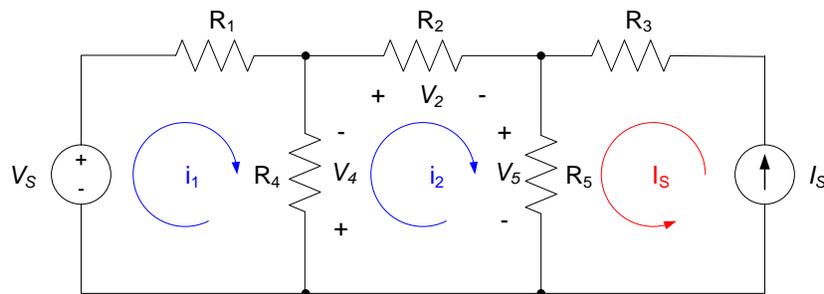
Since only mesh current  $i_1$  passes through the resistor  $R_1$ . Likewise, the voltage drop for the resistor  $R_4$  is

Since mesh currents  $i_1$  and  $i_4$  both pass through  $R_4$  and the current  $i_2$  is in the opposite direction to our assumed polarity for the voltage  $V_4$ .

Using the above expressions for  $V_1$  and  $V_4$ , we can write KVL for the first mesh loop as:

$$-V_S + R_1 i_1 + R_4(i_1 - i_2) = 0$$

When we write KVL for the mesh current  $i_2$  in our example, we choose voltage polarities for resistors  $R_4$ ,  $R_2$ , and  $R_5$  as shown in the figure below – these polarities agree with the passive sign convention for voltages relative to the direction of the mesh current  $i_2$ . Please note that these sign conventions do not need to agree with the sign conventions used in the equations for other mesh currents.



Using the above sign conventions, KVL for the second mesh loop becomes:

$$R_4(i_2 - i_1) + R_2 i_2 + R_5(i_2 + I_S) = 0$$

Please note that the currents  $i_2$  and  $I_S$  are in the same direction in the resistor  $R_5$ , resulting in a summation of these currents in the term corresponding to the voltage drop across the resistor  $R_5$ .

#### Notes:

1. Assumed sign conventions on voltage drops for a particular mesh loop are based on the assumed direction of that loop's mesh current.
2. The current passing through an element is the algebraic sum of all mesh and constraint currents passing through that element. This algebraic sum of currents is used to determine the voltage drop of the element.

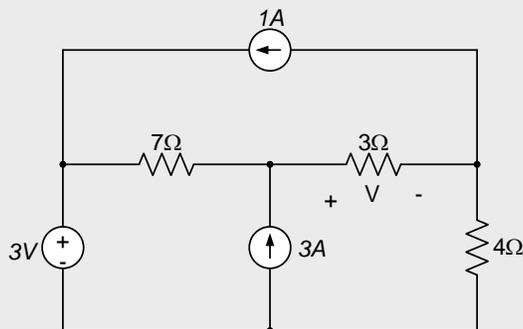
#### Step 4: Solve the System of Equations to Determine the Mesh Currents of the Circuit

Step 3 will always result in  $N$  equations in  $N$  unknowns, where  $N$  is the number of mesh currents identified in Step 1. These equations can be solved for the mesh currents. Any other desired circuit parameters can be determined from the mesh currents.

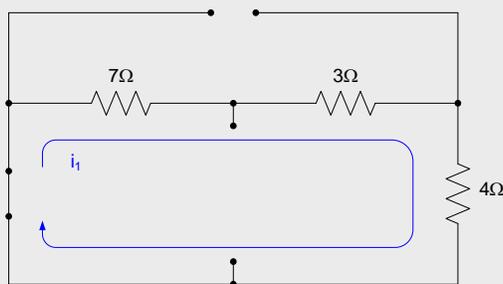
The example below illustrates the above approach.

#### Example 3.5

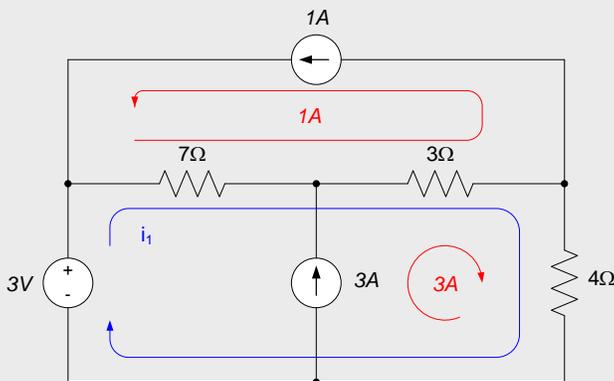
In the circuit below, determine the voltage drop,  $V$ , across the  $3\Omega$  resistor.



Removing the sources results in a single mesh loop with mesh current  $i_1$ , as shown below.



Replacing the sources and defining one constrained loop per source results in the loop definitions shown below (note that each constrained loop goes through only one source and that the amplitude and direction of the constrained currents agrees with source).



Applying KVL around the loop  $i_1$  and using Ohm's law to write voltage drops in terms of currents:

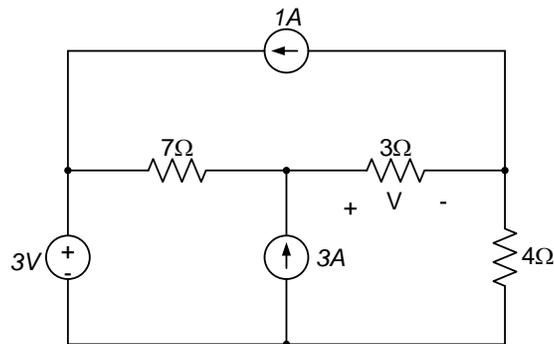
$$-3V + 7\Omega(i_1 + 1A) + 3\Omega(i_1 + 1A + 3A) + 4\Omega(i_1 + 3A) = 0 \Rightarrow i_1 = -2A$$

Thus, the current  $i_1$  is 2A, in the opposite direction to that shown. The voltage across the 3Ω resistor is  $V = 3\Omega(i_1 + 3A + 1A) = 3\Omega(-2A + 3A + 1A) = 3(2A) = 6V$ .

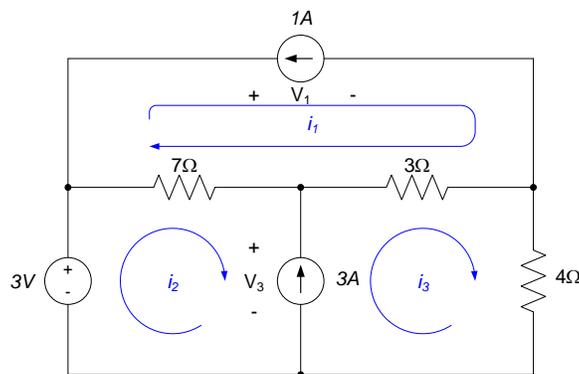
### 3.3.1 Alternate Approach to Constraint Loops: Constraint Equations

In the above examples, the presence of current sources resulted in a reduced number of meshes. Constraint loops were then used to account for current sources. An alternate approach, in which we retain additional mesh currents and then apply *constraint equations* to account for the current sources, is provided here. We use the circuit of the previous example to illustrate this approach.

Example: determine the voltage,  $V$ , in the circuit below.



Define three mesh currents for each of the three meshes in the above circuit and define unknown voltages  $V_1$  and  $V_3$  across the two current sources as shown below.



Applying KVL around the three mesh loops results in three equations with five unknowns:

$$V_1 + 3\Omega \cdot (i_1 - i_3) + 7\Omega \cdot (i_1 - i_2) = 0$$

$$-3V + 7\Omega \cdot (i_2 - i_1) + V_3 = 0$$

$$-V_3 + 3\Omega \cdot (i_3 - i_1) + 4\Omega \cdot i_3 = 0$$

Two additional *constraint equations* are necessary. These can be determined by the requirement that the algebraic sum of the mesh currents passing through a current source must equal the current provided by the source. Thus, we obtain:

$$-i_2 + i_3 = 3A$$

$$-i_1 = 1A$$

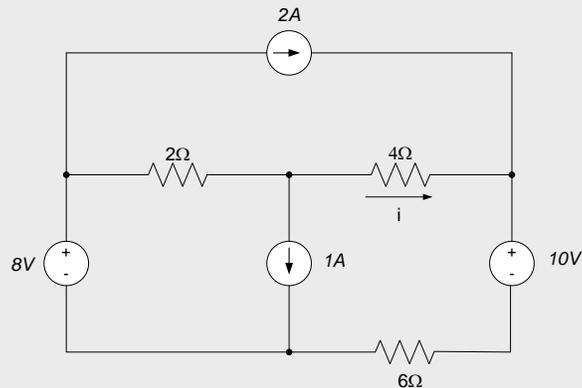
Solving the five simultaneous equations above results in the same answer determined previously.

### 3.3.2 Clarification: Constraint Loops

Previously, it was claimed that the choice of constraint loops is somewhat arbitrary. The requirements are that each source has only one constraint loop passing through it, and that the magnitude and direction of the constrained loop current be consistent with the source. Since constraint loops can overlap other mesh loops without invalidating the mesh analysis approach, the choice of constraint loops is not unique. The examples below illustrate the effect of different choices of constraint loops on the analysis of a particular circuit.

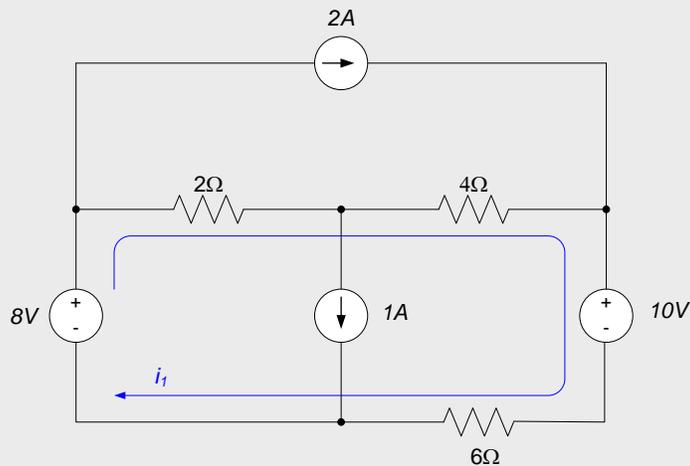
### Example 3.6: Version 1

Using mesh analysis, determine the current,  $i$ , through the  $4\Omega$  resistor.



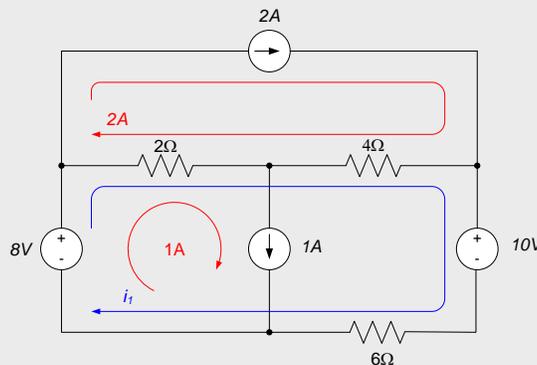
#### Step 1: Define mesh loops

Replacing the two current sources with open circuits and the two voltage sources with short circuits results in a single mesh current,  $i_1$ , as shown below.



#### Step 2: Constrained loops, version 1

Initially, we choose the constrained loops shown below. Note that each loop passes through only one source and has the magnitude and direction imposed by the source.



**Step 3: Write KVL around the mesh loops**

Our example has only one mesh current, so only one KVL equation is required. This equation is:

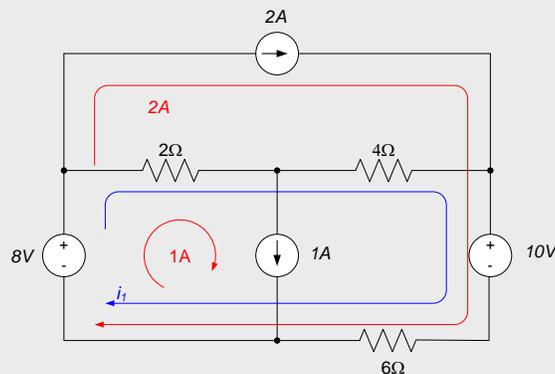
$$-8V + 2\Omega(i_1 + 1A - 2A) + 4\Omega(i_1 - 2A) + 10V + 6\Omega(i_1) = 0$$

**Step 4: Solve the system of equations to determine the mesh currents of the circuit**

Solving the above equation results in  $i_1 = 0.667A$ . The current through the  $4\Omega$  resistor is then, accounting for the  $2A$  constrained loop passing through the resistor,  $i = i_1 - 2A = -1.333A$ .

**Example 3.6: Version 2**

In this version, we choose an alternate set of constraint loops. The alternate set of loops is shown below; all constraint loops still pass through only one current source, and retain the magnitude and direction of the source current.



Now, writing KVL for the single mesh results in:

$$-8V + 2\Omega(i_1 + 1A) + 4\Omega \cdot i_1 + 10V + 6\Omega(i_1 + 2A) = 0$$

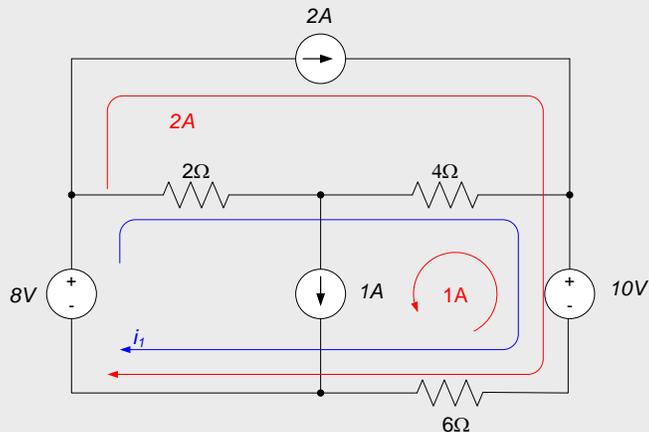
Solving for the mesh current results in  $i_1$ ; note that this result is different than previously. However, we determine the current through the  $4\Omega$  resistor as  $i = i_1 = -1.333A$ , which is the same result as previously.

**Note**

Choice of alternate constrained loops may change the values obtained for the mesh currents. The currents through the circuit elements, however, do not vary with choice of constrained loops.

**Example 3.6: Version 3**

In this version, we choose yet another set of constrained loops. These loops are shown below. Again, each loop passes through one current source and retains that source's current direction and amplitude.



KVL around the mesh loop results in:

$$-8V + 2\Omega \cdot i_1 + 4\Omega(i_1 - 1A) + 10V + 6\Omega(i_1 - 1A + 2A) = 0$$

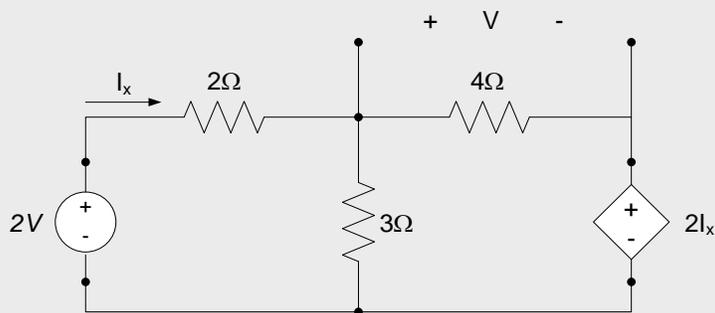
Which results in  $i_1 = -0.333A$ . Again, this is different from the result from our first two approaches. However, the current through the  $4\Omega$  resistor is  $i = i_1 - 1A = -1.333A$ , which is the same result as previously.

### 3.3.3 Dependent Sources

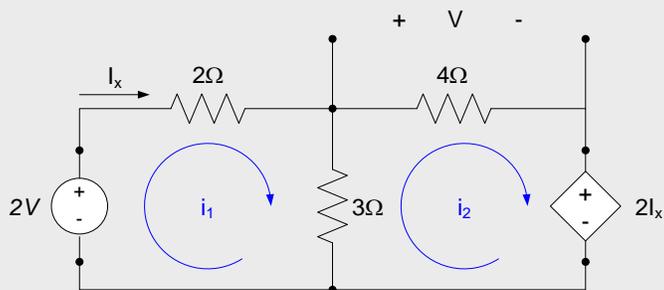
As with nodal analysis, the presence of dependent sources does not significantly alter the overall mesh analysis approach. The primary difference is simply the addition of the additional equations necessary to describe the dependent sources. We discuss the analysis with dependent sources in the context of the following examples.

#### Example 3.7

Determine the voltage  $V$  in the circuit below.



Shorting both of the voltage sources in the circuit above results in two mesh circuits. These are shown in the figure below.



Writing KVL around the two mesh loops results in:

$$-2V + 2\Omega \cdot i_1 + 3\Omega(i_1 - i_2) = 0$$

$$2I_x + 3\Omega(i_2 - i_1) + 4\Omega \cdot i_2 = 0$$

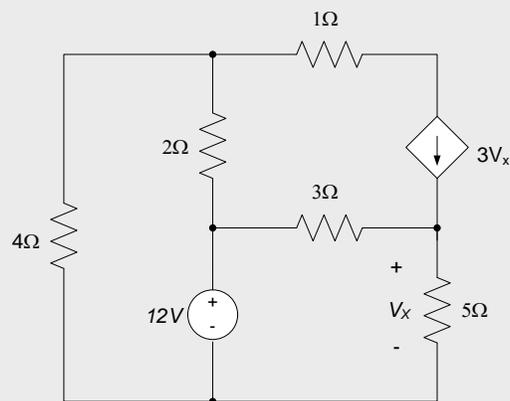
We have two equations and three unknowns. We need an additional equation to solve the system of equations. The third equation is obtained by writing the dependent source's controlling current in terms of the mesh currents:

$$I_x = i_1$$

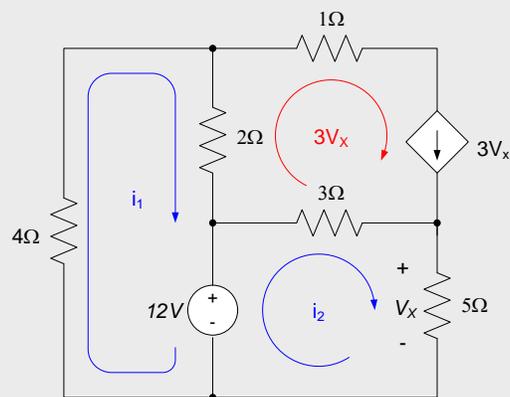
The above three equations can be solved to obtain  $i_1 = 0.4375\text{A}$  and  $i_2 = 0.0625\text{A}$ . The desired voltage  $V = 4i_2 = 0.25\text{V}$ .

### Example 3.8

Write mesh equations for the circuit shown below.



Mesh loops and constraint loops are identified as shown below:



Writing KVL for the two mesh loops results in:

$$4\Omega \cdot i_1 + 2\Omega(i_1 - 3V_x) + 12V = 0$$

$$-12V + 3\Omega(i_2 - 3V_x) + 5\Omega \cdot i_2 = 0$$

Writing the controlling voltage  $V_x$  in terms of the mesh currents results in:

$$V_X = 5\Omega \cdot i_2$$

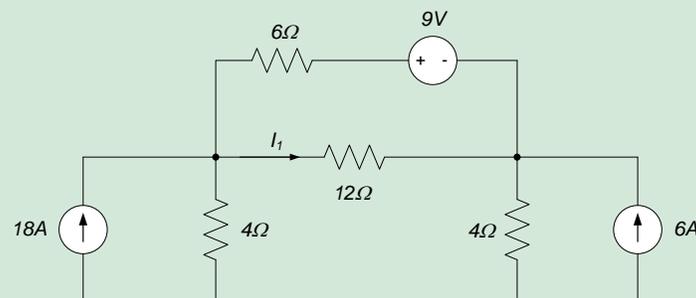
The above consist of three equations in three unknowns, which can be solved to determine the mesh currents. Any other desired circuit parameters can be determined from the mesh currents.

## Section Summary

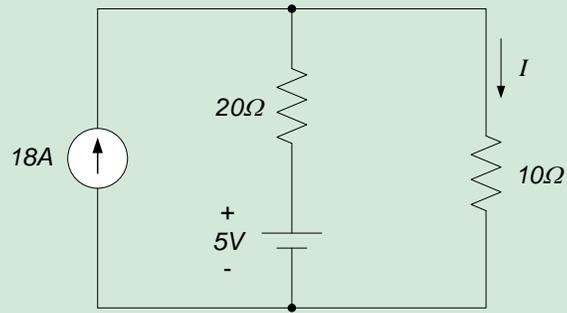
- Basic steps in mesh analysis are:
  - Identify mesh currents. This can be done by short-circuiting voltage sources, open-circuiting current sources, and identifying the enclosed, non-overlapping regions in the circuit. The perimeters of these areas are the circuit meshes. The mesh currents flow around the circuit meshes.
  - Determine constrained loops. The approach in Step 1 will ensure that no mesh currents will pass through the current sources. The current source currents can be accounted for by defining constrained loops. Constrained loops are defined as loop currents which pass through the current sources. Constrained loops are identified by replacing the sources in the circuit schematic, and defining mesh currents which pass through the current sources; these mesh currents form the constrained loops and must match both the magnitude and direction of the current in the current sources.
  - Use Ohm's law to write KVL around each mesh loop, in terms of the mesh currents. This results in N equations in N unknowns, where N is the number of mesh currents. Keep in mind that the voltage difference across each element must correspond to the voltage difference induced by all the mesh currents which pass through that element.
  - Solve the equations of Step 4 to determine the mesh currents.
  - Use the mesh currents to determine any other desired voltages/currents in the circuit.
- The constrained loops in Step 2 above are not unique. Their only requirement is that they must account for the currents through the current sources.
- Modifications to the above approach are allowed. For example, it is not necessary to define constrained loops in Step 3 above. One can define (unknown) mesh currents which pass through the current sources and write KVL for these additional mesh currents. However, the unknown voltage across the current source must be accounted for when writing KVL - this introduces an additional unknown into the governing equations. This added unknown requires an additional equation which is obtained by explicitly writing a constraint equation equating algebraic sum of the mesh currents passing through a current source to the current provided by the source.

## 3.3 Exercises

1. Use mesh analysis to write a set of equations from which you can find  $I_1$ , the current through the  $12\Omega$  resistor. Do not solve the equations.



2. Use mesh analysis to find the current  $I$  flowing through the  $10\Omega$  resistor in the circuit below. Compare your result to your solution to exercise 2 of section 3.2.



## Real Analog Chapter 3: Lab Projects

### 3.2.1: Nodal Analysis

In this lab, we will analyze, build, and test a circuit containing multiple sources. Except in special cases, multiple sources preclude the use of analysis techniques based entirely on circuit reduction approaches. In this assignment, we will use nodal analysis to predict the circuit behavior prior to building and testing the circuit. We will then compare the measured circuit response with our expectations based on our analysis.

It should be noted that analysis techniques other than nodal analysis can be used to predict the circuit's response. The approach used to analyze the circuit does not, however, affect the actual circuit's behavior. Other lab assignments will use this same circuit, in conjunction with other analysis techniques. Please keep in mind that the circuit's physical behavior is not affected by the analysis technique used; the same measurement on the same circuit should provide the same result, regardless of the approach used to predict the response.

#### Before beginning this lab, you should be able to:

- Predict node voltages in a circuit using nodal analysis techniques
- Use node voltages to predict physical voltages and currents in an electric circuit
- Use a digital multimeter to measure resistance, voltage, and current
- Use the Analog Discovery 2 module to apply and measure constant voltages

#### After completing this lab, you should be able to:

- Compare measured voltages and currents in an electrical circuit with predictions based on nodal analysis techniques

#### This lab exercise requires:

- Analog Discovery 2 module
- Digilent Analog Parts Kit
- Digital multimeter

#### Symbol Key:

- |   |   |
|---|---|
|  | Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable. |
|  | Analysis; include principle results of analysis in laboratory report.   |
|  | Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.                      |
|  | Record data in your lab notebook.   |

#### General Discussion:

This lab assignment concerns the circuit shown in Fig. 1 below. Use V+ to apply the 5V voltage source and V- to apply the -5V voltage. The waveform generator can be used to apply the -3V source. (Make sure that you pay attention to the polarity of the applied voltages and use the sources available on the Analog Discovery 2 appropriately!) We wish to determine the voltage differences  $V_1$  and  $V_2$ .

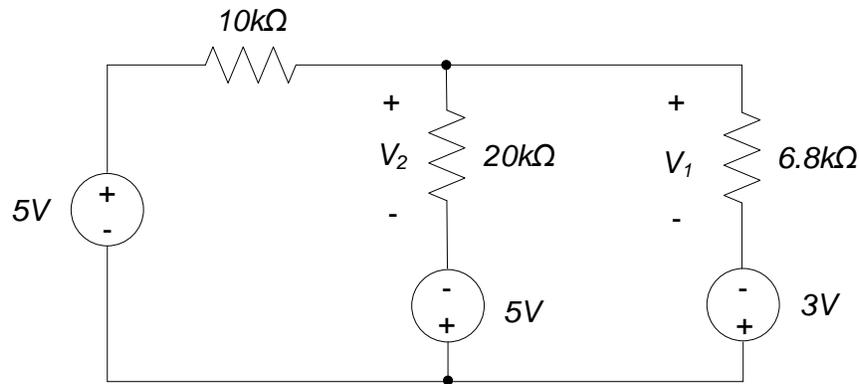


Figure 1. Circuit schematic.

### Pre-lab:

**ANALYSIS**

Use nodal analysis techniques to predict the voltages  $V_1$  and  $V_2$  in the circuit of Fig. 1.

### Lab Procedures:

**DATA**

1. Construct the circuit of Fig. 1; measure and record all actual resistance values. Measure the voltages  $V_1$  and  $V_2$  in the circuit. Calculate a percent error between the measured voltages and your predictions from the pre-lab.

**ANALYSIS****DEMO**

2. Demonstrate operation of your circuit to the Teaching Assistant. Have the TA initial the appropriate page(s) of your lab notebook and the lab checklist.



## Real Analog Chapter 3: Lab Projects

### 3.2.2: Nodal Analysis

In this lab, we will analyze, build, and test a circuit containing multiple sources. Except in special cases, multiple sources preclude the use of analysis techniques based entirely on circuit reduction approaches. In this assignment, we will use nodal analysis to predict the circuit behavior prior to building and testing the circuit. We will then compare the measured circuit response with our expectations based on our analysis.

It should be noted that analysis techniques other than nodal analysis can be used to predict the circuit's response. The approach used to analyze the circuit does not, however, affect the actual circuit's behavior. Other lab assignments will use this same circuit, in conjunction with other analysis techniques. Please keep in mind that the circuit's physical behavior is not affected by the analysis technique used; the same measurement on the same circuit should provide the same result, regardless of the approach used to predict the response.

#### Before beginning this lab, you should be able to:

- Predict node voltages in a circuit using nodal analysis techniques
- Use node voltages to predict physical voltages and currents in an electric circuit
- Use a digital multimeter to measure resistance, voltage, and current
- Use the Analog Discovery 2 module to apply and measure constant voltages

#### After completing this lab, you should be able to:

- Compare measured voltages and currents in an electrical circuit with predictions based on nodal analysis techniques

#### This lab exercise requires:

- Analog Discovery 2 module
- Digilent Analog Parts Kit
- Digital multimeter

#### Symbol Key:

- |   |   |
|---|---|
|  | Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable. |
|  | Analysis; include principle results of analysis in laboratory report.   |
|  | Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.                      |
|  | Record data in your lab notebook.   |

#### General Discussion:

This lab assignment concerns the circuit shown in Fig. 1 below. We want to determine the voltage difference  $V_1$  and the current  $I_1$ .

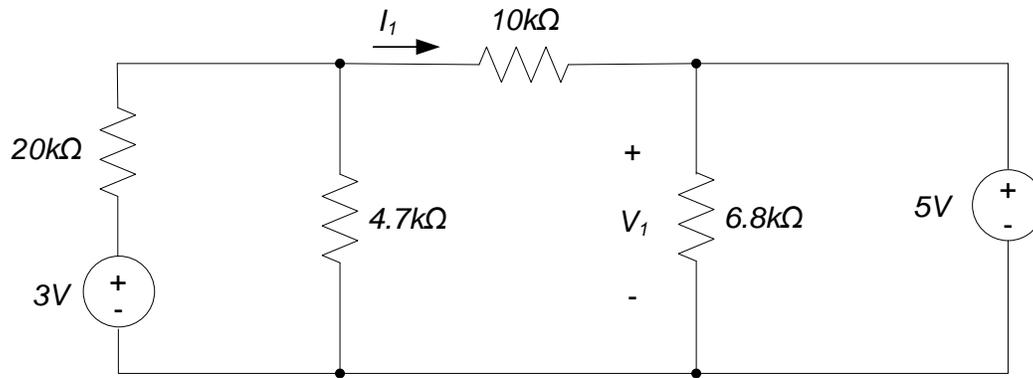


Figure 1. Circuit schematic.

### Pre-lab:

**ANALYSIS**

Use nodal analysis techniques to predict the voltage  $V_1$  and the current  $I_1$  in the circuit of Fig. 1.

### Lab Procedures:

**DATA****ANALYSIS****DEMO**

1. Construct the circuit of Fig. 1; measure and record all actual resistance values. Use whichever voltage supplies you want to implement the voltage sources in the circuit. Measure the voltage  $V_1$  and the current  $I_1$  in the circuit. Calculate a percent error between the measured values and your predictions from the pre-lab.
2. Demonstrate operation of your circuit to the Teaching Assistant. Have the TA initial the appropriate page(s) of your lab notebook and the lab checklist.



## Real Analog Chapter 3: Lab Projects

### 3.2.3: Nodal Analysis

In this lab, we will analyze, build, and test a circuit containing multiple sources. Except in special cases, multiple sources preclude the use of analysis techniques based entirely on circuit reduction approaches. In this assignment, we will use nodal analysis to predict the circuit behavior prior to building and testing the circuit. We will then compare the measured circuit response with our expectations based on our analysis.

It should be noted that analysis techniques other than nodal analysis can be used to predict this circuit's response. The approach used to analyze the circuit does not, however, affect the actual circuit's behavior. Other lab assignments will use this same circuit, in conjunction with other analysis techniques. Please keep in mind that the circuit's physical behavior is not affected by the analysis technique used; the same measurement on the same circuit should provide the same result, regardless of the approach used to predict the response.

#### Before beginning this lab, you should be able to:

- Predict mesh currents in a circuit using mesh analysis
- Use mesh currents to predict physical voltages and currents in an electric circuit
- Use a digital multimeter to measure resistance, voltage, and current
- Use the Analog Discovery 2 module to apply and measure constant voltages

#### After completing this lab, you should be able to:

- Compare measured voltages and currents in an electrical circuit with predictions based on nodal analysis techniques

#### This lab exercise requires:

- Analog Discovery 2 module
- Digilent Analog Parts Kit
- Digital multimeter

#### Symbol Key:

- |   |   |
|---|---|
|  | Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable. |
|  | Analysis; include principle results of analysis in laboratory report.   |
|  | Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.                      |
|  | Record data in your lab notebook.   |

#### General Discussion:

This lab assignment concerns the circuit shown in Fig. 1 below. Use whichever voltage supplies you want to implement the voltage sources in the circuit. We want to determine the voltage difference  $V_1$  and the current  $I_1$ .

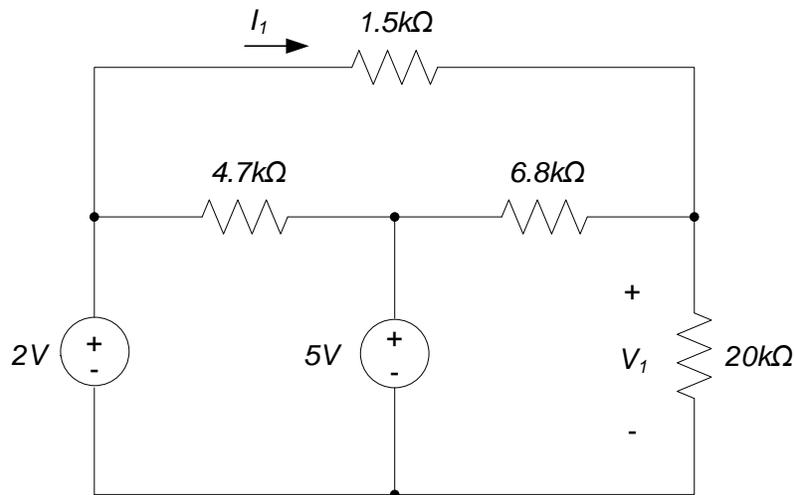


Figure 1. Circuit schematic.

### Pre-lab:

#### ANALYSIS

Use nodal analysis techniques to predict the voltage  $V_1$  and the current  $I_1$  in the circuit of Fig. 1.

### Lab Procedures:

#### DATA

1. Construct the circuit of Fig. 1; measure and record all actual resistance values. Measure the voltage  $V_1$  and the current  $I_1$  in the circuit. Calculate a percent error between the measured values and your predictions from the pre-lab.

#### ANALYSIS

#### DEMO

2. Demonstrate operation of your circuit to the Teaching Assistant. Have the TA initial the appropriate page(s) of your lab notebook and the lab checklist.



## Real Analog Chapter 3: Lab Projects

### 3.3.1: Mesh Analysis

In this lab, we will analyze, build, and test a circuit containing multiple sources. Except in special cases, multiple sources preclude the use of analysis techniques based entirely on circuit reduction approaches. In this assignment, we will use mesh analysis to predict the circuit behavior prior to building and testing the circuit. We will then compare the measured circuit response with our expectations based on our analysis.

It should be noted that analysis techniques other than mesh analysis can be used to predict this circuit's response. The approach used to analyze the circuit does not, however, affect the actual circuit's behavior. Other lab assignments use this same circuit, in conjunction with other analysis techniques. Please keep in mind that the circuit's physical behavior is not affected by the analysis technique used; the same measurement on the same circuit should provide the same result, regardless of the approach used to predict the response.

#### Before beginning this lab, you should be able to:

- Predict mesh currents in a circuit using mesh analysis
- Use mesh currents to predict physical voltages and currents in an electric circuit
- Use a digital multimeter to measure resistance, voltage, and current
- Use the Analog Discovery 2 module to apply and measure constant voltages

#### After completing this lab, you should be able to:

- Compare measured voltages and currents in an electrical circuit with predictions based on mesh analysis techniques

#### This lab exercise requires:

- Analog Discovery 2 module
- Digilent Analog Parts Kit
- Digital multimeter

#### Symbol Key:

- |   |   |
|---|---|
|  | Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable. |
|  | Analysis; include principle results of analysis in laboratory report.   |
|  | Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.                      |
|  | Record data in your lab notebook.   |

#### General Discussion:

This lab assignment concerns the circuit shown in Fig. 1 below. Use V+ to apply the 5V voltage source and V- to apply the -5V voltage. The waveform generator can be used to apply the -3V source. (Pay special attention to the polarity of the sources!) We wish to determine the voltage differences  $V_1$  and  $V_2$ .

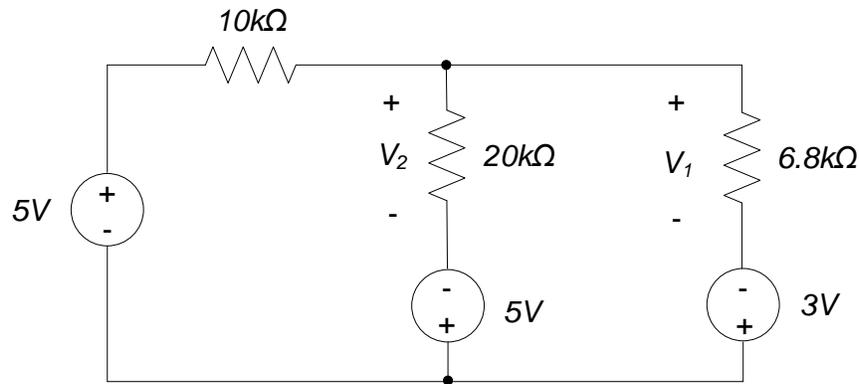


Figure 1. Circuit schematic.

### Pre-lab:

**ANALYSIS**

Use mesh analysis techniques to predict the voltages  $V_1$  and  $V_2$  in the circuit of Fig. 1.

### Lab Procedures:

**DATA**

1. Construct the circuit of Fig. 1; measure and record all actual resistance values. Measure the voltages  $V_1$  and  $V_2$  in the circuit. Calculate a percent error between the measured voltages and your predictions from the pre-lab.

**ANALYSIS****DEMO**

2. Demonstrate operation of your circuit to the Teaching Assistant. Have the TA initial the appropriate page(s) of your lab notebook and the lab checklist.



## Real Analog Chapter 3: Lab Projects

### 3.3.2: Mesh Analysis

In this lab, we will analyze, build, and test a circuit containing multiple sources. Except in special cases, multiple sources preclude the use of analysis techniques based entirely on circuit reduction approaches. In this assignment, we will use mesh analysis to predict the circuit behavior prior to building and testing the circuit. We will then compare the measured circuit response with our expectations based on our analysis.

It should be noted that analysis techniques other than mesh analysis can be used to predict this circuit's response. The approach used to analyze the circuit does not, however, affect the actual circuit's behavior. Other lab assignments use this same circuit, in conjunction with other analysis techniques. Please keep in mind that the circuit's physical behavior is not affected by the analysis technique used; the same measurement on the same circuit should provide the same result, regardless of the approach used to predict the response.

#### Before beginning this lab, you should be able to:

- Predict mesh currents in a circuit using mesh analysis
- Use mesh currents to predict physical voltages and currents in an electric circuit
- Use a digital multimeter to measure resistance, voltage, and current
- Use the Analog Discovery 2 module to apply and measure constant voltages

#### After completing this lab, you should be able to:

- Compare measured voltages and currents in an electrical circuit with predictions based on mesh analysis techniques

#### This lab exercise requires:

- Analog Discovery 2 module
- Digilent Analog Parts Kit
- Digital multimeter

#### Symbol Key:

- |   |   |
|---|---|
|  | Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable. |
|  | Analysis; include principle results of analysis in laboratory report.   |
|  | Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.                      |
|  | Record data in your lab notebook.   |

#### General Discussion:

This lab assignment concerns the circuit shown in Fig. 1 below We wish to determine the current  $I_1$  and the voltage  $V_1$ .

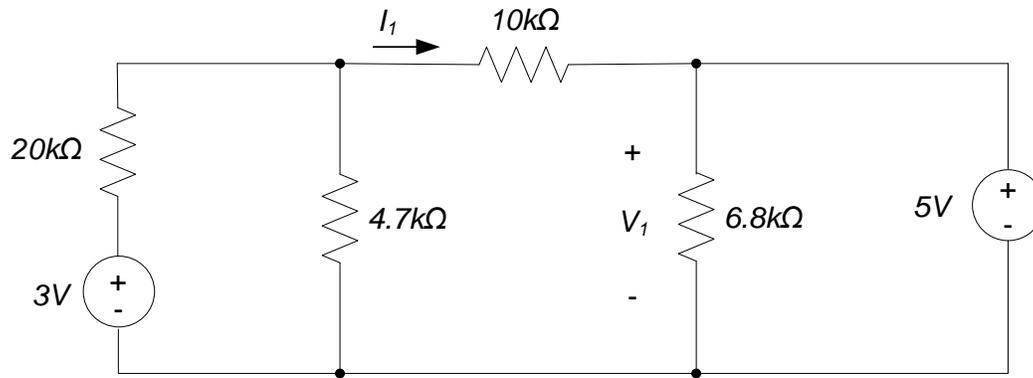


Figure 1. Circuit schematic.

### Pre-lab:

**ANALYSIS**

Use mesh analysis techniques to predict the current  $I_1$  and the voltage  $V_1$  in the circuit of Fig. 1.

### Lab Procedures:

**DATA****ANALYSIS****DEMO**

1. Construct the circuit of Fig. 1; measure and record all actual resistance values. Use whichever voltage supplies you want to implement the voltage sources in the circuit. Measure the voltage  $V_1$  and current  $I_1$  in the circuit. Calculate a percent error between the measured voltages and your predictions from the pre-lab.
2. Demonstrate operation of your circuit to the Teaching Assistant. Have the TA initial the appropriate page(s) of your lab notebook and the lab checklist.

## Real Analog Chapter 3: Lab Worksheets

### 3.3.2: Mesh Analysis (25 points total)

1. In the space below, provide a schematic of the circuit, labeled with meshes and values for mesh currents as determined in the pre-lab. Provide estimates for the voltage  $V_1$  and current  $I_1$ , based on your mesh currents. (7 pts)
2. In the space below, provide a schematic of the circuit being testing, including measured resistance values. (3 pts)
3. Provide your measured values for  $V_1$  and  $I_1$ . Provide a percent error between the measured and expected values of  $V_1$  and  $I_1$ . Comment on any significant differences between the measured and expected values. (8 pts)
4. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuits' operation. (7 pts)
5. **TA Initials:** \_\_\_\_\_

## Real Analog Chapter 3: Lab Projects

### 3.3.3: Mesh Analysis

In this lab, we will analyze, build, and test a circuit containing multiple sources. Except in special cases, multiple sources preclude the use of analysis techniques based entirely on circuit reduction approaches. In this assignment, we will use mesh analysis to predict the circuit behavior prior to building and testing the circuit. We will then compare the measured circuit response with our expectations based on our analysis.

It should be noted that analysis techniques other than mesh analysis can be used to predict this circuit's response. The approach used to analyze the circuit does not, however, affect the actual circuit's behavior. Other lab assignments use this same circuit, in conjunction with other analysis techniques. Please keep in mind that the circuit's physical behavior is not affected by the analysis technique used; the same measurement on the same circuit should provide the same result, regardless of the approach used to predict the response.

#### Before beginning this lab, you should be able to:

- Predict mesh currents in a circuit using mesh analysis
- Use mesh currents to predict physical voltages and currents in an electric circuit
- Use a digital multimeter to measure resistance, voltage, and current
- Use the Analog Discovery 2 module to apply and measure constant voltages

#### After completing this lab, you should be able to:

- Compare measured voltages and currents in an electrical circuit with predictions based on mesh analysis techniques

#### This lab exercise requires:

- Analog Discovery 2 module
- Digilent Analog Parts Kit
- Digital multimeter

#### Symbol Key:

- |   |   |
|---|---|
|  | Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable. |
|  | Analysis; include principle results of analysis in laboratory report.   |
|  | Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.                      |
|  | Record data in your lab notebook.   |

#### General Discussion:

This lab assignment concerns the circuit shown in Fig. 1 below. We wish to determine the current  $I_1$  and the voltage  $V_1$ .

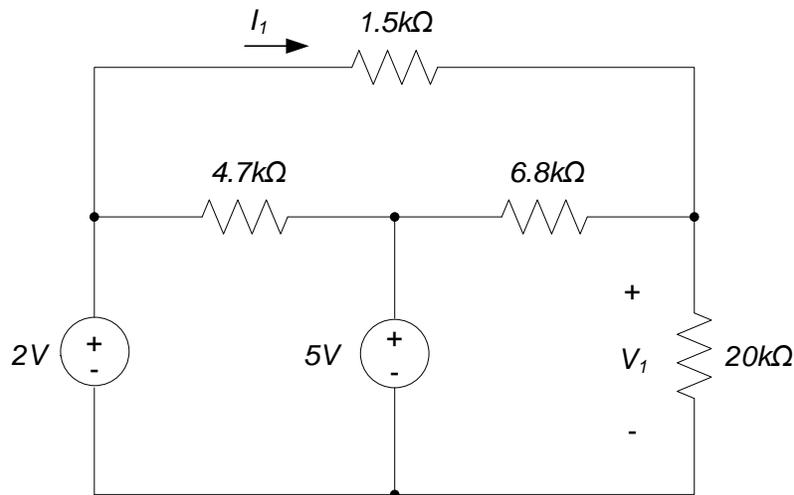


Figure 1. Circuit schematic.

### Pre-lab:

**ANALYSIS**

Use mesh analysis techniques to predict the current  $I_1$  and the voltage  $V_1$  in the circuit of Fig. 1.

### Lab Procedures:

**DATA**

1. Construct the circuit of Fig. 1; measure and record all actual resistance values. Measure the voltage  $V_1$  and current  $I_1$  in the circuit. Calculate a percent error between the measured voltages and your predictions from the pre-lab.

**ANALYSIS****DEMO**

2. Demonstrate operation of your circuit to the Teaching Assistant. Have the TA initial the appropriate page(s) of your lab notebook and the lab checklist.

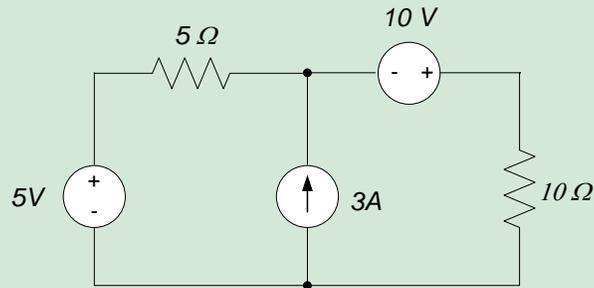
## Real Analog Chapter 3: Lab Worksheets

### 3.3.3: Mesh Analysis (25 points total)

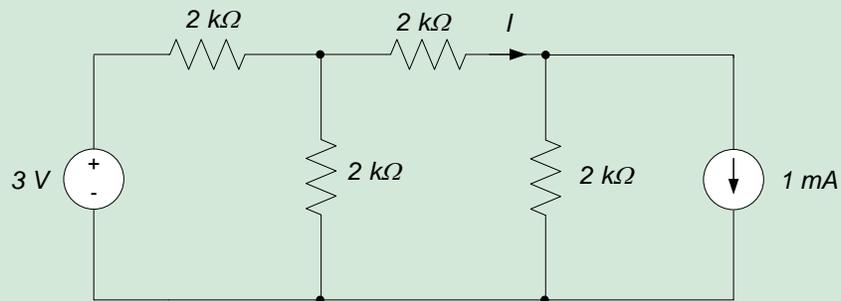
1. In the space below, provide a schematic of the circuit, labeled with meshes and values for mesh currents as determined in the pre-lab. Provide estimates for the voltage  $V_1$  and current  $I_1$ , based on your mesh currents. (7 pts)
2. In the space below, provide a schematic of the circuit being testing, including measured resistance values. (3 pts)
3. Provide your measured values for  $V_1$  and  $I_1$ . Provide a percent error between the measured and expected values of  $V_1$  and  $I_1$ . Comment on any significant differences between the measured and expected values. (8 pts)
4. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuits' operation. (7 pts)
5. **TA Initials:** \_\_\_\_\_

## Real Analog Chapter 3: Homework

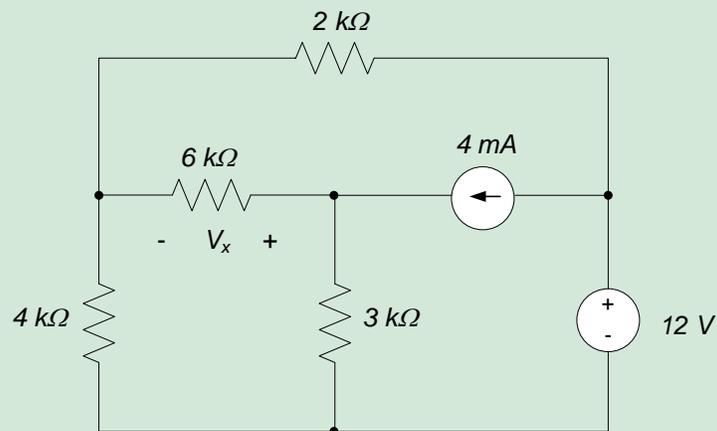
3.1 For the circuit shown, use nodal analysis to determine the current through the  $10\Omega$  resistor.



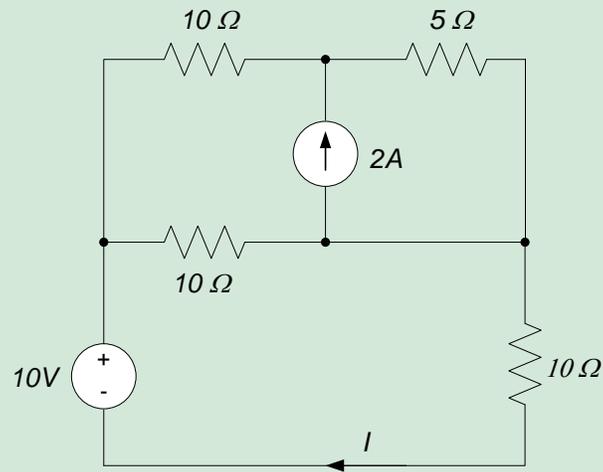
3.2 Find the current  $I$  in the circuit below using nodal analysis.



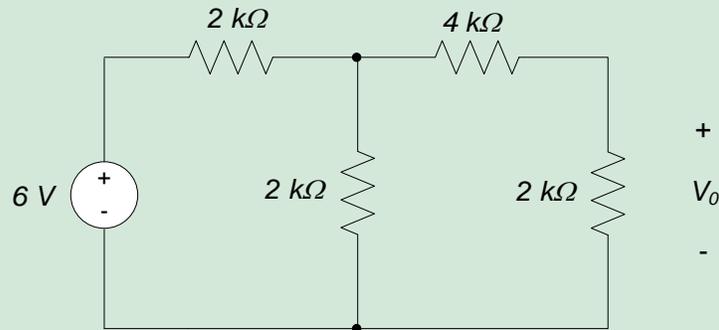
3.3 Use nodal analysis to determine the voltage  $V_x$  in the circuit below.



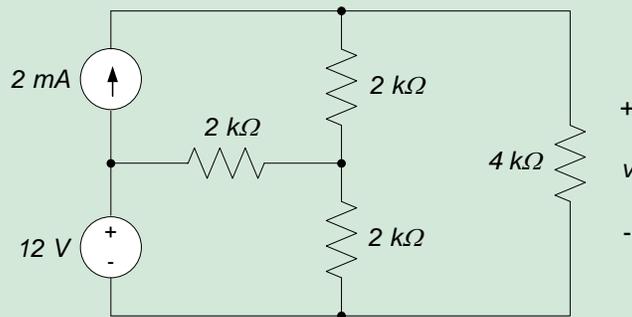
3.4 For the circuit shown, find the current  $I$  using mesh analysis.



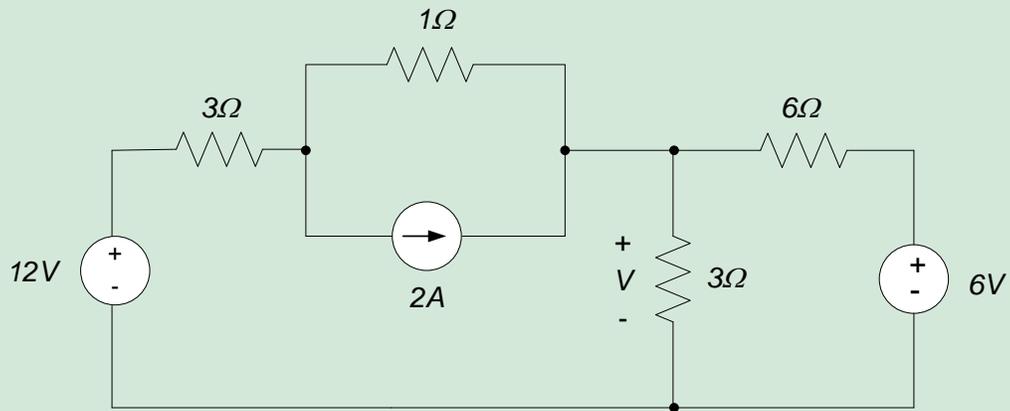
3.5 Use mesh analysis to find the voltage  $V_o$  in the circuit below.



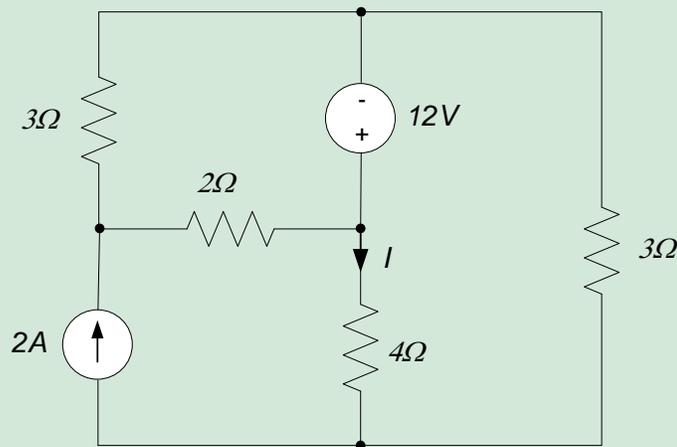
3.6 Find  $v$ , the voltage across the  $4k\Omega$  resistor, in the circuit below using both mesh and nodal analysis.



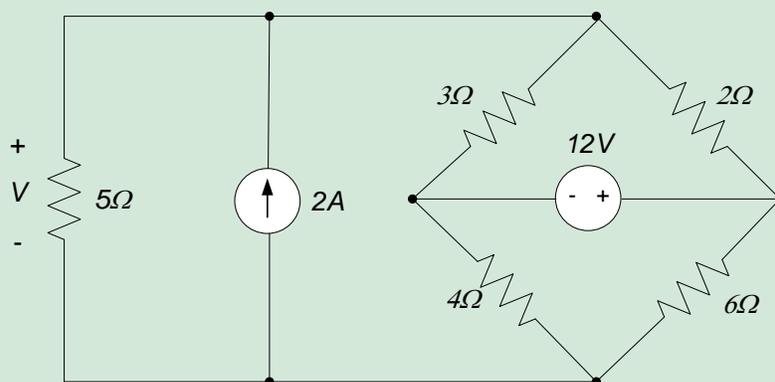
3.7 Using mesh analysis, write a set of equations from which you can find  $\mathbf{V}$ . It is not necessary to simplify or solve the equations for  $\mathbf{V}$ . Clearly show all variable definitions and sign conventions.



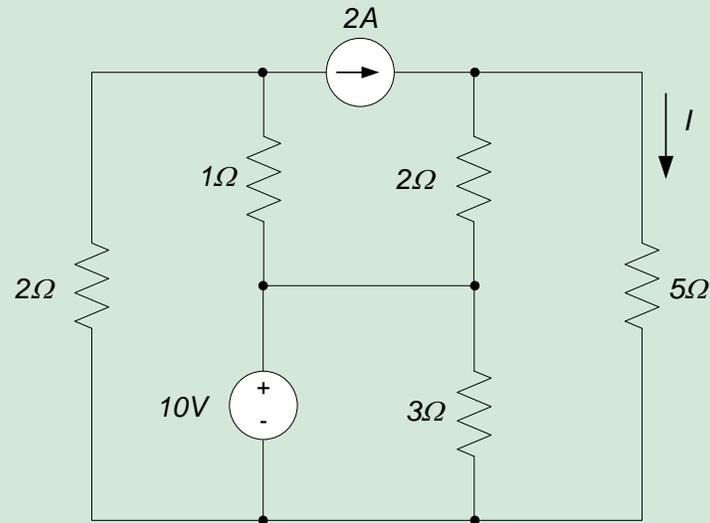
- 3.8 Using nodal analysis, write a set of equations from which you can find  $I$ , the current through the  $4\Omega$  resistor. It is not necessary to simplify or solve the equations for  $I$ . Clearly show all variable definitions and sign conventions.



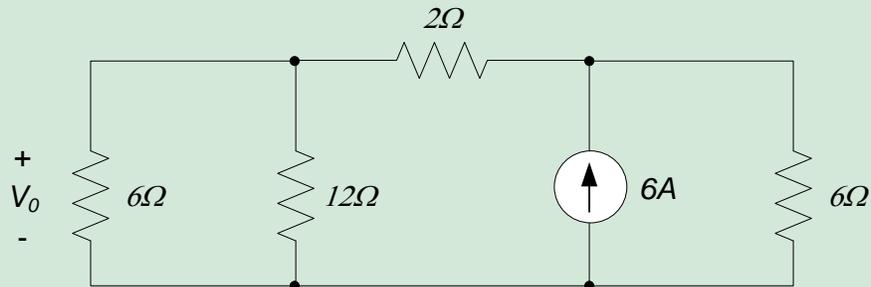
- 3.9 Using mesh analysis, write a set of equations from which you can find  $V$ . It is not necessary to simplify or solve the equations for  $V$ . Clearly show all variable definitions and sign conventions.



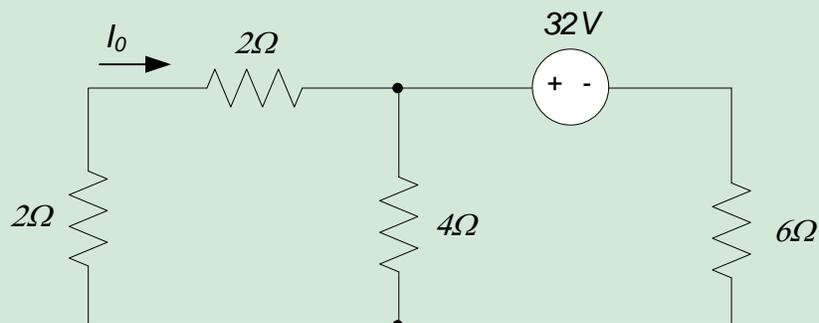
3.10 Using nodal analysis, write a set of equations from which you can find  $I$ , the current through the  $5\Omega$  resistor. It is not necessary to simplify or solve the equations for  $I$ . Clearly show all variable definitions and sign conventions.



3.11 For the circuit shown, find  $V_o$ .



3.12 For the circuit shown, find  $I_o$ .



## Chapter 4: System and Network Theorems

### 4 Introduction and Chapter Objectives

In previous chapters, a number of approaches have been presented for analyzing electrical circuits. In these analysis approaches, we have been provided with a circuit consisting of a number of elements (resistors, power supplies, etc.) and determined some circuit variable of interest (a voltage or current, for example). In the process of determining this variable, we have written equations which allow us to determine any and all variables in the system. For a complex circuit, with many elements, this approach can result in a very large number of equations and a correspondingly large amount of effort expended in the solution of these equations. Unfortunately, much of the physical insight about the overall operation of the circuit may be lost in the detailed description of all of the individual circuit elements. This limitation becomes particularly serious when we attempt to design a circuit to perform some task.

In this chapter, we introduce the concept of a *systems level approach* to circuit analysis. In this type of approach, we represent the circuit as a *system* with some inputs and outputs. We then characterize the system by the mathematical relationship between the system inputs and the system outputs. This relationship is called the *input-output relation* for the system. This representation of a system leads to several network theorems whose use can simplify the analysis of these systems. The network theorems essentially allow us to model a portion of a complicated circuit as a much simpler (but equivalent) circuit. This simplified model can then be used to facilitate the design or analysis of the remainder of the circuit.

The above approach for representing circuits is particularly useful in circuit design; successful design approaches for large circuits typically use a *top-down strategy*. In this design approach, the overall system is broken down into a number of interconnected *subsystems*, each of which performs some specific task. This input-output relationships for these individual subsystems can be determined based on the task performed. The subsystems can then be designed to implement the desired input-output relation. An audio compact disc player, for example, will include subsystems to perform filtering, digital-to-analog conversion, and amplification processes. It is significantly easier to designate the subsystems based on their individual requirements than to attempt to design the entire system all at once. We will thus begin to think of the circuits we analyze as systems which perform some overall task, rather than as a collection of individual circuit elements.

#### After Completing this Chapter, You Should be Able to:

- Define signals and systems
- Represent systems in block diagram form
- Identify system inputs and outputs
- Write input-output equations for systems
- State the defining properties of linear systems
- Determine whether a system is linear
- State conditions under which superposition can be applied to circuit analysis
- Analyze electrical circuits using the principle of superposition
- Define the i-v characteristic for a circuit
- Represent a resistive circuit in terms of its i-v characteristic
- Represent a resistive circuit as a two-terminal network
- Determine Thévenin and Norton equivalent circuits for circuits containing power sources and resistors
- Relate Thévenin and Norton equivalent circuits to i-v characteristics of two-terminal networks
- Determine a load resistance which will maximize the power transfer from a circuit

## 4.1 Signals and Systems

In this section, we introduce basic concepts relative to systems-level descriptions of general physical systems. Later sections will address application of these concepts specifically to electrical circuits.

A system is commonly represented as shown in the block diagram of Fig. 4.1. The system has some input,  $u(t)$ , and some output,  $y(t)$ . In general, both the input and output can be functions of time; the case of constant values is a special case of a time-varying function. The output will be represented as some arbitrary function of the input:

$$y(t) = f\{u(t)\} \quad \text{Eq. 4.1}$$

Equation (4.1) is said to be the *input-output equation* governing the system. The above relationship has only one input and one output - the system is said to be a *single-input-single-output (SISO) system*. Systems can have multiple inputs and multiple outputs, in these cases there will be an input-output equation for each system output and each of these equations may be a function of several inputs. We will concern ourselves only with SISO systems for now.

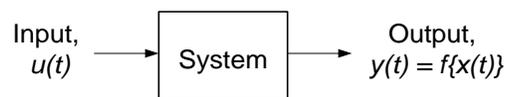


Figure 4.1. Block diagram representation of a system.

One important aspect of the systems-level approach represented by equation (4.1) and Fig. (4.1) is that we are representing our system as a “black box”. We really have no idea what the system itself is, beyond a mathematical dependence of the output variable on the input variable. The physical system itself could be mechanical, thermal, electrical, or fluidic. In fact, it is fairly common to represent a mechanical system as an “equivalent” electrical system (or vice-versa), if doing so increases the physical insight into the system's operation.

The circuits we analyze can now be thought of as *systems* which perform some overall task, rather than as collections of individual circuit elements. We will also think of the inputs and outputs of the system as *signals*, rather than specific circuit parameters such as voltages or currents. This approach is somewhat more abstract than we are perhaps used to, so we will provide some additional discussion of what we mean by these terms.

Generally, most people think of a *system* as a group of interrelated “elements” which perform some task. This viewpoint, though intuitively correct, is not specific enough to be useful from an engineering standpoint. In these chapters, we will define a system as a collection of elements which store and dissipate energy. The system transfers the energy in the system inputs to the system outputs; the process of energy transfer is represented by the input-output equation for the system. Examples of the energy transfer can include mechanical systems (the kinetic energy resulting from using a force to accelerate a mass, or the potential energy resulting from using a force to compress a spring), thermal systems (applying heat to change a mass's temperature), and electrical systems (dissipating electrical power with the filament in a light bulb to produce light).

The task to be performed by the system of Fig. 4.1 is thus the transformation of some input *signal*  $u(t)$  into an output *signal*  $y(t)$ . Signals, for us, will be any waveform which can vary as a function of time. This is an extremely broad definition - examples of signals include:

- The force applied to mass.
- The velocity of the mass as it accelerates in response to the applied force.
- The current applied to a circuit by a power supply.
- The voltage difference across a resistor which is subjected to some current flow.
- The electrical power supplied to a heating element.

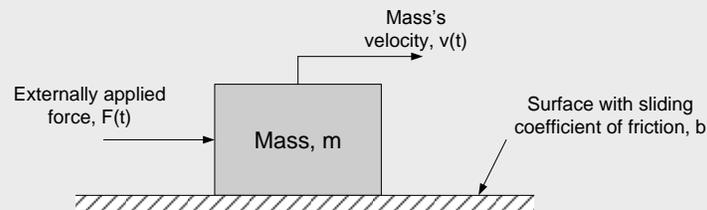
- The temperature of a mass which is being heated by an electric coil.

The transformation of the input signal to the output signal is performed by the input-output relation governing the system. The input-output relation can be a combination of algebraic, differential, and integral equations.

To provide some concrete examples of the above concepts, several examples of system-level representations of common processes are provided below.

### Example 4.1: Mass Subjected to an External Force

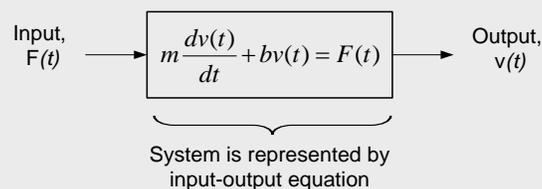
Consider the mass-damper system shown in the figure below. The applied force  $F(t)$  pushes the mass to the right. The mass's velocity resulting from the applied force is  $v(t)$ . The mass slides on a surface with sliding coefficient of friction  $b$ , which induces a force  $F_b = bv(t)$  which opposes the mass's motion. The mass is initially at rest and the applied force is zero for time before time  $t=0$ .



The governing equation for the system (obtained by drawing a free body diagram of the mass and applying  $\sum F = ma$  is:

$$m \frac{dv(t)}{dt} + bv(t) = F(t)$$

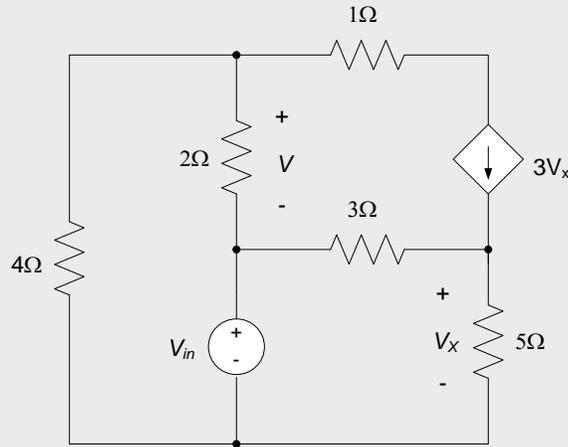
The governing equation for the system is a first order differential equation. Knowledge of the externally applied force  $F(t)$  and the initial velocity of the mass allows us to determine the velocity of the mass at all subsequent times. Thus, we can model the system as having an input signal  $F(t)$  - which is known - and an output velocity  $v(t)$  which can be determined from the input signal and the properties of the system (the mass,  $m$ , and coefficient of friction,  $b$ ). The system can then be represented by the block diagram below:



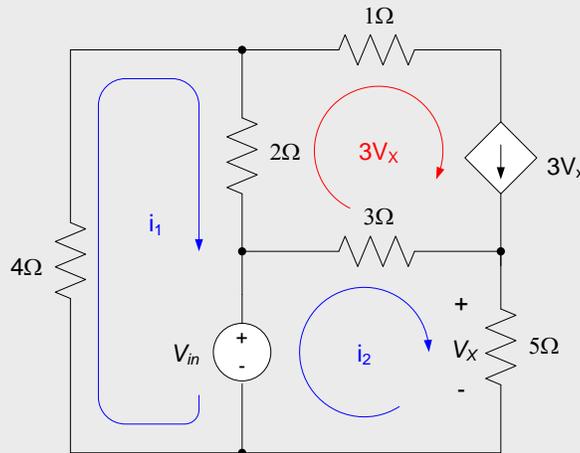
(It is rather unusual to place the system governing equation directly in a block diagram; we do it here to illustrate a point.)

### Example 4.2: Electrical Circuit

For the electrical circuit below, write the equations governing the input-output relationship for the circuit. The applied input to the circuit is the voltage source  $V_{in}$  and the output is the voltage  $V$  across the  $2\Omega$  resistor.



We previously wrote mesh equations for this circuit (for a specific value of  $V_{in}$ ) in Chapter 3.2. We repeat these mesh equations here, along with our definitions of the mesh currents:



$$4i_1 + 2(i_1 - 3V_x) + V_{in} = 0$$

$$-V_{in} + 3(i_2 - 3V_x) + 5i_2 = 0$$

$$V_x = 5i_2$$

The output voltage  $V$  is related to the mesh currents by:

$$V = 2(i_1 - 3V_x)$$

The above four equations provide an input-output description of the circuit. If desired, they can be combined to eliminate all variables except  $V_{in}$  and  $V$  and re-written in the form  $V=f\{V_{in}\}$  per equation (1). Note that all information about the original system, except the relationship between the input and output signals, is lost once we do this.

The system-level block diagram for the circuit might then be drawn as:



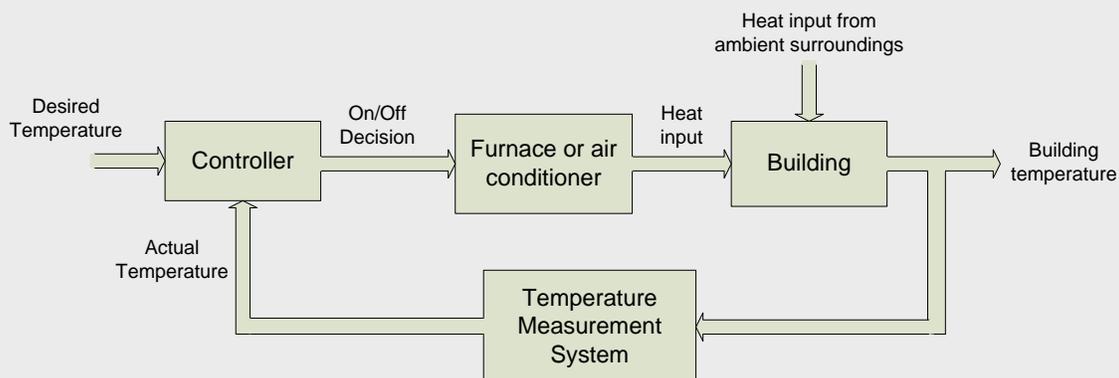
### Example 4.3: Temperature Control System

Our final example is of a temperature control system. This example illustrates the representation of a complex system as a set of interacting subsystems.

A typical temperature control system for a building will have a thermostat which allows the occupants to set a desired temperature, a furnace (or air conditioner) which provides a means of adjusting the building's temperature, some way of measuring the actual building temperature, and a *controller* which decides whether to turn the furnace or air conditioner on or off, based on the difference between the desired and actual temperatures. The block diagram below provides one possible approach toward interconnecting these subsystems into an overall temperature control system. This block diagram can be used to identify individual subsystems, and provide specifications for the subsystems, which can allow the design to proceed efficiently. For example:

1. The temperature measurement system might be required to produce a voltage, which is a function of the temperature in the building. The thermistor-based temperature measurement systems we have designed and constructed in the lab are good examples of this type of system.
2. The controller might operate by comparing the desired temperature (generally represented by a voltage level) with the voltage indicating the actual temperature. For a heating system, if the actual temperature is lower than desired by some minimum amount, the controller will make a decision to switch the furnace on. Design decisions might be made to determine what minimum temperature difference is required to turn the furnace on, and whether to base the decision to turn on the furnace strictly upon a temperature difference or on a rate of change in temperature difference.
3. When the furnace turns on it will apply heat to the building, causing the building's temperature to increase. Once the building temperature is high enough, the controller will then typically turn the furnace back off. The furnace must be designed to provide appropriate heat input to the building, based on the building size and the anticipated heat losses to the building's surroundings. (For example, a larger building or a building in a colder climate will require a larger furnace.)
4. A model of the building's heat losses will generally be necessary in order to size the furnace correctly and choose an appropriate control scheme. Design choices for the building itself may include insulation requirements necessary to satisfy desired heating costs.

Designs for the above subsystems can now proceed somewhat independently, with proper coordination between the design activities.



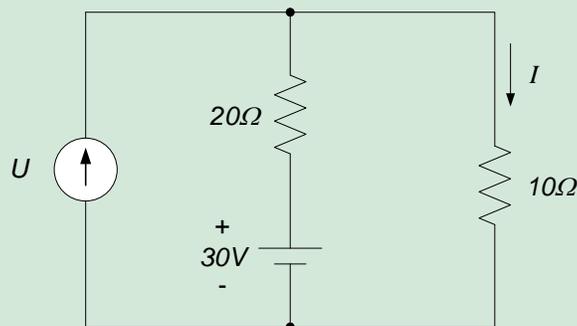
## Section Summary

- *Systems* are a set of components which work together to perform some task. Systems are typically considered to have one or more inputs (which are provided to the system from the external environment) and one or more outputs (which the system provides to the environment).

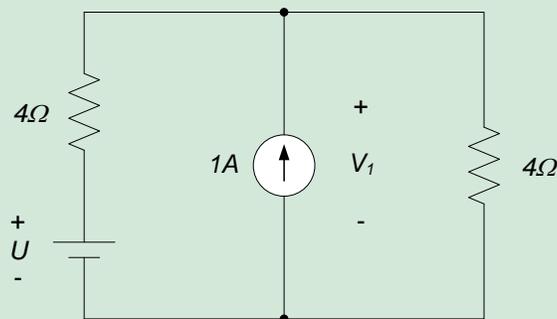
- Generically, the inputs and outputs of systems are *signals*. Signals are simply time-varying functions. They can be voltages, currents, velocities, pressures, etc.
- Systems are often characterized by their *input-output equations*. The input-output equation for a system simply provides a mathematical relationship between the input to the system and the output from the system. Once the input is defined as a particular number or function of time, that value or function can be substituted into the input-output equation to determine the system's response to that input.

## 4.1 Exercises

1. The input to the circuit below is the current,  $U$ . The output is the current through the  $10\Omega$  resistor,  $I$ . Determine an input-output equation for the circuit.



2. The input to the circuit is the voltage  $U$ . The output is the voltage  $V_1$ . Determine an input-output relation for the circuit.



## 4.2 Linear Systems

We have so far introduced a number of approaches for analyzing electrical circuits, including: Kirchhoff's current law, Kirchhoff's voltage law, circuit reduction techniques, nodal analysis, and mesh analysis. When we have applied the above analysis methods, we have generally assumed that any circuit elements operate linearly. For example, we have used Ohm's law to model the voltage-current relationship for resistors. Ohm's law is applicable only for *linear* resistors - that is, for resistors whose voltage-current relationship is a straight line described by the equation  $v=Ri$ . Non-linear resistors have been mentioned briefly; in Lab Assignment 1, for example, we forced a resistor to dissipate an excessive amount of power, thereby causing the resistor to burn out and display nonlinear operating characteristics. All circuit elements will display some degree of non-linearity, at least under extreme operating conditions.

Unfortunately, the analysis of non-linear circuits is considerably more complicated than analysis of linear circuits. Additionally, in subsequent chapters we will introduce a number of analysis methods which are applicable only to

linear circuits. The analysis of linear circuits is thus very pervasive - for example, designing linear circuits is much simpler than the design of non-linear circuits. For this reason, many non-linear circuits are assumed to operate linearly for design purposes; non-linear effects are accounted for subsequently during design validation and testing phases.

The concept of treating an electrical circuit as a *system* was introduced in section 4.1. In systems-level analysis of circuits, we are primarily interested in the relationship between the system's input and output *signals*. Circuits governed by nonlinear equations are considered to be *nonlinear systems*; circuits whose governing input-output relationship is linear are *linear systems*. In this chapter, we formally introduce the concept of linear systems. The analysis of linear systems is extremely common, for the reasons mentioned above: structural systems, fluid dynamic systems, and thermal systems are often analyzed as linear systems, even though the underlying processes are often inherently nonlinear. Linear *circuits* are a special case of linear systems; in which the system consists only of interconnected electrical circuit elements whose voltage-current relationships are linear.

Linear systems are described by linear relations between *dependent variables*. For example, the voltage-current characteristic of a linear resistor is provided by Ohm's law:

$$v = Ri$$

Where  $v$  is the voltage drop across the resistor,  $i$  is the current through the resistor, and  $R$  is the resistance of the resistor. Thus, the dependent variables – current and voltage – are linearly related. Likewise, the equations we have used to describe dependent sources (provided in section 1.2):

- Voltage controlled voltage source:  $v_s = \mu v_1$
- Voltage controlled current source:  $i_s = g v_1$
- Current controlled voltage source:  $v_s = r i_1$
- Current controlled current source:  $i_s = \beta i_1$

All describe linear relationships between the controlled and controlling variables.

All of the above relationships are of the form:

$$y(t) = Kx(t) \tag{Eq. 4.2}$$

Where  $x(t)$  and  $y(t)$  are voltages or currents in the above examples. More generally,  $x(t)$  and  $y(t)$  can be considered to be the *input* and *output signals*, respectively, of a linear *system*. Equation 4.2 is often represented in block diagram form as shown in Fig. 4.2.



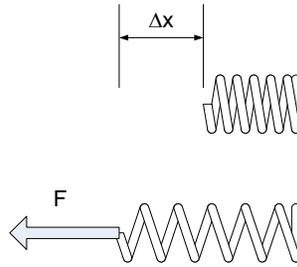
Figure 4.2. Linear system block diagram.

The output is sometimes called the *response* of the system to the input. The multiplicative factor  $K$  relating the input and output is often called the system's *gain*. Elements which are characterized by relationships of the form of equation 4.2 are sometimes called *linear elements*. The equation relating the system's input and output variables is called the *input-output relationship* of the system.

**Aside:** Many types of systems can be described by the relationship of equation (1). For example, Hooke's law, which relates the force applied to a spring to the spring's displacement, is:

$$F = k \cdot \Delta x$$

Where  $k$  is the spring constant,  $F$  is the applied force, and  $\Delta x$  is the resulting displacement as shown below. In this example,  $F$  is the input to the system and  $\Delta x$  is the system output.



Notice that we have allowed the input and output of our system to vary as functions of time. Constant values are special cases of time-varying functions. We will assume that the system gain is not a time-varying quantity.

For our purposes, we will define linearity in somewhat more broad terms than equation (4.2). Specifically, we will define a system as linear if it satisfies the following requirements:

### 4.2.1 Linearity:

1. If the response of a system to some input  $x_1(t)$  is  $y_1(t)$  then the response of the system to some input  $\alpha x_1(t)$  is  $\alpha y_1(t)$ , where  $\alpha$  is some constant. This property is called homogeneity.
2. If the response of the same system to an input  $x_2(t)$  is  $y_2(t)$ , then the response of the system to an input  $x_1(t)+x_2(t)$  is  $y_1(t)+y_2(t)$ . This is called the additive property.

The above two properties defining a linear system can be combined into a single statement, as follows: if the response of a system to an input  $x_1(t)$  is  $y_1(t)$  and the system's response to an input  $x_2(t)$  is  $y_2(t)$ , then the response of the system to an input  $\alpha x_1(t)+\alpha x_2(t)$  is  $\alpha y_1(t)+\alpha y_2(t)$ . This property is illustrated by the block diagram of Fig. 4.3. The Sigma symbol in Fig. 4.3 denotes signal summation; the signs on the inputs to the summation block indicate the signs to be applied to the individual signals.

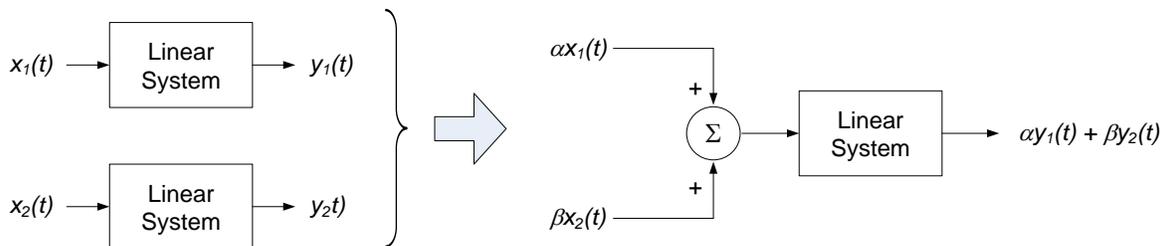


Figure 4.3. Block diagram representation of properties defining a linear system.

The above definition of linearity is more general than the expression of equation (4.2). For example, the processes of differentiation and integration are linear processes according to the above definition. Thus, systems with the input-output relations such as:

$$y = a \int a dt \quad \text{and} \quad y = b \frac{dx}{dt}$$

### 4.2.2 Dependent Variables and Linearity:

Linearity is based on the relationships between *dependent* variables, such as voltage and current. In order for a system to be linear, relationships between dependent variables must be linear – plots of one dependent variable against another are straight lines. This causes confusion among some readers when we begin to talk about *time*

varying signals. Time is not a dependent variable, and plots of voltages or currents as a function of time for a linear system may not be straight lines.

Although the above definitions of linear systems are fundamental, we will not often use them directly. Kirchhoff's voltage law and Kirchhoff's current law rely upon summing multiples of voltages or currents. As long as the voltage-current relations for individual circuit elements are linear, application of KVL and KCL to the circuit will result in linear equations for the system. Therefore, rather than direct application of the above definitions of linear systems, we will simply claim that an electrical circuit containing only linear circuit elements will be linear and will have linear input-output relationships. All circuits we have analyzed so far have been linear.

### 4.2.3 Linearity:

If all elements in a circuit have linear voltage-current relationships, the overall circuit will be linear.

#### Important Note About Power:

A circuit's power is *not a linear* property, even if the voltage-current relations for all circuit elements are linear. Resistors which obey Ohm's law dissipate power according to  $P = iv = \frac{v^2}{R} = i^2R$ . Thus, the power dissipation of a linear resistor is not a linear combination of voltages or currents – the relationship between voltage or current and power is quadratic. Thus, if power is considered directly in the analysis of a linear circuit, the resulting system is nonlinear.

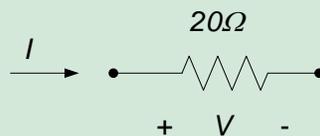
### Section Summary

- *Linear systems* are characterized by linear relationships between dependent variables in the system. For electrical system, this typically means that the relationship between voltage and current for any circuit component is linear – in electrical circuits, for example, this means that a plot of voltage vs. current for every element in the system is a straight line. Ohm's law, for example, describes a linear voltage-current relationship.
- Linear systems have a very important property: the *additive principle* applies to them. Superposition essentially means that the response of a system to some combination of inputs  $x_1+x_2$  will be the same as the sum of the response to the individual inputs  $x_1$  and  $x_2$ .

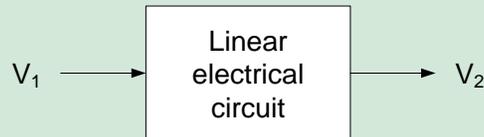
## 4.2 Exercises

1. The  $20\Omega$  resistor below obeys Ohm's law, so that  $V=20I$ . We will consider the input to be the current through the resistor and the output to be the voltage drop across the resistor. Determine:
  - The output  $V$  if the input  $I = 2A$
  - The output  $V$  if the input  $I = 3A$
  - The output  $V$  is the input  $I = 2A + 3A = 5A$

Do your answers above indicate that the additive property holds for this resistor? Why?



2. A linear electrical circuit has an input voltage  $V_1$  and provides an output voltage  $V_2$ , as indicated in the block diagram below. If an input voltage  $V_1=3V$  is applied to the circuit, the measured output voltage  $V_2=2V$ . What is the output voltage if an input voltage  $V_1=6V$  is applied to the circuit?



### 4.3 Superposition

In section 4.2, we stated that, by definition, the input-output relations for linear systems have an additive property. The additive property of linear systems states that:

- If the response of a system to an input  $x_1(t)$  is  $y_1(t)$  and the response of the system to an input  $x_2(t)$  is  $y_2(t)$ , then the response of the system to an input  $x_1(t)+x_2(t)$  is  $y_1(t)+y_2(t)$ .

Thus, if a system has multiple inputs, we can analyze the system's response to each input individually and then obtain the overall response by summing the individual contributions. This property can be useful in the analysis of circuits which have multiple sources. If we consider the sources in a circuit to be the inputs, linear circuits with multiple independent sources can be analyzed by determining the circuit's response to each source individually, and then summing, or *superimposing*, the contributions from each source to obtain the overall response of the circuit to all sources. In general, the approach is to analyze a complicated circuit with multiple sources by determining the responses of a number of simpler circuits – each of which contains only a single source.

We illustrate the overall approach graphically by the block diagram of Fig. 4.4 (which is really just a reversed form of the block diagram of Fig. 4.3).

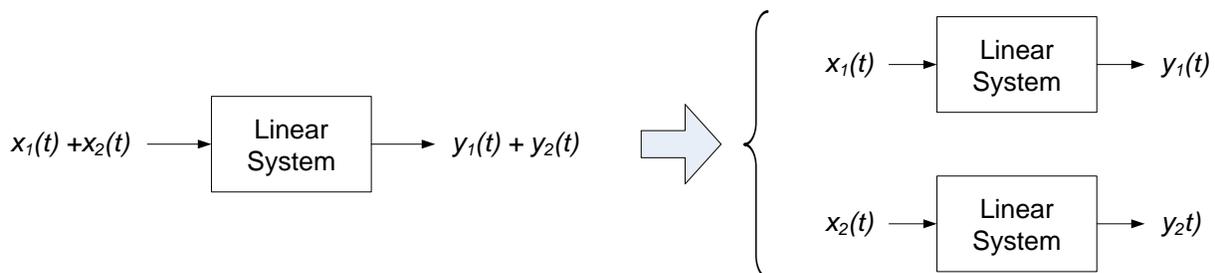


Figure 4.4. Additive property of linear systems.

In Fig. 4.4, we have a linear system with two input signals which are applied by sources in the circuit. We can analyze this circuit by noting that each input signal corresponds to an independent source in the circuit. Thus, if the circuit's overall response to a source  $x_1(t)$  is  $y_1(t)$  and the circuit's response to a source  $x_2(t)$  is  $y_2(t)$ , then the total circuit response will be the sum of the two individual responses,  $y_1(t)+y_2(t)$ . Thus, if we wish to determine the response of the circuit to both sources,  $x_1(t)$  and  $x_2(t)$ , we can determine the individual responses of the circuit,  $y_1(t)$  and  $y_2(t)$  and then sum (or *superimpose*) the responses to obtain the circuit's overall response to both inputs. This analysis method is called *superposition*.

In order to determine a circuit's response to a single source, all other independent sources must be *turned off* (or, in more colorful terminology, *killed*, or made *dead*). To turn off a current source, we must make the input current zero, which corresponds to an open circuit. To turn off a voltage source, we must make the input voltage zero, which corresponds to a short circuit.

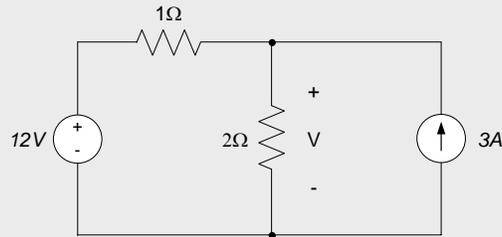
#### Killing Sources:

- To kill a voltage source, replace it with a short circuit
- To kill a current source, replace it with an open circuit

To apply the superposition method, then, the circuit's response to each source in the circuit is determined, with all other sources in the circuit dead. The individual responses are then algebraically summed to determine the total response to all inputs. To illustrate the method, we consider the examples below.

### Example 4.4

Determine the voltage,  $V$ , in the circuit below, using superposition.



The circuit above can be considered to be the superposition of the two circuits shown below, each with a single source (the other source, in both cases, has been killed).

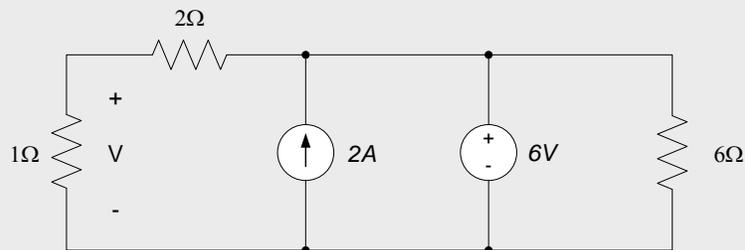


The voltage  $V_1$  above can be determined to be the result of a current division:  $V_1 = \left[ \frac{1\Omega}{1\Omega + 2\Omega} \cdot 3A \right] \times 2\Omega = 2V$ .

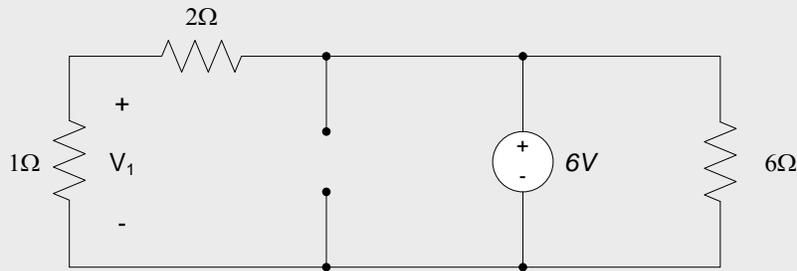
$V_2$  can be determined to be the result of a voltage division:  $V_2 = 12V \cdot \frac{2\Omega}{2\Omega + 1\Omega} = 8V$ . Thus, the voltage:  $V = V_1 + V_2 = 10V$ .

### Example 4.5

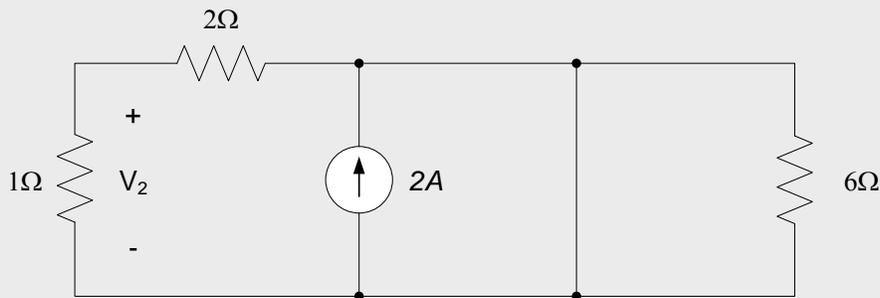
Determine the voltage,  $V$ , in the circuit below, using superposition.



We begin by determining the response  $V_1$  to the 6V source by killing the 2A source, as shown in the figure below.



The voltage  $V_1$  is simply the result of a voltage division:  $V_1 = \frac{1}{2} \cdot 6V = 2V$ . The response  $V_2$  to the 2A source can be determined by killing the 6V source, resulting in the circuit below:



Killing the 6V source places a short circuit in parallel with the 2A source, so no voltage is induced in any of the resistors by the 2A source. Thus,  $V_2 = 0V$ .

The voltage  $V$  is the sum of the two individual voltages:  $V = V_1 + V_2 = 2V + 0V = 2V$ .

### Notes on Superposition:

1. Superposition cannot be used directly to determine power. Previously, we noted that power is not governed by a linear relationship. Thus, you cannot determine the power dissipated by a resistor by determining the power dissipation due to each source and then summing the results. You can, however, use superposition to determine the total voltage or current for the resistor and then calculate the power from the voltage and/or current.
2. When using superposition to analyze circuits with dependent sources, do not kill the dependent sources. You must include the effects of the dependent sources in response to each independent source.
3. Superposition is a powerful circuit analysis tool, but its application can result in additional work. Before applying superposition, examine the circuit carefully to ensure that an alternate analysis approach is not more efficient. Circuits with dependent sources, in particular, tend to be difficult to analyze using superposition

### Section Summary

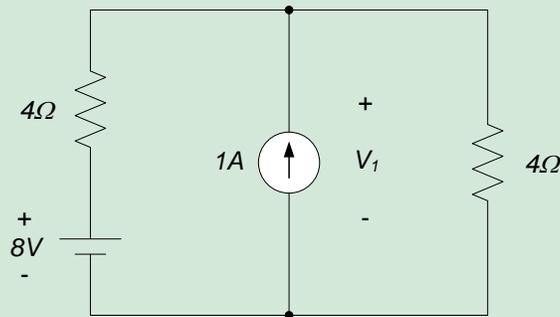
- Superposition is a defining property of linear systems. It essentially means that, for linear systems, we can decompose any input to the system into a number of components, determine the system output resulting from each component of the input, and obtain the overall output by summing up these individual components of the output.
- Superposition can be used directly to analyze circuits which contain multiple independent sources. The responses of the circuit to each source (killing all other sources) are determined individually. The overall

response of the circuit – due to all sources – is then obtained by summing (superimposing) these individual contributions.

- The principle of superposition is a fundamental property of linear systems and has very broad-ranging consequences. We will be invoking it throughout the remainder of this textbook, often without overtly stating that superposition is being used. The fact that superposition applies to linear circuits is the basic reason why engineers make every possible attempt to use linear models when analyzing and designing systems.

### 4.3 Exercises

1. Use superposition to determine the voltage  $V_1$  in the circuit below.



### 4.4 Two-terminal Networks

As noted in section 4.1, it is often desirable, especially during the design process, to isolate different portions of a complex system and treat them as individual subsystems. These isolated subsystems can then be designed or analyzed somewhat independently of one another and subsequently integrated into the overall system in a top-down design approach. In systems composed of electrical circuits, the subsystems can often be represented as *two-terminal networks*. As the name implies, two-terminal networks consist of a pair of terminals; the voltage potential across the terminals and the current flow into the terminals characterizes the network. This approach is consistent with our systems-level approach; we can characterize the behavior of what may be an extremely complex circuit by a relatively simple input-output relationship.

We already have some experience with two-terminal networks; when we determined equivalent resistances for series and parallel resistor combinations, we treated the resistive network as a two-terminal network. For analysis purposes, the network was then replaced with a single equivalent resistance which was indistinguishable from the original circuit by any external circuitry attached to the network terminals. In this chapter, we will formalize some two-terminal concepts and generalize our approach to include networks which contain both sources and resistors.

We will assume that the electrical circuit of interest can be subdivided into two sub-circuits, interconnected at two terminals, as shown in Fig. 4.5. Our goal is to replace circuit A in our overall system with a simpler circuit which is indistinguishable by circuit B from the original circuit. That is, if we disconnect circuit A from circuit B at the terminals and replace circuit A with its equivalent circuit, the voltage  $v$  and the current  $i$  seen at the terminals of the circuits will be unchanged and circuit B's operation will be unaffected. In order to make this substitution, we will need to use the principle of superposition in our analysis of circuit A – thus, circuit A must be a linear circuit. We are not changing circuit B in any way – circuit B can be either linear or nonlinear.

It should be emphasized that circuit A is not being physically changed. We are making the change conceptually in order to simplify our analysis of the overall system. For example, the design of circuit B can now proceed with a simplified model of circuit A's operation, perhaps before the detailed design of circuit A is even finalized. When the

designs of the two circuits are complete, they can be integrated and the overall system has a high probability of functioning as expected.

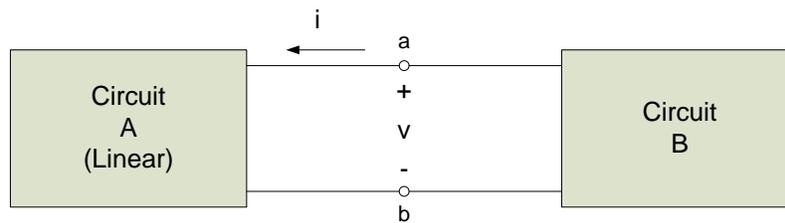


Figure 4.5. Circuit composed of two, two-terminal sub-circuits.

In order to perform the above analysis, we will disconnect the two sub-circuits in Fig. 4.5 at the terminals a – b and determine the current-voltage relationship at the terminals of circuit A. We will generally refer to circuit A's current-voltage relationship as its *i-v characteristic*. Our approach, therefore, is to look at circuit A alone, as shown in Fig. 4.6, and determine the functional relationship between a voltage applied to the terminals and the resulting current. (Equivalently, we could consider that a current is applied at the terminals and look at the resulting voltage.) Fig. 4.6 is at first glance somewhat misleading – the terminals should not be considered to be open-circuited, as a cursory look at the figure might indicate; we are determining the relationship between a voltage difference applied to the circuit and the resulting current flow. (Figure 4.6 indicates a current  $I$  flowing into the circuit, which will, in general, not be zero.)

### System-level Interpretation

When we determine the *i-v* characteristic for the circuit, we are determining the input-output relationship for a system. Either the voltage or the current at the terminals can be viewed as the input to a system; the other parameter is the output. The *i-v* characteristic then provides the output of the system as a function of the input.

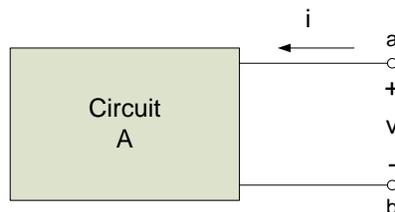


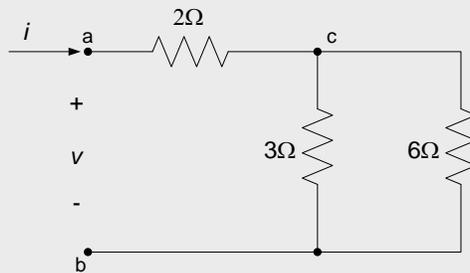
Figure 4.6. Two-terminal representation of circuit.

## 4.4.1 Resistive Networks

We have already (somewhat informally) treated purely resistive circuits as two-terminal networks when we determined equivalent resistances for series and parallel resistors. We will briefly review these concepts here in a systems context in terms of a simple example.

### Example 4.6

Determine the *i-v* characteristic for the circuit below.

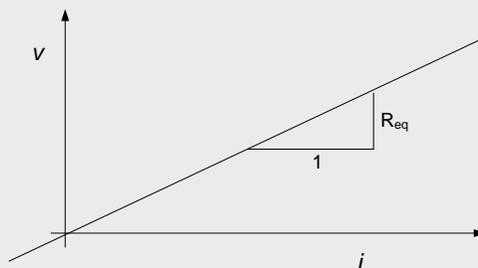


Previously, we would use circuit reduction techniques to solve this problem. The equivalent resistance is  $R_{eq} = 2\Omega + \frac{(3\Omega)(6\Omega)}{3\Omega+6\Omega} = 4\Omega$ . Since  $v = R_{eq}i$ , the circuit's  $i$ - $v$  characteristic is  $v = 4i$ .

We would now, however, like to approach this problem in a slightly more general way and using a systems-level view to the problem. Therefore, we will choose the terminal voltage,  $v$ , to be viewed as the input to the circuit. By default, this means the current  $i$  will be our circuit's output. (We could, just as easily define the current as the input, in which case the voltage would become our output.) Thus, our circuit conceptually looks like a system as shown in the block diagram below.



Applying KCL to node  $c$  in the above circuit results in  $i = \frac{v_c}{6\Omega} + \frac{v_c}{3\Omega}$ . Ohm's law, applied to the  $2\Omega$  resistor, results in  $v - v_c = 2\Omega \cdot i$ . Eliminating  $v_c$  from the above two equations results in  $v = 4i$ , which is the same result we obtained using circuit reduction. The  $i$ - $v$  characteristic for the above circuit is shown graphically below; the slope of the line is simply the equivalent resistance of the network.



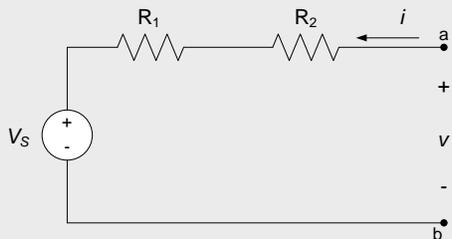
In the above example, viewing the circuit as a general two-terminal network and using a more general systems-level approach to the problem results in additional work relative to using our previous circuit reduction approach. Viewing the circuit as a more general two-terminal network is, however, very profitable if circuit reduction techniques are not applicable or if we allow the circuit to contain voltage or current sources. The latter topic is addressed in the following subsection.

#### 4.4.2 Two-terminal Networks with Sources

When the network consists of resistive elements and independent sources, the circuit's  $i$ - $v$  characteristic can be represented as a single equivalent resistance and a single source-like term. In general, however, we cannot determine this directly by using circuit reduction techniques. The overall approach and typical results are illustrated in the following examples.

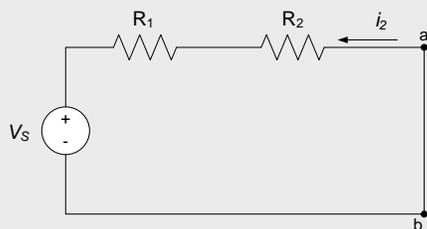
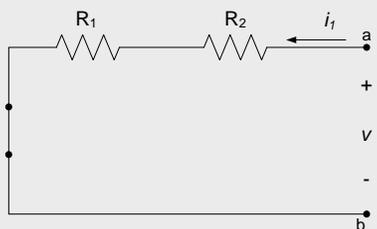
### Example 4.7

Determine the  $i$ - $v$  characteristic of the circuit below.

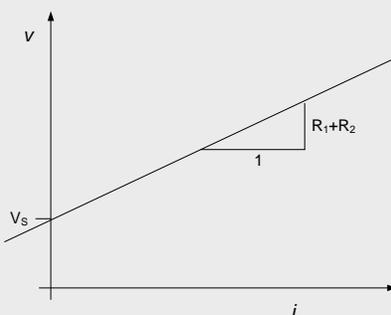


Although it is fairly apparent, by applying Ohm's law across the series combination of resistors, that  $v = (R_1 + R_2)i + V_s$ , we will (for practice) use superposition to approach this problem. The voltage source  $V_s$  will, of course, be one source in the circuit. We will use the voltage across the terminals a-b as a second source in the circuit.

Killing the voltage source  $V_s$  results in the circuit to the left below; the resulting current is  $i_1 = \frac{v}{R_1 + R_2}$ . Killing the "source"  $v$  results in the circuit to the right below; the resulting current is  $i_2 = -\frac{V_s}{R_1 + R_2}$ . The total current is, therefore,  $i = \frac{v}{R_1 + R_2} - \frac{V_s}{R_1 + R_2}$  or  $v = (R_1 + R_2)i + V_s$ .

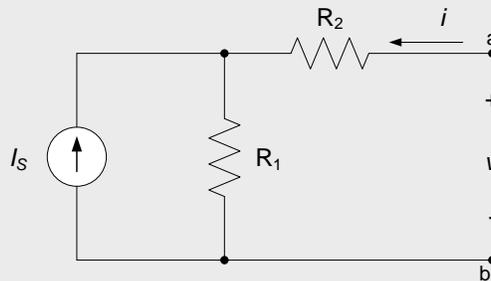


Plotting the above  $i$ - $v$  characteristic results in the figure below.

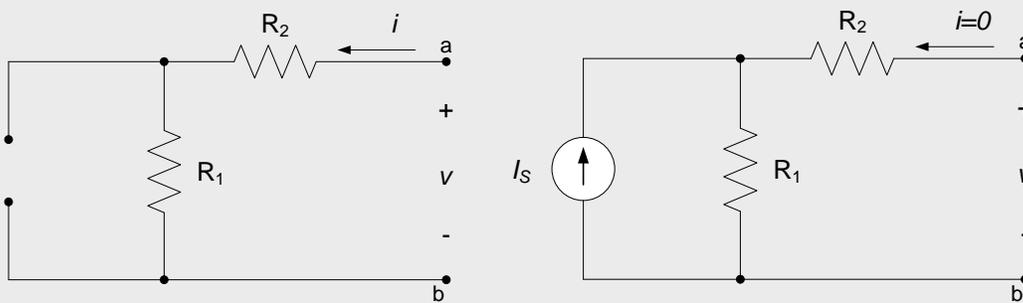


### Example 4.8

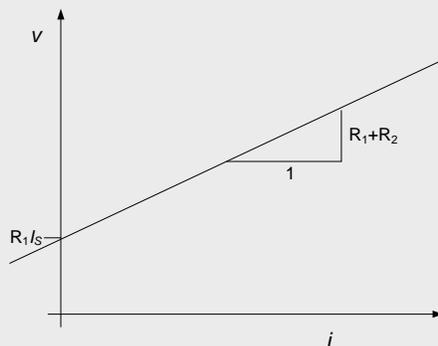
Determine the  $i$ - $v$  characteristic of the circuit below.



Although not the most efficient approach for this problem, we will again use superposition to approach the problem. One source will, of course, be the current source  $I_s$ . We will assume that our second source is the current  $i$  at node a. Killing the current source  $I_s$  results in the circuit to the left below; from this figure the voltage  $v_1$  can be seen to be  $v_1=i(R_1+R_2)$ . Killing the current source  $i$  results in the figure to the right below; from this figure the voltage  $v_2$  is seen to be  $v_2=R_1I_s$  (the dead current source results in an open circuit, so no current flows through the resistor  $R_2$ ). The total voltage across the terminals is, therefore,  $v=(R_1+R_2)i+R_1I_s$ .



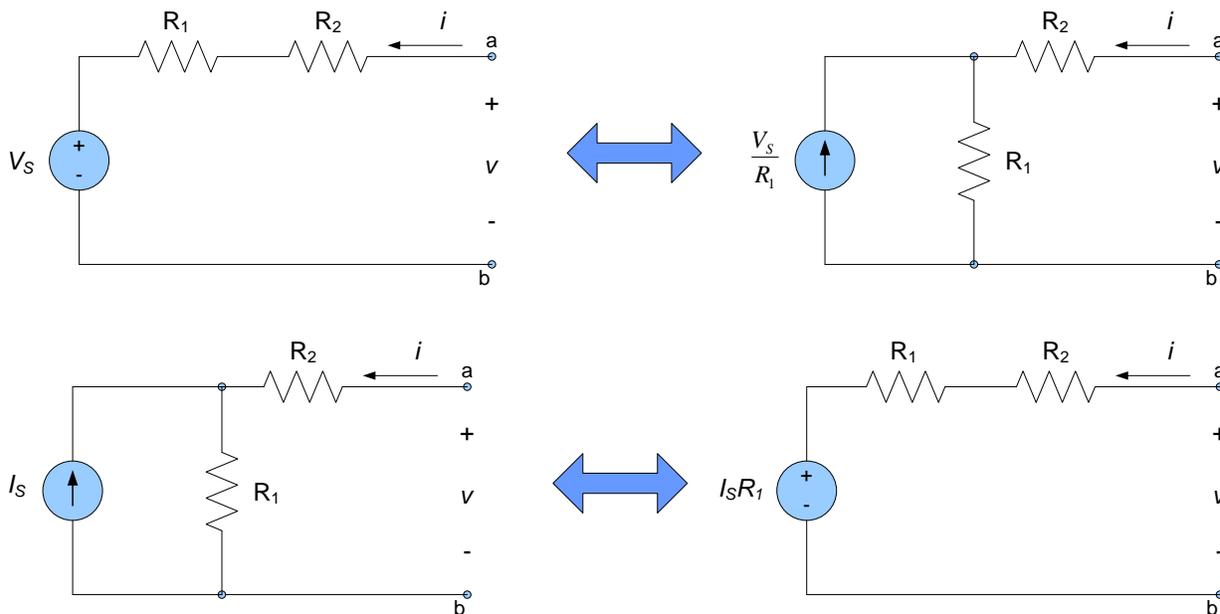
The i-v characteristic for the circuit is, therefore, as shown in the figure below.



**Notes on Linear Circuit i-v Characteristics:**

1. All two-terminal networks which contain only sources and resistors will have i-v relationships of the form shown in examples 1, 2, and 3. That is, they will be straight lines of the form  $v=m \cdot i+b$ . The y-intercept term,  $b$ , is due to sources in the network; if there are no sources in the network,  $b=0$  and the i-v characteristic will pass through the origin.
2. Due to the form of the i-v characteristic provided in note 1 above, any two-terminal network can be represented as a single source and a single resistor.
3. The form of the solution for examples 2 and 3 are the same. Thus, the circuit of example 2 is indistinguishable from a similar circuit with a current source  $\frac{V_S}{R_1}$  in parallel with the resistor  $R_1$ . Likewise,

the circuit of example 3 is indistinguishable from a similar circuit with a voltage source  $I_s R_1$  in series with the resistor  $R_1$ . The equivalent circuits are shown below.

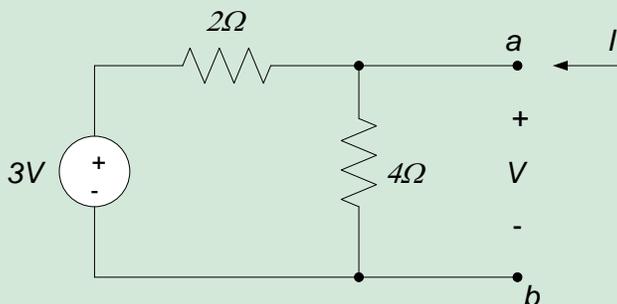


### Section Summary

- Electrical circuits, sub-circuits, and components are often modeled by the relationship between voltage and current at their terminals. For example, we are familiar with modeling resistors by Ohm’s law, which simply relates the voltage to the current at the resistor terminals. In Chapter 2, we used circuit reduction methods to extend this concept by replacing resistive networks with equivalent resistances which provided the same voltage-current relations across their terminals. In this section, we continue to extend this concept to circuits which include sources.
- For linear circuits, the voltage-current relationship across two terminals of the circuit can always be represented as a straight line of the form  $v=m \cdot i+b$ . If we plot this relationship with voltage on the vertical axis and current on the horizontal axis, the slope of the line corresponds to an equivalent resistance seen across the terminals, while the y-intercept of the line is the voltage across the terminals, if the terminals are open-circuited. We will formalize this important result in section 4.5.

### 4.4 Exercises

1. Determine the i-v characteristics of the circuit below, as seen at the terminals a-b.



## 4.5 Thévenin's and Norton's Theorems

In section 4.4, we saw that it is possible to characterize a circuit consisting of sources and resistors by the voltage-current (or  $i$ - $v$ ) characteristic seen at a pair of terminals of the circuit. When we do this, we have essentially simplified our description of the circuit from a detailed model of the internal circuit parameters to a simpler model which describes the overall behavior of the circuit as seen at the terminals of the circuit. This simpler model can then be used to simplify the analysis and/or design of the overall system.

In this section, we will formalize the above result as *Thévenin's* and *Norton's theorems*. Using these theorems, we will be able to represent any linear circuit with an equivalent circuit consisting of a single resistor and a source. Thévenin's theorem replaces the linear circuit with a voltage source in series with a resistor, while Norton's theorem replaces the linear circuit with a current source in parallel with a resistor. In this section, we will apply Thévenin's and Norton's theorems only to purely resistive networks. However, these theorems can be used to represent any circuit made up of linear elements.

Consider the two interconnected circuits shown in Fig. 4.7 below. The circuits are interconnected at the two terminals a and b, as shown. Our goal is to replace circuit A in the system of Fig. 4.7 with a simpler circuit which has the same current-voltage characteristic as circuit A. That is, if we replace circuit A with its simpler equivalent circuit, the operation of circuit B will be unaffected. We will make the following assumptions about the overall system:

- Circuit A is linear
- Circuit A has no dependent sources which are controlled by parameters within circuit B
- Circuit B has no dependent sources which are controlled by parameters within circuit A

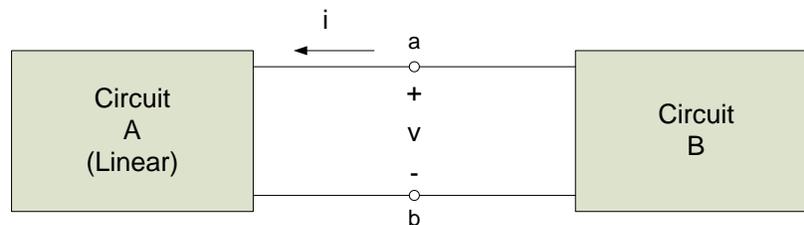


Figure 4.7. Interconnected two-terminal circuits.

In section 4.3, we determined  $i$ - $v$  characteristics for several example two-terminal circuits, using the superposition principle. We will follow the same basic approach here, except for a general linear two-terminal circuit, in order to develop Thévenin's and Norton's theorems.

### 4.5.1 Thévenin's Theorem

First, we will kill all sources in circuit A and determine the voltage resulting from an applied current, as shown in Fig. 4.8 below. With the sources killed, circuit A will look strictly like an equivalent resistance to any external circuitry. This equivalent resistance is designated as  $R_{TH}$  in Fig. 4.8. The voltage resulting from an applied current, with circuit A dead is:

$$v_1 = R_{TH} \cdot i \quad \text{Eq. 4.3}$$

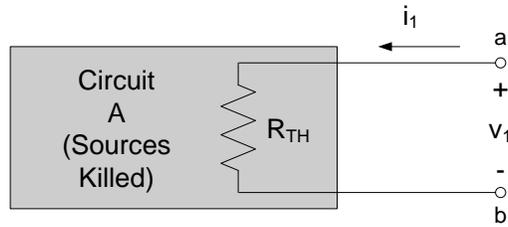


Figure 4.8. Circuit schematic with dead circuit.

Now we will determine the voltage resulting from re-activating circuit A's sources and open-circuiting terminals a and b. We open-circuit the terminals a-b here since we presented equation (4.3) as resulting from a current source, rather than a voltage source. The circuit being examined is as shown in Fig. 4.9. The voltage  $v_{OC}$  is the "open-circuit" voltage.

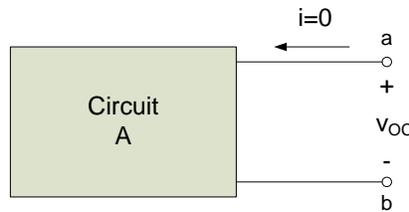


Figure 4.9. Open-circuit response.

Superimposing the two voltages above results in:

$$v = v_1 + v_{OC} \tag{Eq. 4.4}$$

Or

$$v = R_{TH} \cdot i + v_{OC} \tag{Eq. 4.5}$$

Equation (4.5) is Thévenin's theorem. It indicates that the voltage-current characteristic of any linear circuit (with the exception noted below) can be duplicated by an independent voltage source in series with a resistance  $R_{TH}$ , known as the *Thévenin resistance*. The voltage source has the magnitude  $v_{OC}$  and the resistance is  $R_{TH}$ , where  $v_{OC}$  is the voltage seen across the circuit's terminals if the terminals are open-circuited and  $R_{TH}$  is the equivalent resistance of the circuit seen from the two terminals, with all independent sources in the circuit killed. The equivalent Thévenin circuit is shown in Fig. 4.10.

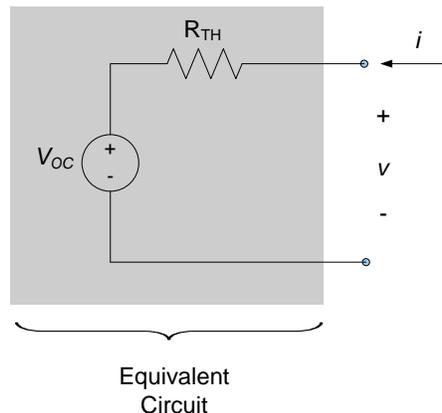


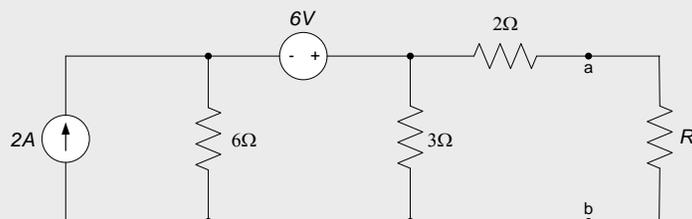
Figure 4.10. Thévenin equivalent circuit.

### Procedure for Determining Thévenin Equivalent Circuit

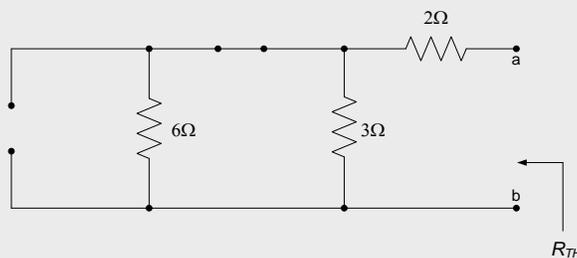
1. Identify the circuit and terminals for which the Thévenin equivalent circuit is desired.
2. Kill the independent sources (do nothing to any dependent sources) in circuit and determine the equivalent resistance  $R_{TH}$  of the circuit. If there are no dependent sources,  $R_{TH}$  is simply the equivalent resistance of the resulting resistive network. Otherwise, one can apply an independent current source at the terminals and determine the resulting voltage across the terminals; the voltage-to-current ratio is  $R_{TH}$ .
3. Re-activate the sources and determine the open-circuit voltage  $V_{OC}$  across the circuit terminals. Use any analysis approach you choose to determine the open-circuit voltage.

### Example 4.9

Determine the Thévenin equivalent of the circuit below, as seen by the load,  $R_L$ .

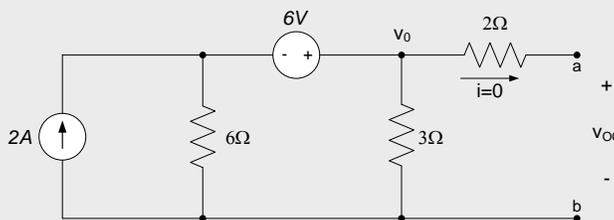


We want to create a Thévenin equivalent circuit of the circuit to the left of the terminals a-b. The load resistor,  $R_L$ , takes the place of “circuit B” in Fig. 1. The circuit has no dependent sources, so we kill the independent sources and determine the equivalent resistance seen by the load. The resulting circuit is shown below.

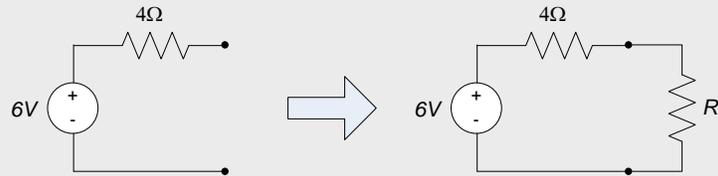


From the above figure, it can be seen that the Thévenin resistance  $R_{TH}$  is a parallel combination of a 3Ω resistor and a 6Ω resistor, in series with a 2Ω resistor. Thus,  $R_{TH} = \frac{(6\Omega)(3\Omega)}{6\Omega+3\Omega} + 2\Omega = 4\Omega$ .

The open-circuit voltage  $v_{OC}$  is determined from the circuit below. We (arbitrarily) choose nodal analysis to determine the open-circuit voltage. There is one independent voltage in the circuit; it is labeled as  $v_0$  in the circuit below. Since there is no current through the 2Ω resistor,  $v_{OC} = v_0$ .



Applying KCL at  $v_0$ , we obtain:  $-2A + \frac{v_0 - 6V}{6\Omega} + \frac{v_0}{3\Omega} = 0 \Rightarrow v_0 = v_{OC} = 6V$ . Thus, the Thévenin equivalent circuit is on the left below. Re-introducing the load resistance, as shown on the right below, allows us to easily analyze the overall circuit.



### 4.5.2 Norton's Theorem

The approach toward generating Norton's theorem is almost identical to the development of Thévenin's theorem, except that we apply superposition slightly differently. In Thévenin's theorem, we looked at the voltage response to an input current; to develop Norton's theorem, we look at the current response to an applied voltage. The procedure is provided below.

Once again, we kill all sources in circuit A, as shown in Fig. 4.8 above but this time we determine the current resulting from an applied voltage. With the sources killed, circuit A still looks like an equivalent resistance to any external circuitry. This equivalent resistance is designated as  $R_{TH}$  in Fig. 4.8. The current resulting from an applied voltage, with circuit A dead is:

$$i_1 = \frac{v}{R_{TH}} \tag{Eq. 4.6}$$

Notice that equation (4.6) can be obtained by rearranging equation (4.3).

Now we will determine the current resulting from re-activating circuit A's sources and short-circuiting terminals a and b. We short-circuit the terminals a-b here since we presented equation (4.4) as resulting from a voltage source. The circuit being examined is as shown in Fig. 4.11. The current  $i_{SC}$  is the "short-circuit" current. It is typical to assume that under short-circuit conditions the short-circuit current enters the node at a; this is consistent with an assumption that circuit A is generating power under short-circuit conditions.

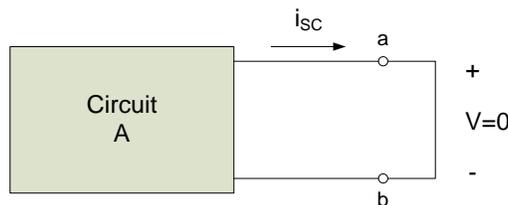


Figure 4.11. Short-circuit response.

Employing superposition, the current into the circuit is (notice the negative sign on the short-circuit current, resulting from the definition of the direction of the short-circuit current opposite to the direction of the current  $i$ ).

$$i = i_1 - i_{SC} \tag{Eq. 4.7}$$

So

$$i = \frac{v}{R_{TH}} - i_{SC} \tag{Eq. 4.8}$$

Equation (4.8) is Norton's theorem. It indicates that the voltage-current characteristic of any linear circuit (with the exception noted below) can be duplicated by an independent current source in parallel with a resistance. The

current source has the magnitude  $i_{SC}$  and the resistance is  $R_{TH}$ , where  $i_{SC}$  is the current seen at the circuit's terminals if the terminals are short-circuited and  $R_{TH}$  is the equivalent resistance of the circuit seen from the two terminals, with all independent sources in the circuit killed. The equivalent Norton circuit is shown in Fig. 4.12.

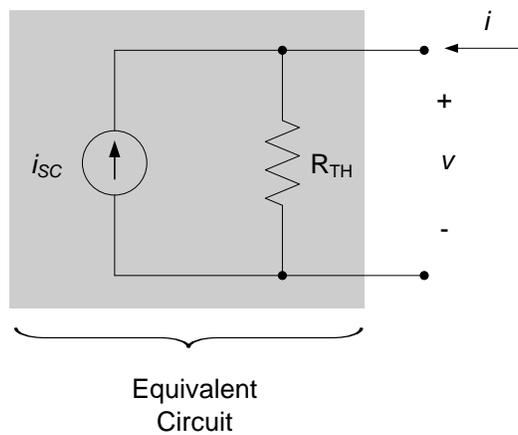


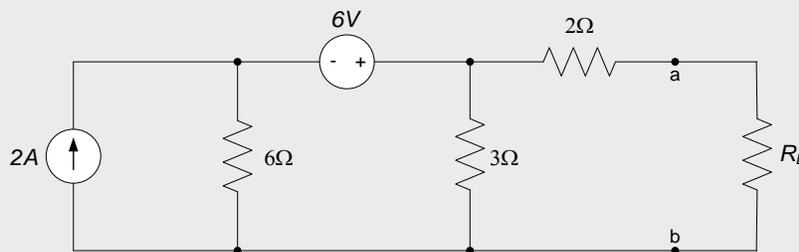
Figure 4.12. Norton equivalent circuit.

**Procedure for Determining Norton Equivalent Circuit:**

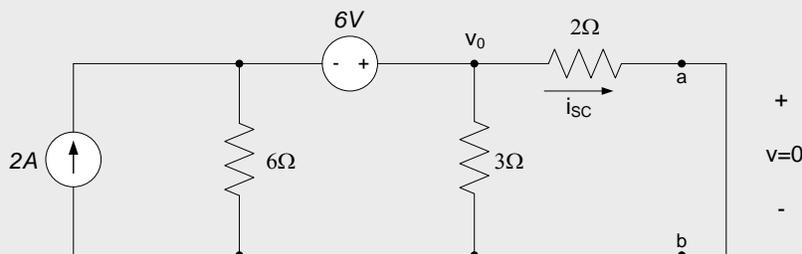
1. Identify the circuit and terminals for which the Norton equivalent circuit is desired.
2. Determine the equivalent resistance  $R_{TH}$  of the circuit. The approach for determining  $R_{TH}$  is the same for Norton circuits as Thévenin circuits.
3. Re-activate the sources and determine the short-circuit current  $i_{SC}$  across the circuit terminals. Use any analysis approach you choose to determine the short-circuit current.

**Example 4.10**

Determine the Norton equivalent of the circuit seen by the load,  $R_L$ , in the circuit below.



This is the same circuit as our previous example. The Thévenin resistance,  $R_{TH}$ , is thus the same as calculated previously:  $R_{TH}=4\Omega$ . Removing the load resistance and placing a short-circuit between the nodes a and b, as shown below, allows us to calculate the short-circuit current,  $i_{sc}$ .



Performing KCL at the node  $v_0$ , results in:

$$\frac{v_0}{2\Omega} + \frac{v_0 - 6V}{6\Omega} + \frac{v_0}{3\Omega} = 2A$$

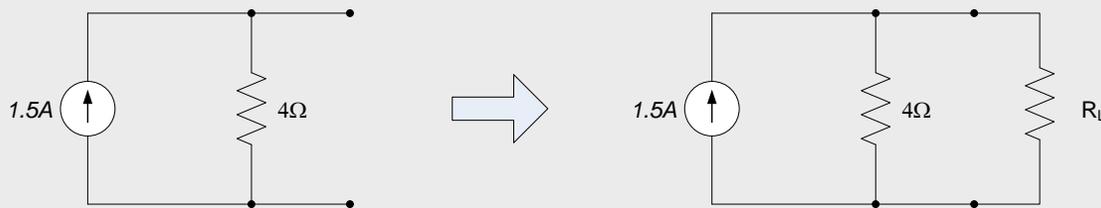
So

$$v_0 = 3V$$

Ohm's law can then be used to determine  $i_{SC}$ :

$$i_{SC} = \frac{3V}{2\Omega} = 1.5A$$

And the Norton equivalent circuit is shown on the left below. Replacing the load resistance in the equivalent overall circuit is shown to the right below.



### Exceptions:

Not all circuits have Thévenin and Norton equivalent circuits. Exceptions are:

1. An ideal current source does not have a Thévenin equivalent circuit. (It cannot be represented as a voltage source in series with a resistance.) It is, however, its own Norton equivalent circuit.
2. An ideal voltage source does not have a Norton equivalent circuit. (It cannot be represented as a current source in parallel with a resistance.) It is, however, its own Thévenin equivalent circuit.

## 4.5.3 Source Transformations

Circuit analysis can sometimes be simplified by the use of *source transformations*. Source transformations are performed by noting that Thévenin's and Norton's theorems provide two different circuits which provide essentially the same terminal characteristics. Thus, we can write a voltage source which is in series with a resistance as a current source in parallel with the same resistance, and vice-versa. This is done as follows.

Equations (4.5) and (4.8) are both representations of the  $i$ - $v$  characteristic of the same circuit. Rearranging equation (4.5) to solve for the current  $i$  results in:

$$i = \frac{v}{R_{TH}} - \frac{v_{OC}}{R_{TH}} \quad \text{Eq. 4.9}$$

Equating equations (4.8) and (4.9) leads to the conclusion that:

$$i_{SC} = \frac{v_{OC}}{R_{TH}} \quad \text{Eq. 4.10}$$

Likewise, rearranging equation (4.8) to obtain an expression for  $v$  gives:

$$v = i \cdot R_{TH} + i_{SC} \cdot R_{TH} \quad \text{Eq. 4.11}$$

Equating equations (4.11) and (4.5) results in:

$$V_{OC} = i_{SC} \cdot R_{TH} \quad \text{Eq. 4.12}$$

Which is the same result as equation (4.10).

Equations (4.10) and (4.12) lead us to the conclusion that any circuit consisting of a voltage source in series with a resistor can be transformed into a current source in parallel with the same resistance. Likewise, a current source in parallel with a resistance can be transformed into a voltage source in series with the same resistance. The values of the transformed sources must be scaled by the resistance value according to equations (4.10) and (4.12). The transformations are depicted in Fig. 4.13.

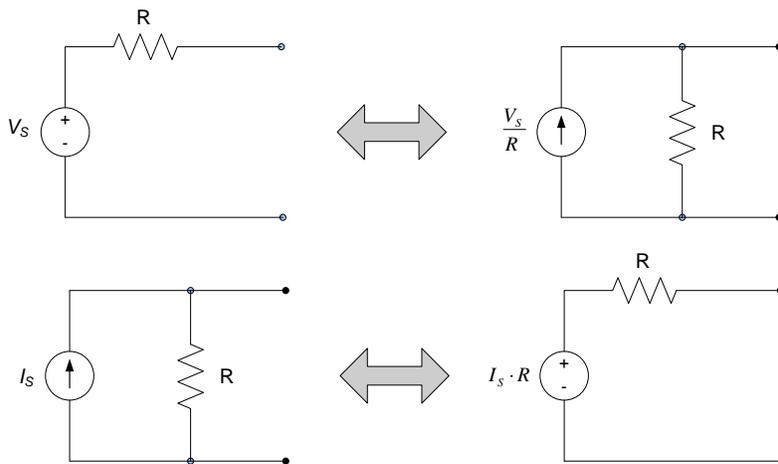
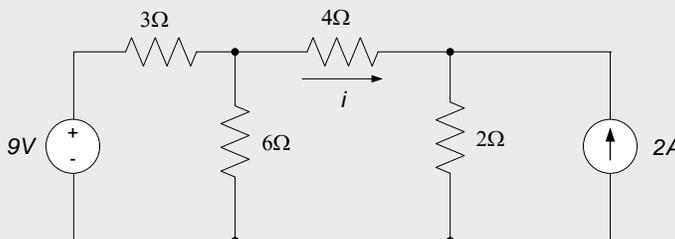


Figure 4.13. Source transformations.

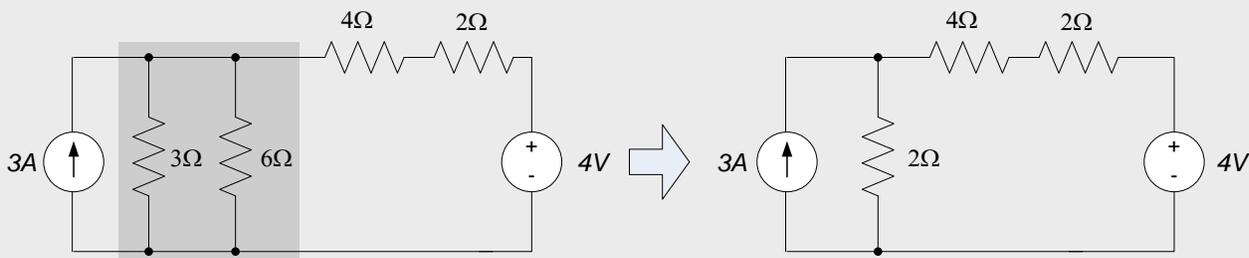
Source transformations can simplify the analysis of some circuits significantly, especially circuits which consist of series and parallel combinations of resistors and independent sources. An example is provided below.

### Example 4.11

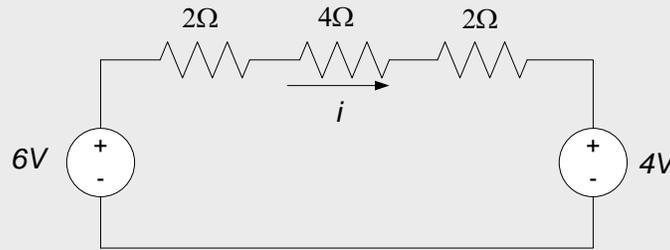
Determine the current  $i$  in the circuit shown below.



We can use a source transformation to replace the 9V source and 3Ω resistor series combination with a 3A source in parallel with a 3Ω resistor. Likewise, the 2A source and 2Ω resistor parallel combination can be replaced with a 4V source in series with a 2Ω resistor. After these transformations have been made, the parallel resistors can be combined as shown in the figure below.



The 3A source and 2Ω resistor parallel combination can be combined to a 6V source in series with a 2Ω resistor, as shown below.



The current  $i$  can now be determined by direct application of Ohm's law to the three series resistors, so that  $i = \frac{6V-4V}{2\Omega+4\Omega+2\Omega} = 0.25A$ .

#### 4.5.4 Voltage-current Characteristics of Thévenin and Norton Circuits

Previously, in section 4.4, we noted that the  $i$ - $v$  characteristics of linear two-terminal networks containing only sources and resistors are straight lines. We now look at the voltage-current characteristics in terms of Thévenin and Norton equivalent circuits.

Equations (4.5) and (4.8) both provide a linear voltage-current characteristic as shown in Fig. 4.14. When the current into the circuit is zero (open-circuited conditions), the voltage across the terminals is the open-circuit voltage,  $v_{OC}$ . This is consistent with equation (4.5), evaluated at  $i=0$ :

$$v = R_{TH} \cdot i_{OC} + v_{OC} = R_{TH} \cdot 0 + v_{OC} = v_{OC}$$

Likewise, under short-circuited conditions, the voltage differential across the terminals is zero and equation (4.8) readily provides:

$$i = \frac{v_{SC}}{R_{TH}} - i_{SC} = \frac{0}{R_{TH}} - i_{SC} = -i_{SC}$$

Which is consistent with Fig. 4.14.

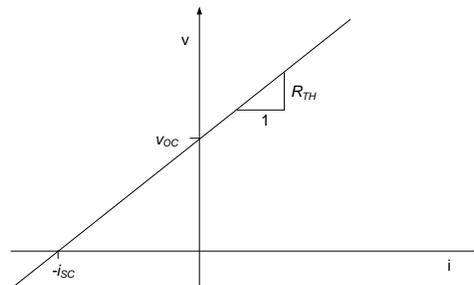


Figure 4.14. Voltage-current characteristic for Thévenin and Norton equivalent circuits.

Figure 4.14 is also consistent with equations (4.10) and (4.12) above, since graphically the slope of the line is obviously  $R_{TH} = \frac{v_{OC}}{i_{SC}}$ .

Figure 4.14 also indicates that there are three simple ways to create Thévenin and Norton equivalent circuits:

1. Determine  $R_{TH}$  and  $v_{OC}$ . This provides the slope and y-intercept of the  $i$ - $v$  characteristic. This approach is outlined above as the method for creating a Thévenin equivalent circuit.

- Determine  $R_{TH}$  and  $i_{SC}$ . This provides the slope and x-intercept of the  $i$ - $v$  characteristic. This approach is outlined above as the method for creating a Norton equivalent circuit.
- Determine  $v_{OC}$  and  $i_{SC}$ . The equivalent resistance  $R_{TH}$  can then be calculated from  $R_{TH} = \frac{v_{OC}}{i_{SC}}$  to determine the slope of the  $i$ - $v$  characteristic. Either a Thévenin or Norton equivalent circuit can then be created. This approach is not commonly used, since determining  $R_{TH}$  – the equivalent resistance of the circuit – is usually easier than determining either  $v_{OC}$  or  $i_{SC}$ .

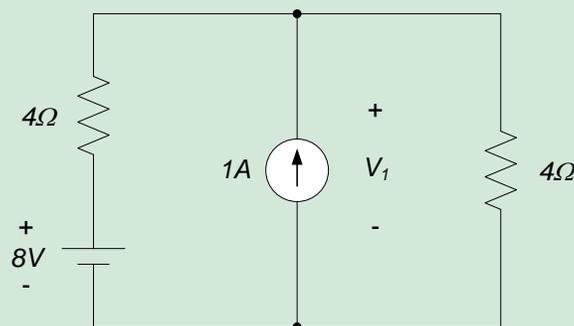
**Note:** It should be emphasized that the Thévenin and Norton circuits are not independent entities. One can always be determined from the other via a source transformation. Thévenin and Norton circuits are simply two different ways of expressing the same voltage-current characteristic.

## Section Summary

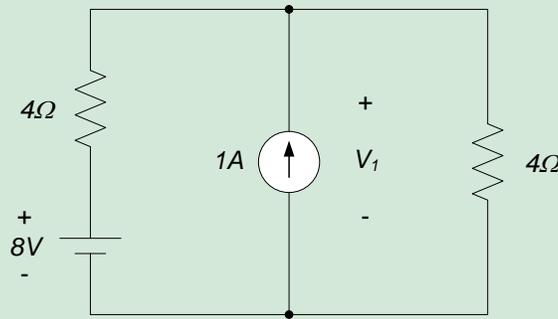
- Thévenin's theorem allows us to replace any linear portion of a circuit with equivalent circuit consisting of a voltage source in series with a resistance. This circuit is called the Thévenin equivalent, and provides the same voltage-current relationship at the terminals as the original circuit. The voltage source in the equivalent circuit is the same as the voltage which would be measured across the terminals of the original circuit, if those terminals were open-circuited. The resistance in the equivalent circuit is called the Thévenin resistance, it is the resistance that would be seen across the terminals of the original circuit, if all sources in the circuit were killed.
- Norton's theorem allows us to replace any linear portion of a circuit with equivalent circuit consisting of a current source in parallel with a resistance. This circuit is called the Norton equivalent, and provides the same voltage-current relationship at the terminals as the original circuit. The current source in the equivalent circuit is the same as the current which would be measured across the terminals of the original circuit, if those terminals were short-circuited. The resistance in the equivalent circuit is the resistance that would be seen across the terminals of the original circuit, if all sources in the circuit were killed; it is the same as the Thévenin resistance.
- Thévenin and Norton's theorems allow us to perform *source transformations* when analyzing circuits. This approach simply allows us to replace any voltage source which is in series with a resistance with a current source in parallel with the same resistance, and vice-versa. The relationship between the voltage and current sources used in these transformations are provided in equations (4.10) and (4.12).

## 4.5 Exercises

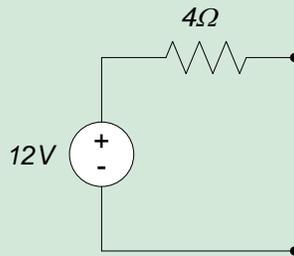
- Replace everything except the 1A current source with its Thévenin equivalent circuit and use the result to find  $V_1$ .



- Replace everything except the 1A current source with its Norton equivalent circuit and use the result to find  $V_1$ .



3. Determine a Norton equivalent circuit for the circuit below.



## 4.6 Maximum Power Transfer

It is often important for our electrical system to transfer as much power as possible to some related system. For example, in an audio system it is important that the amplifier transfer as much power as possible to the loudspeakers. Otherwise, the amplifier generates power which is not used for any productive purpose<sup>6</sup> and the efficiency of the overall system suffers.

In this section, we will develop design guidelines which will ensure that the maximum possible amount of power is transferred from our electrical circuit to the load that the circuit is driving. These guidelines will be based on Thevenin's theorem.

Consider the system shown in Fig. 4.15. The overall system consists of an electrical circuit which is being used to drive a load. Physically, the load can be either another electrical system or some electromechanical system such as an electric motor or a loudspeaker. We will model the load as an electrical resistance,  $R_L$ , though the principles presented here are applicable to more general loading conditions.

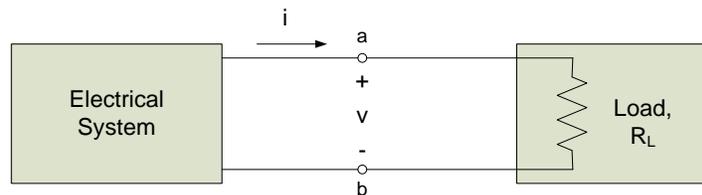


Figure 4.15. General electrical network – load combination.

We will replace our electrical system with its Thevenin equivalent in order to analyze the power delivered by the circuit to the load. The overall circuit that we are analyzing is now modeled as shown in Fig. 4.16.

<sup>6</sup> Other than, perhaps, heating the room it is in.

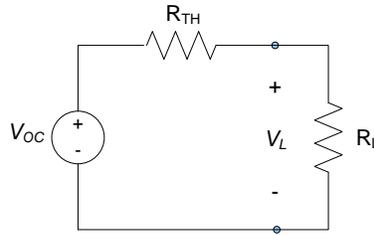


Figure 4.16. Electric circuit – load combination. Electric circuit modeled by its Thevenin equivalent.

From Fig. 4.16, the voltage delivered to the load can be readily determined from a voltage divider relation:

$$V_L = V_{OC} \frac{R_L}{R_L + R_{TH}} \quad \text{Eq. 4.13}$$

Thus, the power delivered to the load is:

$$P_L \frac{V_L^2}{R_L} = \frac{V_{OC}^2}{R_L} \left( \frac{R_L}{R_L + R_{TH}} \right)^2 \quad \text{Eq. 4.14}$$

Figure 4.17 shows a plot of the power delivered to the load, as a function of the load resistance. The power delivered to the load is zero when the load resistance is zero (since there is no voltage drop across the load under this condition) and goes to zero as the load resistance approaches infinity (since there is no current provided to the load under this condition). At some value of  $R_L$  the power transfer will be maximized – our goal will be to determine the value for  $R_L$  which maximizes the power delivered to the load.

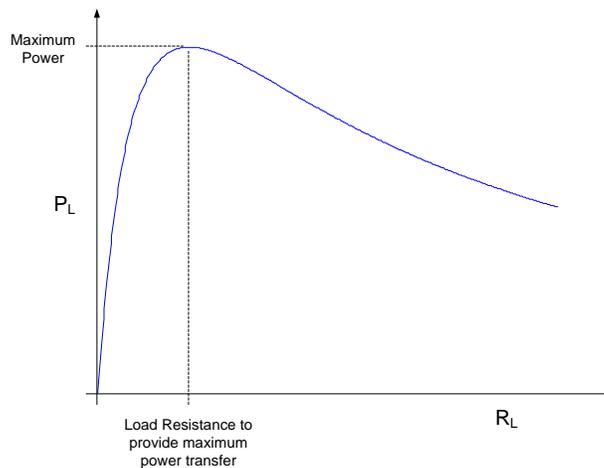


Figure 4.17. Delivered power vs. load resistance.

The maximum value of power on the curve shown in Fig. 4.17 can be determined by differentiating equation (4.14) with respect to the load resistance  $R_L$  and setting the result to zero. This leads to:

$$\frac{\partial P_L}{\partial R_L} = V_{OC}^2 \left[ \frac{(R_L + R_{TH})^2 - 2R_L(R_{TH} + R_L)}{(R_L + R_{TH})^4} \right] = 0 \quad \text{Eq. 4.15}$$

The above condition is satisfied if the numerator of equation (4.15) is zero, so our condition becomes:

$$(R_L + R_{TH})^2 = 2R_L(R_{TH} + R_L)$$

Dividing both sides by  $R_{TH} + R_L$  results in:

$$R_L + R_{TH} = 2R_L$$

Or

$$R_L = R_{TH} \quad \text{Eq. 4.16}$$

Thus, maximum power transfer takes place when the load resistance and the Thevenin resistance of the circuit supplying the power are equal. The above result is sometimes called the *maximum power transfer theorem*. When the conditions of the maximum power transfer theorem are met, the total power delivered to the load is:

$$P_L = \frac{\left(\frac{V_{OC}}{2}\right)^2}{R_{TH}} = \frac{V_{OC}^2}{4R_{TH}} \quad \text{Eq. 4.17}$$

This is one half of the total power generated by the circuit, half the power is absorbed in the resistance  $R_{TH}$ .

**Conclusion:** The power delivered to the load is maximized if the load resistance is equal to the Thevenin resistance of the circuit supplying the power. When this condition is met, the circuit and the load are said to be matched. When the load and the circuit are matched, 50% of the power generated in the circuit can be delivered to the load – under any other circumstances, a smaller percentage of the generated power will be provided to the load.

### 4.6.1 Practical Power Supplies

Practical power supplies were discussed in section 2.4. It was seen that the presence of an internal resistance in a voltage or current source limited the power that could be delivered to a circuit connected to the source. Practical power supplies are a special case of the results presented above; we use them below as examples of the application of the above principles.

In section 2.4, practical voltage sources were modeled as an ideal voltage source  $V_s$  in series with some internal resistance  $R_s$ , as shown in Fig. 4.18. This corresponds exactly to a Thevenin circuit with  $V_{OC}=V_s$  and  $R_{TH}=R_s$ . The practical voltage source provides maximum power to a circuit connected to it when the input resistance of the circuit (the equivalent resistance of the circuit, seen at the terminals to which the power source is connected) is equal to the internal resistance of the voltage source. Under these circumstances, the power delivered to the circuit is:

$$P = \frac{V_s^2}{4R_s}$$

The same amount of power is converted to heat within the power supply; this is the reason many power supplies contain a fan to actively disperse this heat to the atmosphere. If the circuit's input resistance is not equal to the source resistance, less power is transmitted to the circuit and a correspondingly greater amount is dissipated within the power supply.

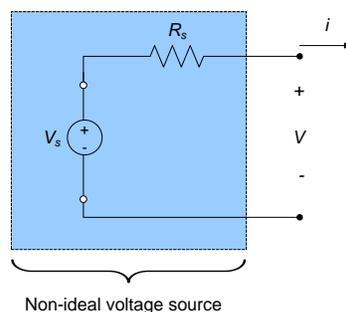


Figure 4.18. Practical voltage source model.

Practical current sources were modeled in section 2.4 as an ideal current source  $I_S$  in parallel with some internal resistance  $R_S$ , as shown in Fig. 4.19. This corresponds directly to a Norton equivalent circuit with  $I_{SC}=I_S$  and  $R_{TH}=R_S$ . The current source provides maximum power to a circuit connected to it when the input resistance of the circuit is equal to the internal resistance of the source. A source transformation in conjunction with equation (5) indicates that the power delivered to the circuit by the current source is:

$$P = \frac{R_S I_S^2}{4}$$

Again, a reduced percentage of the power generated by the source will be delivered to the circuit when the circuit and source are not well matched.

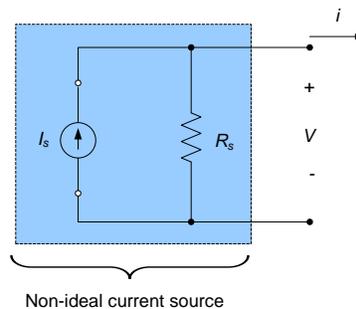


Figure 4.19. Non-ideal current source model.

Often, it may not be feasible to match the load with the power supply. For example, when we are testing circuits in our lab assignments we do not generally attempt to maximize the power delivered to the circuit – this is typical when prototype circuits are being tested. One simply recognizes that excessive power is being dissipated within the power supply and that the overall system is not functioning efficiently. If, however, the power supply and associated circuit are being designed as part of an integrated overall system one will generally attempt to match the power supply to the rest of the system.

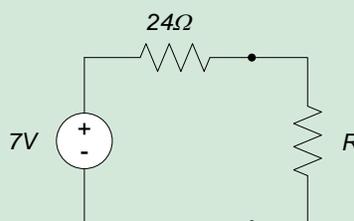
One problem which can occur during circuit testing is that extremely poorly matched power supply-load combinations may result in so much power being dissipated within the power supply that insufficient power is available to drive the load. This can result in the load apparently behaving abnormally, unless power delivery effects are considered.

## Section Summary

- The maximum power that a circuit can deliver to a load resistor occurs when the load resistance is equal to the Thévenin equivalent resistance of the circuit.

## 4.6 Exercises

1. Determine the resistance  $R$  which will absorb the maximum power from the 7V source.



## Real Analog Chapter 4: Lab Projects

### 4.3.1: Superposition

In this lab, we will analyze, build, and test a circuit containing multiple sources. Except in special cases, multiple sources preclude the use of analysis techniques based entirely on circuit reduction approaches. We can, however, use circuit reduction techniques in conjunction with superposition to determine the response of a circuit with multiple sources.

It should be noted that alternate analysis techniques can be used to predict this circuit's response. The approach used to analyze the circuit does not, however, affect the actual circuit's behavior. Other lab assignments will use this same circuit, in conjunction with other analysis techniques. Please keep in mind that the circuit's physical behavior is not affected by the analysis technique used; the same measurement on the same circuit should provide the same result, regardless of the approach used to predict the response.

#### Before beginning this lab, you should be able to:

- Calculate circuit voltages, currents using circuit reduction techniques
- Use superposition techniques to determine circuit voltage, current responses in the presence of multiple sources

#### After completing this lab, you should be able to:

- Compare measured voltages and currents in an electrical circuit with predictions based on superposition techniques

#### This lab exercise requires:

- Analog Discovery 2 module
- Digilent Analog Parts Kit
- Digital multimeter (optional)

#### Symbol Key:

-  DEMO Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
-  ANALYSIS Analysis; include principle results of analysis in laboratory report.
-  SIM Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
-  DATA Record data in your lab notebook.

#### General Discussion:

This lab assignment concerns the circuit shown in Figure 1 below. Use V+ to apply the 5V voltage source and W1 to apply the 3V source. We wish to determine the voltage difference  $V$ .

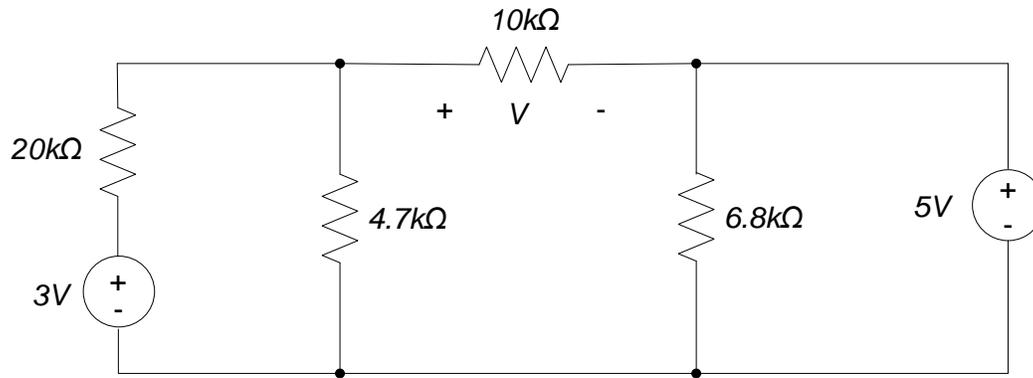


Figure 1. Circuit schematic.

**Pre-lab:****ANALYSIS**

Use superposition to predict the voltage  $V$  in the circuit of Figure 1. (e.g. determine the response of the voltage across the  $10\text{k}\Omega$  resistor resulting from the  $3\text{V}$  source, determine the response of the voltage across the  $10\text{k}\Omega$  resistor resulting from the  $5\text{V}$  source, and sum the results to obtain the total voltage across the  $10\text{k}\Omega$  resistor from both sources.)

**Lab Procedures:****DATA**

1. Choose the four resistors for the circuit of Figure 1. Record the actual resistance values.

**DATA**

2. Build the circuit of Figure 1, with the  $5\text{V}$  source replaced with a short circuit. Measure the resulting voltage across the  $10\text{k}\Omega$  resistor.

**DATA**

3. Build the circuit of Figure 1, with the  $3\text{V}$  source replaced with a short circuit. Measure the resulting voltage across the  $10\text{k}\Omega$  resistor.

**DATA**

4. Build the circuit of Figure 1, with both sources in place. Measure the resulting voltage across the  $10\text{k}\Omega$  resistor.

**ANALYSIS**

5. Calculate a percent difference between the sum of the voltages of step 2 and step 3 and the voltage measured in step 4. Comment on the agreement between the principle of superposition and your results.

**ANALYSIS**

6. Create a table summarizing the results of your pre-lab analysis and your experimental results. Your table should list, at a minimum:

- The measured and expected voltage  $V$  due to the  $3\text{V}$  source alone
- The measured and expected voltage  $V$  due to the  $5\text{V}$  source alone
- The measured and expected voltage  $V$  due to both sources
- The percent differences between the measured and expected values for the above three cases.



## Real Analog Chapter 4: Lab Projects

### 4.3.2: Superposition

In this lab, we will analyze, build, and test a circuit containing multiple sources. Except in special cases, multiple sources preclude the use of analysis techniques based entirely on circuit reduction approaches. We can, however, use circuit reduction techniques in conjunction with superposition to determine the response of a circuit with multiple sources.

It should be noted that alternate analysis techniques can be used to predict this circuit's response. The approach used to analyze the circuit does not, however, affect the actual circuit's behavior. Other lab assignments will use this same circuit, in conjunction with other analysis techniques. Please keep in mind that the circuit's physical behavior is not affected by the analysis technique used; the same measurement on the same circuit should provide the same result, regardless of the approach used to predict the response.

#### Before beginning this lab, you should be able to:

- Calculate circuit voltages, currents using circuit reduction techniques
- Use superposition techniques to determine circuit voltage, current responses in the presence of multiple sources

#### After completing this lab, you should be able to:

- Compare measured voltages and currents in an electrical circuit with predictions based on superposition techniques

#### This lab exercise requires:

- Analog Discovery 2 module
- Digilent Analog Parts Kit
- Digital multimeter (optional)

#### Symbol Key:

- DEMO** Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
- ANALYSIS** Analysis; include principle results of analysis in laboratory report.
- SIM** Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
- DATA** Record data in your lab notebook.

#### General Discussion:

This lab assignment concerns the circuit shown in Figure 1 below. We wish to determine the voltage  $V$  across the  $6.8\text{k}\Omega$  resistor.

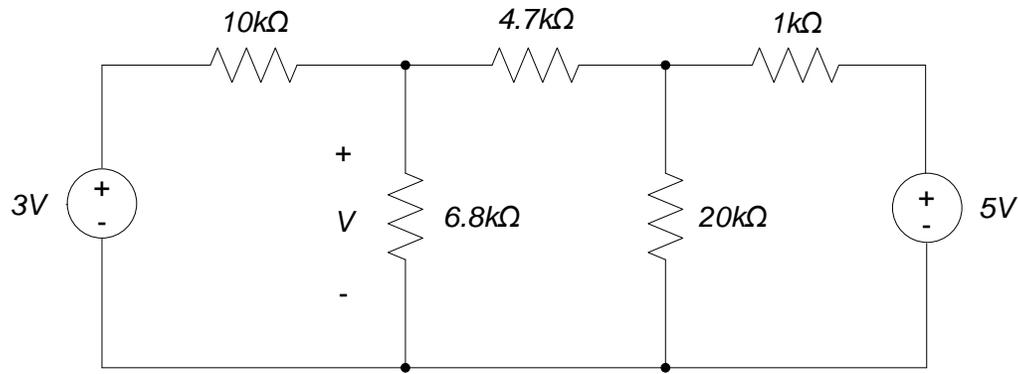


Figure 1. Circuit schematic.

**Pre-lab:****ANALYSIS**

Use superposition to predict the voltage  $V$  in the circuit of Figure 1. (e.g. determine the response of the voltage across the  $6.8\text{k}\Omega$  resistor resulting from the  $5\text{V}$  source, determine the response of the voltage across the  $6.8\text{k}\Omega$  resistor resulting from the  $3\text{V}$  source, and sum the results to obtain the total voltage across the  $6.8\text{k}\Omega$  resistor from both sources.)

**Lab Procedures:****DATA**

1. Choose the five resistors for the circuit of Figure 1. Record the actual resistance values.

**DATA**

2. Build the circuit of Figure 1, with the  $3\text{V}$  source replaced with a short circuit. Measure the resulting voltage across the  $6.8\text{k}\Omega$  resistor.

**DATA**

3. Build the circuit of Figure 1, with the  $5\text{V}$  source replaced with a short circuit. Measure the resulting voltage across the  $6.8\text{k}\Omega$  resistor.

**DATA**

4. Build the circuit of Figure 1, with both sources in place. Measure the resulting voltage across the  $6.8\text{k}\Omega$  resistor.

**ANALYSIS**

5. Calculate a percent difference between the sum of the voltages of step 2 and step 3 and the voltage measured in step 4. Comment on the agreement between the principle of superposition and your results.

**ANALYSIS**

6. Create a table summarizing the results of your pre-lab analysis and your experimental results. Your table should list, at a minimum:

- The measured voltage  $V$  due to the  $5\text{V}$  source alone
- The expected value of the voltage due to the  $5\text{V}$  source from your pre-lab
- The measured voltage  $V$  due to the  $3\text{V}$  source alone
- The expected value of the voltage due to the  $3\text{V}$  source from your pre-lab
- The measured voltage  $V$  due to both sources
- The expected value of the voltage due to both sources from your pre-lab



## Real Analog Chapter 4: Lab Projects

### 4.4.1: Two-terminal Characteristics

In this lab, we will estimate the voltage-current characteristics of a simple circuit containing a power supply from measured data. The voltage-current data estimated from experimental data will be compared to analytical estimates.

#### Before beginning this lab, you should be able to:

- Calculate voltage-current characteristics for circuits containing sources
- Perform least-squares curve fitting to data (Lab 1.3.2)

#### After completing this lab, you should be able to:

- Estimate voltage-current characteristics from measured data and compare the result with analytical estimates

#### This lab exercise requires:

- Analog Discovery 2 module
- Digilent Analog Parts Kit
- Digital multimeter

#### Symbol Key:

- DEMO** Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
- ANALYSIS** Analysis; include principle results of analysis in laboratory report.
- SIM** Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
- DATA** Record data in your lab notebook.

#### General Discussion:

This lab assignment concerns the circuit shown in Figure 1 below. (Pay special attention to the polarity of the 3V source relative to the polarity of  $V_{ab}$ !)

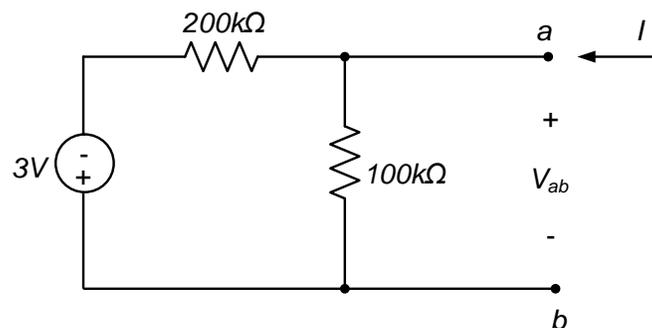


Figure 1. Circuit schematic.

**Pre-lab:****ANALYSIS**

1. Calculate the functional relationship between the voltage  $V_{ab}$  and the current  $I$  for the circuit of Figure 1. Plot the voltage  $V_{ab}$  as a function of the current,  $I$  (the plot should have voltage on the vertical axis and current on the horizontal axis). Calculate the slope of the curve and the y-intercept of the curve.

**ANALYSIS**

2. Calculate the expected voltage  $V_{ab}$  if the terminals a-b are open-circuited. Compare this voltage to the y-intercept of the curve you calculated in part (a) of the pre-lab. Kill the 3V source and determine the equivalent resistance of the circuit seen across the terminals a-b. Compare this value to the slope of the curve you calculated in part (a) of the pre-lab.

**Lab Procedures:****DATA**

1. Build the circuit of Figure 1. Use W1 to apply the 3V voltage source and W2 to apply the voltage  $V_{ab}$ . (Please pay attention to the polarity on the 3V source!) Record the actual resistance values.

**DATA**

2. Record the current,  $I$ , resulting from values  $V_{ab} = 0.2V, 0.5V, 1.0V, 2.0V, 4.0V,$  and  $5.0V$ . Tabulate the voltage vs. current data. Plot the data, with current on the x-axis and voltage on the y-axis. Perform a least-squares curve fit of a straight line to the data and determine the slope and y-intercept of the line.

**ANALYSIS****DATA**

3. Replace the  $V_{ab}$  voltage source with an open-circuit and measure the resulting voltage  $V_{ab}$ .

**DATA**

4. With  $V_{ab}$  still open-circuited as in step 4, replace the 3V source with a short-circuit and use your DMM to measure the resistance seen across terminals a-b.



## Real Analog Chapter 4: Lab Projects

### 4.5.1: Thevenin's Theorem

In this lab, we experimentally investigate Thevenin's Theorem. We will analytically determine a Thevenin equivalent for a given circuit; we will then experimentally determine the Thevenin resistance and the open-circuit voltage necessary to create the Thevenin equivalent circuit.

We will use the Thevenin equivalent to the given circuit to analytically predict the voltage which will result from loading the circuit with a resistor. We will then test our Thevenin equivalent circuit by:

1. connecting the load resistor to the original circuit and measuring the resulting load voltage, and
2. physically creating the Thevenin equivalent circuit, connecting the load resistor to this circuit, and measuring the resulting load voltage and comparing the results obtained with the Thevenin circuit and the original circuit.

Finally, we will vary the value of the load resistance and measure the power dissipated by the load resistor as a function of the load resistance. We will use this result to check our expectations based on the maximum power transfer theorem.

#### Before beginning this lab, you should be able to:

- Create a Thevenin equivalent of an arbitrary linear circuit
- Determine the load resistance necessary to draw the maximum power from a circuit

#### After completing this lab, you should be able to:

- Compare measured voltages and currents in an electrical circuit with predictions based on superposition techniques

#### This lab exercise requires:

- Analog Discovery 2 module
- Digilent Analog Parts Kit
- Digital multimeter

#### Symbol Key:

- DEMO** Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
- ANALYSIS** Analysis; include principle results of analysis in laboratory report.
- SIM** Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
- DATA** Record data in your lab notebook.

#### General Discussion:

This lab assignment concerns the circuit shown in Figure 1 below. We want to determine the equivalent Thevenin circuit seen at the terminals a-b.

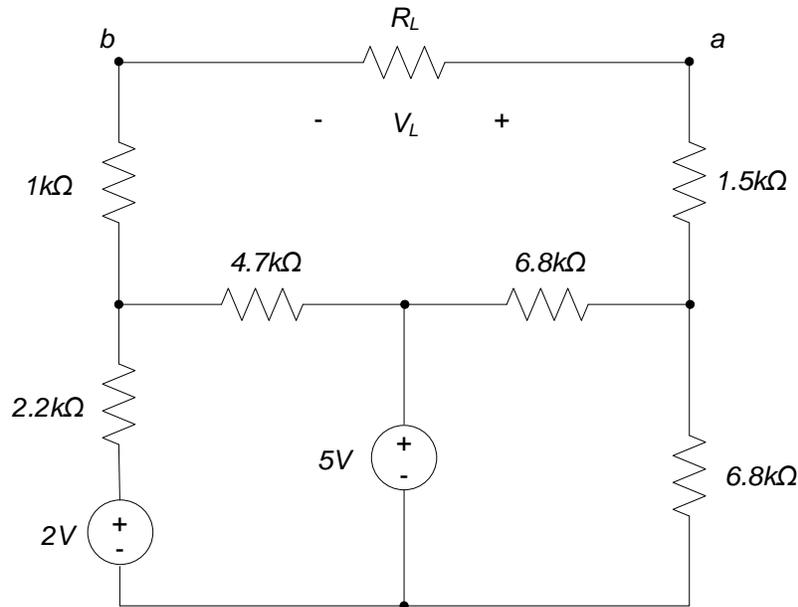


Figure 1. Circuit schematic.

**Pre-lab:****ANALYSIS**

Determine the Thevenin equivalent circuit (the open circuit voltage  $V_{OC}$  and the Thevenin resistance  $R_{TH}$ ) seen by a resistor connected to the terminals a-b of the circuit of Figure 1. (e.g. determine the Thevenin equivalent circuit seen by the load resistor  $R_L$ .) Sketch the Thevenin equivalent circuit.

**Lab Procedures:****DATA**

1. Choose the six resistors (all resistors except for the resistor  $R_L$ ) in the circuit of Figure 1. Record the actual resistance values.

**DATA**

2. Assemble the circuit of Figure 1, except for the resistor  $R_L$ . Measure the open-circuit voltage ( $V_{OC}$ ) across the terminals a-b. Also measure the Thevenin resistance ( $R_{TH}$ ) seen at the terminals a-b. (You will need to replace the voltage sources in the circuit with short circuits in order to determine the Thevenin resistance of the circuit.) Compare these measured values with your expectations based on the pre-lab analyses, by calculating a percent error between the actual and theoretical values.

**ANALYSIS****DATA**

3. Pick a random resistor for the load resistance  $R_L$ , in the range  $4k\Omega < R_L < 10k\Omega$ , and connect it between the terminals a and b in the circuit of Figure 1. Measure the voltage across the load resistance and compare this voltage to your expectation based on the Thevenin equivalent circuit from your pre-lab.

**DATA**

4. Build the Thevenin equivalent circuit you determined in the pre-lab. Load the Thevenin circuit with the resistor  $R_L$  you used in part 4 above. Measure the voltage across the load resistor and compare it to the voltage you measured in part 4. (e.g. compute a percent difference between the load voltage produced by the original circuit and the Thevenin equivalent circuit.)

**ANALYSIS****DATA**

5. Connect a potentiometer between the terminals a-b. Measure and record the load voltage as a function of potentiometer resistance. Calculate the power delivered to the load (

**ANALYSIS**

$$P_{Load} = \frac{V_{ab}^2}{R_L}) \text{ as a function of the load resistance. Plot the load power as a function of the}$$

load resistance and estimate the load resistance which results in maximum power dissipation by the load. Compare this value from your expectations based on your Thevenin resistance from the pre-lab and the maximum power theorem.





## Real Analog Chapter 4: Lab Projects

### 4.6.1: Maximum Power Transfer

This assignment involves the maximum power transfer theorem. We will inappropriately attempt to maximize the power delivered to a load resistor by, in some sense, re-defining the source circuit.

#### Before beginning this lab, you should be able to:

- Create a Thevenin equivalent of an arbitrary linear circuit
- Determine the load resistance necessary to draw the maximum power from a circuit

#### After completing this lab, you should be able to:

- Compare measured voltages and currents in an electrical circuit with predictions based on superposition techniques

#### This lab exercise requires:

- Analog Discovery 2 module
- Digilent Analog Parts Kit
- Digital multimeter

#### Symbol Key:

- DEMO** Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
- ANALYSIS** Analysis; include principle results of analysis in laboratory report.
- SIM** Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
- DATA** Record data in your lab notebook.

#### General Discussion:

Thévenin's theorem tells us that the power delivered to a load is maximized if the load resistance is equal to the Thevenin resistance of the circuit supplying the power. It is often desirable to design our load resistance to draw the maximum power from a source circuit – this results in the minimum power being lost as heat dissipated within the source circuit. In this case, the load is said to be *matched* to the source.

In this lab exercise, we will attempt to maximize power transfer to a load by modifying the source circuit. (We will, in essence, attempt to match the source to the load rather than the other way around.) We will see that this approach is inappropriate; the maximum power transfer theorem does not apply in “reverse”.

#### Pre-lab:

Our goal is to transfer the maximum amount of power to a 2.2kΩ load resistor. The circuit schematic shown in Figure 1 illustrates connection of the load resistor directly to a non-ideal source. Assuming that the source resistance  $R_s$  is much less than the load resistance  $R_L$  (e.g.  $R_s \ll R_L$ ), calculate the power delivered to the load resistor.

**ANALYSIS**

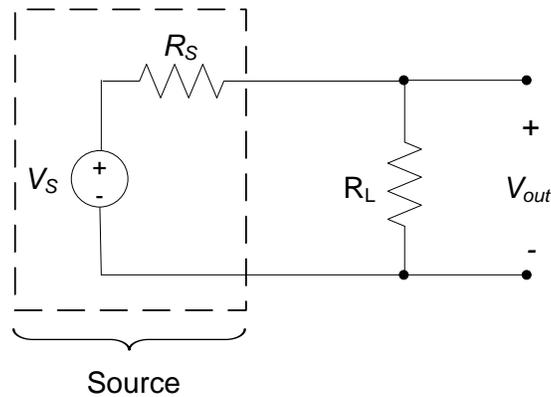


Figure 1. “Loaded” non-ideal source.

We know that the maximum power will be delivered to the load resistor of Figure 1 if  $R_L = R_S$ . In this lab assignment, we do not have the luxury of changing our load, but we can re-define our “source”. Add a resistance to the “source” as shown in Figure 2; note that the “source” resistance and the load resistance are now the same. Calculate the power delivered to the load resistor for this case and compare this power delivery to that of Figure 1.

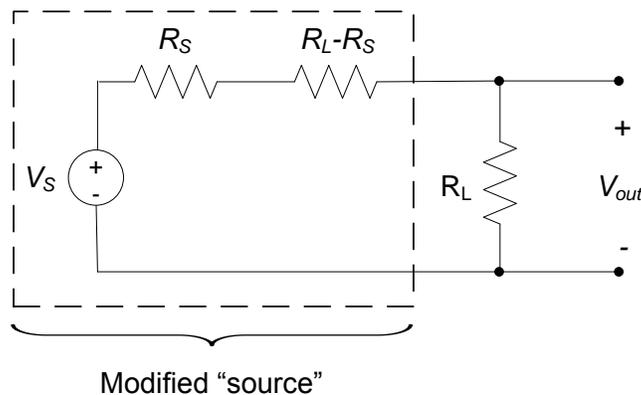


Figure 2. “Loaded” source with modified source resistance.

### Lab Procedures:

#### DATA

1. Implement the circuit of Figure 1 with  $R_L = 2.2\text{k}\Omega$  and  $V_S = 5\text{V}$ . Use a  $5\text{k}\Omega$  potentiometer to implement  $R_S$ ; we will vary the load resistance in step 3 below. Measure  $V_{OUT}$  and calculate the power delivered to the load. Compare this power delivery with the power delivered to the  $25\Omega$  resistor in lab assignment 2.4. Are the relative powers delivered consistent with your estimate of the source resistance determined in lab assignment 2.4 and the maximum power theorem?

#### DATA

2. Modify the “source” according to Figure 2, with  $R_L = 2.2\text{k}\Omega$  and  $V_S = 5\text{V}$ . You may assume that  $R_S \ll R_L$ , so that  $R_L - R_S \approx R_L = 2.2\text{k}\Omega$ . You may use a fixed  $2.2\text{k}\Omega$  resistor to implement the added source resistance  $R_L - R_S$ . Measure  $V_{OUT}$  and calculate the power delivered to the load. Is this power consistent with your previous results and the maximum power theorem?
3. In step 1 above, we (hopefully) realized that attempting to modify the source resistance does not result in the maximum overall power transfer to the load. In this step, we will see that the modified circuit of step 2 does provide the maximum power transfer for that value of source resistance. (That maximum power is just less than the maximum possible power transfer if we vary the source resistance.)

#### DATA

In order to do this, measure  $V_{OUT}$  and calculate the power delivered to the load for at least five values of load resistance over a range of (approximately)  $1\text{k}\Omega < R_L < 4\text{k}\Omega$ . Use  $V_S = 5\text{V}$ , as in steps

1 and 2. (This is easy if the load resistance is implemented with a potentiometer, as recommended in step 1 above.) Plot the power delivered to the load vs. the load resistance. Are your results consistent with the maximum power theorem?

**DEMO**

4. Demonstrate operation of your circuit to the Teaching Assistant. Have the TA initial the appropriate page(s) of your lab notebook and the lab checklist.

## Real Analog Chapter 4: Lab Worksheets

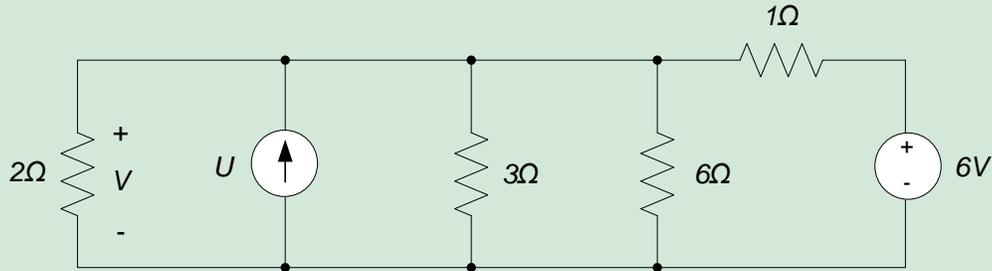
### 4.6.1: Maximum Power Transfer (30 points total)

1. Provide below your measured  $V_{OUT}$  and power delivered to the load. (Be sure to note actual load resistance value.) Comparison with power delivered to the  $25\Omega$  resistor in lab 2.4. Comment on results vs. expectations based on estimated source resistance from lab 2.4. (6 pts)
2. In the space below, provide your measured  $V_{OUT}$  and the power delivered to the load with modified source. (Provide actual values for added source resistance and load resistance.) Briefly discuss your results relative to previous results and the maximum power theorem. (7 pts)
3. Attach to this worksheet a table of load resistances,  $V_{OUT}$ , and power delivered to load for at least five values of load resistance over a range of (approximately)  $1k\Omega < R_L < 4k\Omega$ . Also attach to this worksheet a plot of power delivered to the load vs. load resistance. In the space below, comment on your plot result relative to previous results. Does the plot agree with your expectations based on the maximum power theorem? (12 pts)
4. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuits' operation. (5 pts)

TA Initials: \_\_\_\_\_

## Real Analog Chapter 4: Homework

4.1 Use superposition to determine the voltage  $V$  in the circuit below. The current  $U$  is an arbitrary value.

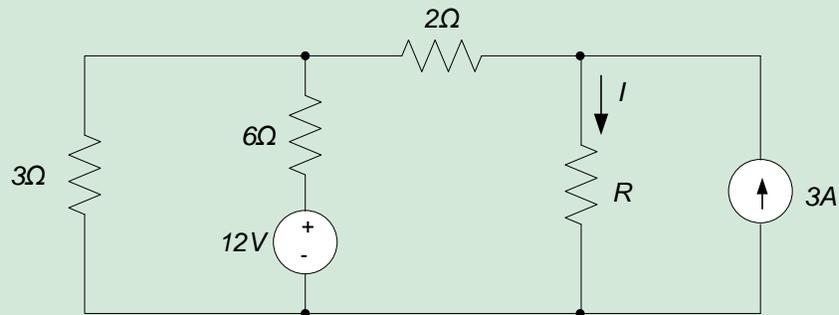


4.2 For the circuit of problem 1, determine an input-output relationship between the current  $U$  and the voltage  $V$ . Plot the relationship between  $U$  and  $V$ , using  $U$  as the independent variable (e.g.  $U$  is on the horizontal axis.)

4.3 For the circuit of problem 1, replace everything but the current source with its Thevenin equivalent circuit. Replace the current source in your Thevenin circuit and determine the input-output relationship between the current  $U$  and the voltage  $V$ . Does your result agree with the input-output relation you obtained in problem 2?

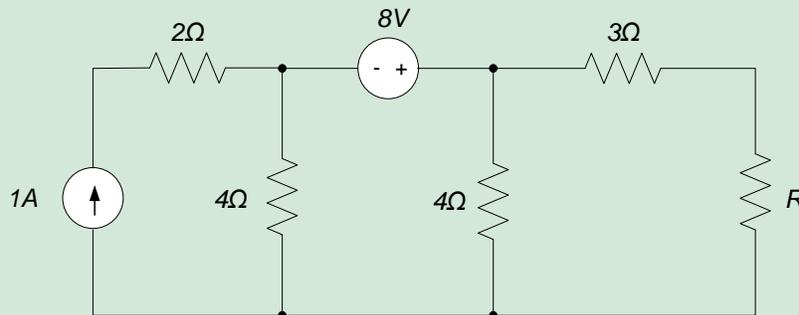
4.4 For the circuit below

- Replace everything except the resistor  $R$  in the circuit below with its Thevenin equivalent circuit.
- Using your result from part (a), determine the current  $I$  if  $R = 3\Omega$ .
- Determine the value of  $R$  that draws maximum power from the circuit.

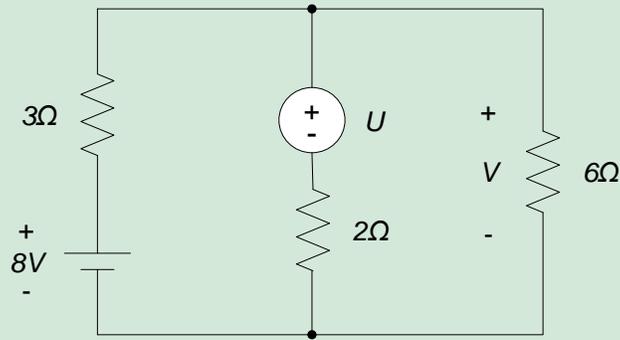


4.5 For the circuit below

- Replace everything except the resistor  $R$  in the circuit below with its Thevenin equivalent circuit.
- Determine the value of  $R$  that draws maximum power from the circuit.

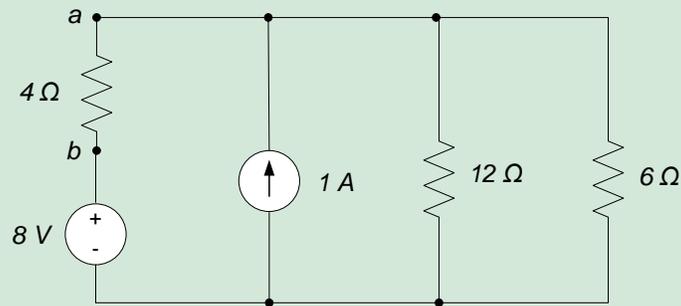


4.6 Use superposition to determine the voltage  $V$  in the circuit below. The voltage  $U$  is an arbitrary value.

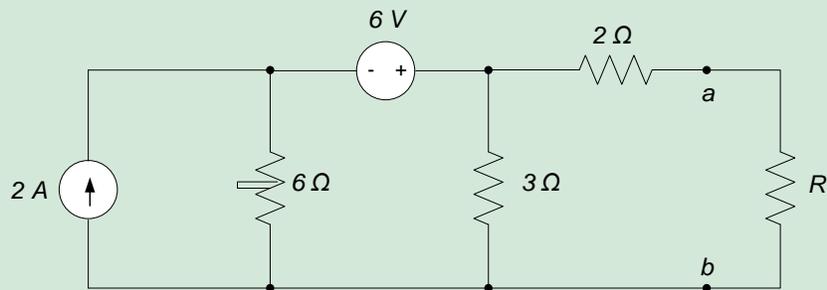


4.7 For the circuit of problem 1, determine an input-output relationship between the voltage  $U$  and the voltage  $V$ . Plot the relationship between  $U$  and  $V$ , using  $U$  as the independent variable (e.g.  $U$  is on the horizontal axis.)

4.8 Replace everything in the circuit below, *except* the  $4\Omega$  resistor, with its Thevenin equivalent circuit. Use the result to determine  $v_{ab}$ .



4.9 Replace everything to the left of the terminals a-b with its Thevenin equivalent circuit. Determine the value of  $R$  that will draw the maximum power from the circuit.



## Chapter 5: Operational Amplifiers

### 5 Introduction and Chapter Objectives

*Operational amplifiers* (commonly abbreviated as *op-amps*) are extremely useful electronic devices. Some argue, in fact, that operational amplifiers are the single most useful integrated circuit in analog circuit design. Operational amplifier-based circuits are commonly used for *signal conditioning*, performing *mathematical operations*, and *buffering*. These topics are discussed briefly below.

*Signal conditioning* is the process of manipulating a given signal (such as a voltage) to improve its properties or usefulness. Examples of common signal conditioning processes are:

- **Level adjustment:** the overall level of a signal may be too small to be usable. For example, the voltage output from a thermocouple (an electrical component used to measure temperature) may be only a few thousandths of a volt. It is often desirable to *amplify* the signal to increase the output voltage – this is often done using circuits containing operational amplifiers.
- **Noise reduction:** electrical signals are susceptible to noise; an undesirable component of a signal. (For example, static on a radio signal.) Operational amplifier circuits can be used to remove, or filter out undesirable components of a voltage signal.
- **Signal manipulation:** Electrical signals are often used to transmit information. For example, the voltage output of a thermocouple changes as the temperature of the thermocouple changes. The sensitivity of the thermocouple output to temperature changes may be changed by an operational amplifier circuit to provide a more readily usable output voltage-to-temperature relationship.

A common use of electrical circuits is to perform mathematical operations. So far, we have focused on developing mathematical models of existing circuits – we have been performing analysis tasks. The design process, conversely, can be considered to consist of implementing an electrical circuit that will perform a desired mathematical operation. (Of course, a large part of the design process consists of determining what mathematical operation is to be performed by the circuit.) Operational amplifier circuits are readily developed to perform a wide range of mathematical operations, including addition, subtraction, multiplication, differentiation, and integration.

*Buffers* allow us to electrically isolate one section of an electrical circuit from another. For example, using an electrical circuit to supply power to a second electrical circuit may result in undesirable loading effects, in which the power requirements of the second circuit exceed the power that the first circuit can provide. In this case, a buffer can be used to isolate the two circuits and thus simplifying design problems associated with integrating the two circuits. Operational amplifier circuits are commonly used for this purpose.

Operational amplifiers (or *op-amps*) are active devices. This differs from *passive* devices, such as resistors, in that an external power source must be provided to the operational amplifier in order to make it function properly. Op-amps are rather complex devices, consisting of a number of interconnected transistors and resistors. We will not be interested at this point in a detailed description of the internal operation of operational amplifiers – instead, we will use an op-amp model which provides us with relatively simple input-output relations for the overall circuit. In fact, our op-amp model will most often take the form of a dependent source<sup>7</sup>. In their most basic form, operational amplifiers are most readily modeled as voltage controlled voltage sources, but it can be used within other circuits to create devices which act as other types of dependent sources. This simplified model will be adequate for many analysis and design purposes.

---

<sup>7</sup> Op-amp circuits will be our first exposure to physical devices which act as dependent sources.

The operational amplifier symbol which we will most often use is shown in Fig. 5.1. Operational amplifiers are essentially three-terminal devices (ignoring the power supply connections previously mentioned for the moment), having two input terminals and one output terminal. The inputs are called the inverting terminal (indicated by the  $-$  sign) and the non-inverting terminal (indicated by the  $+$  sign). We will use  $v_n$  and  $i_n$  to denote the voltage and current at the inverting terminal, and  $v_p$  and  $i_p$  to denote the voltage and current at the non-inverting terminal. The voltage and current at the output terminal are denoted as  $v_{OUT}$  and  $i_{OUT}$ . The voltages  $v_p$ ,  $v_n$ , and  $v_{OUT}$  are all measured relative to some common reference voltage level, such as ground.

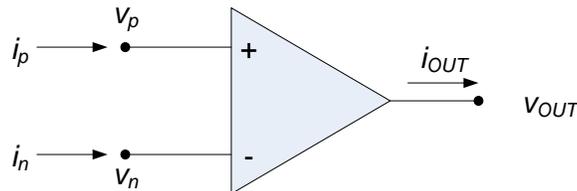


Figure 5.1. Operational amplifier symbol.

### After Completing this Chapter, You Should be Able to:

- State ideal operational amplifier modeling rules
- State constraints on the operational amplifier output voltage
- Represent operational amplifiers as dependent voltage sources
- Be able to identify standard operational amplifier pin connections
- Analyze electrical circuits containing ideal operational amplifiers and resistors
- Sketch op-amp based circuits which perform the following operations:
  - Inverting voltage amplification
  - Non-inverting voltage amplification
  - Summation (addition)
  - Differencing (subtraction)
  - Buffering
- Describe the operation of a comparator
- Briefly describe the effect of the following non-ideal op-amp parameters, relative to ideal op-amp performance:
  - Finite input resistance
  - Finite output resistance
  - Finite op-amp gain

## 5.1 Ideal Operational Amplifier Model

We will begin by summarizing the rules governing ideal operational amplifiers. In the following section, we will provide some background material relative to these rules and some additional criteria which the operational amplifier must satisfy. It should be emphasized that these rules govern ideal operational amplifiers; modeling of non-ideal operational amplifiers will most likely be presented in later electronics courses.

### Ideal Op-amp Modeling Rules

1. No current flows into the input terminals:  $i_n = i_p = 0$
2. The voltages at the input terminals are the same:  $v_n = v_p$  (when sufficient negative feedback is applied).

No requirements are placed on the output voltage and current. One may not conclude that  $i_{OUT} = 0$  simply because the input currents are zero. It may appear, from the input-output relations governing the op-amp, that

the op-amp violates Kirchhoff's current law – this is because we are not examining the details of the internal operation of the op-amp. Since the op-amp is an active device with its own power supply, it can provide an output current with no input current. Operational amplifiers, unlike passive devices, are capable of adding power to a signal. The presence of the external power supplies raises some additional constraints relative to op-amp operation; we address these issues next.

A more complete schematic symbol for an operational amplifier, including the op-amp's external power supplies, is shown in Fig. 5.2. Figure 5.2 shows two additional op-amp terminals. One is connected to a voltage source  $V^+$  and the other is connected to a voltage source  $V^-$ . These terminals are sometimes called the *positive* and *negative power supply terminals*. We must set the external voltage supplies so that the positive power supply voltage is greater than the negative power supply voltage:  $V^+ > V^-$ . In our discussions, it will be assumed that the power supply voltages are relative to the same reference voltage as all other voltages on the schematic.

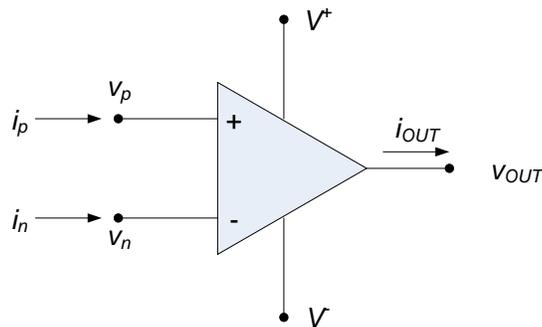


Figure 5.2. Operational amplifier schematic, including external power supplies.

The power supply voltages provide a constraint on the range of allowable output voltages, as provided below:

#### Output Voltage Constraint:

The output voltage is constrained to be between the positive and negative power supply voltages:  $V^- < v_{OUT} < V^+$

The above constraint is based on pure inequalities – in general, the output voltage range will be somewhat less than the range specified by  $V^-$  and  $V^+$ . The margin between the output and the supply voltages will vary depending on the specific op-amp. Any attempt to drive the output voltage beyond the range specified by the supply voltages will cause the output to *saturate* at the appropriate supply voltage. Similarly, it makes sense that the power supply voltages will constrain the range of allowable input voltages, as provided below:

#### Input Voltage Constraint:

- The input voltages,  $v_p$  and  $v_n$ , are constrained to be between the positive and negative power supply voltages:  $V^- < v_p, v_n < V^+$ .

The above constraint is based on pure inequalities – in general, the input voltage range will be somewhat less than the range specified by  $V^-$  and  $V^+$ . The margin between the inputs and the supply voltages will vary depending on the specific op-amp. Any attempt to drive the input voltages beyond the range specified by the supply voltages will cause the op-amp to no longer operate as we describe in this simple ideal model.

It is important to keep in mind, when analyzing operational amplifier circuits, that all of the terminal voltages shown in Fig. 5.2 should be taken as having the same reference voltage<sup>8</sup>. Figure 5.3 provides an explicit illustration of what is implied by this statement.

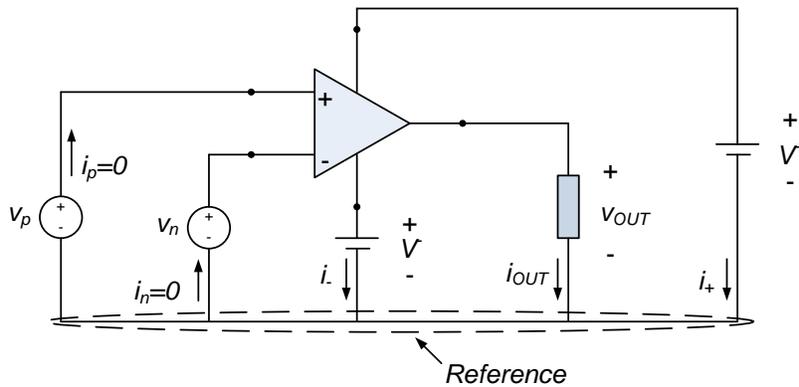


Figure 5.3. Op-amp voltages with reference node defined.

All voltages in Fig. 5.3, including the power supply voltages  $V_+$  and  $V_-$ , have the same reference. It is obvious from Fig. 5.3 that KCL at the reference node provides:

$$i_p + i_n = i_- + i_{OUT} + i_+ = 0$$

So that the positive and negative power supplies provide the current to the output. However, it is common to leave the power supply terminals off of the op-amp diagram (as in Fig. 5.1). If one interprets these types of diagrams literally, the figure corresponding to Fig. 5.3 will be as shown in Fig. 5.4.

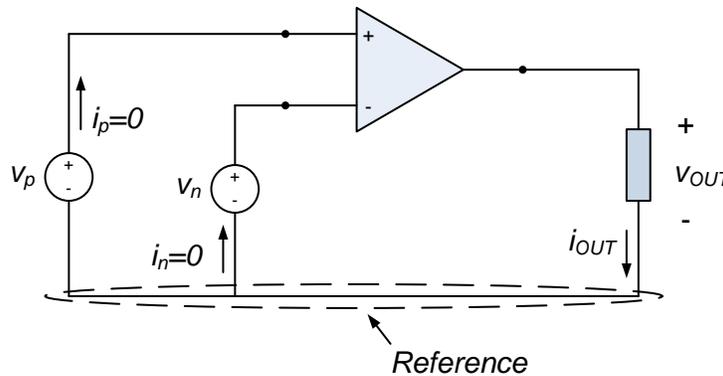


Figure 5.4. Op-amp voltages with reference node defined, but without supply voltages explicitly noted.

Figure 5.4. Op-amp voltages with reference node defined, but without supply voltages explicitly noted.

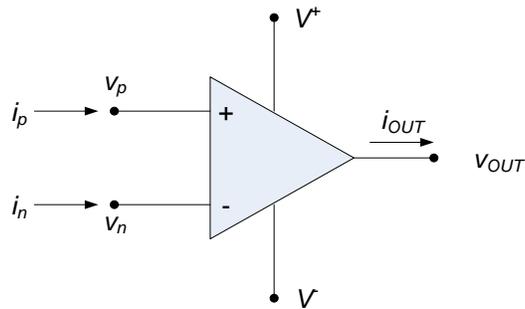
$$i_p + i_n = i_{OUT} = 0$$

And it is tempting to infer that the current out of the op-amp must be zero. This is not true; it is a misconception based upon an attempt to literally interpret a somewhat incomplete schematic.

## Section Summary

<sup>8</sup> This can be difficult at times, since circuit schematics containing operational amplifiers often do not emphasize this point. It is common to assume that the person reading the schematic understands op-amp operation.

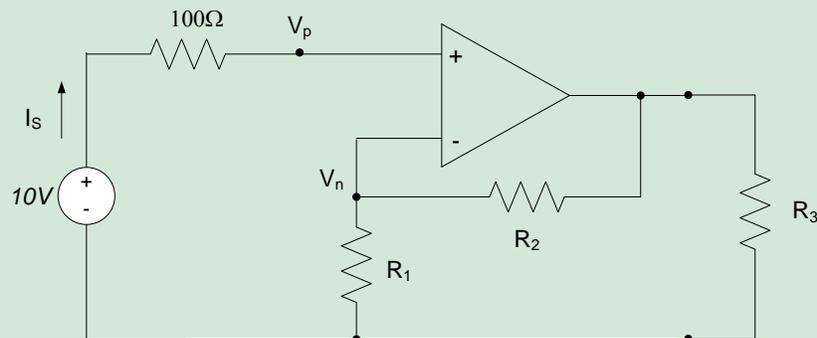
- The operational amplifier symbol is:



- The operation of ideal operational amplifiers follows the rules below:
  - No current flows into the input terminals:  $i_n = i_p = 0$
  - The voltages at the input terminals are the same:  $v_n = v_p$
  - The output voltage is constrained to be between the positive and negative power supply voltages:  $V^- < v_{OUT} < V^+$
  - Nothing is known about the current out of the op-amp,  $i_{OUT}$
- All voltages on the above diagram are relative to the same reference.

## 5.1 Exercises

- The op-amp in the circuit with negative feedback below is ideal. Find:
  - The current  $I_S$
  - The voltage  $V_p$
  - The voltage  $V_n$



## 5.2 Operational Amplifier Model Background

The rules provided in section 5.1 governing our ideal operational amplifier model can be applied directly to operational amplifier circuits, but some background information will allow more insight into the basis for these rules. We will still treat the operational amplifier as a single circuit element with some input-output relationship, but our more complete description will model the op-amp as a dependent source. Certain assumptions relative to this dependent source allow us to recover the op-amp rules presented in section 5.2, but our more complete model will allow us to later introduce some basic non-ideal operational amplifier effects.

An operational amplifier operates as a differential amplifier with a very high gain. That is, the output of the amplifier is the difference between the input voltages, multiplied by a large gain factor,  $K$ . Figure 5.5 shows the operation of the op-amp, from a systems-level standpoint:

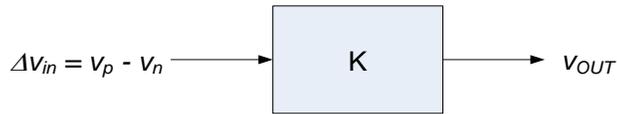


Figure 5.5. Block diagram of op-amp operation.

Thus, the input-output relation for an operational amplifier is:

$$V_{OUT} = K(v_p - v_n) = K \cdot \Delta v_{in} \quad \text{Eq. 5.1}$$

Where in  $\Delta v_{in}$  is the difference between the voltages at the input terminals and  $K$  is a very large number. (Values of  $K$  for typical commercially available operational amplifiers can be on the order of  $10^6$  or higher.) Since the output voltage is constrained to be less than the supply voltages,

$$V^- < K \cdot \Delta v_{in} < V^+$$

So

$$\frac{V^-}{K} < \Delta v_{in} < \frac{V^+}{K} \quad \text{Eq. 5.2}$$

If the voltage supplies are finite and  $K$  is very large, the difference in the input voltages must be very small. Thus,

$$\Delta v_{in} \approx 0$$

And  $\Delta v_p \approx v_n$ . This is of course only true when  $V^- < v_{OUT} < V^+$ .

The second operational amplifier modeling rule is a result of the high input resistance of operational amplifiers. We assume that any difference in the input terminal voltages is due to the operational amplifier's input resistance,  $R_{in}$ , times the current at the input terminals. This is illustrated conceptually in Fig. 5.6.

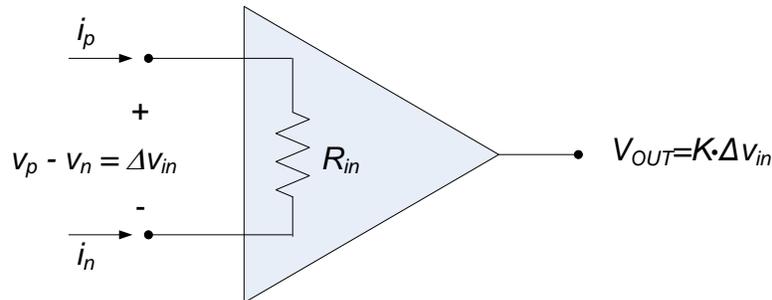


Figure 5.6. Operational amplifier symbol with "input resistance" indicated.

From Fig. 5.6, we see that the voltage difference between the input terminals can be considered to result from an input current passing through this input resistance.

$$v_p - v_n = R_{in} \cdot i_p$$

We will also assume that KCL applies across the input terminals of Fig. 5.6, so that:

$$i_p = -i_n$$

The above equations can be combined to give:

$$i_p = -i_n = \frac{v_p - v_n}{R_{in}} \tag{Eq. 5.3}$$

Since the input resistance of operational amplifiers is very large (commercial operational amplifiers have input resistances of several mega-ohms or higher) and the voltage difference across the input terminals is very small,

$$i_p = -i_n \approx 0$$

The above results suggest that an operational amplifier operates as a voltage-controlled-voltage source as shown in Fig. 5.7. Typically, commercially available operational amplifiers have very high gains,  $K$ , very high input resistances,  $R_{in}$ , and very low output resistances,  $R_{OUT}$ .

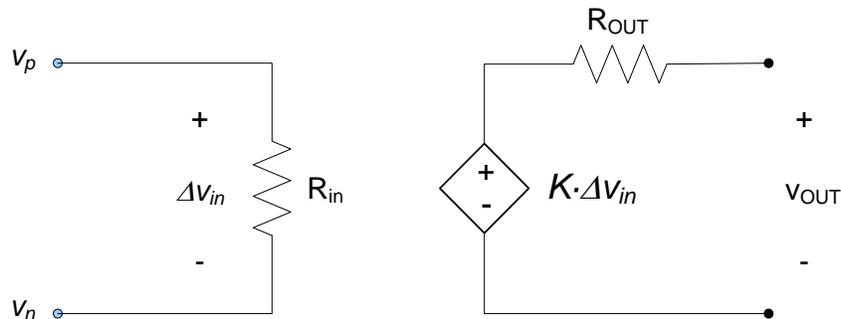


Figure 5.7. Equivalent circuit for operational amplifier model.

Combining the criteria provided by equations (5.1) and (5.2) results in the input-output relationship shown graphically in Fig. 5.8 below. The circuit operates linearly only when the output is between the supply voltages. When the output attempts to go outside this range, the circuit saturates and the output remains at the appropriate supply voltage. Notice that the negative supply voltage in Fig. 5.8 is indicated as a negative number; this is fairly typical, though not a requirement.

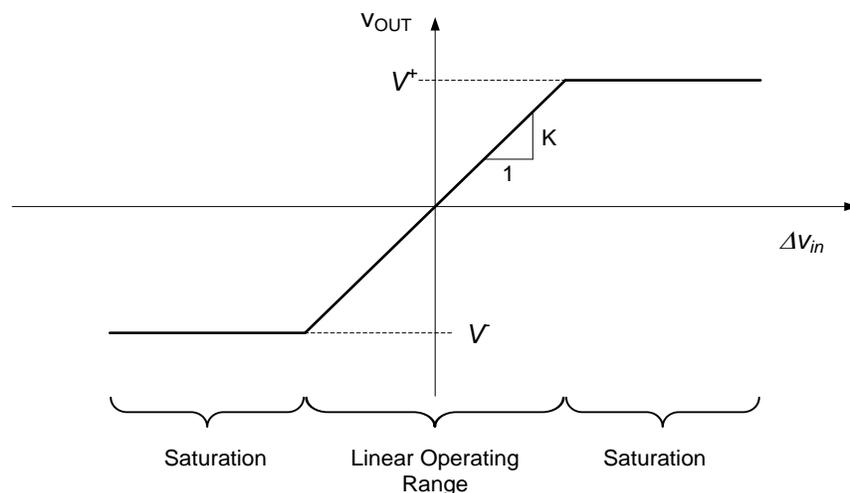


Figure 5.8. Op-amp input-output relationship.

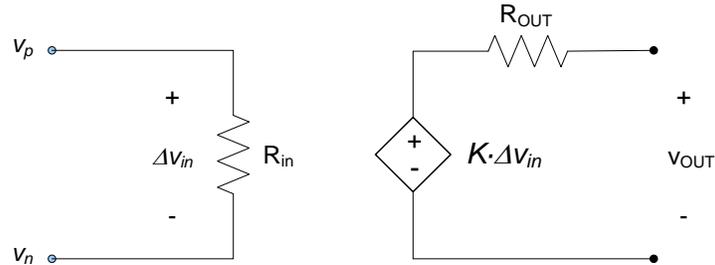
Our ideal operational amplifier model rules are based on the above, more general, operational amplifier relationships. The assumptions relative to ideal operational amplifier operation, along with their associated conclusions, are provided below:

- The output voltage is bounded by the power supply voltages:  $V^- < v_{OUT} < V^+$

- $K \rightarrow \infty$ . This, in conjunction with equation (5.2) implies that  $\Delta v_{in} = 0$  and  $v_p = v_n$ .
- $R_{in} \rightarrow \infty$ . This, in conjunction with equation (5.3) implies that  $i_p = -i_n = 0$ .
- $R_{OUT} = 0$ .

## Section Summary

- A circuit modeling the behavior of an operational amplifier is:



- For ideal operational amplifiers, the parameters in the circuit above are:
  - $K \rightarrow \infty$ . This implies that  $\Delta v_{in} = 0$  and  $v_p = v_n$ .
  - $R_{in} \rightarrow \infty$ . This implies that  $i_p = -i_n = 0$ .
  - $R_{OUT} = 0$ . This implies that the operational amplifier can provide infinite power as its output.
- In the operational amplifier model above, it is still assumed that  $V^- < v_{OUT} < V^+$ .

## 5.2 Exercises

1. An operational amplifier has a gain  $K = 10,000$ . The voltage supplies are  $V^+ = 20V$  and  $V^- = -10V$ . Determine the output voltage if the voltage difference between the input terminals ( $v_p - v_n$ ) is:
  - a. 1mV
  - b. 2mV
  - c. 4mV
  - d. -0.2mV
  - e. -2mV

## 5.3 Commercially Available Operational Amplifiers

Operational amplifiers are available commercially as integrated circuits (ICs). They are generally implemented as *dual in-line packages (DIPs)*, so called because the terminals (pins) on the package are in pairs and line-up with one another. A typical DIP is shown in Fig. 5.9. The pins on DIPs are numbered; in order to correctly connect the DIP, pin 1 must be correctly oriented. Pin 1 is commonly located by looking for a notch at one end of the IC – pin 1 will be to the immediate left of this notch, if you are looking at the IC from the top. Alternate methods of indicating pin 1 are also used: sometimes the corner of the IC nearest pin 1 is shaved off or a small indentation or dot is located at the corner of the IC nearest pin 1.

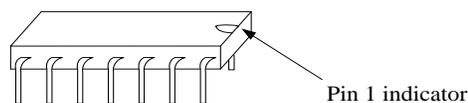


Figure 5.9. Dual in-line transistor package.

One common op-amp device is the 741 op-amp. The 741 is an eight-lead DIP; a top view of the package, with the leads labeled, is shown in Fig. 5.10. Key features of the package are as follows:

- Orientation of the pins is determined by the location of a semicircular notch on the package, as shown in Fig. 5.10. (Recall that Fig. 5.10 is a top view of the device.) Alternately, some packages place a circular indentation near pin 1 in order to provide the orientation of the pins.
- Inverting and non-inverting inputs are pins 2 and 3, respectively in Fig. 5.10
- The output terminal is pin 6 on the package.
- The positive and negative power supplies are labeled as  $V_{CC+}$  and  $V_{CC-}$  in Fig. 5.10. They are pins 7 and 4, respectively.  $V_{CC+}$  should be less than +15 volts and  $V_{CC-}$  should be more than -15 volts. A larger range of power supply voltages may destroy the device.
- The pins labeled OFFSET NULL 1, OFFSET NULL 2, and NC (pins 1, 5, and 8) will not be used for this class. The offset null pins are used to improve the op-amp’s performance. The NC pin is never used. (NC stands for “not connected”).

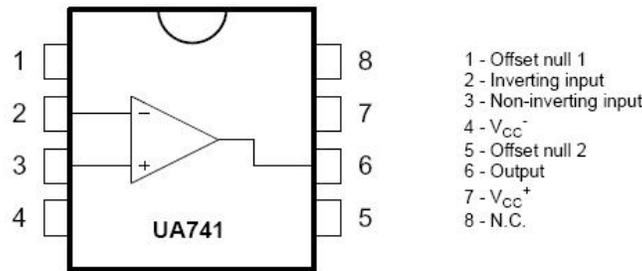


Figure 5.10. 741-type operational amplifier pin connections.

Most commercially available operational amplifiers will conform to a relatively standard pin connection layout. However, there will tend to be variations to one extent or another. For example, pin connections for an OP27 operational amplifier are shown in Fig. 5.11(a). Some operational amplifier chips will also contain more than one operational amplifier on the chip. For example, Fig. 5.11(b) provides pin connections for an OP282 package, in which two operational amplifiers are included. (A chip with two operational amplifiers is commonly called a “dual package”. Chips with four operational amplifiers are also common; they are often called “quad packages”). In Fig. 5.11,  $V+$  and  $V-$  are the positive and negative power supplies, respectively. (Both operational amplifiers on the OP282 chip share the same power supplies.)  $+IN$  and  $-IN$  are the positive and negative input terminals, and  $OUT$  is the output terminal. In the OP27 amplifier, the  $V_{OS}$  TRIM terminals perform the same purpose as the Offset Null pins on the 741-type operational amplifier. The OP282 chip contains two amplifiers, “A” and “B”. In Fig. 5.11(b), the inputs and outputs for the two amplifiers are identified as being associated with the “A” or “B” amplifier by appending the appropriate letter.



Figure 5.11. Additional operational amplifier pin connection examples.

Note: Always check the manufacturer’s data sheet for the specific operational amplifier you are using. This can eliminate irritating and time-consuming errors when wiring your circuits!

### 5.3 Exercises:

1. Go to the Analog Devices website, <http://www.analog.com>, and look up the pin connections for the OP482 operational amplifier package. Sketch the package and label the pin connections. Briefly describe your interpretation of the various pin connections.

## 5.4 Analysis of Op-amp Circuits

Operational amplifiers can be used as either linear or nonlinear circuit elements. When used as nonlinear circuit elements, the op-amp is deliberately operated so that the output voltage from the op-amp is driven to the power supply voltages. In this mode of operation, the output of the op-amp is said to be saturated and does not necessarily change as the input to the system changes. When the operational amplifier is used as a linear circuit element, the output is maintained within the range of the power supply voltages and the output voltage is a linear function of some input voltage or voltages.

In this chapter, we will be concerned with the use of operational amplifiers as linear circuit elements. When used in this mode, the overall circuit is generally constructed to provide *negative feedback* around the operational amplifier itself. When operated in a negative feedback mode, the output of the operational amplifier is connected to the inverting input terminal, generally through some other circuit elements. Negative feedback tends to make the overall circuit less sensitive to the specific value of the op-amp gain and reduces the likelihood of saturation at the op-amp output. We will not be concerned here with the details of why this is true, beyond noting that these devices will generally not operate linearly without feedback. (Later electronics courses will discuss why this is true.)

Nodal analysis is often the most efficient way to approach the analysis of an operational amplifier-based circuit. When applying nodal analysis to a circuit containing an ideal operational amplifier, the first step should be to apply the basic op-amp rules to the overall circuit. These were presented in the previous chapter, and are repeated here for convenience:

1. The voltages at the input terminals of the operational amplifier are the same.
2. The currents into the input terminals of the operational amplifier are zero.

It should be emphasized that application of rule 1 above does not imply that both of the op-amp input terminals can be treated as being part of the same node. The op-amp input terminals should be treated as being two separate nodes, with the same voltage potential. After applying the basic op-amp rules, it is generally appropriate to apply Kirchoff's current law at the input terminals of the operational amplifier. Additional nodes in the circuit may necessitate application of KCL at other points, but the above approach is generally an extremely good starting point.

**Important Tip:** Applying KCL at the output node of an operational amplifier is often not productive. Since no information is available about the current out of an operational amplifier (due to the active nature of the device, as noted in the previous section) application of KCL at the output node generally provides an additional equation, at the expense of introducing an additional unknown. Application of KCL at an op-amp output node is generally only productive if one must determine the current output of the op-amp.

When analyzing an operational amplifier as a linear circuit element, the external power supply voltages will generally be ignored. We will assume that the output voltage is within the voltage range specified by the external power supplies. If the output voltage is not within this range, the circuit will not behave linearly, and our analysis will be invalid. The final step of any analysis of an operational amplifier circuit is to determine whether the output voltage is within the external power supply voltage range; meeting this constraint often results on a constraint on the input voltages applied to the circuit.

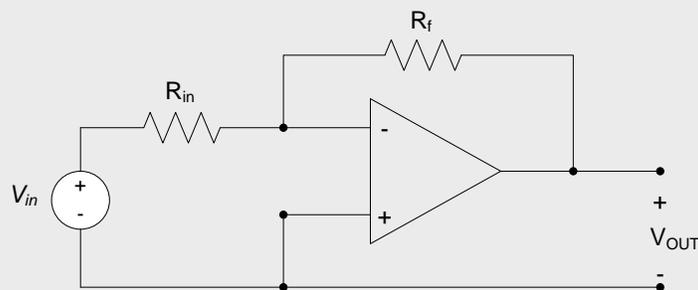
### Suggested Analysis Approach:

1. Apply ideal operational amplifier rules to circuit. (Voltage potentials at op-amp input terminals are the same; no current enters the op-amp input terminals.)
2. Apply KCL at op-amp input terminals.
3. Apply KCL at other circuit nodes, if necessary.
4. Check to ensure that output voltage remains within range specified by op-amp power supply voltages.

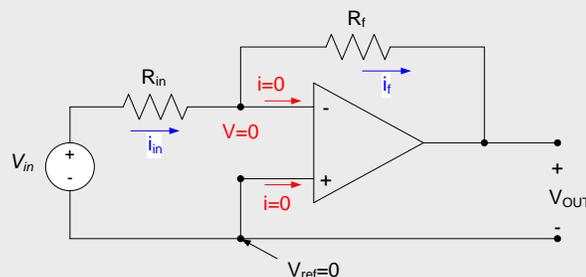
We illustrate the above analysis approach with several examples. The example circuits provided below illustrate the use of operational amplifier circuits to perform the mathematical operations of scaling (multiplication by a constant), addition, and subtraction. We also provide an example circuit which performs a buffering operation – this circuit can be useful for isolating different parts of a circuit from one another.

### Example 5.1

Determine  $V_{OUT}$  as a function of  $V_{IN}$  for the circuit shown below.



Choosing the non-inverting terminal voltage as our reference voltage and applying the ideal operational amplifier rules allows us to label the voltages and currents shown in red below.



Applying KCL at the non-inverting input terminal provides no information (we know the current and voltage at the non-inverting input). Applying KCL at the inverting input terminal results in:

$$i_{in} = i_f$$

Using Ohm's law to write these currents in terms of node voltages and taking advantage of the fact that the voltage at the inverting terminal of the op-amp is zero (because there is no voltage difference across the input terminals of the op-amp and we have chosen the non-inverting terminal voltage as our reference) results in:

$$\frac{V_{in} - 0}{R_{in}} = \frac{0 - V_{OUT}}{R_f}$$

Solving for  $V_{OUT}$  results in:

$$V_{OUT} = -\left(\frac{R_f}{R_{in}}\right)V_{in}$$

**Comments:**

- This circuit is called an *inverting voltage amplifier*. The output voltage is a scaled version of the input voltage, hence the term “voltage amplifier”. The change in sign between the output and input voltage makes the amplifier “invert”.
- The output voltage must be between the op-amp power supply voltages. Depending on the values of  $R_f$  and  $R_{in}$ , this sets limits on the magnitude of the input voltage to avoid saturation.

It is worthwhile at this point to make a few comments relative to some concepts presented sections 5.1 and 5.2, in the context of the op-amp circuit of Example 5.1.

- The input-output relationship governing the circuit of Example 5.1 can be represented conceptually as a dependent source-based circuit. The input-output relationship for the circuit, as determined in Example 5.1, is:

$$V_{OUT} = -\left(\frac{R_f}{R_{in}}\right)V_{in}$$

While the current provided by the source to the circuit is:

$$i_{in} = \frac{V_{in} - 0}{R_{in}}$$

These two relationships are satisfied by the voltage controlled voltage source (VCVS) shown in Fig. 5.12 below. The input resistance of this circuit is the resistance  $R_{in}$ ; the input resistance governs the relationship between the voltage applied by the source and the source current necessary to maintain that voltage. Thus, increasing  $R_{in}$  reduces the power which the source must provide to maintain the output voltage  $V_{OUT}$ .

**Note:** The input resistance of the circuit of Example 5.1 is not the same as the input resistance of the operational amplifier itself, as shown in Figs. 5.6 and 5.7.

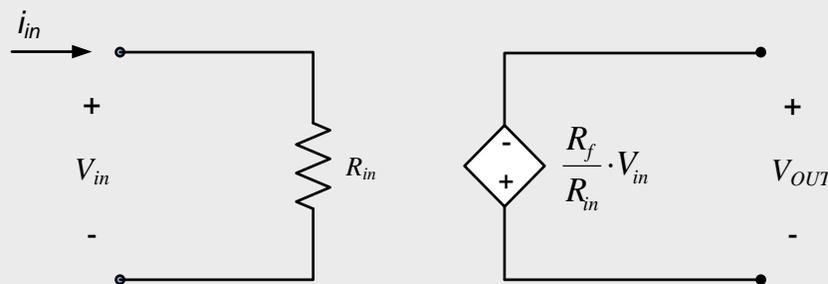


Figure 5.12. VCVS to model circuit of Example 5.1.

- The representation given in Fig. 5.12 of the circuit of Example 5.1 is not unique. For example, the current controlled voltage source (CCVS) model shown in Fig. 5.13 is also a valid model for the circuit.

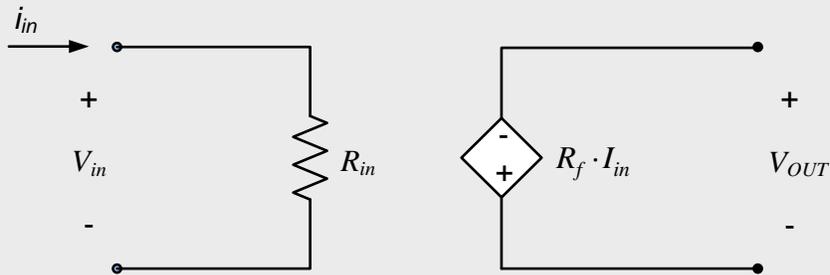


Figure 5.13. CCVS model for circuit of Example 5.1.

- The circuit of Example 5.1 has an open-circuit at the op-amp output terminal. Thus, no current is provided to the output. However, this does not imply that the current at the output terminal of the op-amp is zero! KCL at the op-amp output terminal indicates that the current through the feedback resistor goes into the op-amp output terminal, where (from our diagram) it seems to disappear! Recall, however, from our discussion in section 5.1 (Fig. 5.3 in particular) that this current will pass through the supply voltages and then to the reference node. The supply voltages and their associated path to the reference node are not shown on the circuit diagram in Example 5.1, but they do exist.
- If we apply a load resistor  $R_L$  to the output terminal of the operational amplifier in Exercise 5.1, we obtain the circuit shown in Fig. 5.14. The analysis of the circuit proceeds exactly as in Exercise 5.1, and we again obtain:

$$V_{OUT} = -\left(\frac{R_f}{R_{in}}\right)V_{in}$$

This result does not depend on the current through the load,  $i_L$ ! This is because we made no assumptions relative to the current out of the operational amplifier output; the operational amplifier adjusts its output current as necessary to provide the current required to maintain the output voltage as in the above expression.

**Note:** Our ideal operational amplifier will draw power from the supply voltages to provide whatever output current is necessary to satisfy the rules governing operational amplifier provided in section 5.1. Real operational amplifiers, of course, have current limitations. We will discuss these limitations in section 5.6.

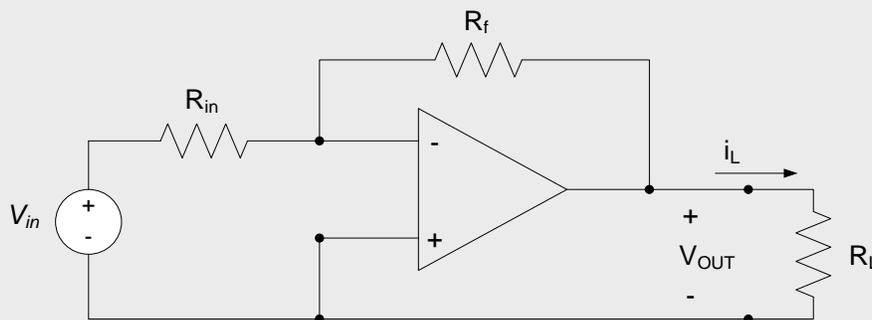
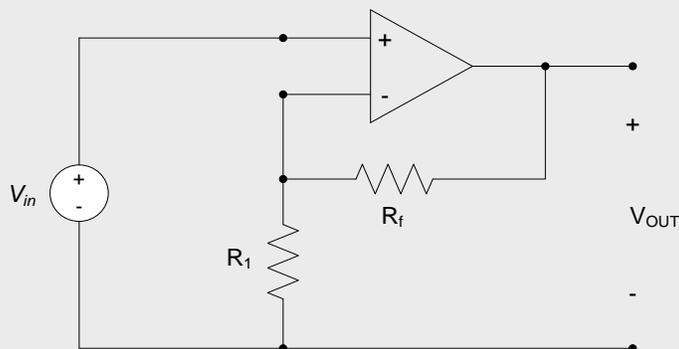


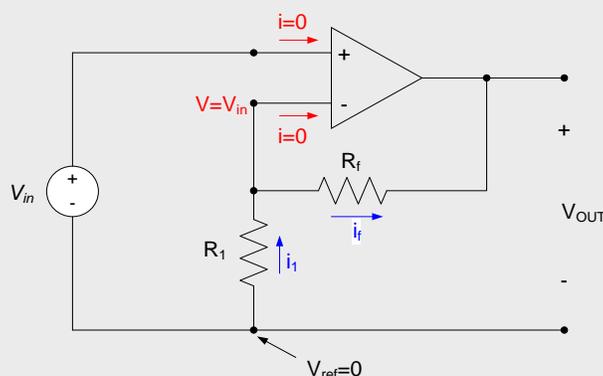
Figure 5.14. Inverting voltage amplifier with load resistor.

### Example 5.2

Determine  $V_{OUT}$  as a function of  $V_{IN}$  for the circuit shown below.



Choosing our reference voltage at the negative terminal of both  $V_{in}$  and  $V_{OUT}$  and applying the ideal operational amplifier rules allows us to label the voltages and currents shown in red below. (Note that since the input voltage sets the voltage of the non-inverting op-amp terminal, it also indirectly sets the voltage at the inverting terminal of the op-amp.)



Applying KCL at the inverting terminal of the op-amp results in  $i_1 = i_f$ . Using Ohm's law to write these in terms of voltages provides:

$$\frac{0 - V_{in}}{R_1} = \frac{V_{in} - V_{OUT}}{R_f}$$

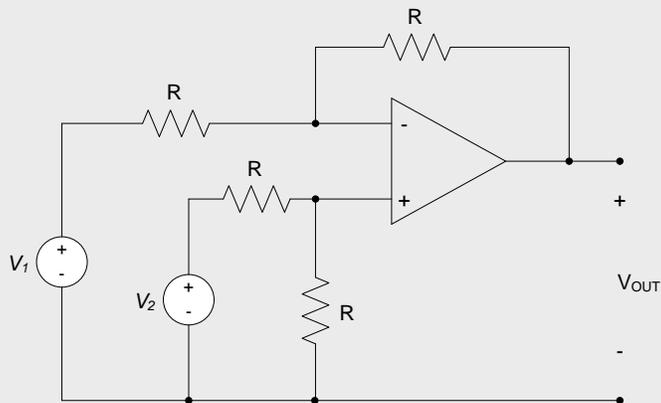
Solving this for  $V_{OUT}$  gives  $V_{OUT} = \left(1 + \frac{R_f}{R_1}\right) V_{in}$

**Comments:**

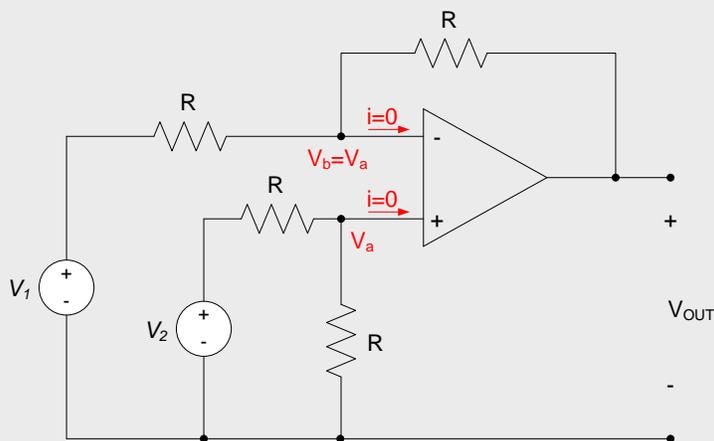
- The output voltage can be expressed as a gain (a multiplicative factor) times the input voltage; the circuit is a “voltage amplifier”. Since there is no sign change between the input and output voltage, the circuit is a *non-inverting voltage amplifier*.
- The output voltage must be between the op-amp power supply voltages. Depending on the values of  $R_f$  and  $R_1$ , this sets limits on the magnitude of the input voltage to avoid saturation.

**Example 5.3**

Determine  $V_{OUT}$  as a function of  $V_1$  and  $V_2$  for the circuit shown below.



Denoting the non-inverting terminal of the op-amp as node a and the inverting terminal as node b, and applying the ideal op-amp rules results in the figure below:



The voltage  $V_a$  can be determined from a voltage divider relation (or by applying KCL at node a) as  $V_a = \frac{V_2}{2}$ .

Thus, the voltage at the inverting terminal is  $V_b = V_a = \frac{V_2}{2}$ . Applying KCL at node b results in:

$$\frac{V_1 - V_b}{R} = \frac{V_b - V_{OUT}}{R} \Rightarrow \frac{V_1 - \frac{V_2}{2}}{R} = \frac{\frac{V_2}{2} - V_{OUT}}{R}$$

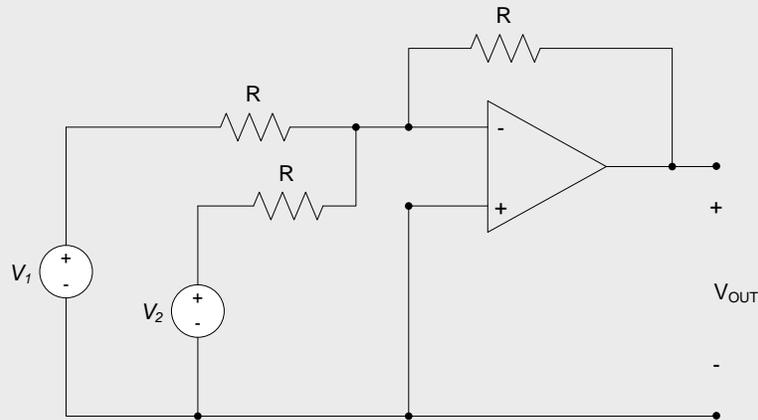
Simplification of the above results in  $V_{OUT} = V_2 - V_1$ .

**Comments:**

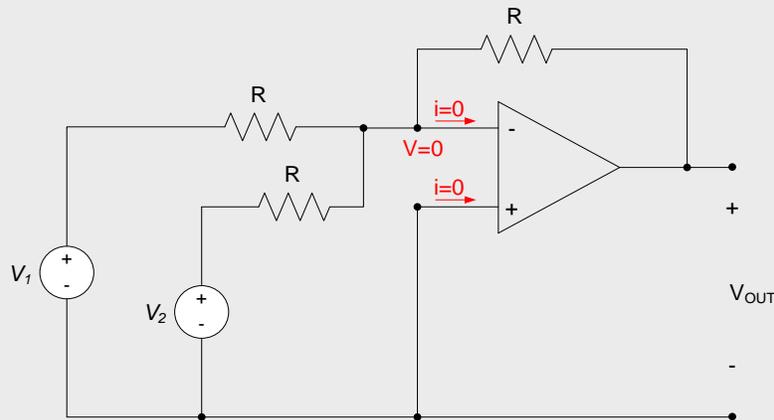
- The above circuit performs a subtraction operation. The voltage  $V_1$  is subtracted from the voltage  $V_2$ .
- The inverting and non-inverting terminals of the op-amp are treated as separate nodes in this analysis, even though the op-amp constrains the voltages at these nodes to be the same. Thus, we apply KCL at each input terminal of the op-amp.

**Example 5.4**

Determine  $V_{OUT}$  as a function of  $V_1$  and  $V_2$  for the circuit shown below.



Choosing the non-inverting terminal voltage as our reference voltage and applying the ideal operational amplifier rules allows us to label the voltages and currents shown in red below:



Applying KCL at the inverting terminal of the op-amp results in:

$$\frac{V_1 - 0}{R} + \frac{V_2 - 0}{R} = \frac{0 - V_{OUT}}{R}$$

Or

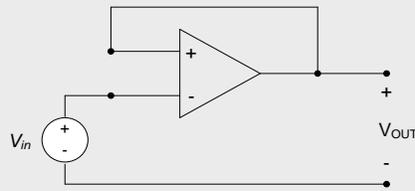
$$V_{OUT} = -(V_1 + V_2)$$

#### Comments:

- The circuit inverts the sum of the inputs. One can use an inverting amplifier with a gain of one in conjunction with the above circuit to obtain a non-inverted sum of the inputs.
- An arbitrary number of inputs can be summed, by simply increasing the number of input signals and resistors applied at the inverting terminal of the op-amp, which is often referred to as the “summing node”.

#### Example 5.5

Determine  $V_{OUT}$  as a function of  $V_{IN}$  for the circuit shown below.

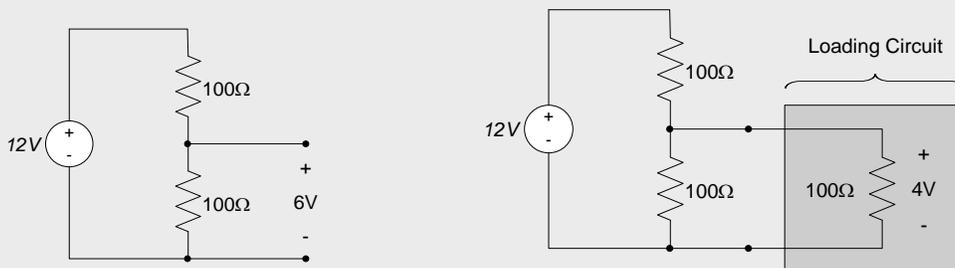


Since there is no circuit element in the feedback loop, the inverting terminal voltage is identical to the output voltage,  $V_{OUT}$ . The ideal op-amp rules require that the inverting and non-inverting terminal voltages are the same, so:

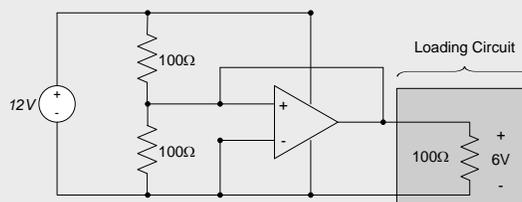
$$V_{OUT} = V_{in}$$

The circuit is called a *voltage follower*, since the output voltage simply “follows” the input voltage. This circuit, though it appears to do nothing, is actually extremely useful. Since the input voltage is applied directly to an op-amp input terminal, the input resistance to the circuit is infinite and no current is drawn from the source. Thus, the source provides no power in order to generate the output voltage - all power provided to the load comes from the op-amp power supplies. This can be extremely useful in isolating different portions of a circuit from one another.

Consider, as an example, the following case. We have a loading circuit with an equivalent resistance of  $100\Omega$ . We wish to apply 6V to the circuit, but only have access to a 12V source. It is decided that we will use a voltage divider containing two  $100\Omega$  resistors in series to reduce the supply voltage to the desired 6V level as shown in the circuit to the left below. However, adding the loading circuit to the voltage divider changes the voltage provided to the load, as shown to the right below.



Addition of a voltage follower to the circuit isolates the voltage divider from the load, as shown below. Power to the op-amp can be provided by connecting the 12V source to  $V+$  and grounding  $V-$ , as shown, since the desired op-amp output is between 0V and 12V.



## Section Summary

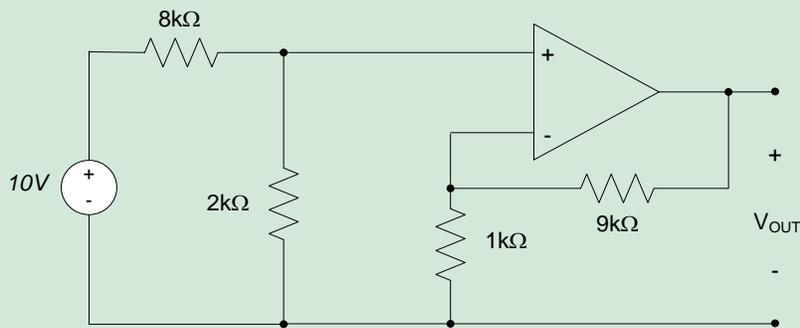
- Analysis of linear operational amplifier circuits typically consists of the following components:
  - Assume that the voltage difference across the input terminals is zero.
  - Assume that the currents into the input terminals is zero.
  - Apply KCL at op-amp input terminals.
  - Apply KCL at other circuit nodes, if necessary.

- Check to ensure that output voltage remains within range specified by op-amp power supply voltages.
- Op-amp circuits which perform the following functions are presented in this section:
  - Inverting voltage amplification
  - Non-inverting voltage amplification
  - Summation (addition)
  - Differencing (subtraction)
  - Buffering

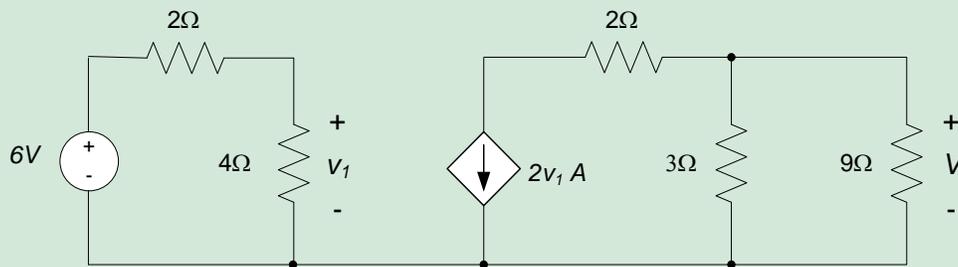
The reader should be able to sketch circuits, which perform the functions above.

### 5.4 Exercises

1. Represent the circuit of Example 5.2 as a voltage controlled voltage source.
2. Represent the circuit of Example 5.2 as a voltage controlled current source.
3. Find  $V_{out}$  for the circuit below.



4. Find  $V$  in the circuit below.



### 5.5 Comparators

Operational amplifiers are intended to be incorporated into circuits which *feeds back* the op-amp output to one or both of the input terminals. That is, the output voltage is connected in some way to the op-amp inputs. Typically, for stable operation, the output is fed back to the inverting input terminal for stable operation (as in all of our circuit examples in section 5.4). If the output is not fed back to the input of the op-amp, the op-amp may not function as expected.

Comparators are operational amplifier –like devices which are intended to be operated without feedback from the output to the input. The circuit symbol for a comparator looks like an op-amp symbol, reflecting their similarities. Figure 5.15 provides a typical comparator symbol, with applicable voltages labeled.

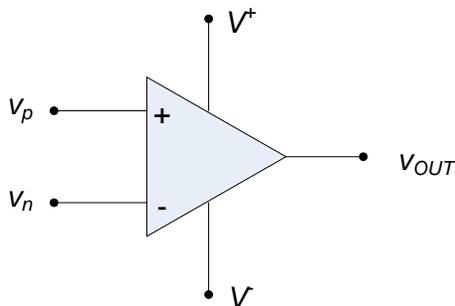


Figure 5.15. Comparator circuit symbol.

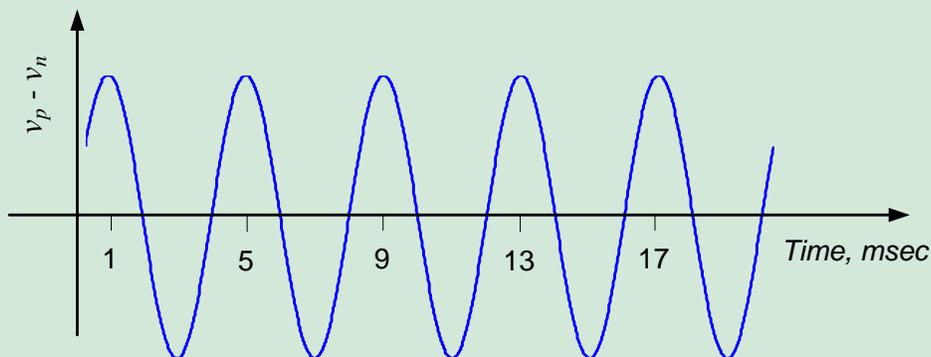
Operation of the comparator is simple: if  $v_p$  is greater than  $v_n$ , the output goes to the high supply voltage,  $V^+$ . If  $v_p$  is less than  $v_n$ , the output goes to the low supply voltage,  $V^-$ . The comparator is essentially checking the sign between the voltage at the inverting and non-inverting inputs, and adjusting the output voltage accordingly.

Mathematically, the operation of a comparator can be expressed as:

$$V_{OUT} = \begin{cases} V^+, & v_p - v_n > 0 \\ V^-, & v_p - v_n < 0 \end{cases}$$

### 5.5 Exercises

1. A comparator like that shown in Fig. 5.14 has the sinusoidal signal below applied across the input terminals. (E.g. the plot below is  $v_p - v_n$  vs. time.) Sketch the output voltage  $v_{out}(t)$ .



### 5.6 A Few Non-ideal Effects

In section 5.2, we indicated that operational amplifiers are designed to have high input resistances, low output resistances and high gains between the input voltage difference and the output voltage. Figure 5.7 of section 5.2 provided a model of an operational amplifier as a dependent source, including input and output resistances. This model is repeated below as Fig. 5.16 for convenience.

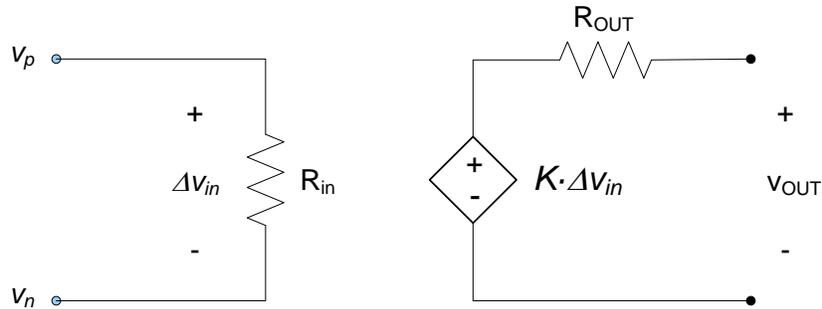


Figure 5.16. Operational Amplifier model.

In section 5.2, we also provided the assumptions applicable to ideal operational amplifier operation, along with their associated conclusions, and are provided below:

- The output voltage is bounded by the power supply voltages:  $V^- < V_{OUT} < V^+$
- $K \rightarrow \infty$ . Thus,  $\Delta v_{in} = 0$  and  $v_p = v_n$ .
- $R_{in} \rightarrow \infty$ . Thus,  $i_p = -i_n = 0$ , and the operational amplifier draws no power at its input.
- $R_{OUT} = 0$ . Thus, there is no limit on the output current (or power) which can be provided by the op-amp.

Practical operational amplifiers have finite gains ( $K$  for most amplifiers is in the range  $10^5 - 10^7$ ), finite input resistances (typical values are on the order of a few mega-ohms to hundreds or thousands of mega-ohms) and non-zero output resistances (generally on the order of 10 to 100 ohms). In this section, we will very briefly discuss a few of the ramifications of these non-ideal parameters.

### 5.6.1 Input Resistance Effects

The high input resistance of the operational amplifier means that circuits connected to the op-amp input do not have to provide much power to the op-amp circuit. This is the op-amp property that is employed in buffer amplifiers and instrumentation amplifiers. Instrumentation systems, for example, have very limited power output capabilities; these limitations are typically modeled as high output resistances in the instrumentation systems. Thermocouples, for example, provide low voltage levels, and very small power output – they can be modeled as a voltage source with a fairly high output resistance. When a system of this type is connected to the input terminals of an op-amp, the situation is as shown in Fig. 5.17.

It is apparent from Fig. 5.17 that the output resistance of the system, if it is large enough, can have an effect on the voltage difference across the op-amp input terminals, since:

$$\Delta v_{in} = V_s \left( \frac{R_{in}}{R_{in} + R_s} \right)$$

Since we generally want to amplify  $V_s$  directly, any difference between  $V_s$  and  $\Delta v_{in}$  will degrade our output voltage from its desired value.

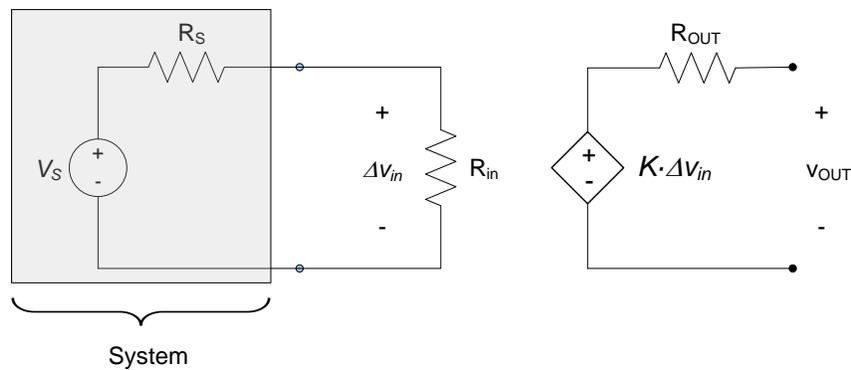


Figure 5.17. Effect of input resistance on output.

### 5.6.2 Output Resistance Effects

The op-amp output resistance essentially limits the amount of power the op-amp can provide at its output terminal. This can become a problem if we want to connect very low resistance loads to the output of an operational amplifier. For example, audio speakers commonly have an 8Ω resistance. Figure 5.18 shows an 8Ω speaker connected to the output of an operational amplifier which has an 80Ω output resistance. In this case, we expect the maximum output voltage to be:

$$v_{OUT} = K \cdot V_S \left( \frac{8\Omega}{8\Omega + 80\Omega} \right) = 0.09KV_S$$

If the maximum output voltage of the op-amp is low, we may not have nearly enough power to operate the speaker.

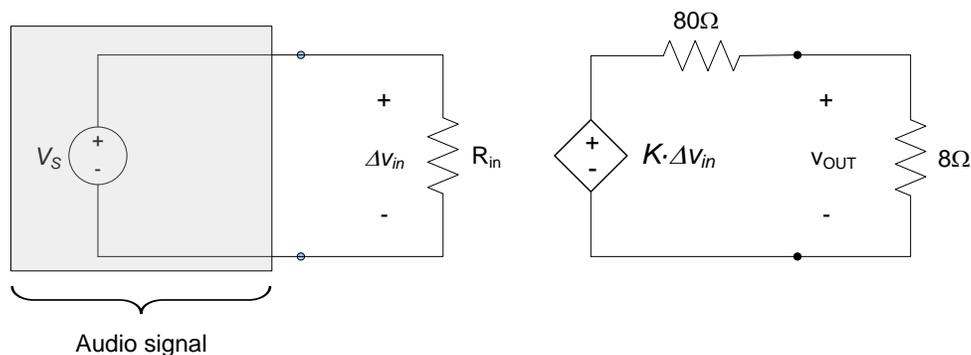


Figure 5.18. Audio amplifier.

### 5.6.3 Finite Gain Effects

As an example of the effects of a finite voltage gain, let us assume that an operational amplifier has a gain of  $K = 10,000$  and supply voltages  $V^+ = 10V$  and  $V^- = -10V$ . From equation (5.2), the linear operating range of the operational amplifier is over the range of input voltage differences:

$$\frac{V^-}{K} \leq \Delta \leq \frac{V^+}{K}$$

The non-ideal operational amplifier of interest can then allow input terminal voltage differences of up to  $-1mV \leq \Delta v_{in} \leq 1mV$ . Although voltage differences of a millivolt will be considered to be essentially zero for any of the voltage levels we will deal with in this class, these voltages are definitely not zero for some applications.

## Section Summary

- The effect of a finite input resistance on an operational amplifier's operation is that the current into the input terminals will not be identically zero. Thus, a real operational amplifier with finite input resistance will always draw some power from a circuit connected to it. Whether this has a significant effect on the overall circuit's operation is generally a function of the output resistance of the circuit to which the amplifier is connected.
- The effect of a non-zero output resistance on an operational amplifier's operation is that the power output of the amplifier is limited. Thus, a realistic operational amplifier will not be able to provide any arbitrary current to a load. Whether this has a significant effect on the overall circuit's operation is primarily dependent upon the value of the load resistance.
- The effect of a finite op-amp gain is that the voltage difference across the input terminals may not be identically zero.

## Real Analog Chapter 5: Lab Projects

### 5.4.1: Inverting Voltage Amplifier

In this assignment, we implement a simple operational amplifier-based circuit. Since operational amplifiers are used commonly in circuits used to implement mathematical operations, we implement the processes of multiplication by a negative constant.

#### Before beginning this lab, you should be able to:

- Analyze operational amplifier-based circuits

#### After completing this lab, you should be able to:

- Design and build an operational amplifier-based inverting voltage amplifier

#### This lab exercise requires:

- Analog Discovery 2 module
- Digilent Analog Parts Kit
- Digital multimeter (optional)

#### Symbol Key:

- DEMO** Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
- ANALYSIS** Analysis; include principle results of analysis in laboratory report.
- SIM** Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
- DATA** Record data in your lab notebook.

#### General Discussion:

The circuit shown in Fig. 1 is called an inverting amplifier. Appropriate pin numbers for the OP27 operational amplifier are provided on Fig. 1.  $v_{in}$  is the applied (input) voltage to the circuit.  $v_{out}$  is the output voltage from the circuit. The relationship between  $v_{in}$  and  $v_{out}$  for this circuit is:

$$v_{out} = -\frac{R_2}{R_1} v_{in}$$

Thus, the output voltage is an *inverted* (due to the sign change) and *amplified* or *scaled* (due to the multiplicative factor  $\frac{R_2}{R_1}$ ) version of the input voltage. The scaling factor  $\frac{R_2}{R_1}$  is sometimes called the *gain* of the amplifier. The

ground symbol, , is used to denote the reference voltage from which all other voltages are measured. Note that if  $R_1$  and  $R_2$  are the same, the output voltage is simply the negative of the input voltage.

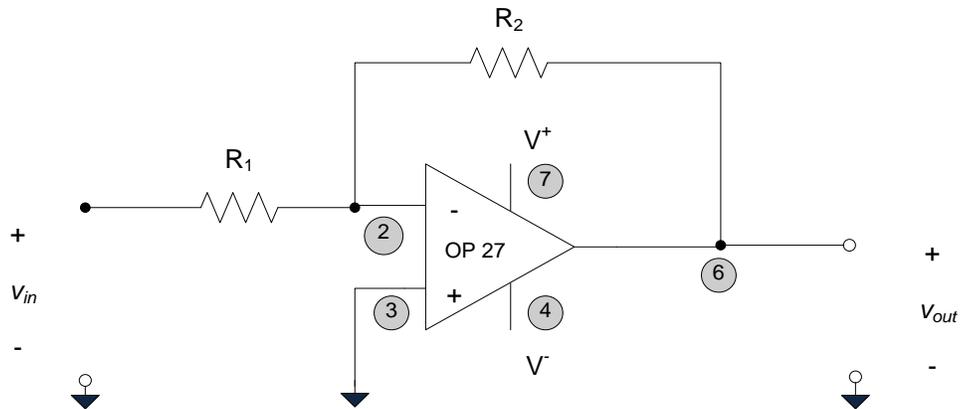


Figure 1. Inverting amplifier circuit.

### Pre-lab:

#### ANALYSIS

Design an inverting amplifier which provides a gain of approximately 2 and an input resistance,  $R_1$ , of approximately  $2\text{k}\Omega$ . (The input resistance is defined as the input voltage divided by the input current. Since pin 2 provides a “virtual” ground, the input resistance is simply  $R_1$ .)

### Lab Procedures:

#### DATA

1. Implement the amplifier design you generated in the pre-lab. Create a schematic of the circuit in your lab notebook, record actual resistance values, and label supply voltages on your schematic. Recommended connections are as follows:
  - Use  $V_+$  as positive supply rail to the op-amp and  $V_-$  as the negative supply rail to the op-amp.
  - Use one of the waveform generator channels on your Analog Discovery 2 to provide the input voltage  $v_{in}$  to your circuit.
  - Measure both the input voltage,  $v_{in}$  and the output voltage,  $v_{out}$ , using your DMM and/or the scope channels on your Analog Discovery 2.

#### DATA

2. Test your design with input voltages of approximately  $-3\text{V}$  to  $+4\text{V}$  by step sizes of  $.5\text{V}$ . Tabulate your results ( $v_{in}$  and  $v_{out}$ ) in your lab notebook. Also in your lab notebook, create a plot of  $v_{in}$  vs.  $v_{out}$  and comment on your results (make sure that you calculate a circuit gain – the rate of change of output to input – and compare it to your expectations based on your pre-lab). Note in your lab notebook the range of output voltages over which the circuit response is linear.

#### DEMO

3. Demonstrate operation of your circuit to the Teaching Assistant. Have the TA initial the appropriate page(s) of your lab notebook and the lab checklist.



## Real Analog Chapter 5: Lab Projects

### 5.4.2: Summing Amplifier

In this assignment, we implement a simple operational amplifier-based circuit. Since operational amplifiers are used commonly in circuits used to implement mathematical operations, we implement the processes of summing two voltages.

**Before beginning this lab, you should be able to:**

- Analyze operational amplifier-based circuits

**After completing this lab, you should be able to:**

- Design and build an operational amplifier-based inverting voltage amplifier

**This lab exercise requires:**

- Analog Discovery 2 module
- Digilent Analog Parts Kit
- Digital multimeter (optional)

#### Symbol Key:

- DEMO** Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
- ANALYSIS** Analysis; include principle results of analysis in laboratory report.
- SIM** Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
- DATA** Record data in your lab notebook.

#### General Discussion:

The circuit shown in Fig. 1 is a summing amplifier circuit. Appropriate pin numbers for the OP27 operational amplifier are provided on Fig. 1. The output voltage,  $v_{out}$ , is an inverted and scaled version of the sum of the input voltages,  $v_a$  and  $v_b$ . If  $R_1 = R_2$ , the input voltages are not individually scaled and the output voltage is:

$$v_{out} = -\frac{R_3}{R_1}(v_a + v_b)$$

Note that if, in addition,  $R_3 = R_1$ , the output voltage is simply the sum of the two input voltages.

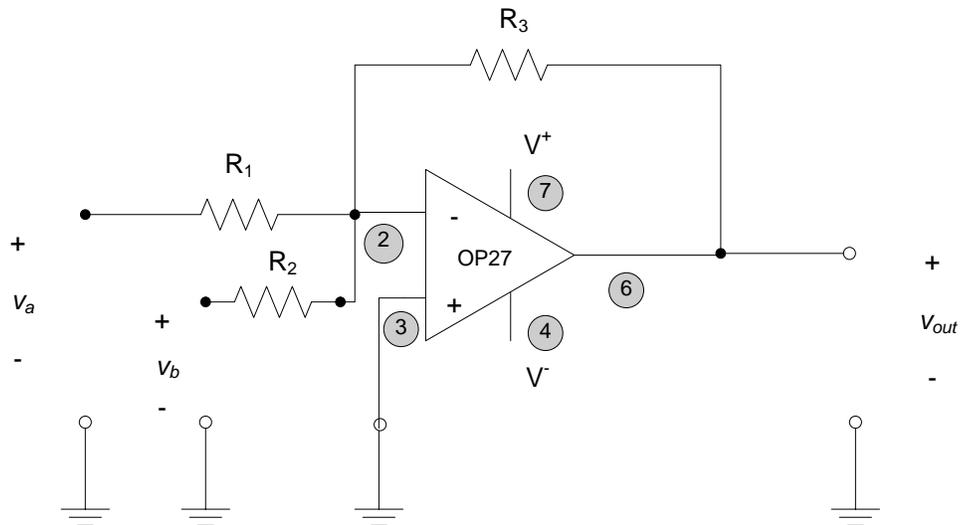


Figure 1. Summing amplifier circuit.

### Pre-lab:

#### ANALYSIS

Design an inverting summing circuit which performs an addition of two signals. The input resistance seen by the two voltage sources ( $v_a$  and  $v_b$ ) should be at least  $1\text{k}\Omega$ . (The input resistance of a circuit is defined as the input voltage divided by the input current. Since the inverting input terminal of the operational amplifier in Fig. 1 is a “virtual” ground, the input resistance seen by  $v_a$  and  $v_b$  are  $R_1$  and  $R_2$ , respectively.)

### Lab Procedures:

#### DATA

1. Implement the design you generated in the pre-lab. Create a schematic of the circuit in your lab notebook, record actual resistance values, and label supply voltages on your schematic. Recommended connections are as follows:

- Use V- on the Analog Discovery 2 to set the negative op-amp supply  $V^-$  to  $-5\text{V}$ ; set the positive op-amp supply  $V^+$  to  $5\text{V}$  using V+ on the Analog Discovery 2.
- Use one waveform generator channel to provide the input voltage  $v_a$  to your circuit, and use the other waveform generator channel to set  $v_b = 1\text{V}$ .
- Measure the output voltage,  $v_{out}$ .

#### DATA

2. Test your design with  $v_a = -4\text{V}, -2\text{V}, -1\text{V}, 0\text{V}, 1\text{V}, 2\text{V}, 3\text{V},$  and  $5\text{V}$ . Tabulate your results ( $v_a$ ,  $v_b$  and  $v_{out}$ ) in your lab notebook (you may assume that the values you set for  $v_a$  and  $v_b$  are correct – you do not need to measure these). Also in your lab notebook, comment on your results (make sure that you compare your results with your expectations based on your pre-lab).

#### DEMO

3. Demonstrate operation of your circuit to the Teaching Assistant. Have the TA initial the appropriate page(s) of your lab notebook and the lab checklist.



# Real Analog Chapter 5: Lab Projects

## 5.4.3: Non-inverting Voltage Amplifier

In this assignment, we implement a simple operational amplifier-based circuit. Since operational amplifiers are used commonly in circuits used to implement mathematical operations, we implement the process of multiplication by a positive constant.

**Before beginning this lab, you should be able to:**

- Analyze operational amplifier-based circuits

**After completing this lab, you should be able to:**

- Design and build an operational amplifier-based non-inverting voltage amplifier

**This lab exercise requires:**

- Analog Discovery 2 module
- Digilent Analog Parts Kit
- Digital multimeter (optional)

**Symbol Key:**

- DEMO** Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
- ANALYSIS** Analysis; include principle results of analysis in laboratory report.
- SIM** Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
- DATA** Record data in your lab notebook.

**General Discussion:**

The circuit shown in Fig. 1 is called a non-inverting amplifier. Appropriate pin numbers for the OP27 operational amplifier are provided on Fig. 1.  $v_{in}$  is the applied (input) voltage to the circuit.  $v_{out}$  is the output voltage from the circuit.

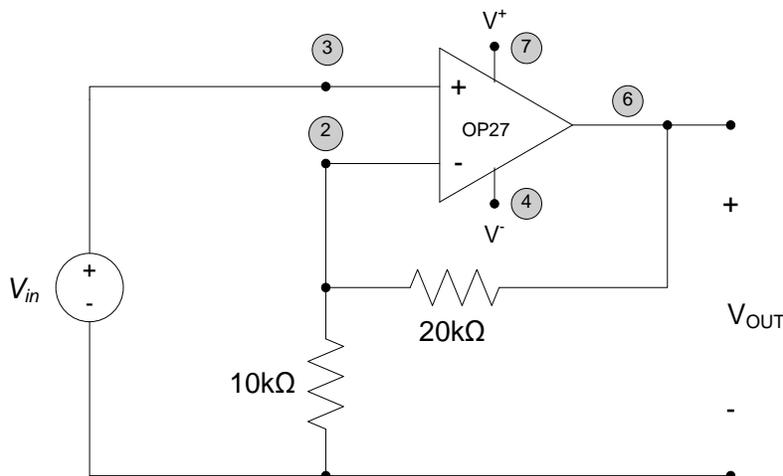


Figure 1. Inverting amplifier circuit.

**Pre-lab:****ANALYSIS**

Determine the relationship between  $V_{in}$  and  $V_{out}$  for the circuit shown in Fig. 1. Why is the circuit called a non-inverting voltage amplifier?

**Lab Procedures:****DATA**

1. Implement the amplifier shown in Fig. 1. Create a schematic of the circuit in your lab notebook, record actual resistance values, and label supply voltages on your schematic. Recommended connections are as follows:
  - Use V- on the Analog Discovery 2 to set the negative op-amp supply  $V^-$  to -5V; use V+ on the Analog Discovery 2 to set the positive op-amp supply  $V^+$  to +5V.
  - Use W1 on your Analog Discovery 2 to provide the input voltage  $v_{in}$  to your circuit.
  - Measure  $V_{OUT}$ .

**DATA**

2. Test your design with input voltages of approximately -3V to +3V with steps of .5V. Tabulate your results ( $v_{in}$  and  $v_{out}$ ) in your lab notebook. Also in your lab notebook, create a plot of  $V_{in}$  vs.  $V_{out}$  and comment on your results (make sure that you calculate a circuit gain – the rate of change of output to input – and compare it to your expectations based on your pre-lab.

**DEMO**

3. Demonstrate operation of your circuit to the Teaching Assistant Have the TA initial the appropriate page(s) of your lab notebook and the lab checklist.



## Real Analog Chapter 5: Lab Projects

### 5.4.3: Difference Amplifier

In this assignment, we implement a simple operational amplifier-based circuit. Since operational amplifiers are used commonly in circuits used to implement mathematical operations, we implement the process of taking the difference between two voltages.

#### Before beginning this lab, you should be able to:

- Analyze operational amplifier-based circuits

#### After completing this lab, you should be able to:

- Design and build an operational amplifier-based difference amplifier

#### This lab exercise requires:

- Analog Discovery 2 module
- Digilent Analog Parts Kit
- Digital multimeter (optional)

#### Symbol Key:

- DEMO** Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
- ANALYSIS** Analysis; include principle results of analysis in laboratory report.
- SIM** Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
- DATA** Record data in your lab notebook.

#### General Discussion:

The circuit shown in Fig. 1 is called a difference amplifier.  $v_a$  and  $v_b$  are the applied (input) voltages to the circuit.  $v_{out}$  is the output voltage from the circuit.

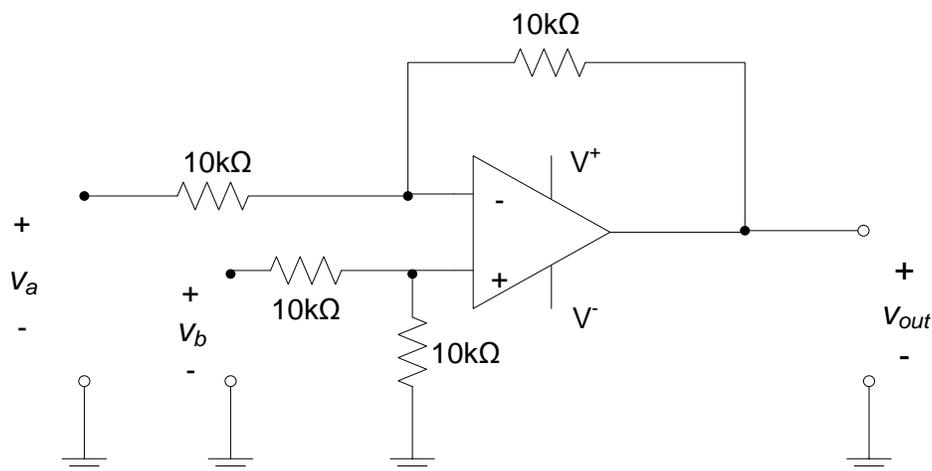


Figure 1. Difference amplifier circuit.

**Pre-lab:****ANALYSIS**

Determine the relationship between  $v_a$ ,  $v_b$  and  $V_{out}$  for the circuit shown in Fig. 1. Why is the circuit called a difference amplifier?

**Lab Procedures:****DATA**

1. Implement the amplifier shown in Fig. 1, using an OP27 operational amplifier. Create a schematic of the circuit in your lab notebook, record actual resistance values, and label supply voltages on your schematic. Recommended connections are as follows:

- Use V- on the Analog Discovery 2 to set the negative op-amp supply  $V^-$  to -5V; use V+ on the Analog Discovery 2 to set the positive op-amp supply  $V^+$  to +5V.
- Use AWG1 on your Analog Discovery 2 to provide the input voltage  $v_a$  to your circuit. Use AWG2 on your Analog Discovery 2 to provide the input voltage  $v_b$  to your circuit.
- Measure  $V_{OUT}$ .

**DATA**

2. Set  $v_b = 1V$ . Test your design with  $v_a = -4V, -2V, -1V, 0V, 1V, 3V,$  and  $5V$ . Tabulate your results ( $v_a$ ,  $v_b$  and  $v_{out}$ ) in your lab notebook. You may assume that the set values for  $v_a$  and  $v_b$  are correct, and that you do not need to measure them. Also in your lab notebook, comment on your results (including a comparison between your test results and your expectations based on your pre-lab).

**DATA**

3. Repeat the above tests with  $v_b = -1V$ . Tabulate your results ( $v_a$ ,  $v_b$  and  $v_{out}$ ) in your lab notebook. Also in your lab notebook, comment on your results (including a comparison between your test results and your expectations based on your pre-lab).

**DEMO**

4. Demonstrate operation of your circuit to the Teaching Assistant. Have the TA initial the appropriate page(s) of your lab notebook and the lab checklist.



## Real Analog Chapter 5: Lab Projects

### 5.4.5: Temperature Measurement System Design

In this lab, we will design a simple temperature measurement system which outputs a DC voltage which indicates temperature. Our system will use a *thermistor* to indicate the temperature; the electrical resistance of the thermistor changes as the temperature changes. We will use a *Wheatstone bridge* circuit to convert this resistance change to a voltage change. The voltage output of the Wheatstone bridge circuit will be small relative to the amount of temperature change (the measurement is said to have low *sensitivity*), so we will use a *difference amplifier* to increase the overall sensitivity of the temperature measurement system.

#### Before beginning this lab, you should be able to:

- State rules governing ideal op-amps
- Analyze electrical circuits which include ideal op-amps
- Describe the operation of a thermistor (Background information for Lab assignments 1.4.4, 2.1.1)

#### After completing this lab, you should be able to:

- Design and balance a Wheatstone bridge circuit
- Design and implement an operational amplifier-based difference amplifier
- Integrate the above subsystems to create an overall temperature measurement system

#### This lab exercise requires:

- Analog Discovery 2 module
- Digiilent Analog Parts Kit
- Digital multimeter (optional)

#### Symbol Key:

-  Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
-  Analysis; include principle results of analysis in laboratory report.
-  Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
-  Record data in your lab notebook.

#### General Discussion:

In this lab assignment, we will design and implement a measurement system which outputs a voltage which is indicative of temperature. A thermistor will be used to measure temperature. The resistance of the thermistor changes with temperature.

The design requirements for the system are as follows:

1. The output voltage from the system is  $0V \pm 20mV$  at room temperature (approximately  $25^{\circ}C$ ).
2. Output voltage is positive for temperatures above room temperature, negative for temperatures below room temperature.
3. Output voltage increases by a minimum of 2V over a temperature range of  $25^{\circ}C$  to  $37^{\circ}C$ . (These temperatures correspond approximately to room temperature and body temperature, respectively.)

A common approach to this problem (and the one we will implement) is to use a Wheatstone bridge circuit in conjunction with a difference amplifier circuit to achieve the necessary sensitivity between temperature and output voltage. A block diagram of the overall system is shown in Fig. 1. The input to the overall system is the temperature of the thermistor. The thermistor converts this temperature into an output resistance. This resistance change is used in a Wheatstone bridge circuit, which converts this resistance change to a voltage change. The voltage difference output by the Wheatstone bridge is generally smaller than desired, so an amplifier is used to increase the amplitude of the overall output voltage from the system.

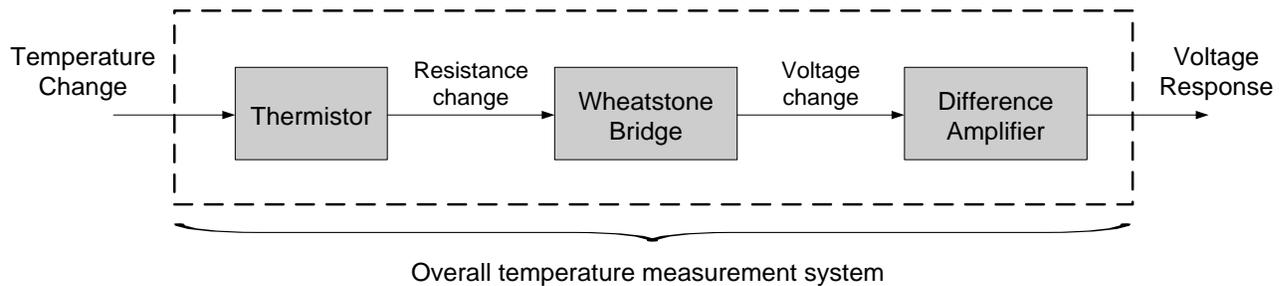


Figure 1. Overall temperature measurement system block diagram.

### Pre-lab:

#### ANALYSIS

Read the information relative to Wheatstone bridge circuits provided in Appendix A of this lab assignment. It is not necessary to exhaustively follow all derivations provided in Appendix A, but you should be able to summarize, in a few sentences, the overall approach toward setting up and balancing a Wheatstone bridge circuit prior to coming to lab. Pay particular attention to equation (A9), which can be used to choose nominal resistances for the bridge circuit, and the practical note on adjusting the nominal values to balance the circuit. Please be sure to note that Appendix A recommends using a potentiometer to balance your Wheatstone bridge circuit.

Read the information relative to difference amplifiers provided in Appendix B of this lab assignment.

### Lab Procedures:

Design approaches tend to vary from individual to individual, however, the recommended lab procedures for this assignment consist of the three discrete steps provided below. Feel free to modify these steps if you wish, but be prepared to explain your design approach to a teaching assistant.

#### DATA

#### 1. Thermistor Characterization:

Measure the nominal resistance of the thermistor (when the thermistor is at room temperature) and the resistance variation from this value when the thermistor is approximately at body temperature (37°C). Apply the 37°C temperature to the thermistor by firmly grasping the thermistor between two fingers. Record these values in your lab notebook.

#### ANALYSIS

#### 2. Wheatstone bridge design and balancing:

Design and build a Wheatstone bridge circuit which converts the resistance variation of the thermistor to a voltage variation. The output of this circuit should be (approximately) zero volts when the thermistor is at room temperature. Provide a schematic of your Wheatstone bridge circuit in your lab notebook, along with the desired and actual resistance values used in the circuit. Also in your lab notebook, record the voltage variation provided by the Wheatstone bridge circuit, resulting from the full range of temperature change.

#### DATA

### Design Hint:

You may not wish to spend a lot of time balancing the bridge at this stage. It is probably more productive to roughly balance the bridge at this point, and then do a final balance after the amplification stage described in step (c) has been implemented.

#### DEMO

*Demonstrate your Wheatstone bridge operation to a teaching assistant and have them initial your lab notebook and the lab checklist.*

#### ANALYSIS

#### 3. Difference amplifier design and implementation:

Design a difference amplifier which amplifies the output voltage difference from the Wheatstone bridge to the levels specified in the design requirements. The circuit schematic and governing equations for a difference amplifier are provided in Appendix B. Implement the circuit. Provide a schematic of your circuit, along with desired and actual resistance values used in your circuit in your lab notebook.

#### DATA

#### DATA

Connect your circuit to the thermistor/Wheatstone bridge assembly and measure the output voltage resulting from the temperature range provided in the specifications (approximately 25°C to 37°C). Verify that voltage increases for as temperature increases and decreases as temperature decreases. Record the range of voltages in your lab notebook, corresponding to the full range of temperature change provided in the design requirements. Compare your measured voltage response to the original design requirements. You do not need to re-design your system if the design requirements are not met, but you should provide comments in your lab notebook as to why you think the circuit behaves differently than expected.

#### DEMO

*Demonstrate your Wheatstone bridge operation to a teaching assistant and have them initial your lab notebook and the lab checklist.*

### Design Hint:

1. Your amplifier circuit in step (c) may “load” your Wheatstone bridge circuit in a different manner than was done in step (b) above. (In step (b), you measured the output voltage from the Wheatstone bridge with a DMM; your difference amplifier may require the Wheatstone bridge to provide different power levels than the DMM does. These effects can be mitigated by using relatively high resistance values in your difference amplifier. (Recall the discussion of input resistance provided in Lab 2. The DMM has a very high input resistance; an alternate circuit may have lower input resistance, and thus make different power demands on the Wheatstone bridge. The change in power requirements may affect the operation of the Wheatstone bridge.
2. After implementing your amplifier circuit in step (c), you may wish to re-balance the Wheatstone bridge circuit. The added sensitivity of the overall system will make the balancing process simpler.
3. The amplifier circuit of Lab 4 (the inverting voltage amplifier) is not appropriate for the implementation of step (c) of this lab assignment. The inverting voltage amplifier of Lab 4 amplifies a voltage which is relative to the ground of the amplifier. In our current application, we have two voltages which are both measured relative to ground; we need to amplify the difference between these voltages.

## Appendix A: Wheatstone Bridge Circuits

Wheatstone bridge circuits are most often used to convert variations in resistance to variations in voltage. Wheatstone bridges are commonly used in measurement systems, as a number of common sensors provide a resistance variation in response to some external influence. For example, *thermistors* change resistance in response to temperature changes, *strain gages* change resistance in response to deformations, and *photoconductive transducers* change resistance in response to changes in light intensity. Wheatstone bridges are generally used in conjunction with these sensors in order to convert these resistance changes to voltage changes since voltages are generally easier to record and transmit than resistances.

### Wheatstone bridge sensitivity to resistance variations:

A Wheatstone bridge circuit is shown in Fig. A1. The bridge is generally presented as shown in the figure to the left; we will use the equivalent circuit shown to the right in our analysis. A Wheatstone bridge is commonly used to convert a variation in resistance to a variation in voltage. A constant supply voltage  $V_S$  is applied to the circuit. The resistors in the circuit all have a nominal resistance of  $R$ ; the variable resistor has a variation  $\Delta R$  from this nominal value. The output voltage  $v_{ab}$  indicates the variation  $\Delta R$  in the variable resistor. The variable resistor in the network is often a transducer whose resistance varies dependent upon some external variable such as temperature.

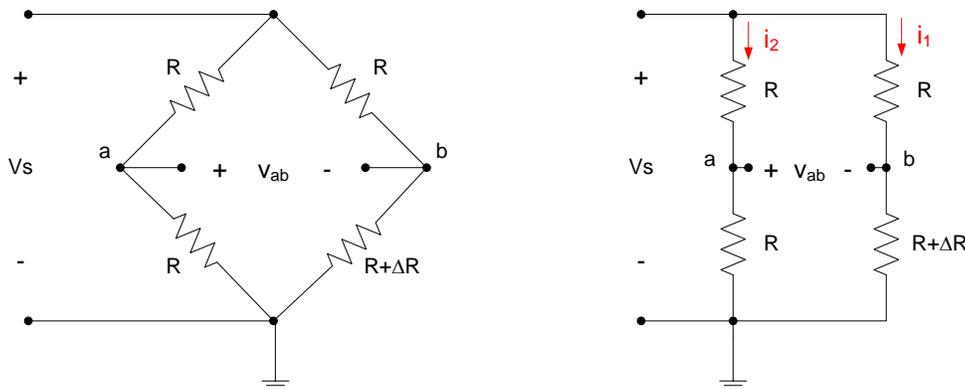


Figure A1. Wheatstone bridge circuit.

By voltage division, the voltages  $v_b$  and  $v_a$  (relative to ground) are

$$v_b = \frac{(R+\Delta R)}{2R+\Delta R} V_S \quad \text{and} \quad v_a = R i_2 = \frac{V_S \cdot R}{2R} = \frac{V_S}{2} \quad \text{Eq. A1}$$

The voltage  $v_{ab}$  is then

$$v_{ab} = v_a - v_b = \left( \frac{1}{2} - \frac{R+\Delta R}{2R+\Delta R} \right) V_S = \left( \frac{(2R+\Delta R) - 2(R+\Delta R)}{2(2R+\Delta R)} \right) V_S = -\frac{\Delta R}{2(2R+\Delta R)} \cdot V_S \quad \text{Eq. A2}$$

For the case in which  $\Delta R \ll 2R$ , this simplifies to:

$$v_{ab} \approx -\frac{V_S}{4R} \Delta R \quad \text{Eq. A3}$$

and the output voltage is proportional to the change in resistance of the variable resistor.

## Balancing the Wheatstone Bridge Circuit:

In Fig. A1, it is assumed that all four resistances in the Wheatstone bridge have identical nominal values. In the case when the Wheatstone bridge output voltage is to result from a varying resistance sensor, this requires one to obtain three resistors with resistance exactly equal to the sensor's nominal resistance. In general, this is not possible. In this chapter, we present an approach for balancing a Wheatstone bridge so that the output voltage is zero when the variable resistance is at its nominal value (i.e.  $\Delta R$  in Fig. A1 is zero), even if the other three resistances in the bridge are not identical.

A schematic of a Wheatstone bridge with non-equivalent resistances is shown in Fig. A2.  $R_{Nom}$  is the nominal value of a variable resistance; it is desired that the output voltage from the bridge circuit is zero for this resistance value.  $R_2$  and  $R_3$  are fixed resistors;  $R_1$  is a variable resistor which will be used to *balance* the bridge circuit.

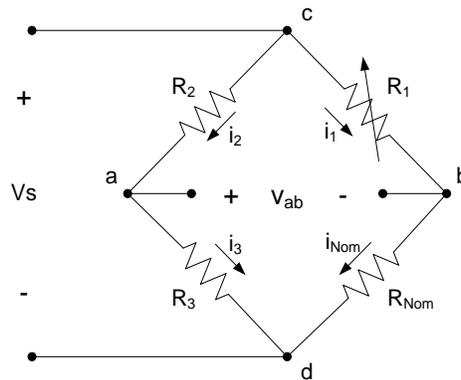


Figure A2. Wheatstone bridge.

## Governing equations for balanced circuit:

If the Wheatstone bridge of Fig. A2 is balanced,  $v_{ab} = 0$ , and

$$i_2 R_2 = i_1 R_1 \quad \text{Eq. A4}$$

KCL at nodes a and b tells us that  $i_2 = i_3$  and  $i_1 = i_{Nom}$ . Using this along with Ohm's law gives:

$$V_S = i_2 (R_2 + R_3) \quad \text{Eq. A5}$$

And

$$V_S = i_1 (R_1 + R_{Nom}) \quad \text{Eq. A6}$$

Equating (A4) and (A5) and taking advantage of equation (A3) provides

$$i_3 R_3 = i_1 R_{Nom} \quad \text{Eq. A7}$$

Or

$$\frac{i_1}{i_2} = \frac{R_3}{R_{Nom}} \quad \text{Eq. A8}$$

From equation (A3),  $\frac{i_1}{i_2} = \frac{R_2}{R_1}$ , so equation (A7) becomes

$$\frac{R_2}{R_1} = \frac{R_3}{R_{Nom}} \quad \text{Eq. A9}$$

and the variable resistance can be set according to:

$$R_1 = \frac{R_2 R_{Nom}}{R_3} \quad \text{Eq. A10}$$

Where  $R_{Nom}$  is the variable resistance for which the circuit is balanced.

### Practical Note:

The value of the resistance  $R_1$  must be set very accurately, so it is common to use a variable-resistance potentiometer to set the resistance  $R_1$ . Specifically, a relatively large-resistance potentiometer can be placed in parallel with a fixed resistor with a resistance slightly higher than the value specified by equation (9) in order to provide the ability to provide very fine adjustments to the value of the resistor  $R_1$ .

Balancing the bridge circuit is commonly performed by setting the variable resistance to its nominal value  $R_{Nom}$ , and monitoring the voltage  $v_{ab}$  while adjusting the resistance  $R_1$ . The resistance  $R_1$  is at its desired value when the voltage  $v_{ab}$  is zero. (The actual value of  $R_1$  required to balance the circuit is generally not important – setting the output voltage to zero at the nominal variable resistance is the ultimate goal.)

## Appendix B: Difference Amplifier

The circuit shown in Fig. B1 is called a difference amplifier. The output of the circuit ( $v_{out}$  in Fig. B1) is proportional to the difference between the two inputs,  $v_a$  and  $v_b$ . With the four independent resistances ( $R_1$ ,  $R_2$ ,  $R_3$ , and  $R_4$ ) shown in Fig. B1, the input voltages can be scaled independently. In order to apply the same scaling factor to both inputs, we can apply a requirement to the choice of the resistances. If we choose

$$\frac{R_1}{R_2} = \frac{R_3}{R_4}$$

then the expression for the output voltage becomes:

$$v_{out} = \frac{R_2}{R_1} (v_b - v_a)$$

The output voltage is, then, a scaled version of the difference between the two input voltages. Note that if  $v_a$  is zero, the circuit is a *non-inverting* amplifier. (The output is a scaled version of the single input  $v_b$ , with no sign change.) Also note that if  $R_1 = R_2 = R_3 = R_4$ , the circuit simply subtracts the voltage  $v_a$  from the voltage  $v_b$ .

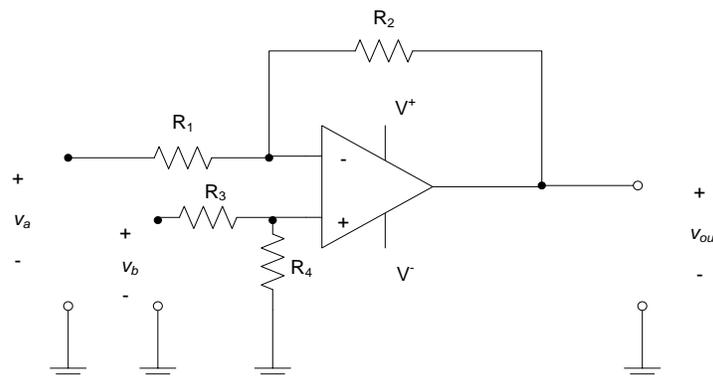


Figure B1. Difference amplifier circuit.

## Real Analog Chapter 5: Lab Worksheets

### 5.4.5: Temperature Measurement System (100 points total)

1. Thermistor Characterization (10 pts total)
  - a. In the space below, provide the measured sensor resistance variation over the specified temperature range. (10 pts)
  
2. Wheatstone Bridge Design (45 pts total)
  - a. Sketch below the circuit to convert resistance variation to voltage variation. Include desired and actual values for circuit components (e.g. resistor values). (15 pts)
  
  - b. Provide below the measured output voltage for nominal (approximately 25°C) temperature. (5 pts)
  
  - c. Provide below the measured range of output voltage over specified temperature range (25°C to 37°C). (10 pts)
  
  - d. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuits' operation. (15 pts)

**TA Initials:** \_\_\_\_\_
  
3. Difference Amplifier Design (45 pts total)
  - a. Provide a sketch of difference amplifier circuit, including desired and actual values for circuit components. (10 pts)

- b. Verify that output voltage increases as temperature increases and decreases as temperature decreases. Provide below the measured range of output voltage over specified temperature range (25°C to 37°C) (10 pts)

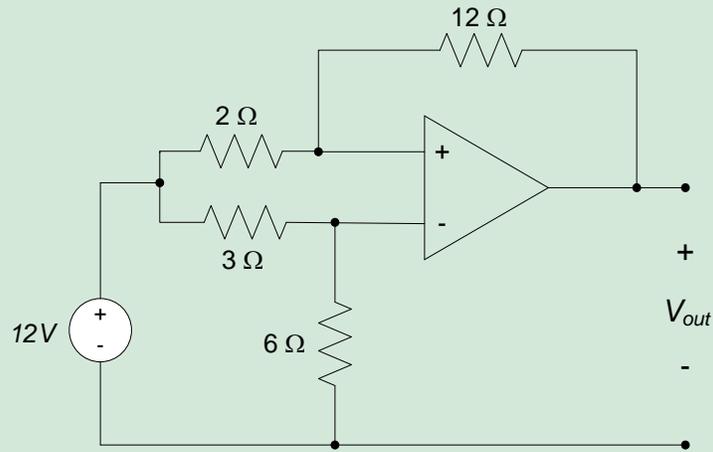
- c. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuits' operation. (15 pts)

**TA Initials:** \_\_\_\_\_

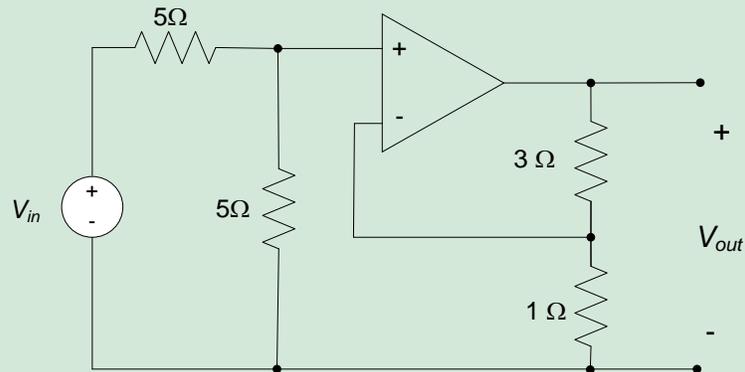
- d. Discuss below the performance of your temperature measurement system performance and comparison with design specifications. (10 pts)

## Real Analog Chapter 5: Homework

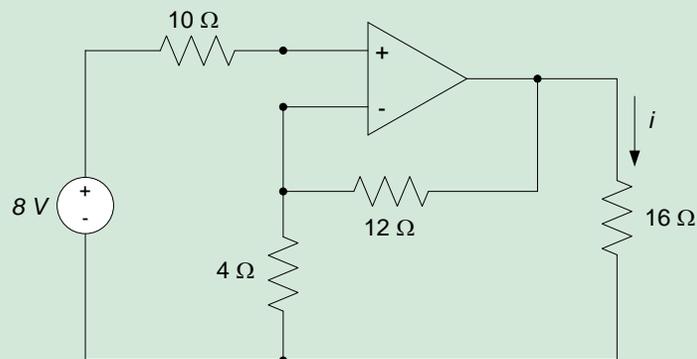
5.1 For the circuit shown, find  $V_{out}$ .



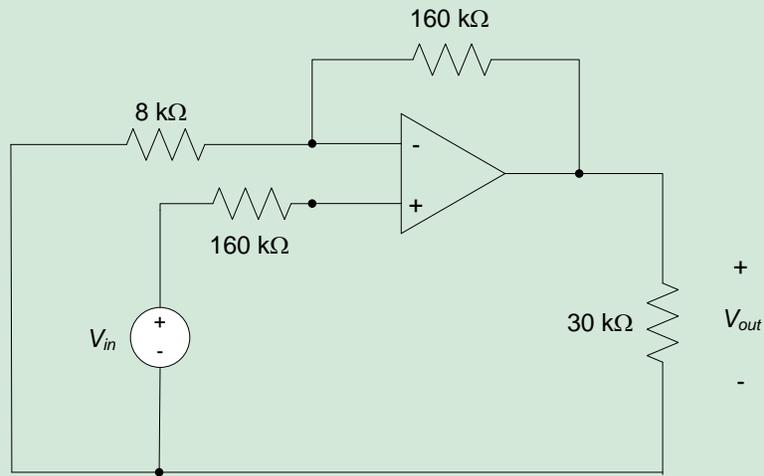
5.2 For the circuit shown, find the relationship between  $V_{out}$  and  $V_{in}$ .



5.3 For the circuit below find  $i$ , the current through the 16 Ω resistor.

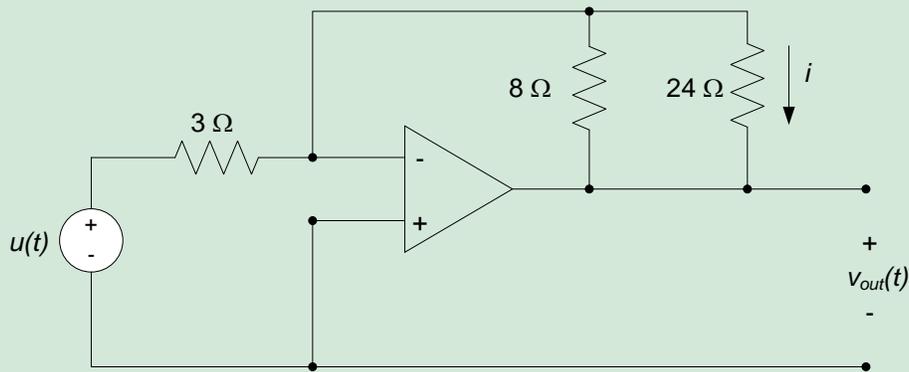


5.4 For the circuit shown, find  $V_{out}$ , the voltage across the 30kΩ resistor.

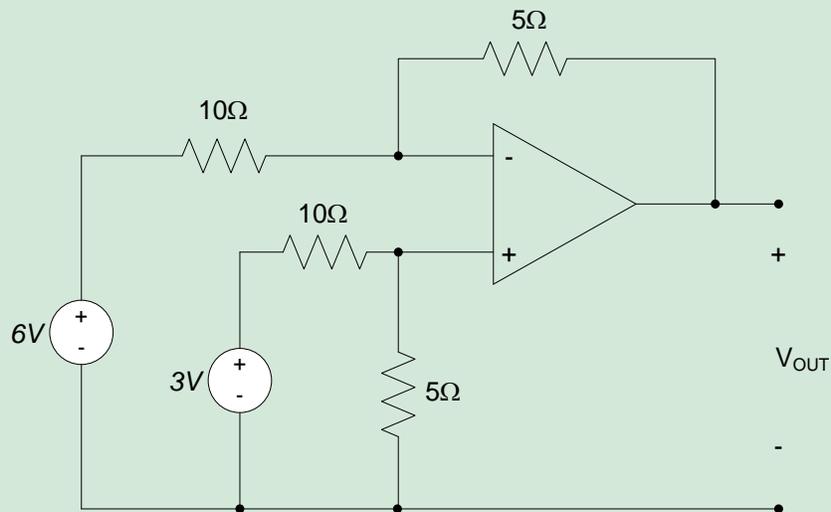


5.5 For the circuit below, determine:

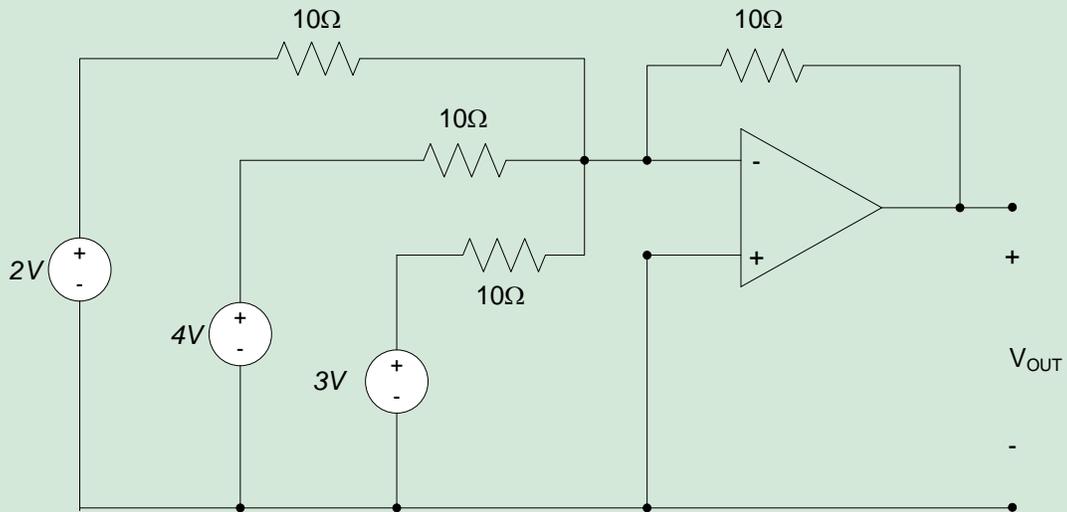
- a. The current,  $i$ , and voltage  $v_{out}$  if  $u(t) = 4V$
- b. The current,  $i$ , and voltage  $v_{out}$  if  $u(t) = 2\cos(5t)$



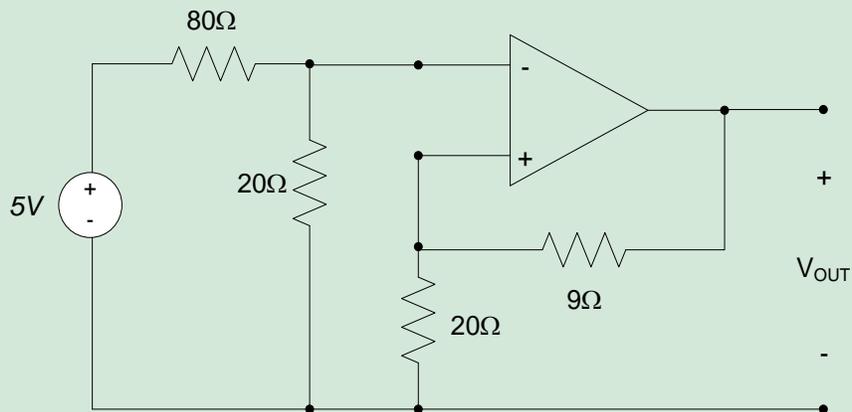
5.6 For the circuit shown, find  $V_{out}$ .



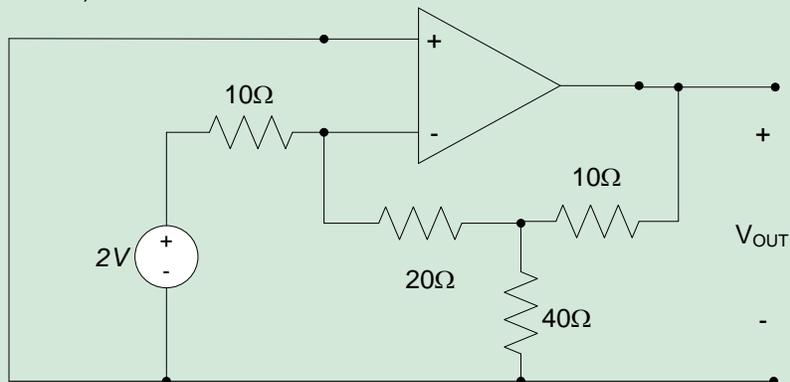
5.7 For the circuit shown, find  $V_{out}$ .



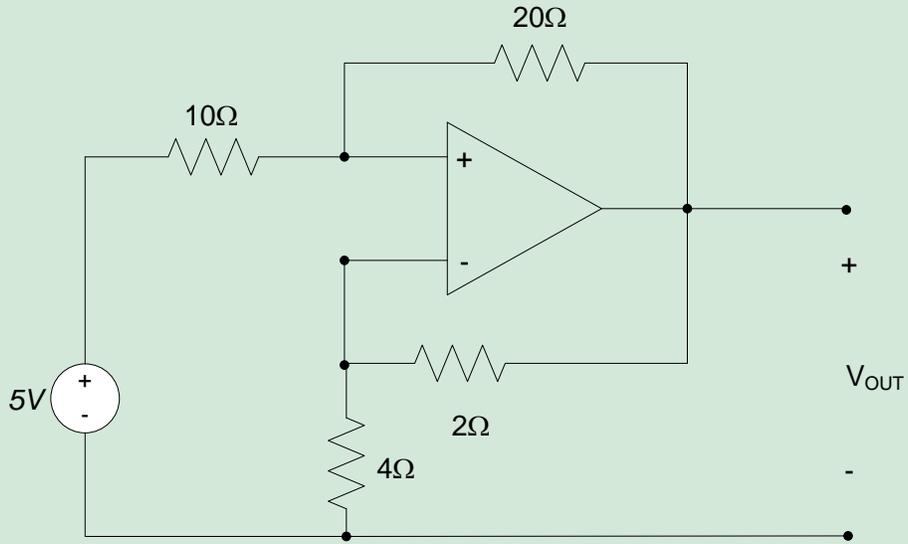
5.8 For the circuit shown, find  $V_{out}$ .



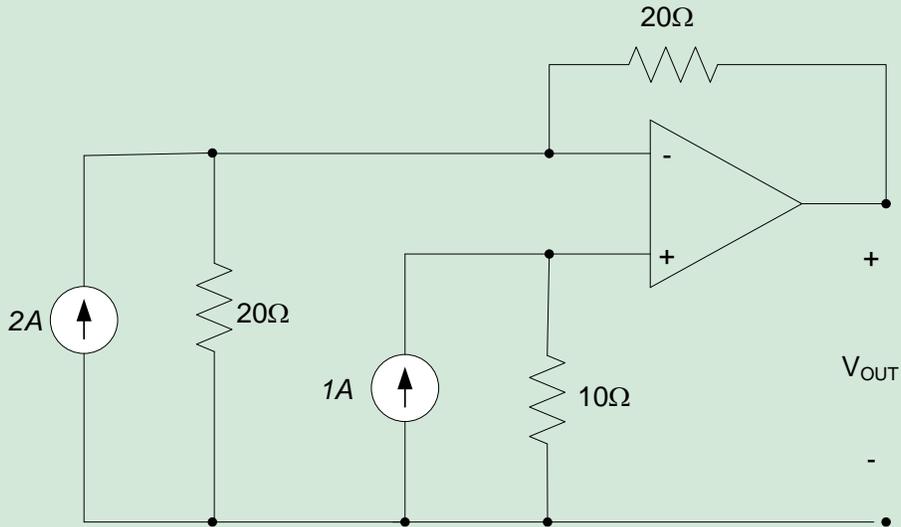
5.9 For the circuit shown, find  $V_{out}$ .



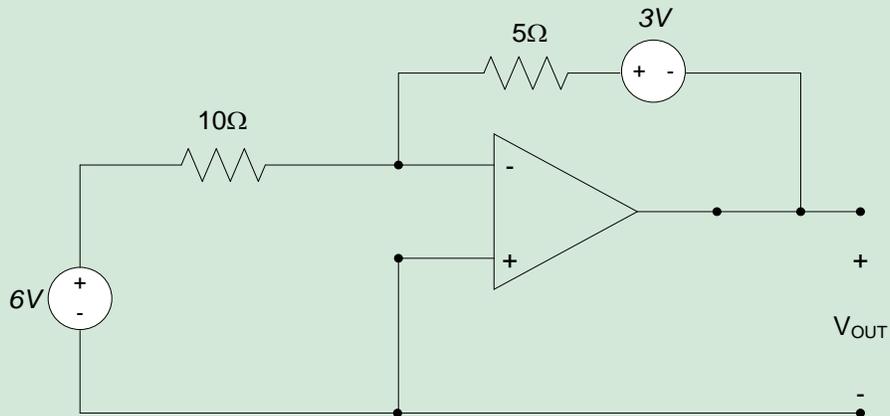
5.10 For the circuit shown, find  $V_{out}$ .



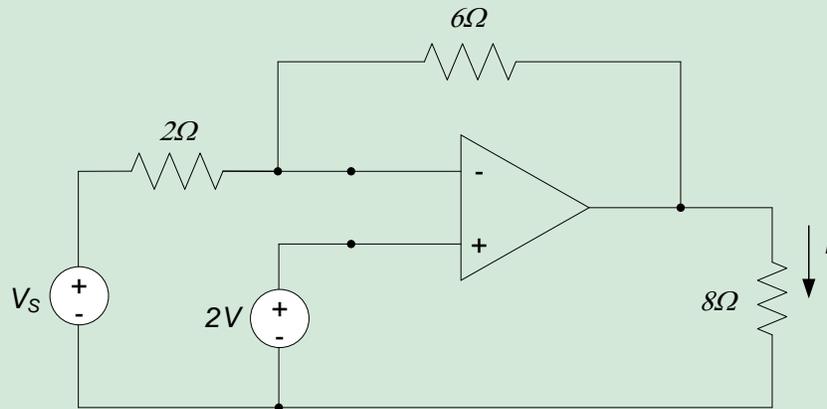
5.11 For the circuit shown, find  $V_{out}$ .



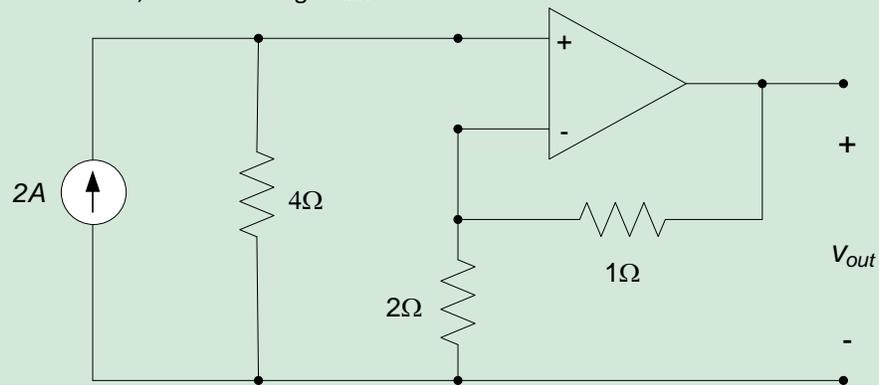
5.12 For the circuit shown, find  $V_{out}$ .



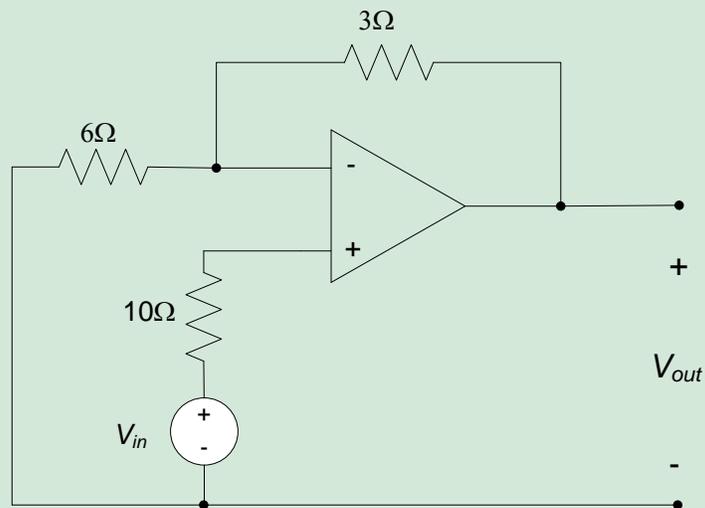
5.13 Find the current  $I$  in the circuit below. Your answer may be a function of the voltage supply  $V_s$ . Clearly show all your work.



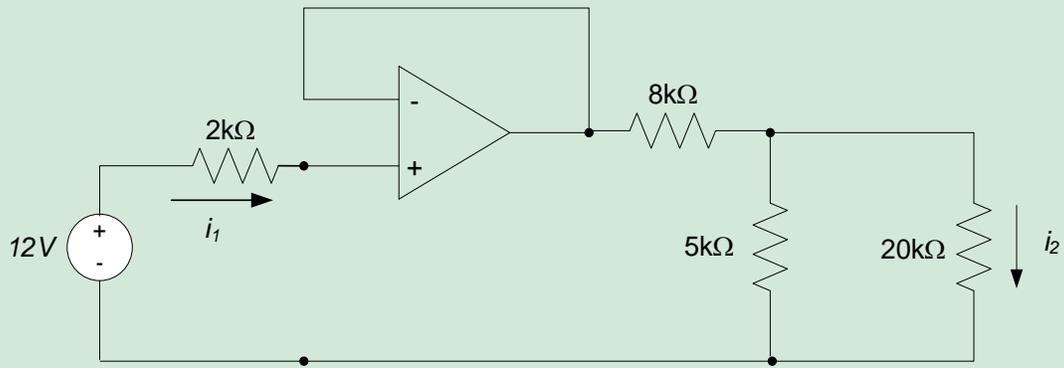
5.14 For the circuit shown, find the voltage  $V_{out}$ .



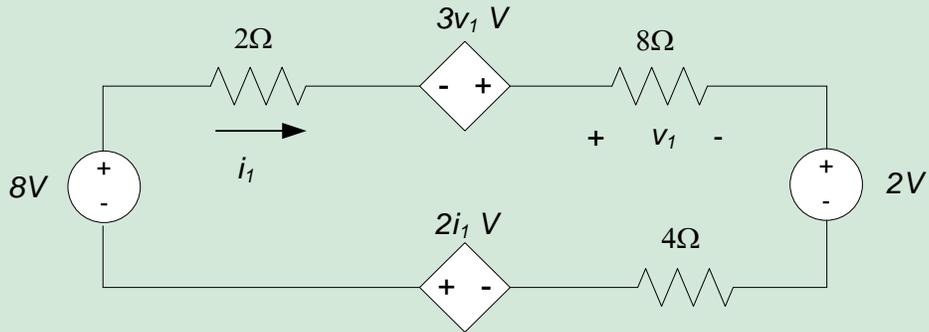
5.15 For the circuit shown, find  $V_{out}$  as a function of  $V_{in}$ .



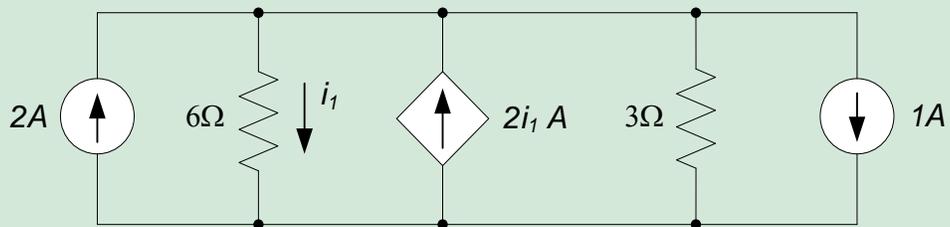
5.16 Determine the currents  $i_1$  and  $i_2$  in the circuit below.



5.17 Determine the voltage  $v_1$  in the circuit below.



5.18 Determine the current  $i_2$  in the circuit below.



## Chapter 6: Energy Storage Elements

### 6 Introduction and Chapter Objectives

So far, we have considered circuits that have been governed by algebraic relations. These circuits have, in general, contained only power sources and resistive elements. All elements in these circuits, therefore, have either supplied power from external sources or dissipated power. For these resistive circuits, we can apply either time-varying or constant signals to the circuit without really affecting our analysis approach. Ohm's law, for example, is equally applicable to time-varying or constant voltages and currents:

$$V = I \cdot R \Leftrightarrow v(t) = i(t) \cdot R$$

Since the governing equation is algebraic, it is applicable at every point in time – voltages and currents at a point in time are affected only by voltages and currents at the same point in time.

We will now begin to consider circuit elements, which are governed by differential equations. These circuit elements are called *dynamic circuit elements* or *energy storage elements*. Physically, these circuit elements store energy, which they can later release back to the circuit. The response, at a given time, of circuits that contain these elements is not only related to other circuit parameters at the same time; it may also depend upon the parameters at other times.

This chapter begins with an overview of the basic concepts associated with energy storage. This discussion focuses not on electrical systems, but instead introduces the topic qualitatively in the context of systems with which the reader is already familiar. The goal is to provide a basis for the mathematics, which will be introduced subsequently. Since we will now be concerned with time-varying signals, section 6.2 introduces the basic signals that we will be dealing with in the immediate future. This chapter concludes with presentations of the two electrical energy storage elements that we will be concerned with: capacitors and inductors. The method by which energy is stored in these elements is presented in sections 6.3 and 6.4, along with the governing equations relating voltage and current for these elements.

#### After completing this chapter, you should be able to:

- Qualitatively state the effect of energy storage on the type of mathematics governing a system
- Define transient response
- Define steady-state response
- Write the mathematical expression for a unit step function
- Sketch the unit step function
- Sketch shifted and scaled versions of the unit step function
- Write the mathematical expression for a decaying exponential function
- Define the time constant of an exponential function
- Sketch a decaying exponential function, given the function's initial value and time constant
- Use a unit step function to restrict an exponential function to times greater than zero
- Write the circuit symbol for a capacitor
- State the mechanism by which a capacitor stores energy
- State the voltage-current relationship for a capacitor in both differential and integral form
- State the response of a capacitor to constant voltages and instantaneous voltage changes
- Write the mathematical expression describing energy storage in a capacitor
- Determine the equivalent capacitance of series and parallel combinations of capacitors
- Sketch a circuit describing a non-ideal capacitor

- Write the circuit symbol for an inductor
- State the mechanism by which an inductor stores energy
- State the voltage-current relationship for an inductor in both differential and integral form
- State the response of an inductor to constant voltages and instantaneous current changes
- Write the mathematical expression describing energy storage in an inductor
- Determine the equivalent inductance of series and parallel combinations of inductors
- Sketch a circuit describing a non-ideal inductor

## 6.2 Fundamental Concepts

This section provides a brief overview of what it meant by energy storage in terms of a system-level description of some physical process. Several examples of energy storage elements are presented, for which the reader should have an intuitive understanding. These examples are intended to introduce the basic concepts in a qualitative manner; the mathematical analysis of dynamic systems will be provided in later chapters.

We have previously introduced the concept of representing a physical process as a *system*. In this viewpoint, the physical process has an input and an output. The input to the system is generated from sources external to the system – we will consider the input to the system to be a known function of time. The output of the system is the system's response to the input. The *input-output equation* governing the system provides the relationship between the system's input and output. A general input-output equation has the form:

$$y(t) = f\{u(t)\} \quad \text{Eq. 6.1}$$

The process is shown in block diagram form in Fig. 6.1.

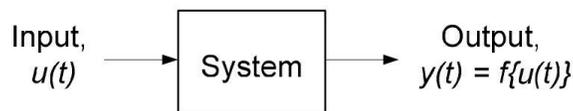


Figure 6.1. Block diagram representation of a system.

The system of Fig. 6.1 transfers the energy in the system input to the system output. This process transforms the input signal  $u(t)$  into the output signal  $y(t)$ . In order to perform this energy transfer, the system will, in general, contain elements that both store and dissipate energy. To date, we have analyzed systems which contain only energy dissipation elements. We review these systems briefly below in a systems context. Subsequently, we introduce systems that store energy; our discussion of energy storage elements is mainly qualitative in this chapter and presents systems for which the reader should have an intuitive understanding.

### 6.2.1 Systems with no Energy Storage

In previous chapters, we considered cases in which the input-output equation is algebraic. This implies that the processes being performed by the system involve only sources and components which dissipate energy. For example, output voltage of the inverting voltage amplifier of Fig. 6.2 is:

$$V_{OUT} = -\left(\frac{R_f}{R_{in}} V_{in}\right) \quad \text{Eq. 6.2}$$

This circuit contains only resistors (in the form of  $R_f$  and  $R_{in}$ ) and sources (in the form of  $V_{in}$  and the op-amp power supplies) and the equation relating the input and output is algebraic. Note that the op-amp power supplies do not appear in equation (6.2), since linear operation of the circuit of Fig. 6.2 implies that the output voltage is independent of the op-amp power supplies.

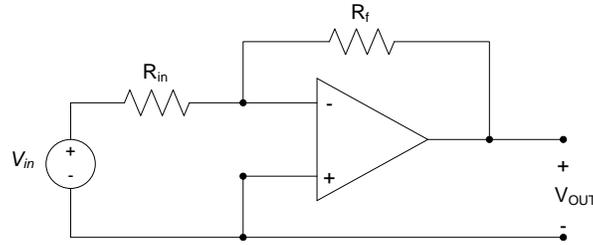


Figure 6.2. Inverting voltage amplifier.

One side effect of an algebraic input-output equation is that the output responds instantaneously to any changes in the input. For example, consider the circuit shown in Fig. 6.3. The input voltage is based on the position of a switch; when the switch closes, the input voltage applied to the circuit increases instantaneously from 0V to 2V. Fig. 6.3 indicates that the switch closes at time  $t = 5$  seconds; thus, the input voltage as a function of time is as shown in Fig. 6.4(a). For the values of  $R_f$  and  $R_{in}$  shown in Fig. 6.3, the input-output equation becomes:

$$V_{OUT}(t) = -5V_{in}(t) \tag{Eq. 6.3}$$

And the output voltage as a function of time is as shown in Fig. 6.4(b). The output voltage responds immediately to the change in the input voltage.

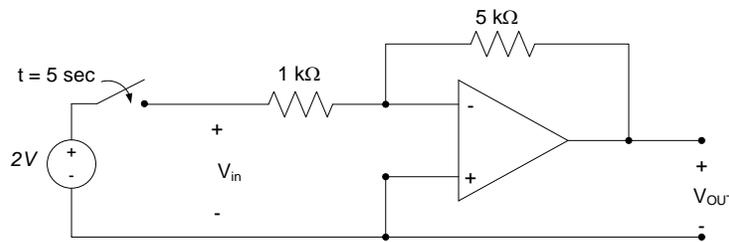


Figure 6.3. Switched voltage amplifier.

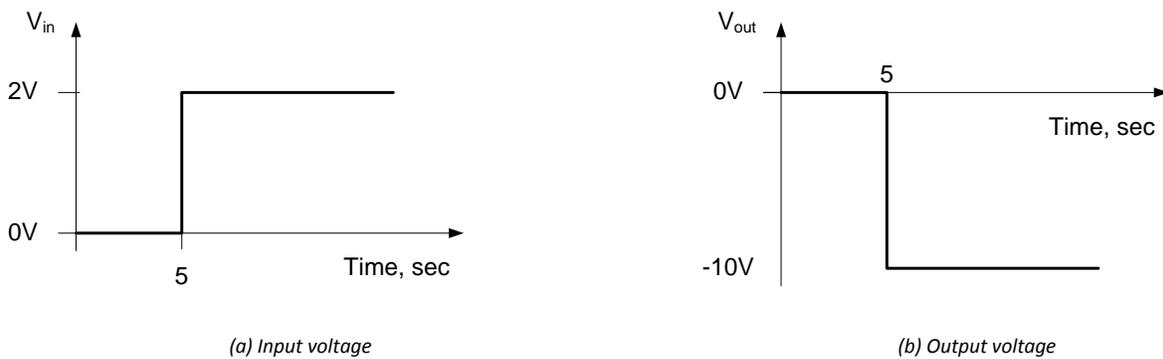


Figure 6.4. Input and output signals for circuit of Figure 3.

## 6.2.2 Systems with Energy Storage

We now consider systems, which contain energy storage elements. The inclusion of energy storage elements results in the input-output equation for the system, which is a differential equation. We present the concepts in terms of two examples for which the reader most likely has some expectations based on experience and intuition.

### Example 6.1: Mass-damper system

As an example of a system, which includes energy storage elements, consider the mass-damper system shown in Fig. 6.5. The applied force  $F(t)$  pushes the mass to the right. The mass's velocity is  $v(t)$ . The mass slides on a surface with sliding coefficient of friction  $b$ , which induces a force, which opposes the mass's motion. We will consider the applied force to be the input to our system and the mass's velocity to be the output, as shown by the block diagram of Fig. 6.6. This system models, for example, pushing a stalled automobile.

The system of Fig. 6.5 contains both energy storage and energy dissipation elements. Kinetic energy is stored in the form of the velocity of the mass. The sliding coefficient of friction dissipates energy. Thus, the system has a single energy storage element (the mass) and a single energy dissipation element (the sliding friction). In section 4.1, we determined that the governing equation for the system was the first order differential equation:

$$m \frac{dv(t)}{dt} + bv(t) = F(t) \quad \text{Eq. 6.4}$$

The presence of the energy storage element causes the input-output equation to be a differential equation.

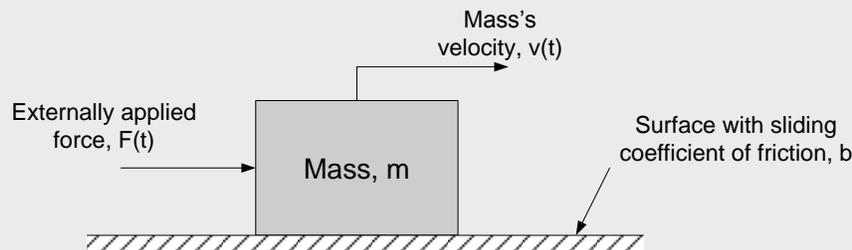


Figure 6.5. Sliding mass on surface with friction coefficient,  $b$ .



Figure 6.6. Mass-damper system represented as a block diagram.

We will examine the effect that the energy storage element has upon the system response in qualitative terms, rather than explicitly solving equation (6.4). If we increase the force applied to the mass, the mass will accelerate and the velocity of the mass increases. The system, therefore, is converting the energy in the input force to a kinetic energy of the mass. This energy transfer results in a change in the output variable, velocity.

The energy storage elements of the system of Fig. 6.5 do not, however, allow an instantaneous change in velocity to an instantaneous change in force. For example, say that before time  $t = 0$  no force is applied to the mass and the mass is at rest. At time  $t = 0$  we suddenly apply a force to the mass, as shown in Fig. 6.7(a) below. At time  $t = 0$  the mass begins to accelerate but it takes time for the mass to approach its final velocity, as shown in Fig. 6.7(b). This transitory stage, when the system is in transition from one constant operating condition to another is called the *transient response*. After a time, the energy input from the external force is balanced by the energy dissipated by the sliding friction, and the velocity of the mass remains constant. When the operating conditions are constant, the energy input is exactly balanced by the energy dissipation, and the system's response is said to be in *steady-state*. We will discuss these terms in more depth in later chapters when we perform the mathematical analysis of dynamic systems.

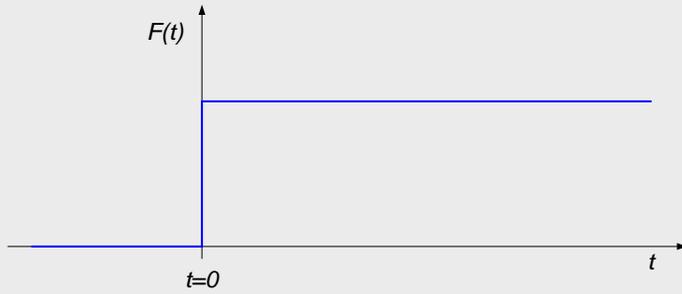


Figure 6.7(a). Force applied to mass.

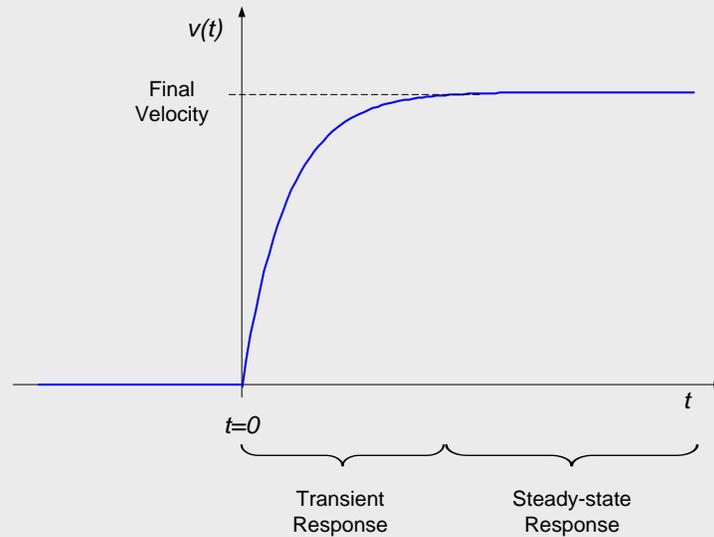


Figure 6.7(b). Velocity of mass.

### Example 6.2

Our second example of a system, which includes energy storage elements, is a body that is subjected to some heat input. The overall system is shown in Fig. 6.8. The body being heated has some mass  $m$ , specific heat  $C_p$ , and temperature  $T_B$ . Some heat input  $q_{in}$  is applied to the body from an external source, and the body transfers heat  $q_{out}$  to its surroundings. The surroundings are at some ambient temperature  $T_0$ . We will consider the input to our system to be the applied heat input  $q_{in}$  and the output to be the temperature of the body  $T_B$ , as shown in the block diagram of Fig. 6.9. This system is a model, for example, of the process of heating a frying pan on a stove. Heat input is applied by the stove burner and the pan dissipates heat by transferring it to the surroundings.

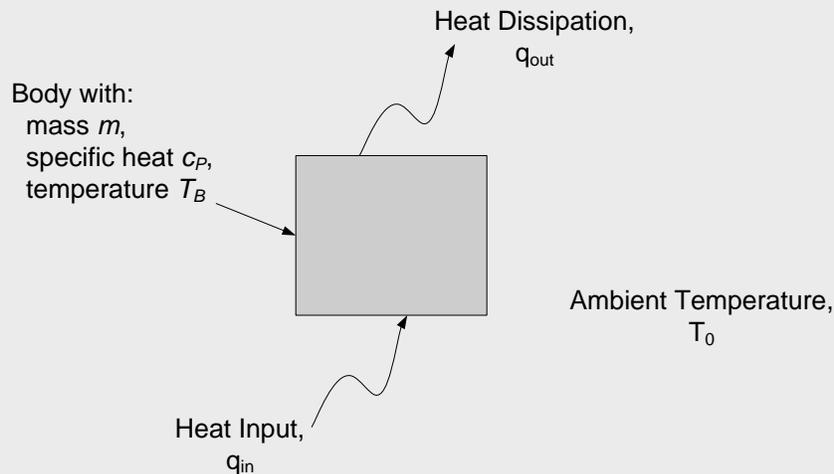


Figure 6.8. Body subjected to heating.

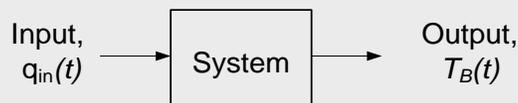


Figure 6.9. System block diagram.

The system of Fig. 6.8 contains both energy storage and energy dissipation elements. Energy is stored in the form of the temperature of the mass. Energy is dissipated in the form of heat transferred to the surroundings. Thus, the system has a single energy storage element (the mass) and a single energy dissipation element (the heat dissipation). The governing equation for the system is the first order differential equation:

$$mc_p \frac{d(T_B - T_0)}{dt} + q_{OUT} = q_{in} \quad \text{Eq. 6.5}$$

The presence of the energy storage element causes the input-output equation to be a differential equation.

We again examine the response of this system to some input in qualitative rather than quantitative terms in order to provide some insight into the overall process before immersing ourselves in the mathematics associated with analyzing the system quantitatively. If the heat input to the system is increased instantaneously (for example, if we suddenly turn up the heat setting on our stove burner) the mass's temperature will increase. As the mass's temperature increases, the heat transferred to the ambient surroundings will increase. When the heat input to the mass is exactly balanced by the heat transfer to the surroundings, the mass's temperature will no longer change and the system will be at a *steady-state* operating condition. Since the mass provides energy storage, the temperature of the mass will not respond instantaneously to a sudden change in heat input – the temperature will rise relatively slowly to its steady-state operating condition. (We know from experience that changing the burner setting on the stove does not immediately change the temperature of our pan, particularly if the pan is heavy.) The process of changing the body's temperature from one steady state operating condition to another is the system's *transient response*.

The process of changing the body's temperature by instantaneously increasing the heat input to the body is illustrated in Fig. 6.10. The signal corresponding to the heat input is shown in Fig. 6.10(a), while the resulting temperature response of the body is shown in Fig. 6.10(b).

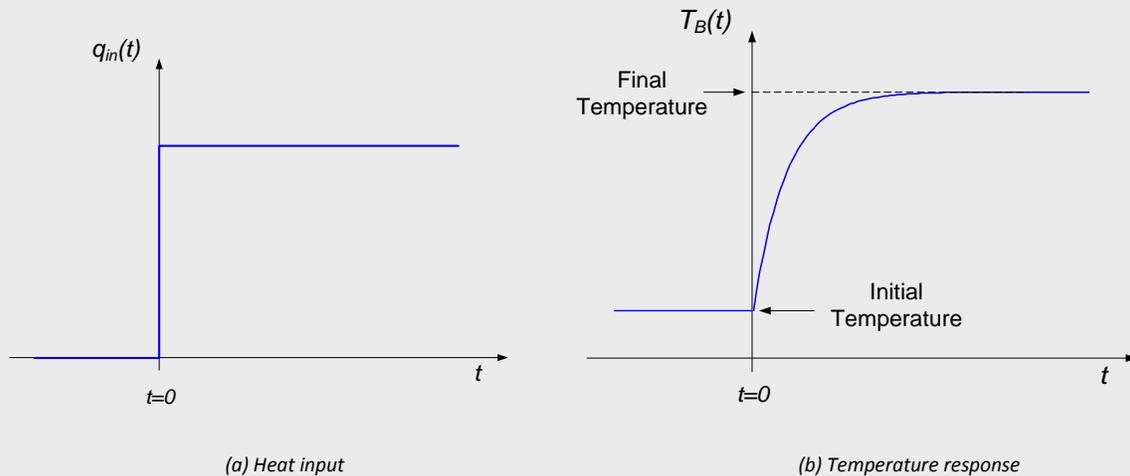


Figure 6.10. Temperature response to instantaneous heat input.

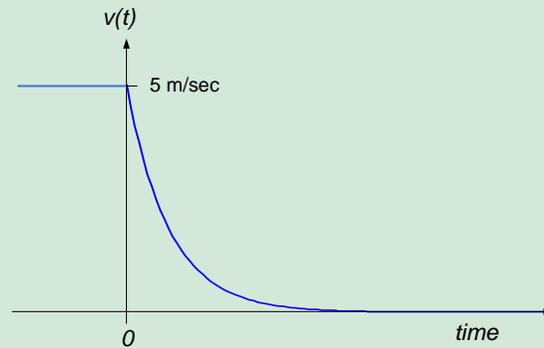
## Section Summary

- Systems with energy storage elements are governed by differential equations. Systems that contain only energy dissipation elements (such as resistors) are governed by algebraic equations.
- The responses of systems governed by algebraic equations will typically have the same “shape” as the input. The output at a given time is simply dependent upon the input at that same time – the system does not “remember” any previous conditions.
- The responses of systems governed by differential equations will not, in general, have the same “shape” as the forcing function applied to the system. The system “remembers” previous conditions – this is why the solution to a differential equation requires knowledge of initial conditions.
- The response of a system that stores energy is generally considered to consist of two parts: the *transient* response and the *steady-state* response. These are described as follows:
  - The transient response typically is shaped differently from the forcing function. It is due to initial energy levels stored in the system.
  - The steady-state response is the response of the system as  $t \rightarrow \infty$ . It is the same “shape” as the forcing function applied to the system.

In differential equations courses, the transient response corresponds (approximately) to the homogeneous solution of the governing differential equation, while the steady-state response corresponds to the particular solution of the governing differential equation.

## 6.1 Exercises

1. A mass is sliding on a surface with an initial velocity of 5 meters/seconds. All external forces (except for the friction force on the surface) are removed from the mass at time  $t = 0$  seconds. The velocity of the mass as a function of time is shown below. What is the steady-state velocity of the mass?



## 6.2 Basic Time-varying Signals

Since the analysis of dynamic systems relies upon time-varying phenomenon, this chapter section presents some common time-varying signals that will be used in our analyses. Specific signals that will be presented are step functions and exponential functions.

### 6.2.1 Step Function

We will use a *step function* to model a signal, which changes suddenly from one constant value to another. These types of signals can be very important. Examples include digital logic circuits (which switch between low and high voltage levels) and control systems (whose design specifications are often based on the system's response to a sudden change in input).

We define a *unit step function*,  $u_0(t)$  as follows:

$$u_0(t) = \begin{cases} 0, & t < 0 \\ 1, & t > 0 \end{cases} \quad \text{Eq. 6.6}$$

The unit step function is illustrated in Fig. 6.11 below. For now, it will not be necessary to define a value for the step function at time  $t=0$ .

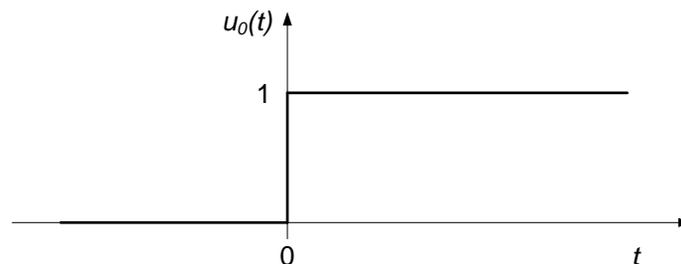


Figure 6.11. Unit step function.

Physically, the step function models a switching process. For example, the output voltage  $V_{out}$  of the circuit shown in Fig. 6.12, in which a constant 1V source supplies voltage through a switch which closes at time  $t=0$ , is a unit step function.

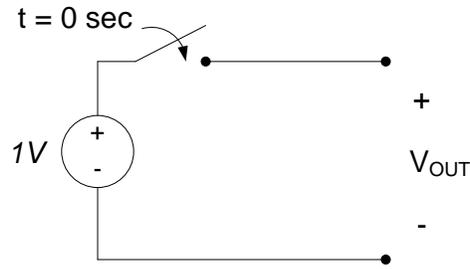


Figure 6.12. Circuit to realize a unit step function.

The unit step function can be *scaled* to provide different amplitudes. Multiplication of the unit step function by a constant  $K$  results in a signal which is zero for times less than zero and  $K$  for times greater than zero, as shown in Fig. 6.13.

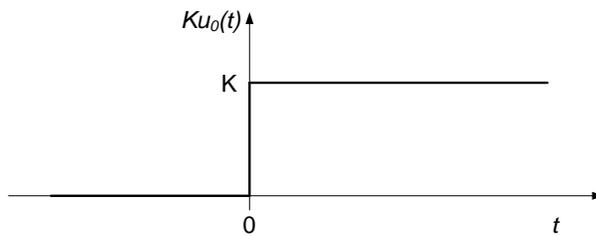


Figure 6.13. Scaled step function  $Ku_0(t)$ ;  $K>0$ .

The step function can also be *shifted* to model processes which switch at times other than  $t=0$ . A step function with amplitude  $K$  which occurs at time  $t=a$  can be written as  $Ku_0(t-a)$ :

$$Ku_0(t - a) = \begin{cases} 0, & t < a \\ K, & t > a \end{cases} \tag{Eq. 6.7}$$

The function is zero when the argument  $t-a$  is less than zero and  $K$  when the argument  $t-a$  is greater than zero, as shown in Fig. 6.14. If  $a>0$ , the function is shifted to the right of the origin; if  $a<0$ , the function is shifted to the left of the origin.

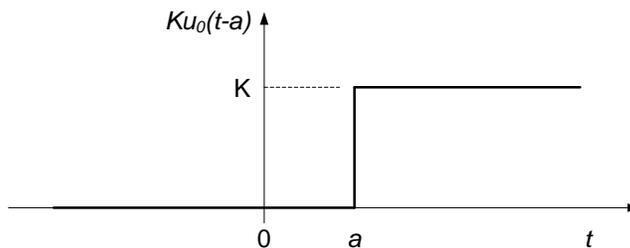


Figure 6.14. Shifted and scaled step function  $Ku_0(t-a)$ ;  $K>0$  and  $a>0$ .

Switching the sign of the above argument in equation (6.7) results in:

$$Ku_0(-t + a) = Ku_a(a - t) = \begin{cases} K, & t < a \\ 0, & t > a \end{cases} \tag{Eq. 6.8}$$

And the value of the function is  $K$  for  $t<a$  and zero for  $t>a$ , as shown in Fig. 6.15. As above, the transition from  $K$  to zero is to the right of the origin if  $a>0$  and to the left of the origin if  $a<0$ .

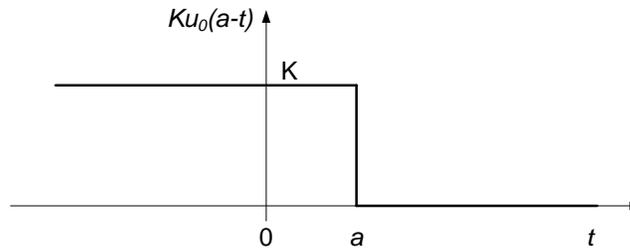


Figure 6.15. The function step function  $Ku_0(a-t)$ ;  $K>0$  and  $a>0$ .

Step functions can also be used to describe finite-duration signals. For example, the function:

$$f(t) = \begin{cases} 0, & t < 0 \\ 1, & 0 < t < 2 \\ 0, & t > 2 \end{cases}$$

Illustrated in Fig. 6.16, can be written in terms of sums or products of unit step functions as follows:

$$f(t) = u_0(t) - u_0(t - 1)$$

Or

$$f(t) = u_0(t) \cdot u_0(2 - t)$$

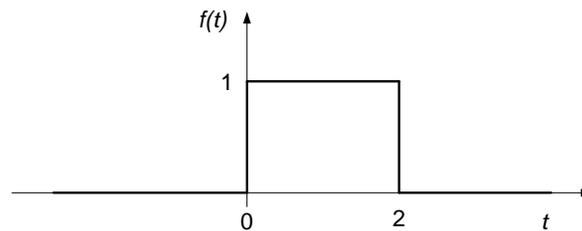


Figure 6.16. Finite-duration signal.

The step function can also be used to create other finite-duration functions. For example, the finite-duration *ramp* function:

$$f(t) = \begin{cases} 0, & t < 0 \\ t, & 0 < t < 1 \\ 0, & t > 1 \end{cases}$$

Shown in Fig. 6.17, can be written as a single function over the entire range  $-\infty < t < \infty$  by using unit step functions, as follows:

$$f(t) = t \cdot [u_0(t) - u_0(t - 1)]$$

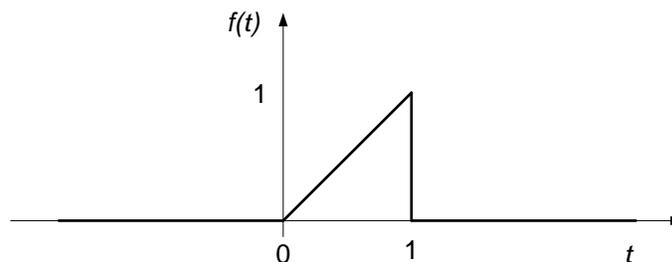


Figure 6.17. Finite-duration "ramp" signal.

## 6.2.2 Exponential Functions

A function that appears commonly in the analysis of linear systems is the *decaying exponential*:

$$f(t) = Ae^{-at}$$

Where  $a > 0$ . The function  $f(t)$  is illustrated in Fig. 6.18. The value of the function is  $A$  at  $t=0$  and decreases to zero as  $t \rightarrow \infty$ . As  $t \rightarrow -\infty$  the function increases without bound. The constant  $a$  dictates the rate at which the function decreases as time increases.

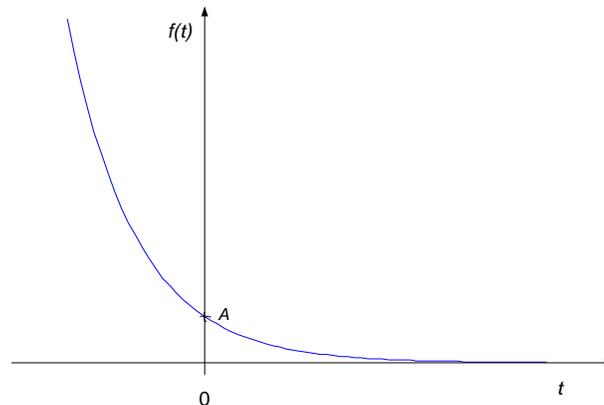


Figure 6.18. Decaying exponential function.

We will usually be interested in this function only for positive values of time. We will also commonly write our exponential function in terms of a time constant,  $\tau$ , rather than the constant  $a$ . Thus, the decaying exponential function we will generally use is:

$$f(t) = \begin{cases} 0, & t < 0 \\ Ae^{-\frac{t}{\tau}}, & t > 0 \end{cases} \quad \text{Eq. 6.10}$$

Or, using the unit step function to limit the function to positive values of time:

$$f(t) = Ae^{-\frac{t}{\tau}} \cdot u_0(t) \quad \text{Eq. 6.11}$$

The function of equations (6.10) and (6.11) is illustrated in Fig. 6.19. The time constant,  $\tau$ , is a positive number which dictates the rate at which the function will decay with time. When the time  $t = \tau$ ,  $f(t)Ae^{-1} = 0.368A$  and the function has decayed to 36.8% of its original value. In fact, the function decreases by 36.8% every  $\tau$  seconds. Therefore, a signal with a small time constant decays more rapidly than a signal with a large time constant, as illustrated in Fig. 6.20.

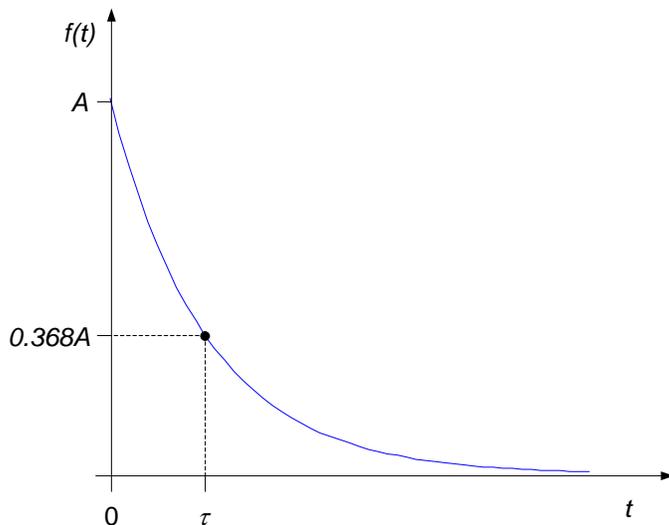


Figure 6.19. Exponential function  $f(t) = Ae^{-\frac{t}{\tau}}u_0(t)$ .

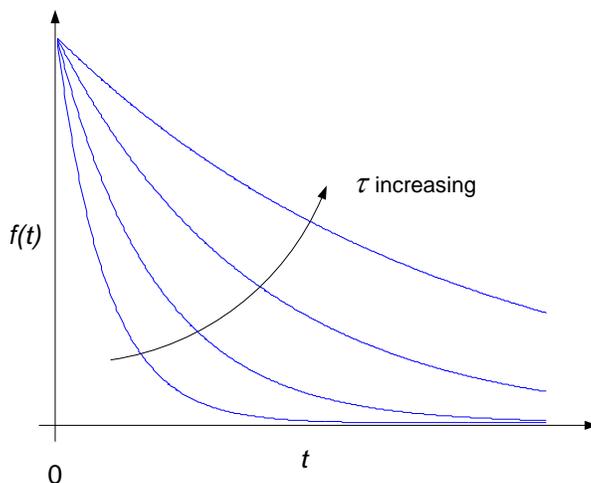


Figure 6.20. Exponential function variation with time cons.

### Section Summary

- Step functions are useful for representing conditions (generally inputs), which change from one value to another instantaneously. In electrical engineering, they are commonly used to model the opening or closing of a switch that connects a circuit to a source, which provides a constant voltage or current. Mathematically, an arbitrary step function can be represented by:

$$Ku_0(-t + a) = Ku_0(a - t) = \begin{cases} K, & t < a \\ 0, & t > a \end{cases}$$

So that the step function turns “on” at time  $t=a$ , and has an amplitude  $K$ .

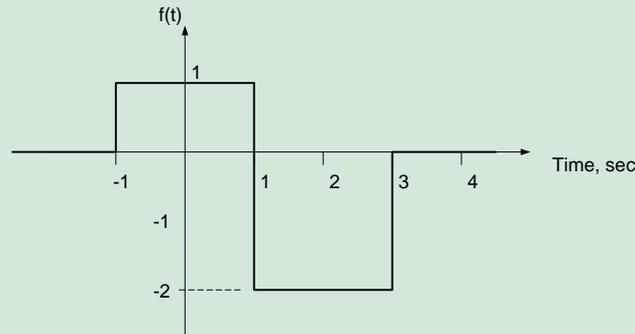
- An exponential function, defined for  $t>0$ , is mathematically defined as:

$$f(t) = Ae^{-\frac{t}{\tau}} \cdot u_0(t)$$

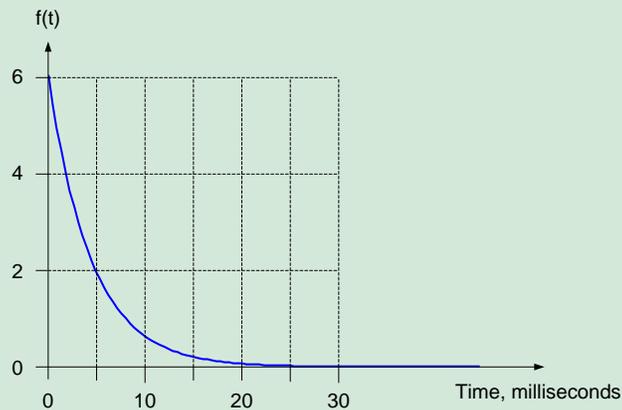
The function has an initial value,  $A$ , and a time constant,  $\tau$ . The time constant indicates how quickly the function decays; the value of the function decreases by 63.2% every  $\tau$  seconds. Exponential functions are important to use because the solutions of linear, constant coefficient, ordinary differential equations typically take the form of exponentials.

## 6.2 Exercises

- Express the signal below in terms of step functions.



- The function shown below is a decaying exponential. Estimate the function from the given graph.



## 6.3 Capacitors

We begin our study of energy storage elements with a discussion of capacitors. Capacitors, like resistors, are passive two-terminal circuit elements. That is, no external power supply is necessary to make them function. Capacitors consist of a non-conductive material (or *dielectric*) which separates two electrical conductors; capacitors store energy in the form of an electric field set up in the dielectric material.

In this section, we describe physical properties of capacitors and provide a mathematical model for an ideal capacitor. Using this ideal capacitor model, we will develop mathematical relationships for the energy stored in a capacitor and governing relations for series and parallel connections of capacitors. The section concludes with a brief discussion of practical (non-ideal) capacitors.

### 6.3.1 Capacitors

Two electrically conductive bodies, when separated by a non-conductive (or *insulating*) material, will form a *capacitor*. Figure 6.21 illustrates the special case of a *parallel plate capacitor*. The non-conductive material

between the plates is called a dielectric; the material property of the dielectric, which is currently important to us, is its *permittivity*,  $\epsilon$ . When a voltage potential difference is applied across the two plates, as shown in Fig. 6.21, charge accumulates on the plates – the plate with the higher voltage potential will accumulate positive charge  $q$ , while the plate with the lower voltage potential will accumulate negative charge,  $-q$ . The charge difference between the plates induces an *electric field* in the dielectric material; the capacitor stores energy in this electric field. The *capacitance* of the capacitor is a quantity that tells us, essentially, how much energy can be stored by the capacitor. Higher capacitance means that more energy can be stored by the capacitor. Capacitance has units of *Farads*, abbreviated F.

The amount of capacitance a capacitor has is governed by the geometry of the capacitor (the shape of the conductors and their orientation relative to one another) and the permittivity of the dielectric between the conductors. These effects can be complex and difficult to quantify mathematically; rather than attempt a comprehensive discussion of these effects, we will simply claim that, in general, capacitance is dependent upon the following parameters:

- The spacing between the conductive bodies (the distance  $d$  in Fig. 6.21). As the separation between the bodies increases, the capacitance decreases.
- The surface area of the conductive bodies. As the surface area of the conductors increases, the capacitance increases. The surface area referred to here is the area over which both the conductors and the dielectric overlap.
- The permittivity of the dielectric. As the permittivity increases, the capacitance increases.

The parallel-plate capacitor shown in Fig. 6.21, for example, has capacitance:

$$C = \frac{\epsilon \cdot A}{d}$$

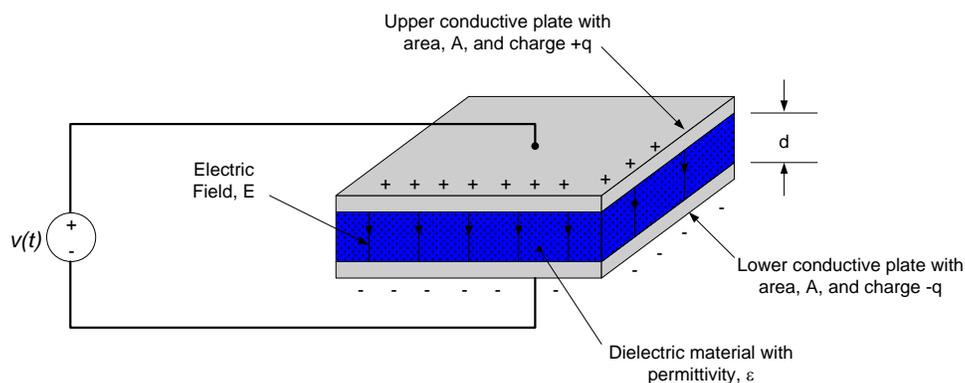


Figure 6.21. Parallel plate capacitor with applied voltage across conductors.

Mathematically, the capacitance of the device relates the voltage difference between the plates and the charge accumulation associated with this voltage:

$$q(t) = CV(t) \tag{Eq. 6.12}$$

Capacitors that obey the relationship of equation (6.12) are *linear capacitors*, since the potential difference between the conductive surfaces is linearly related to the charge on the surfaces. Please note that the charges on the upper and lower plate of the capacitor in Fig. 6.21 are equal and opposite – thus, if we increase the charge on one plate, the charge on the other plate must decrease by the same amount. This is consistent with our previous assumption electrical circuit elements cannot accumulate charge, and current entering one terminal of a capacitor must leave the other terminal of the capacitor.

Since current is defined as the time rate of change of charge,  $i(t) = \frac{dq(t)}{dt}$ , equation (6.12) can be re-written in terms of the current through the capacitor:

$$i(t) = \frac{d}{dt} [Dv(t)] \quad \text{Eq. 6.13}$$

Since the capacitance of a given capacitor is constant, equation (6.13) can be written as:

$$i(t) = C \frac{dv(t)}{dt} \quad \text{Eq. 6.14}$$

The circuit symbol for a capacitor is shown in Fig. 6.22, along with the sign conventions for the voltage-current relationship of equation (6.14). We use our passive sign convention for the voltage-current relationship – positive current is assumed to enter the terminal with positive voltage polarity.

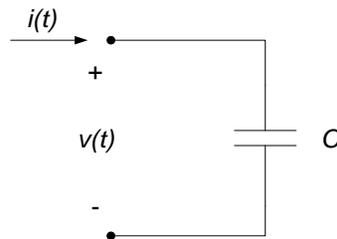


Figure 6.22. Capacitor circuit symbol and voltage-current sign convention.

Integrating both sides of equation (6.14) results in the following form for the capacitor's voltage-current relationship:

$$v(t) = \frac{1}{C} \int_{t_0}^t i(\xi) d\xi + v(t_0) \quad \text{Eq. 6.15}$$

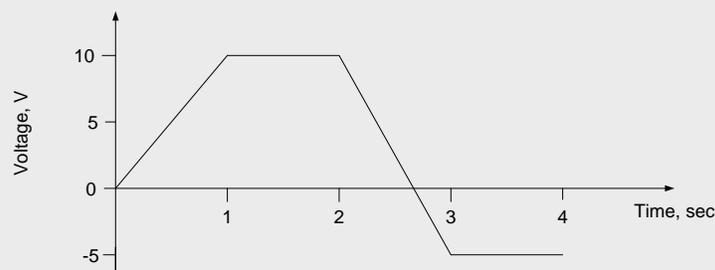
Where  $v(t_0)$  is a known voltage at some initial time,  $t_0$ . We use a dummy variable of integration,  $\xi$ , to emphasize that the only "t" which survives the integration process is the upper limit of the integral.

**Important result:** The voltage-current relationship for an ideal capacitor can be stated in either differential or integral form, as follows:

- $i(t) = C \frac{dv(t)}{dt}$
- $v(t) = \frac{1}{C} \int_{t_0}^t i(\xi) d\xi + v(t_0)$

### Example 6.3

If the voltage as a function of time across a capacitor with capacitance  $C=1\mu F$  is as shown below, determine the current as a function of time through the capacitor.



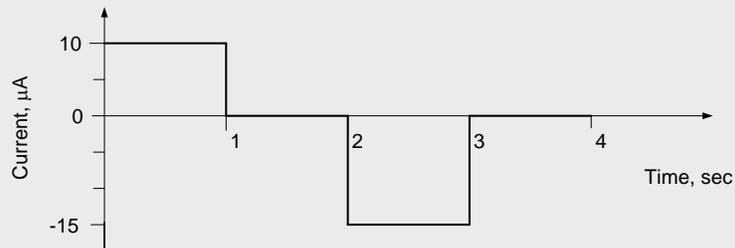
$0 < t < 1$ : The voltage rate of change is 10 V/s. Thus,  $C \frac{dv(t)}{dt} = (1 \times 10^{-6} F) \left(10 \frac{V}{s}\right) = 10 \mu A$ .

$1 < t < 2$ : The voltage is constant. Thus,  $C \frac{dv(t)}{dt} = 0 A$ .

$2 < t < 3$ : The voltage rate of change is -15V/s. Thus,  $C \frac{dv(t)}{dt} = (1 \times 10^{-6} F) \left(-15 \frac{V}{s}\right) = -15 \mu A$ .

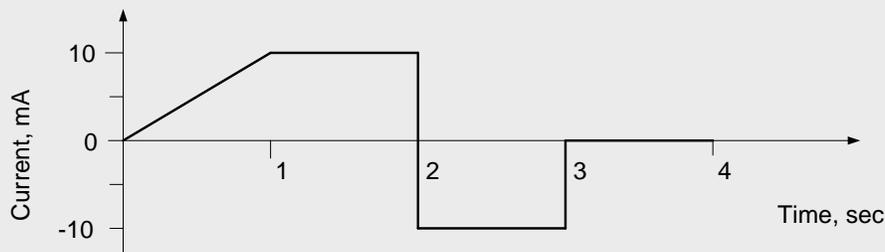
$3 < t < 4$ : The voltage is constant. Thus,  $C \frac{dv(t)}{dt} = 0 A$

A plot of the current through the capacitor as a function of time is shown below.



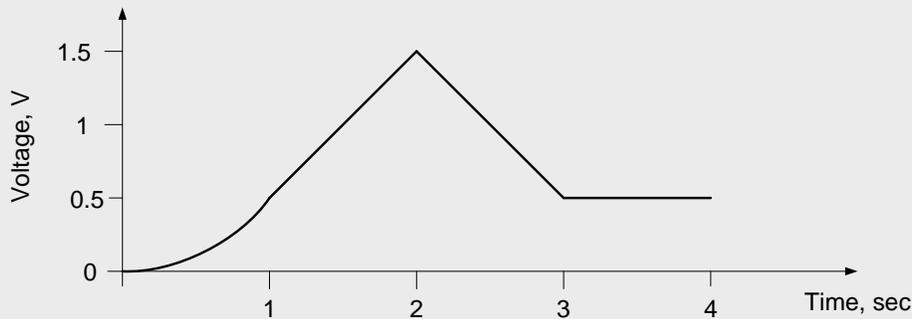
### Example 6.4

If the current as a function of time through a capacitor with capacitance  $C=10mF$  is as shown below, determine the voltage as a function of time across the capacitor. Assume that the voltage across the capacitor is 0V at time  $t=0$ .



- At time  $t=0$ , the voltage is given to be 0V.
- In the time period  $0 < t < 1$  second, the current increases linearly and the voltage will increase quadratically. The total voltage change during this time period is the integral of the current, which is simply the area under the current curve divided by the capacitance:  $\frac{1}{2} \frac{(10 \times 10^{-3} A)(1s)}{0.01 F} = 0.5V$ .
- In the time period  $1 < t < 2$  seconds, the current is constant at 10 mA. The voltage change is the area under the current curve divided by the capacitance:  $(10 \times 10^{-3} A) \frac{(1s)}{0.01 F} = 1V$ . The total voltage at  $t=2$  seconds is, then,  $0.5V + 1V = 1.5V$ .
- In the time period  $2 < t < 3$  seconds, the current is constant at -10 mA. The voltage change is the negative of the voltage change from  $1 < t < 2$  sec. The total voltage at  $t=3$  seconds is, then,  $1.5V - 1V = 0.5V$ .
- In the time period  $3 < t < 4$  seconds, the current is zero. The integral of zero over any time period is zero, so there is no change in voltage during this time range and the voltage remains constant at 0.5V.

A plot of the voltage across the capacitor as a function of time is shown below.



It is often useful, when analyzing circuits containing capacitors, to examine the circuit's response to constant operating conditions and to instantaneous changes in operating condition. We examine the capacitor's response to each of these operating conditions below:

- Capacitor response to constant voltage:
  - If the voltage across the capacitor is constant, equation (6.14) indicates that the current through the capacitor is zero. Thus, if the voltage across the capacitor is constant, the capacitor is equivalent to an open circuit.
  - This property can be extremely useful in determining a circuit's steady-state response to constant inputs. If the inputs to a circuit change from one constant value to another, the transient components of the response will eventually die out and all circuit parameters will become constant. Under these conditions, capacitors can be replaced with open circuits and the circuit analyzed relatively easily. As we will see later, this operating condition can be useful in determining the response of circuits containing capacitors and in double-checking results obtained using other methods.
- Capacitor response to instantaneous voltage changes:
  - If the voltage across the capacitor changes instantaneously, the rate of change of voltage is infinite. Thus, by equation (6.14), if we wish to change the voltage across a capacitor instantaneously, we must supply infinite current to the capacitor. This implies that infinite power is available, which is not physically possible. Thus, in any practical circuit, the voltage across a capacitor cannot change instantaneously.
  - Any circuit that allows an instantaneous change in the voltage across an ideal capacitor is not physically realizable. We may sometimes assume, for mathematical convenience, that an ideal capacitor's voltage changes suddenly; however, it must be emphasized that this assumption requires an underlying assumption that infinite power is available and is thus not an allowable operating condition in any physical circuit.

#### Important Capacitor Properties:

- Capacitors can be replaced by open-circuits, under circumstances when all operating conditions are constant.
- Voltages across capacitors cannot change instantaneously. No such requirement is placed on currents.

### 6.3.2 Energy Storage

The power dissipated by a capacitor is:

$$p(t) = v(t) \cdot i(t) \quad \text{Eq. 6.16}$$

Since both voltage and current are functions of time, the power dissipation will also be a function of time. The power as a function of time is called the *instantaneous power*, since it provides the power dissipation at any instant in time.

Substituting equation (6.14) into equation (6.16) results in:

$$p(t) = C \cdot v(t) \frac{dv(t)}{dt} \quad \text{Eq. 6.17}$$

Since power is, by definition, the rate of change of energy, the energy is the time integral of power. Integrating equation (6.17) with respect to time gives the following expression for the energy stored in a capacitor:

$$W_c(t) = \int_{-\infty}^t C v(\xi) \frac{dv(\xi)}{dt} dt = \int_{-\infty}^t C v(\xi) dv(\xi) = \frac{1}{2} C v^2(\xi) \Big|_{-\infty}^t$$

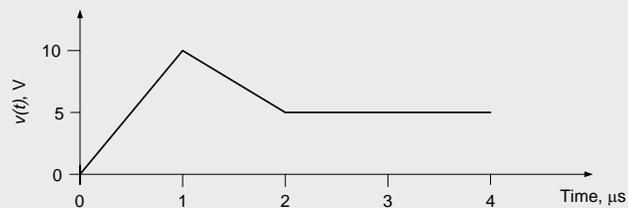
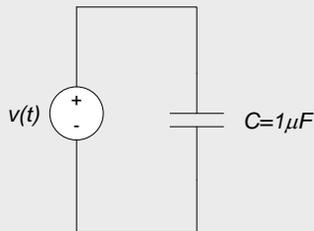
Where we have set our lower limits of integration at  $t = -\infty$  to avoid issues relative to initial conditions. We assume that no energy is stored in the capacitor at time  $t = -\infty$  so that:

$$W_c(t) = \frac{1}{2} C v^2(t) \quad \text{Eq. 6.18}$$

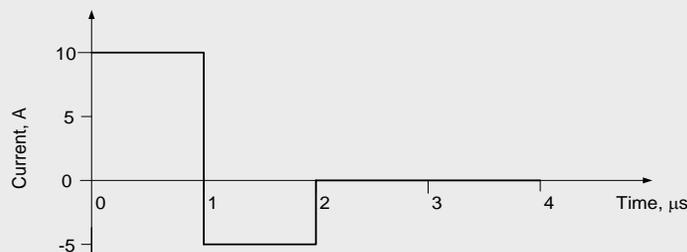
From equation (6.18) we see that the energy stored in a capacitor is always a non-negative quantity, so  $W_c(t) \geq 0$ . Ideal capacitors do not dissipate energy, as resistors do. Capacitors store energy when it is provided to them from the circuit; this energy can later be recovered and returned to the circuit.

### Example 6.5

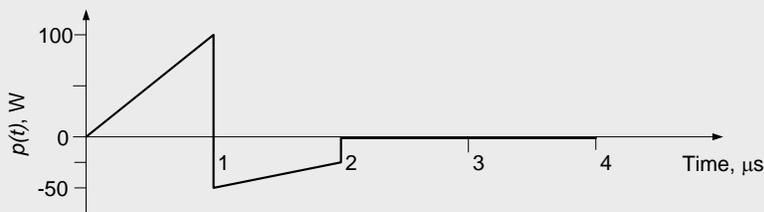
Consider the circuit shown below. The voltage applied to the capacitor by the source is as shown. Plot the power absorbed by the capacitor and the energy stored in the capacitor as functions of time.



Power is most readily computed by taking the product of voltage and current. The current can be determined from equation (6.14). The current as a function of time is plotted below.

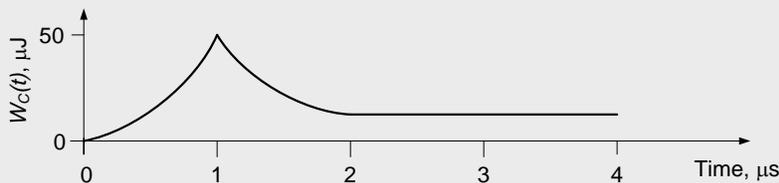


The power absorbed by the capacitor is determined by taking a point-by-point product between the voltage and current.



Recall that power is absorbed or generated based on the passive sign convention. If the relative signs between voltage and current agree with the passive sign convention, the circuit element is absorbing power. If the relative signs between voltage and current are opposite to the passive sign convention, the element is generating power. Thus, the capacitor in this example is absorbing power for the first microsecond. It generates power (returns power to the voltage source) during the second microsecond). After the second microsecond, the current is zero and the capacitor neither absorbs nor generates power.

The energy stored in the capacitor can be determined either from integrating the power or from application of equation (6.18) to the voltage curve provided in the problem statement. The energy in the capacitor as a function of time is shown below:



During the first microsecond, while the capacitor is absorbing power, the energy in the capacitor is increasing. The maximum energy in the capacitor is 50  $\mu\text{J}$ , at 1  $\mu\text{s}$ . During the second microsecond, the capacitor is releasing power back to the circuit and the energy in the capacitor is decreasing. At 2  $\mu\text{s}$ , the capacitor still has 12.5  $\mu\text{J}$  of stored energy. After 2  $\mu\text{s}$ , the capacitor neither absorbs nor generates energy and the energy stored in the capacitor remains at 12.5  $\mu\text{J}$ .

### 6.3.3 Capacitors in Series

Consider the series connection of  $N$  capacitors shown in Fig. 6.23.

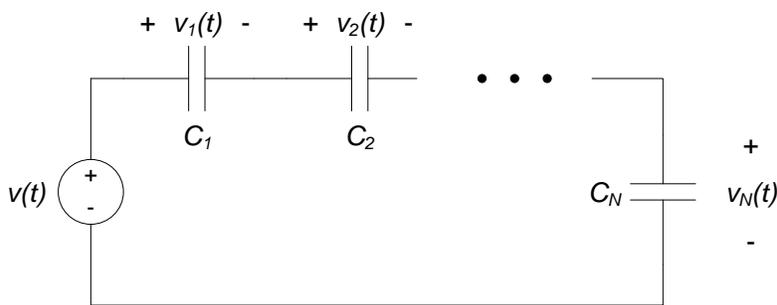


Figure 6.23. Series connection of  $N$  capacitors.

Applying Kirchhoff's voltage law around the loop results in:

$$v(t) = v_1(t) + v_2(t) + \dots + v_N(t) \tag{Eq. 6.19}$$

Using equation (6.15) to write the capacitor voltage drops in terms of the current through the loop gives:

$$\begin{aligned}
 v(t) &= \left[ \frac{1}{C_1} \int_{t_0}^t i(\xi) d\xi + v_1(t_0) \right] + \left[ \frac{1}{C_2} \int_{t_0}^t i(\xi) d\xi + v_2(t_0) \right] + \dots + \left[ \frac{1}{C_N} \int_{t_0}^t i(\xi) d\xi + v_N(t_0) \right] \\
 &= \left[ \frac{1}{C_1} \int_{t_0}^t i(\xi) d\xi + \frac{1}{C_2} \int_{t_0}^t i(\xi) d\xi + \dots + \frac{1}{C_N} \int_{t_0}^t i(\xi) d\xi \right] + [v_1(t_0) + v_2(t_0) + \dots + v_N(t_0)] \\
 &= \left( \frac{1}{C_1} + \frac{1}{C_2} + \dots + \frac{1}{C_N} \right) \int_{t_0}^t i(\xi) d\xi + v(t_0)
 \end{aligned}$$

This can be re-written using summation notation as:

$$v(t) = \left( \sum_{k=1}^N \frac{1}{C_k} \right) \int_{t_0}^t i(\xi) d\xi + v(t_0) \tag{Eq. 6.21}$$

Thus, the circuits of Fig. 6.23 and Fig. 6.24 are equivalent circuits, if the equivalent capacitance is chosen according to equation (6.21).

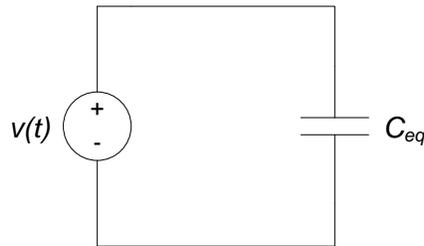


Figure 6.24. Equivalent circuit to Figure 3.

For the special case of two capacitors  $C_1$  and  $C_2$  in series, equation (6.21) simplifies to:

$$C_{eq} = \frac{C_1 C_2}{C_1 + C_2} \tag{Eq. 6.22}$$

Equations (6.21) and (6.22) are analogous to the equations, which provide the equivalent resistance of parallel combinations of resistors.

### 6.3.4 Capacitors in Parallel

Consider the parallel combination of  $N$  capacitors, as shown in Fig. 6.25.

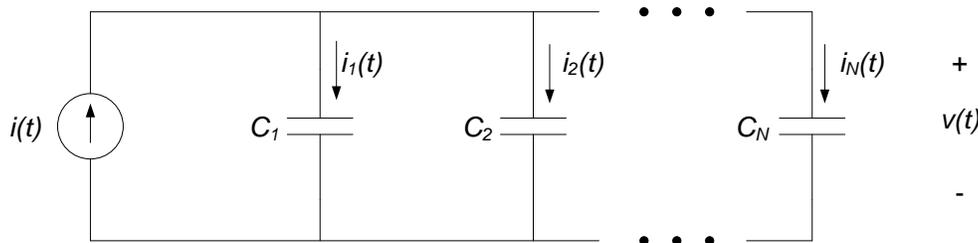


Figure 6.25. Series connection of  $N$  capacitors.

Applying Kirchoff's current law at the upper node results in:

$$i(t) = i_1(t) + i_2(t) + \dots + i_N(t) \tag{Eq. 6.23}$$

Using equation (6.14) to write the capacitor currents in terms of their voltage drop gives:

$$\begin{aligned}
 i(t) &= C_1 \frac{dv(t)}{dt} + C_2 \frac{dv(t)}{dt} + \cdots + C_N \frac{dv(t)}{dt} \\
 &= (C_1 + C_2 + \cdots + C_N) \frac{dv(t)}{dt}
 \end{aligned}$$

Using summation notation results in:

$$i(t) = \left( \sum_{k=1}^N C_k \right) \frac{dv(t)}{dt} \quad \text{Eq. 6.24}$$

This is the same equation that governs the circuit of Fig. 6.26, if:

$$C_{eq} = \sum_{k=1}^N C_k \quad \text{Eq. 6.25}$$

Thus, the equivalent capacitance of a parallel combination of capacitors is simply the sum of the individual capacitances. This result is analogous to the equations, which provide the equivalent resistance of a series combination of resistors.

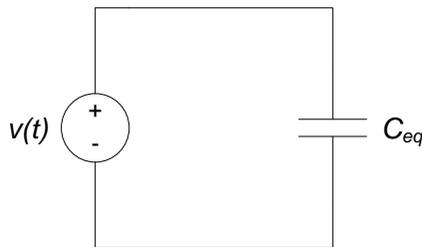


Figure 6.26. Equivalent circuit to Figure 5.

### Summary: Series and Parallel Capacitors

- The equivalent capacitance of a series combination of capacitors  $C_1, C_2, \dots, C_N$  is governed by a relation which is analogous to that providing the equivalent resistance of a parallel combination of resistors:

$$\frac{1}{C_{eq}} = \sum_{k=1}^N \frac{1}{C_k}$$

- The equivalent of a parallel combination of capacitors  $C_1, C_2, \dots, C_N$  is governed by a relation which is analogous to that providing the equivalent resistance of a series combination of resistors:

$$C_{eq} = \sum_{k=1}^N C_k$$

### 6.3.5 Practical Capacitors

Commercially available capacitors are manufactured in a wide range of both conductor and dielectric materials and are available in a wide range of capacitances and voltage ratings. The voltage rating of the device is the maximum voltage, which can be safely applied to the capacitor; using voltages higher than the rated value will damage the capacitor. The capacitance of commercially available capacitors is commonly measured in microfarads ( $\mu\text{F}$ ; one microfarad is  $10^{-6}$  of a Farad) or pico-farads ( $\text{pF}$ ; one picofarad is  $10^{-12}$  of a Farad). Large capacitors are available, but are relatively infrequently used. These are generally called “super-capacitors” or “ultra-capacitors”

and are available in capacitances up to tens of Farads. For most applications, however, using one would be comparable to buying a car with a 1000 gallon gas tank.

Several approaches are used for labeling a capacitor with its capacitance value. Large capacitors often have their value printed plainly on them, such as “10 $\mu$ F” (for 10 microfarads). Smaller capacitors, appearing as small disks or wafers, often have their values printed on them in an encoded manner. For these capacitors, a three-digit number indicates the capacitor value in pico-farads. The first two digits provides the “base” number, and the third digit provides an exponent of 10 (so, for example, “104” printed on a capacitor indicates a capacitance value of 10 x 10<sup>4</sup> or 100000 pF). Occasionally, a capacitor will only show a two-digit number, in which case that number is simply the capacitor value in pF. (For completeness, if a capacitor shows a three-digit number and the third digit is 8 or 9, then the first two digits are multiplied by .01 and .1 respectively).

Capacitors are generally classified according to the dielectric material used. Common capacitor types include mica, ceramic, Mylar, paper, Teflon and polystyrene. An important class of capacitors which require special mention are *electrolytic* capacitors. Electrolytic capacitors have relatively large capacitances relative to other types of capacitors of similar size. However, some care must be exercised when using electrolytic capacitors – they are *polarized* and must be connected to a circuit with the correct polarity. The positive lead of the capacitor must be connected to the positive lead of the circuit. Connecting the positive lead of the capacitor to the negative lead of a circuit can result in unwanted current “leakage” through the capacitor or, in extreme cases, destroy the capacitor. Polarized capacitors either have a dark stripe near the pin that must be kept at the higher voltage, or a “-” near the pin that must be kept at a lower voltage.

Practical capacitors, unlike ideal capacitors, will dissipate some power. This power loss is primarily due to *leakage currents*. These currents are due to the fact that real dielectric materials are not perfect insulators – some small current will tend to flow through them. The overall effect is comparable to placing a high resistance in parallel with an ideal capacitor, as shown in Fig. 6.27. Different types of capacitors have different leakage currents. Mica capacitors tend to have low leakage currents, the leakage currents of ceramic capacitors vary according to the type of capacitor, and electrolytic capacitors have high leakage currents.

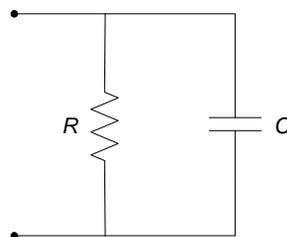


Figure 6.27. Model of practical capacitor including leakage current path.

## Section Summary

- Capacitors store electrical energy. This energy is stored in an electric field between two conductive elements, separated by an insulating material.
- Capacitor energy storage is dependent upon the voltage across the capacitor, if the capacitor voltage is known, the energy in the capacitor is known.
- The voltage-current relationship for a capacitor is:

$$i(t) = C \frac{dv(t)}{dt}$$

Where  $C$  is the capacitance of the capacitor. Units of capacitance are Farads (abbreviated F). The capacitance of a capacitor, very roughly speaking, gives an indication of how much energy it can store

- The above voltage-current relation results in the following important properties of capacitors:
  - If the capacitor voltage is constant, the current through the capacitor is zero. Thus, if the capacitor voltage is constant, the capacitor can be modeled as an open circuit.
  - Changing the capacitor voltage instantaneously requires infinite power. Thus (for now, anyway) we will assume that capacitors cannot instantaneously change their voltage.
- Capacitors placed in series or parallel with one another can be modeled as a single equivalent capacitance. Thus, capacitors in series or in parallel are not “independent” energy storage elements.

### 6.3 Exercises

1. Determine the maximum and minimum capacitances that can be obtained from four  $1\mu\text{F}$  capacitors. Sketch the circuit schematics that provide these capacitances.
2. Determine voltage divider relationships to provide  $v_1$  and  $v_2$  for the two uncharged series capacitors shown below. Use your result to determine  $v_2$  if  $C_1=C_2=10\mu\text{F}$ .

## 6.4 Inductors

We continue our study of energy storage elements with a discussion of *inductors*. Inductors, like resistors and capacitors, are passive two-terminal circuit elements. That is, no external power supply is necessary to make them function. Inductors commonly consist of a conductive wire wrapped around a core material; inductors store energy in the form of a magnetic field set up around the current-carrying wire.

In this section, we describe physical properties of inductors and provide a mathematical model for an ideal inductor. Using this ideal inductor model, we will develop mathematical relationships for the energy stored in an inductor and governing relations for series and parallel connections of inductors. The section concludes with a brief discussion of practical (non-ideal) inductors.

### 6.4.1 Inductors

Passing a current through a conductive wire will create a *magnetic field* around the wire. This magnetic field is generally thought of in terms of as forming closed loops of *magnetic flux* around the current-carrying element. This physical process is used to create *inductors*. Figure 6.28 illustrates a common type of inductor, consisting of a coiled wire wrapped around a core material. Passing a current through the conducting wire sets up lines of magnetic flux, as shown in Fig. 6.28; the inductor stores energy in this magnetic field. The *inductance* of the inductor is a quantity, which tells us how much energy can be stored by the inductor. Higher inductance means that the inductor can store more energy. Inductance has units of *Henrys*, abbreviated H.

The amount of inductance an inductor has is governed by the geometry of the inductor and the properties of the core material. These effects can be complex; rather than attempt a comprehensive discussion of these effects, we will simply claim that, in general, inductance is dependent upon the following parameters:

- The number of times the wire is wrapped around the core. More coils of wire results in a higher inductance.
- The core material’s type and shape. Core materials are commonly ferromagnetic materials, since they result in higher magnetic flux and correspondingly higher energy storage. Air, however, is a fairly commonly used core material – presumably because of its ready availability.
- The spacing between turns of the wire around the core.

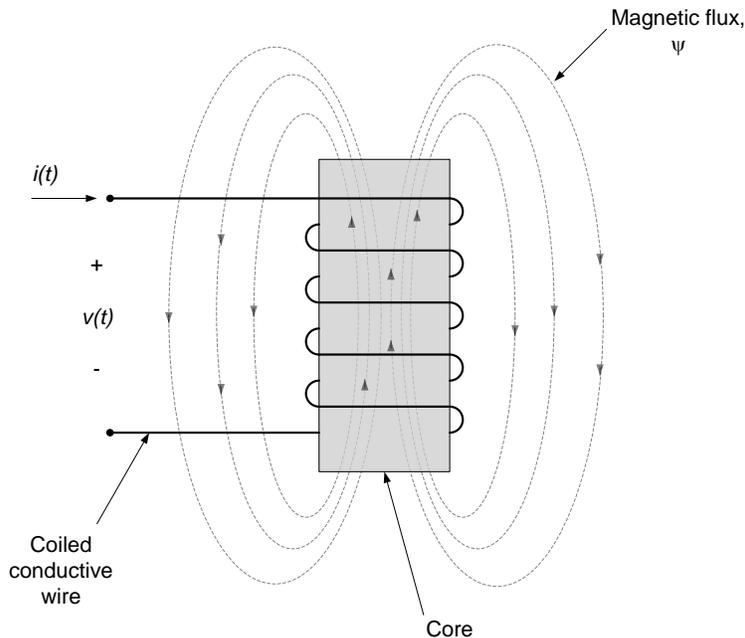


Figure 6.28. Wire-wrapped inductor with applied current through conductive wire.

We will denote the total magnetic flux created by the inductor by  $\psi$ , as shown in Fig. 6.28. For a linear inductor, the flux is proportional to the current passing through the wound wires. The constant of proportionality is the inductance,  $L$ :

$$\psi(t) = Li(t) \tag{Eq. 6.26}$$

Voltage is the time rate of change of magnetic flux, so:

$$v(t) = \frac{d\psi(t)}{dt} \tag{Eq. 6.27}$$

Combining equations (6.26) and (6.27) results in the voltage-current relationship for an ideal inductor:

$$v(t) = L \frac{di(t)}{dt} \tag{Eq. 6.28}$$

The circuit symbol for an inductor is shown in Fig. 6.29, along with the sign conventions for the voltage-current relationship of equation (6.28). The passive sign convention is used in the voltage-current relationship, so positive current is assumed to enter the terminal with positive voltage polarity.

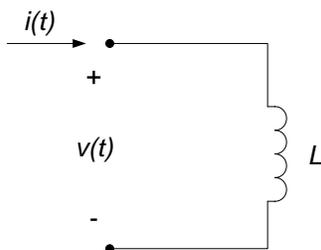


Figure 6.29. Inductor circuit symbol and voltage-current sign convention.

Integrating both sides of equation (6.28) results in the following form for the inductor’s voltage-current relationship:

$$i(t) = \frac{1}{L} \int_{t_0}^t v(\xi) d\xi + i(t_0)$$

Eq. 6.29

In equation (6.29),  $i(t_0)$  is a known current at some initial time  $t_0$  and  $\xi$  is used as a dummy variable of integration to emphasize that the only “ $t$ ” which survives the integration process is the upper limit of the integral.

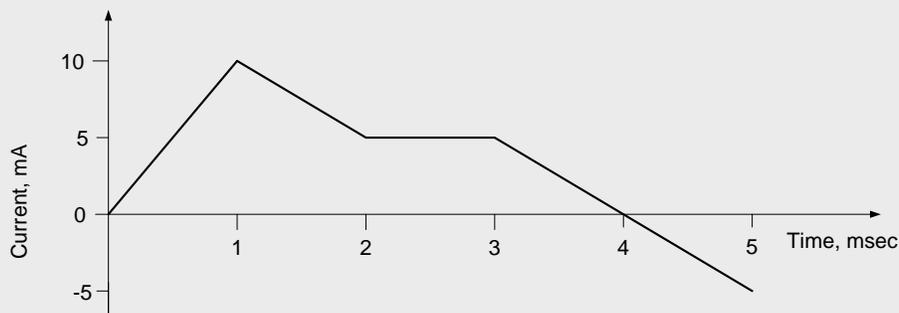
### Important Result

The voltage-current relationship for an ideal inductor can be stated in either differential or integral form, as follows:

- $v(t) = L \frac{di(t)}{dt}$
- $i(t) = \frac{1}{L} \int_{t_0}^t v(\xi) d\xi + i(t_0)$

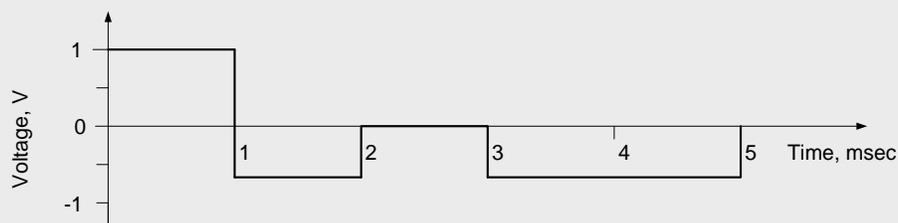
### Example 6.6

A circuit contains a 100mH inductor. The current as a function of time through the inductor is measured and shown below. Plot the voltage across the inductor as a function of time.

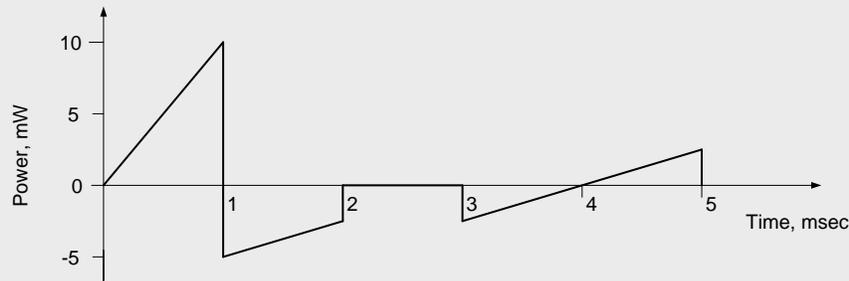


- In the time range  $0 < t < 1\text{ms}$ , the rate of change of current is 10 A/sec. Thus, from equation (3), the voltage is  $v(t) = (0.1\text{H})(10\text{A} / \text{s}) = 1\text{V}$ .
- In the time range  $1\text{ms} < t < 2\text{ms}$ , the rate of change of current is -5A/sec. The voltage is -0.5V.
- In the time range  $2\text{ms} < t < 3\text{ms}$ , the current is constant and there is no voltage across the inductor.
- In the time range  $3\text{ms} < t < 5\text{ms}$ , the rate of change of current is -5A/sec. The voltage is -0.5V

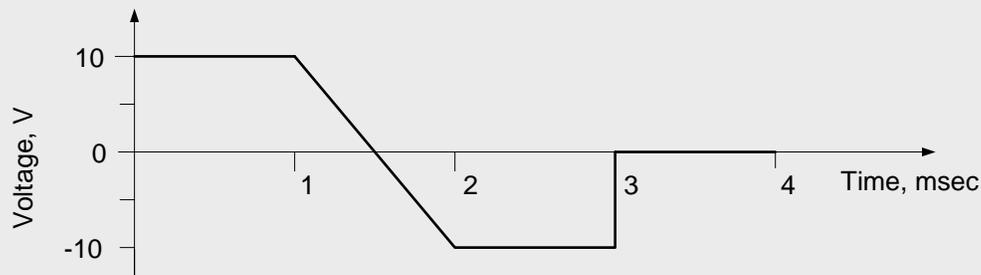
The plot of voltage vs. time is shown below:



Power is the product of voltage and current. If the signs of voltage and current are the same according to the passive sign convention, the circuit element absorbs power. If the signs of voltage and current are not the same, the circuit element generates power. From the above voltage and current curves, the inductor is absorbing power from the circuit during the times  $0 < t < 1\text{ms}$  and  $4\text{ms} < t < 5\text{ms}$ . The inductor returns power to the circuit during the times  $1\text{ms} < t < 2\text{ms}$  and  $3\text{ms} < t < 4\text{ms}$ .

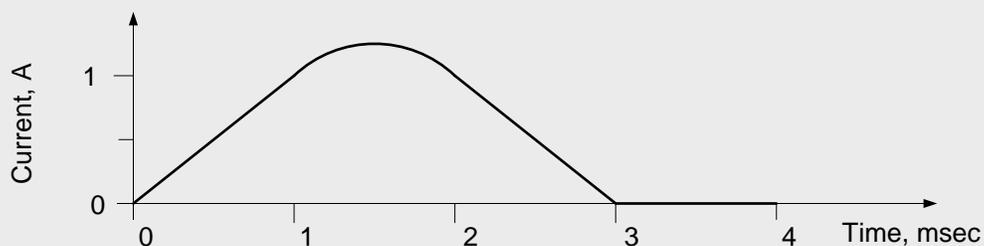


Example 2: If the voltage as a function of time across an inductor with inductance  $L = 10 \text{ mH}$  is as shown below, determine the current as a function of time through the capacitor. Assume that the current through the capacitor is  $0 \text{ A}$  at time  $t=0$ .



- At time  $t=0$ , the current is given to be  $0 \text{ A}$ .
- In the time period  $0 < t < 1 \text{ msec}$ , the voltage is constant and positive so the current will increase linearly. The total current change during this time period is the area under the voltage curve, divided by the inductance:  $\frac{1}{0.01} (10 \text{ V})(1 \times 10^{-3} \text{ s}) = 1 \text{ A}$
- In the time period  $1 < t < 2 \text{ msec}$ , the voltage is decreasing linearly. The current during this time period is a quadratic curve, concave downward. The maximum value of current is  $1.25 \text{ A}$ , at  $t=1.5 \text{ msec}$ . The current at the end of this time period is  $1 \text{ A}$ .
- In the time period  $2 < t < 3 \text{ msec}$ , the voltage is constant at  $-10 \text{ V}$ . The current change during this time period is the area under the voltage curve, divided by the inductance:  $\frac{1}{0.01} (-10 \text{ V})(1 \times 10^{-3} \text{ s}) = -1 \text{ A}$ . The total current at  $t=3 \text{ msec}$  is, then,  $1 \text{ A} - 1 \text{ A} = 0 \text{ A}$ .
- In the time period  $3 < t < 4 \text{ msec}$ , the voltage is zero. The integral of zero over any time period is zero, so there is no change in current during this time range and the current remains constant at  $0 \text{ A}$ .

A plot of the current through the inductor as a function of time is shown below.



It is often useful, when analyzing circuits containing inductors, to examine the circuit's response to constant operating conditions and to instantaneous changes in operating condition. We examine the inductor's response to each of these operating conditions below:

- Inductor response to constant current:
  - If the current through the inductor is constant, equation (6.28) indicates that the voltage across the inductor is zero. Thus, if the current through the inductor is constant, the inductor is equivalent to a short circuit.
- Inductor response to instantaneous current changes:
  - If the current through the inductor changes instantaneously, the rate of change of current is infinite. Thus, by equation (6.28), if we wish to change the current through an inductor instantaneously, we must supply infinite voltage to the inductor. This implies that infinite power is available, which is not physically possible. Thus, in any practical circuit, the current through an inductor cannot change instantaneously.
  - Any circuit that allows an instantaneous change in the current through an ideal inductor is not physically realizable. We may sometimes assume, for mathematical convenience, that an ideal inductor's current changes suddenly; however, it must be emphasized that this assumption requires an underlying assumption that infinite power is available and is thus not an allowable operating condition in any physical circuit.

### Important Inductor Properties

- Inductors can be replaced by short-circuits, under circumstances when all operating conditions are constant.
- Currents through inductors cannot change instantaneously. No such requirement is placed on voltages.

## 6.4.2 Energy Storage

The instantaneous power dissipated by an electrical circuit element is the product of the voltage and current:

$$p(t) = v(t) \cdot i(t) \quad \text{Eq. 6.30}$$

Using equation (6.28) to write the voltage in equation (6.30) in terms of the inductor's current:

$$p(t) = L \cdot i(t) \frac{di(t)}{dt} \quad \text{Eq. 6.31}$$

As was previously done for capacitors, we integrate the power with respect to time to get the energy stored in the inductor:

$$W_L(t) = \sum_{-\infty}^t Li(\xi) \frac{di(\xi)}{dt} dt$$

Which, after some manipulation (comparable to the approach taken when we calculated energy storage in capacitors), results in the following expression for the energy stored in an inductor:

$$W_L(t) = \frac{1}{2} Li^2(t) \quad \text{Eq. 6.32}$$

## 6.4.3 Inductors in Series

Consider the series connection of  $N$  inductors shown in Fig. 6.30.

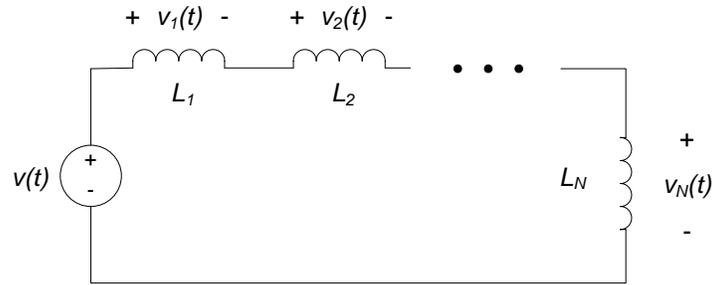


Figure 6.30. Series connection of N inductors.

Applying Kirchoff's voltage law around the loop results in:

$$v(t) = v_1(t) + v_2(t) + \dots + v_N(t) \tag{Eq. 6.33}$$

Using equation (6.28) to write the inductor voltage drops in terms of the current through the loop gives:

$$\begin{aligned} v(t) &= L_1 \frac{di(t)}{dt} + L_2 \frac{di(t)}{dt} + \dots + L_N \frac{di(t)}{dt} \\ &= (L_1 + L_2 + \dots + L_N) \frac{di(t)}{dt} \end{aligned}$$

Using summation notation results in:

$$v(t) = \left( \sum_{k=1}^N L_k \right) \frac{di(t)}{dt} \tag{Eq. 6.34}$$

This is the same equation that governs the circuit of Fig. 6.31, if:

$$L_{eq} = \sum_{k=1}^N L_k$$

Thus, the equivalent inductance of a series combination of inductors is simply the sum of the individual inductances. This result is analogous to the equations which provide the equivalent resistance of a series combination of resistors.

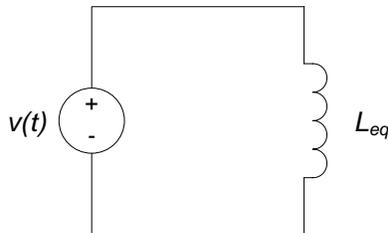


Figure 6.31. Equivalent circuit to Figure 3.

### 6.4.4 Inductors in Parallel

Consider the parallel combination of N inductors, as shown in Fig. 6.32.

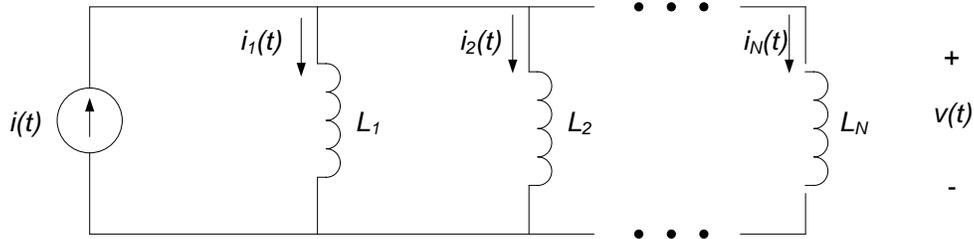


Figure 6.32. Parallel combination of  $N$  inductors.

Applying Kirchoff's current law at the upper node results in:

$$i(t) = i_1(t) + i_2(t) + \dots + i_N(t) \tag{Eq. 6.36}$$

Using equation (6.29) to write the inductor currents in terms of their voltage drops gives:

$$\begin{aligned} i(t) &= \left[ \frac{1}{L_1} \int_{t_0}^t v(\xi) d\xi + i_1(t_0) \right] + \left[ \frac{1}{L_2} \int_{t_0}^t v(\xi) d\xi + i_2(t_0) \right] + \dots + \left[ \frac{1}{L_N} \int_{t_0}^t v(\xi) d\xi + i_N(t_0) \right] \\ &= \left[ \frac{1}{L_1} \int_{t_0}^t v(\xi) d\xi + \frac{1}{L_2} \int_{t_0}^t v(\xi) d\xi + \dots + \frac{1}{L_N} \int_{t_0}^t v(\xi) d\xi \right] + [i_1(t_0) + i_2(t_0) + \dots + i_N(t_0)] \\ &= \left( \frac{1}{L_1} + \frac{1}{L_2} + \dots + \frac{1}{L_N} \right) \int_{t_0}^t v(\xi) d\xi + i(t_0) \end{aligned}$$

This can be re-written using summation notation as:

$$i(t) = \left( \sum_{k=1}^N \frac{1}{L_k} \right) \int_{t_0}^t v(\xi) d\xi + i(t_0) \tag{Eq. 6.37}$$

This is the same equation that governs the circuit of Fig. 6.31, if:

$$\frac{1}{L_{eq}} = \frac{L_1 L_2}{L_1 + L_2} \tag{Eq. 6.39}$$

Equations (6.38) and (6.39) are analogous to the equations which provide the equivalent resistance of parallel combinations of resistors.

### Summary: Series and Parallel Inductors

- The equivalent inductance of a series combination of inductors  $L_1, L_2, \dots, L_N$  is governed by a relation which is analogous to that providing the equivalent resistance of a series combination of resistors:

$$L_{eq} = \sum_{k=1}^N L_k$$

- The equivalent inductance of a parallel combination of inductors  $L_1, L_2, \dots, L_N$  is governed by a relation which is analogous to that providing the equivalent resistance of a parallel combination of resistors:

$$\frac{1}{L_{eq}} = \sum_{k=1}^N \frac{1}{L_k}$$

## 6.5 Practical Inductors

Most commercially available inductors are manufactured by winding wire in various coil configurations around a core. Cores can be a variety of shapes; Fig. 6.28 in this chapter shows a core, which is basically a cylindrical bar. Toroidal cores are also fairly common – a closely wound toroidal core has the advantage that the magnetic field is confined nearly entirely to the space inside the winding.

Inductors are available with values from less than 1 micro-Henry ( $1\mu H = 10^{-6}$  Henries) up to tens of Henries. A 1H inductor is very large; inductances of most commercially available inductors are measured in millihenries ( $1mH = 10^{-3}$  Henries) or microhenries. Larger inductors are generally used for low-frequency applications (in which the signals vary slowly with time).

Attempts at creating inductors in integrated-circuit form have been largely unsuccessful; therefore many circuits that are implemented as integrated circuits do not include inductors. Inclusion of inductance in the analysis stage of these circuits may however, be important. Since any current-carrying conductor will create a magnetic field, the *stray inductance* of supposedly non-inductive circuit elements can become an important consideration in the analysis and design of a circuit.

Practical inductors, unlike the ideal inductors discussed in this chapter, dissipate power. An equivalent circuit model for a practical inductor is generally created by placing a resistance in series with an ideal inductor, as shown in Fig. 6.33.

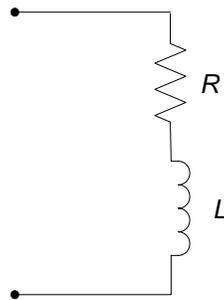


Figure 6.33. Equivalent circuit model for a practical inductor.

### Section Summary

- Inductors store magnetic energy. This energy is stored in a magnetic field (typically) generated by a coiled wire wrapped around a core material.
- Inductor energy storage is dependent upon the current through the inductor, if the inductor current is known, the energy in the inductor is known.
- The voltage-current relationship for an inductor is:

$$v(t) = L \frac{di(t)}{dt}$$

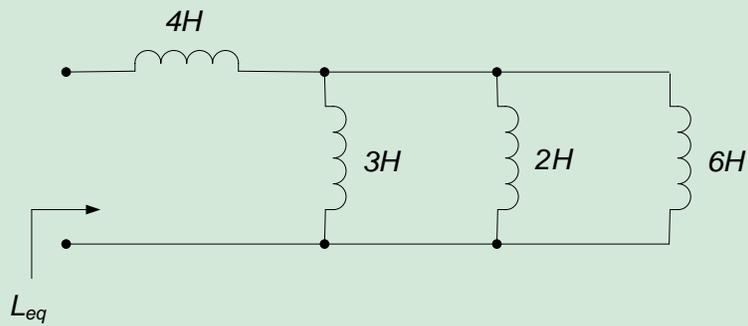
Where L is the inductance of the inductor. Units of inductance are Henries (abbreviated H). The inductance of an inductor, very roughly speaking, gives an indication of how much energy it can store.

- The above voltage-current relation results in the following important properties of inductors:
  - If the inductor current is constant, the voltage across the inductor is zero. Thus, if the inductor current is constant, the inductor can be modeled as a short circuit.

- Changing the inductor current instantaneously requires infinite power. Thus (for now, anyway) we will assume that inductors cannot instantaneously change their current.
- Inductors placed in series or parallel with one another can be modeled as a single equivalent inductance. Thus, inductors in series or in parallel are not “independent” energy storage elements.

## 6.4 Exercises

1. Determine the equivalent inductance of the network below:



## Real Analog Chapter 6: Lab Projects

### 6.2.1: Time-varying Signals

This assignment will focus on using an arbitrary waveform generator to generate time-varying signals and using an oscilloscope to measure time varying signals.

In chapter 6 of the text book, we deal analytically only with step functions and exponential functions. This lab will, however introduce us to a larger class of time-varying waveforms.

The ability to apply and measure time varying signals will be crucial throughout the remainder of your career. It is strongly recommended that you not only complete the specific steps outlined in this assignment, but that you spend some additional time “playing with” the tools we introduce in this assignment – it is guaranteed to be time well spent!

#### Before beginning this lab, you should be able to:

- Define a step function.
- State Ohm’s law for time-varying signals

#### After completing this lab, you should be able to:

- Use a switch to create a step function
- Use the Analog Discovery 2 waveform generator to apply square, triangular, and sinusoidal waveforms
- Use the Analog Discovery 2 oscilloscope to measure and display time-varying waveforms

#### This lab exercise requires:

- Analog Discovery 2 module
- Digilent Analog Parts Kit

#### Symbol Key:

-  Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
-  Analysis; include principle results of analysis in laboratory report.
-  Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
-  Record data in your lab notebook.

#### General Discussion:

Once we begin to deal in earnest with systems which include energy storage elements, it will be crucial apply time-varying power to our electrical circuits and measure the circuits’ responses as functions of time. This lab introduces the concepts necessary for application, measurement, and interpretation of time-varying signals.

Since we have not yet been introduced to dynamic systems, the electrical circuit of interest in this assignment will be the voltage divider shown in Figure 1.

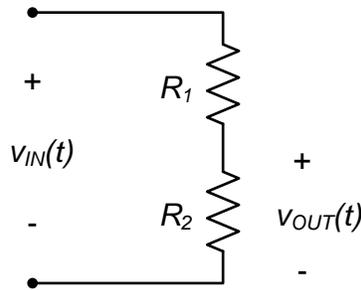


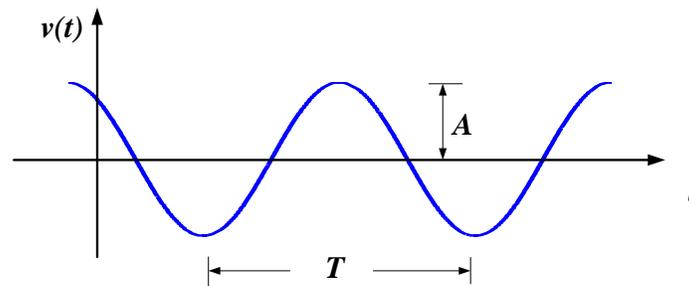
Figure 1. Voltage divider circuit.

In Figure 1, the output voltage,  $v_{OUT}(t)$  is related to the input voltage  $v_{IN}(t)$  via the voltage divider relation:

$$v_{OUT}(t) = v_{IN}(t) \frac{R_2}{R_1 + R_2} \quad \text{Eq. 1}$$

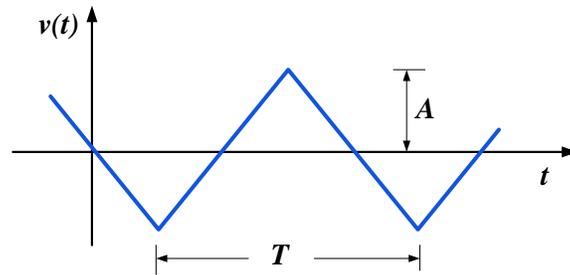
Notice that the relationship between  $v_{IN}(t)$  and  $v_{OUT}(t)$  is *algebraic* – the value of  $v_{OUT}$  at a particular time depends only upon the value of  $v_{IN}$  at that same time.

In order to familiarize ourselves with the fundamentals of applying and measuring time-varying signals, we will restrict ourselves to some of the most common signals encountered in engineering applications: sinusoidal waves, square waves, and triangular waves. The basic shapes of these signals are shown in Fig. 2. The signals of Fig. 2 are all *periodic* signals – that is, they repeat themselves at regular intervals. This interval is called the *period* (commonly denoted mathematically as  $T$ ). The period of each of the signals of interest to us is indicated on Fig. 2. The other primary attribute of the signals we will be dealing with is their amplitude (which we will denote as  $A$ ). The amplitude of the signal is essentially the maximum (and minimum) value that the signal achieves<sup>9</sup>.

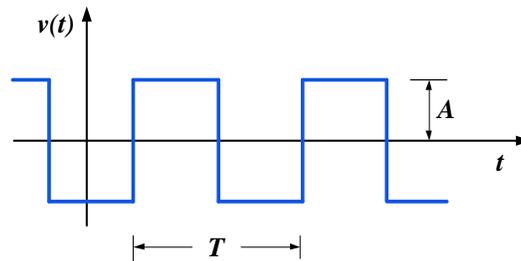


(a) Sinusoidal wave.

<sup>9</sup> For now, our signals will be symmetric with respect to the time axis. That is, their average value (also called the *offset*) will be zero. For the signals of immediate interest to us, this means that their minimum value will be the negative of their maximum value. Later labs will explore the effects of a non-zero offset to the signal, and signals which are not symmetric with respect to the time axis.



(b) Triangular wave



(c.) Square wave

Figure 2. Basic signal shapes.

Although we have used the signal's period as a fundamental parameter defining the signal, it is more common for electrical instruments to use the *frequency* of the signal as a defining characteristic. The frequency provides essentially the same information as the period; the frequency is just the inverse of the period:

$$f = \frac{1}{T} \quad \text{Eq. 2}$$

As defined in equation (2), the units of frequency are in Hertz (abbreviated Hz) or cycles per second. Sinusoidal signals, however, are more accurately defined mathematically in terms of their *radian frequency*, denoted as  $\omega$ . Since there are  $2\pi$  radians in one cycle, the conversion between frequency and radian frequency is:

$$\omega = 2\pi f = \frac{2\pi}{T} \quad \text{Eq. 3}$$

Mathematically, the sinusoidal wave of Fig. 2(a) can be represented as:

$$v(t) = A\cos(\omega t + \theta) = A\cos(2\pi f t + \theta) \quad \text{Eq. 4}$$

Where  $\theta$  is the *phase angle* of the signal; it translates the sinusoid in time. We will concern ourselves with phase later in the course.

### Pre-lab:

In the circuit of Fig. 1, if  $R_1 = R_2$ , overlay sketches using the input and output voltages ( $v_{IN}(t)$  and  $v_{OUT}(t)$ ) for the following cases:

**ANALYSIS**

(a)  $v_{IN}(t)$  is a sinusoidal wave with amplitude  $A$  and period  $T$ .

**ANALYSIS**

(b)  $v_{IN}(t)$  is a triangular wave with amplitude  $A$  and period  $T$ .

**ANALYSIS**

(c)  $v_{IN}(t)$  is a square wave with amplitude  $A$  and period  $T$ .

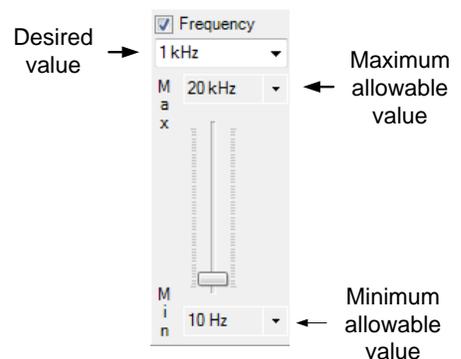
Label the amplitude and period of both the input and output waveforms on your sketch. These values may be functions of  $A$ ,  $T$ ,  $R_1$  and  $R_2$ .

### Lab Procedures:

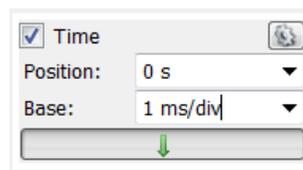
- (a) Test the response of the circuit to a sinusoidal input voltage with 2kHz frequency and 2V amplitude. Details are below:
- Set  $v_{IN}(t)$  in the circuit of Fig. 1 to be a sinusoidal voltage with amplitude 2V and frequency 1kHz across the voltage divider. The average value of the sinusoid should be zero volts. To do this, open the *WaveGen* instrument in the waveforms file. Click on the **Basic** drop-down option (if it is not already selected). There should be a series of icons in a column below this option, indicating the shape of the associated waveform. Click on the  icon to select a sinusoidal waveform. Choose 1kHz as the frequency (you can choose the desired frequency by selecting it from the drop-down menu, typing the desired value in the text box, or using the slider bar) and 2V as the amplitude<sup>10</sup>. The plot window on the waveform generator instrument will display one period the waveform you have set. Use this plot window to double check that your signal has the correct frequency and amplitude.

### Note on selecting parameters:

When choosing parameters describing signals (e.g. frequency, amplitude, offset, and symmetry) the allowable values are limited to the range specified by the values above and below the slider bar, as indicated on the figure to the right for the frequency parameter. When selecting a value, the desired value must be between the maximum and minimum values shown. If you want a value outside the displayed range, simply re-set the range using the appropriate drop-down menus. If the waveform generator will not let you set a desired value, be sure to check that the desired value is within the allowable range.



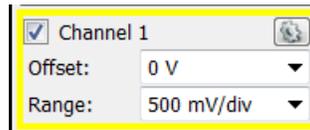
- Use the oscilloscope to display the voltages  $v_{IN}(t)$  and  $v_{OUT}(t)$  of Fig. 1. To do this, open the *Scope* instrument. Set the horizontal scale (or the time axis scale) to be 1ms/div. Horizontal axis settings are set in the time axis settings box on the oscilloscope window; this box and the desired settings for this lab are shown below:



Set the vertical axis settings on both channel 1 and channel 2 (C1 and C2) to 500mv/div. Vertical axis settings are set in the channel axis settings boxes on the oscilloscope window;

<sup>10</sup> The offset should be zero, the symmetry 50%, and the phase 0 degrees. These are the default values, and should not need to be re-set.

the settings box for channel 1 and its desired settings are shown below. Use the same settings for channel 2.

**DATA**

Click on “Run” to acquire and display the data. Record an image of the oscilloscope main time window to a file for later documentation.

- iii. From the time plots displayed in the oscilloscope window, determine the period and amplitude of  $v_{IN}(t)$  and  $v_{OUT}(t)$ . From your measured period, calculate the signal’s frequency in Hertz. Create a table, showing the expected amplitude and frequency of  $v_{IN}(t)$  and  $v_{OUT}(t)$  and your measured amplitude and frequency of  $v_{IN}(t)$  and  $v_{OUT}(t)$ .
- iv. Click on the “Measure” button on the oscilloscope window to open a measurements window. Use the measurement window to measure the amplitude, period, and frequency of  $v_{IN}(t)$  and  $v_{OUT}(t)$ . Record the image of the oscilloscope window, showing the waveforms and their measured amplitudes, periods, and frequencies<sup>11</sup>. Comment on the agreement between the oscilloscope’s measurements and the measurements you made in part iii above.

**DATA****ANALYSIS****DEMO**

- v. Demonstrate operation of your circuit to the Teaching Assistant. Have the TA initial the appropriate page(s) of your lab notebook and the lab checklist.
- vi. Vary the amplitude and frequency of the sinusoidal waveform using the waveform generator. Change the horizontal and vertical axis scales in the oscilloscope. Verify that the changes result in data that agree with your expectations. Familiarizing yourself with these instruments now will be rewarded in later experiments – you can only interpret the results of future experiments if you are comfortable with measuring the data upon which the results depend!

**ANALYSIS**

- (b) Test the response of the circuit to a triangular input voltage with 1kHz frequency and 3V amplitude.
  - i. Perform all the steps you did above for the sinusoidal input.
  - ii. Demonstrate operation of your circuit to the Teaching Assistant. Have the TA initial the appropriate page(s) of your lab notebook and the lab checklist.
- (c) Test the response of the circuit to a square wave input voltage with 500Hz frequency and 2.5V amplitude.
  - i. Perform all the steps you did above for the sinusoidal input.
  - ii. Demonstrate operation of your circuit to the Teaching Assistant. Have the TA initial the appropriate page(s) of your lab notebook and the lab checklist.

**DEMO****DEMO**

<sup>11</sup> Holding down the “Alt” key and pressing “Print Screen” (commonly labeled as “PrtScn” or “PrtSc” on computer keyboards) will copy the currently active window to the clipboard. You can then paste this image to a document. The “Export” button on the oscilloscope instrument allows you to copy an image of the main time window to the clipboard or save it to a file in a variety of formats. This option will not, however, display the measurement window – if you use this approach, you will want to record the measured values elsewhere.

## Real Analog Chapter 6: Lab Worksheets

### 6.2.1: Time-varying Signals (40 points total)

1. Attach to this worksheet the input and output voltage sketches you created in the pre-lab for sinusoidal, triangular, and square waves. (7 pts)
2. Attach to this worksheet an image of the oscilloscope window, showing the sinusoidal waveforms and their measured amplitudes, periods, and frequencies. In the space below, provide the amplitudes, periods, and frequencies determined directly from the time plot in the oscilloscope window. Comment on the agreement between the two sets of data. (8 pts)
3. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuit's operation for sinusoidal inputs. (3 pts)

**TA Initials:** \_\_\_\_\_

4. Attach to this worksheet an image of the oscilloscope window, showing the triangular waveforms and their measured amplitudes, periods, and frequencies. In the space below, provide the amplitudes, periods, and frequencies determined directly from the time plot in the oscilloscope window. Comment on the agreement between the two sets of data. (8 pts)
5. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuit's operation for triangular inputs. (3 pts)

**TA Initials:** \_\_\_\_\_

6. Attach to this worksheet an image of the oscilloscope window, showing the square waveforms and their measured amplitudes, periods, and frequencies. In the space below, provide the amplitudes, periods, and frequencies determined directly from the time plot in the oscilloscope window. Comment on the agreement between the two sets of data. (8 pts)
7. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuit's operation for square wave inputs. (3 pts)

**TA Initials:** \_\_\_\_\_

## Real Analog Chapter 6: Lab Projects

### 6.3.1: Capacitor Voltage-current Relations

In this assignment, we will measure the relationship between the voltage difference across a capacitor and the current passing through it. We will apply several types of time-varying signals to a series combination of a resistor and a capacitor. The voltage difference across the resistor, in conjunction with Ohm's law, will provide an estimate of the current through the capacitor. This current can be related to the voltage difference across the capacitor.

The results of our voltage-current measurements will be compared to analytical expectations.

#### Before beginning this lab, you should be able to:

- State voltage-current relationships for capacitors in both differential and integral form
- Apply the capacitor voltage-current relations to calculate a capacitor's voltage from its current and vice-versa
- Use the Analog Discovery 2's arbitrary waveform generator and oscilloscope to apply and measure time-varying waveforms (Lab 6.2.1)

#### After completing this lab, you should be able to:

- Use the Analog Discovery 2 oscilloscope's math function to calculate the current through a known resistor from the measured voltage difference.
- Verify a capacitor's voltage-current relations using measured data

#### This lab exercise requires:

- Analog Discovery 2 module
- Digilent Analog Parts Kit

#### Symbol Key:

##### DEMO

Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.

##### ANALYSIS

Analysis; include principle results of analysis in laboratory report.

##### SIM

Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.

##### DATA

Record data in your lab notebook.

#### General Discussion:

We will use the circuit of Fig. 1 in this lab assignment. Both the voltage difference across the capacitor and the resistor ( $v_C(t)$  and  $v_R(t)$ ) will be measured. From this data, we can readily compare the voltage across the capacitor with the current through the capacitor. Since the voltage across the resistor is measured, we can readily infer the current through the resistor via Ohm's law:

$$i_R(t) = \frac{v_R(t)}{R} \quad \text{Eq. 1}$$

The resistor and capacitor are in series, so the current through the capacitor is the same as the current through the resistor, so:

$$i_C(t) = \frac{v_R(t)}{R} \tag{Eq. 2}$$

Since we are also measuring the voltage difference across the capacitor,  $v_C(t)$ , we can readily compare these parameters with our expectations based on our mathematical models of the capacitor voltage-current relationships.

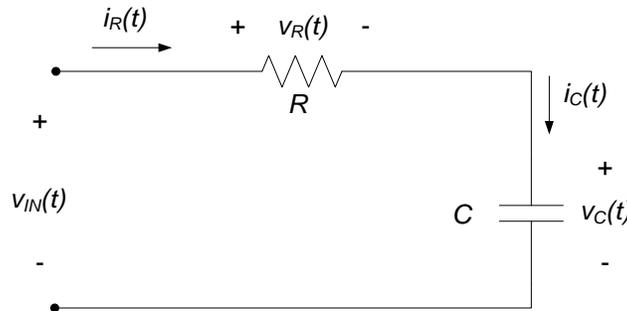


Figure 1. Series RC circuit.

**Pre-lab:**

For the circuit of Fig. 1, use the inductor voltage-current relations to overlay sketches of the capacitor voltage and the capacitor current ( $v_C(t)$  and  $i_C(t)$ ) if the capacitor voltage is:

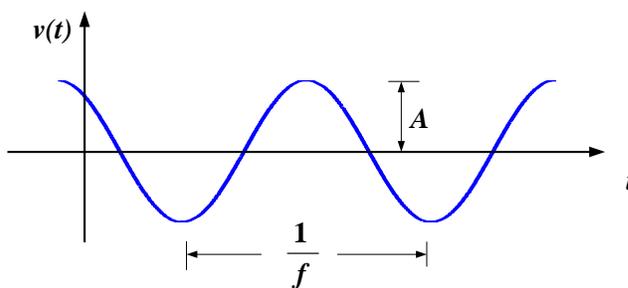
**ANALYSIS**

(a) A sinusoidal wave,  $v(t)$ , with frequency ( $f$ ) and amplitude ( $A$ ) as shown in Fig. 2 (a)

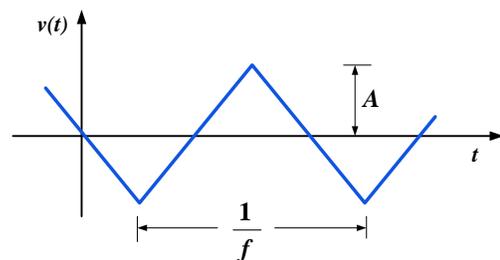
**ANALYSIS**

(b) A triangular wave,  $v(t)$ , with frequency ( $f$ ) and amplitude ( $A$ ) as shown in Fig. 2 (b).

Label your sketch to show the amplitude and period of the capacitor current for both of the above cases. Your results may be dependent up on the parameters  $A$ ,  $f$ ,  $R$ , and  $C$ . Be sure that your sketches of voltage and current share the same time axis!



(a) Sinusoidal waveform



(b) Triangular waveform

**Lab Procedures:**

Construct the circuit of Fig. 1, using  $R = 100\Omega$  and  $C = 1\mu F$ . Use channel 1 of your oscilloscope to measure the resistor voltage difference, and channel 2 of your oscilloscope to measure the capacitor voltage difference. Use channel 1 of your waveform generator (W1) to apply the voltage  $v_{in}(t)$  in Fig. 1. Set up a math channel to calculate the current through the capacitor per equation (2) in the pre-

lab<sup>12</sup>. Set the oscilloscope measurements to provide at least the amplitude of each of the three displayed waveforms.

**DATA**

1. Apply a sinusoidal input voltage with frequency = 1kHz, amplitude = 2V, and offset = 0V to the circuit of Fig. 1. Use your oscilloscope to display the data listed above (waveforms corresponding to C1, C2, and M1; measurement window displaying amplitudes of C1, C2, and M1). Record the image of the oscilloscope window, showing the waveforms and their measured amplitudes.

**DATA**

2. Apply a sinusoidal input voltage with frequency = 2 kHz, amplitude = 2V, and offset = 0V to the circuit of Fig. 1. Use your oscilloscope to display the data listed above (waveforms corresponding to C1, C2, and M1; measurement window displaying amplitudes of C1, C2, and M1). Record the image of the oscilloscope window, showing the waveforms and their measured amplitudes.

**DATA****DEMO**

3. Apply a triangular input voltage with frequency = 100 Hz, amplitude = 4V, and offset = 0V to the circuit of Fig. 1. Use your oscilloscope to display the data listed above (waveforms corresponding to C1, C2, and M1; measurement window displaying amplitudes of C1, C2, and M1). Record the image of the oscilloscope window, showing the waveforms and their measured amplitudes.

4. Demonstrate operation of your circuit to the Teaching Assistant. Have the TA initial the appropriate page(s) of your lab notebook and the lab checklist.

### Post-lab Exercises:

**ANALYSIS**

For the three cases in the lab procedures (1kHz sinusoid, 2kHz sinusoid, 100Hz triangular wave), use your pre-lab results to sketch the expected capacitor current waveforms corresponding to the capacitor voltage waveforms you measured in the lab procedures. Comment briefly on the agreement between the measured and expected capacitor currents for each of these cases. In your comments, be sure to include a quantitative comparison (including percent difference) between the expected and measured amplitudes of the capacitor

---

<sup>12</sup> Detailed instructions for doing this are provided in Appendix A.

## Real Analog Chapter 6: Lab Worksheets

### 6.3.1: Capacitor Voltage-current Relations (35 points total)

1. Attach to this worksheet the sketches of the capacitor voltage and current for both sinusoidal and triangular inputs. (6 pts)
2. Attach to this worksheet an image of the oscilloscope window, showing the capacitor voltage and current waveforms and the measured amplitudes of the waveforms for a 1kHz sinusoidal input. (8 pts)
3. Attach to this worksheet an image of the oscilloscope window, showing the capacitor voltage and current waveforms and the measured amplitudes of the waveforms for a 2kHz sinusoidal input. (8 pts)
4. Attach to this worksheet an image of the oscilloscope window, showing the capacitor voltage and current waveforms and the measured amplitudes of the waveforms for a 100Hz triangular input. (8 pts)
5. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuit's operation for the triangular input. (5 pts)

**TA Initials:** \_\_\_\_\_

## Real Analog Chapter 6: Lab Projects

### 6.3.2: Leakage Currents and Electrolytic Capacitors

Voltage-current relationships for ideal capacitors do not always adequately explain measured capacitor behavior. In this assignment, we will focus on the effects of *leakage currents* on capacitor behavior. As we saw in our discussion of non-ideal capacitors in section 6.3 of the text, models for realistic capacitors often include a resistor in parallel with an ideal capacitor; this resistor allows us to model leakage currents, which explain – among other effects – the inability of a capacitor to hold a charge indefinitely, even if the capacitor terminals are open-circuited.

An important effect of leakage currents is in the case of *electrolytic* capacitors. These capacitors are attractive in many cases, since a relatively large capacitance can be provided in a small package. However, one must be aware, when using electrolytic capacitors, that their leakage currents can be significant and that they are not symmetric relative to the capacitor's polarity. Thus, reversing the polarity of the capacitor in a circuit can alter the behavior of the capacitor – this makes electrolytic capacitors undesirable in some applications (such as filtering) in which the behavior of the capacitor should be independent of the polarity of the voltage applied to the capacitor.

#### Before beginning this lab, you should be able to:

- State voltage-current relationships for capacitors in both differential and integral form
- Define the time constant of an exponential waveform
- Use the Analog Discovery 2 to apply and measure time-varying waveforms (Lab 6.2.1)

#### After completing this lab, you should be able to:

- Model a non-ideal capacitor as an ideal capacitor in parallel with a resistor
- Identify some effects of non-ideal capacitors
- Describe how the polarity of an electrolytic capacitor affects the capacitor's leakage current.

#### This lab exercise requires:

- Analog Discovery 2 module
- Digilent Analog Parts Kit

#### Symbol Key:

	Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
	Analysis; include principle results of analysis in laboratory report.
	Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
	Record data in your lab notebook.

#### General Discussion:

In large part, this lab will be concerned with *electrolytic* capacitors. Electrolytic capacitors are polarized – that is, one of their terminals is intended to always be at a higher voltage than the other. The terminal which is intended to be at the higher voltage is called the *anode*, while the terminal which is to be at the lower polarity is the *cathode*. A symbol for an electrolytic capacitor is shown in Fig. 1 – the cathode side is indicated as a curved line. Physically, electrolytic capacitors are readily identifiable: the lead connected to the anode is a longer wire than

that of the cathode, and (if the capacitor is physically large enough) a bar is printed on the cathode side of the capacitor<sup>13</sup>.

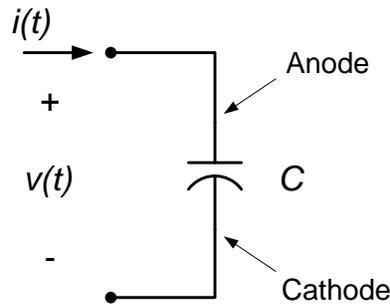


Figure 1. Electrolytic capacitor circuit symbol.

Electrolytic capacitors are desirable in that their capacitance can be large relative to their volume. However, they also have some undesirable qualities. Chief among these is that they can fail rather spectacularly if the cathode voltage is significantly higher than the anode voltage for an extended period of time. More subtle drawbacks include the fact that leakage currents can be large if the polarity of the capacitor is reversed<sup>14</sup>. It is this latter characteristic that we will explore in this assignment.

### Caution:

In order to explore leakage effects, we will be applying voltages with the opposite polarity as required by the electrolytic capacitor. Due to the voltage levels we will use, it is unlikely that we will cause a failure of the capacitor. However, it is recommended that you wear eye protection while doing this lab assignment.

In this lab assignment, we will measure the voltage across an electrolytic capacitor for both of the cases shown in Fig. 2. In Fig. 2(a), the polarity of the capacitor is correct; the anode is always at the higher voltage. In Fig. 2(b), the polarity of the capacitor is reversed – the cathode is now at the higher voltage. We will use a “switch” to change the voltage applied to the capacitor – our switch will be implemented simply by unplugging the positive voltage terminal of our power supply from the rest of the circuit.

The resistor  $R$  in the circuit of Fig. 2 simply limits the amount of current the capacitor demands when it is being initially charged. Capacitors require a large amount of current to charge rapidly; without the resistor, the capacitor will attempt to draw more current from the power supply than is available.

<sup>13</sup> In addition, the bar may have a negative sign printed on it, which further indicates that the cathode is to be at the lower (or negative) voltage.

<sup>14</sup> The non-symmetry of the leakage currents relative to capacitor voltage makes electrolytic capacitors poor choices for filter circuits. Filter circuits typically require their operation to be identical for positive and negative voltage inputs, unless special biasing schemes are introduced to ensure that the voltage polarity does not change.

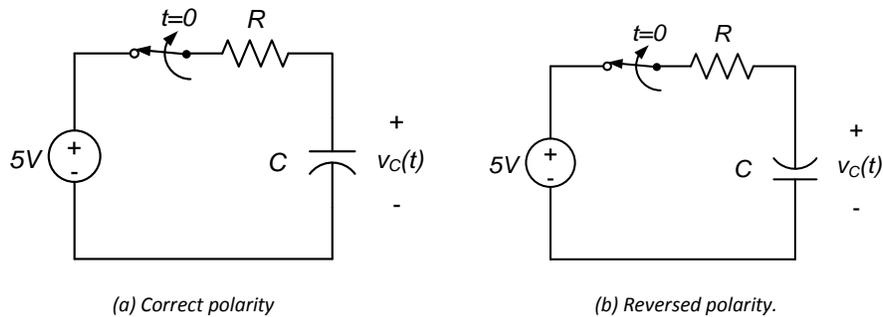


Figure 2. Capacitor configurations used in this lab.

**Pre-lab:**

None

**Lab Procedures:**

1. Using  $R = 100\Omega$  and  $C = 10\mu\text{F}$ , implement the circuit of Fig. 2(a). (Recall that, in Fig. 2(a), the anode is at the higher voltage. Thus, the capacitor terminal with the longer lead is connected to the resistor  $R$  and the shorter lead is connected to ground.) Use  $V+$  to apply the 5V supply.
    - i. Use channel 1 of your oscilloscope to measure the voltage across the capacitor,  $v_C(t)$ . We will be monitoring the amount of time required for the capacitor to discharge once we open the switch in Fig. 2(a); this will take a relatively long time, so set the time scale on your oscilloscope to 5 s/div. Set the vertical scale of your scope to 1 V/div, with a -2V offset.
    - ii. Turn on the power supply and click “Run” to start acquisition of data. The oscilloscope should indicate a +5V voltage across the capacitor.
    - iii. Open the “switch” in Fig. 2(a) by unplugging the power supply terminal from the circuit. (Simply pull the  $V+$  connector out of the breadboard.) The capacitor voltage displayed on the oscilloscope screen should decay exponentially. Measure the time constant of the waveform<sup>15</sup>. Record the image of the oscilloscope window, showing the waveform.
- DATA**
- DEMO**
- iv. Demonstrate operation of your circuit to the Teaching Assistant. Have the TA initial the appropriate page(s) of your lab notebook and the lab checklist.
2. Still using  $R = 100\Omega$  and  $C = 10\mu\text{F}$ , implement the circuit of Fig. 2(b). (In Fig. 2(b), the cathode is at the higher voltage. Thus, the capacitor terminal with the shorter lead is connected to the resistor  $R$  and the longer lead is connected to ground.) Use  $V+$  to apply the 5V supply. Note that this circuit can be easily created from the circuit of Fig. 2(a) by removing the capacitor, reversing the leads, and replacing it again.
    - i. Measure the voltage across the capacitor,  $v_C(t)$ , as in part 1 Turn on the power supply and click “Run” to start acquisition of data. The oscilloscope should indicate a +5V voltage across the capacitor.
    - ii. Open the “switch” in Fig. 2(b) by unplugging the power supply terminal from the circuit. (Simply pull the  $V+$  connector out of the breadboard.) The capacitor voltage displayed on the oscilloscope screen should decay approximately exponentially. Measure the time constant of the waveform. Record the image of the oscilloscope window, showing the waveform.
- DATA**
- DEMO**
- iii. Demonstrate operation of your circuit to the Teaching Assistant. Have the TA initial the appropriate page(s) of your lab notebook and the lab checklist.

<sup>15</sup> Recall that the time constant is the amount of time required for an exponential waveform to decay to 36.8% of its initial value.

3. In the circuits of Fig. 2, there is no way for the capacitor's voltage to decay – the charge difference on the plates cannot “leak” away, since there is (in our model, anyway) no path for the charge to get from one plate to the other. The circuits of Fig. 2 are not realistic, based on the capacitor behavior we observed in parts 1 and 2 above. We must modify how we think about capacitors' behavior in order to explain our data!

In section 6.3 of the textbook, we modeled non-ideal capacitors as a resistance in parallel with an ideal capacitor. Using this model, the circuits of Fig. 2 can be modified to become Fig. 3 below. Now there is a path – the resistor  $R_C$  – which allows the capacitor voltage to decay after the switch opens, allowing us to explain our previous data!

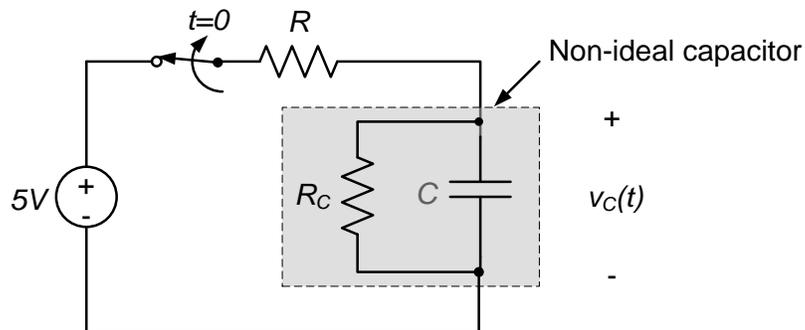


Figure 3. Circuit of Figures 2, with non-ideal capacitor.

As we observed in parts 1 and 2 above, the leakage rate in electrolytic capacitors changes, based on the polarity of the capacitor voltage. In essence, this means that the capacitor resistance,  $R_C$ , in the model of Fig. 3 depends on the polarity of the capacitor voltage<sup>16</sup>!

To obtain an idea as to the variation in the capacitor resistance when the capacitor polarity is reversed, let's normalize the time constants we measured in parts 1 and 2 above. To do this, simply divide the time constant by the capacitance value:

$$R_C \approx \frac{\tau}{C} \quad \text{Eq. 1}$$

As we will see later, the units of equation (1) are consistent.

#### ANALYSIS

- i. Tabulate the results you obtained in parts 1, 2, and 3 above – for each polarity, list the measured time constant, and the estimated capacitor resistance as determined by equation (1). Briefly comment on the magnitude of these resistances and the differences between the capacitor resistances for the two cases. (Include a percent change in resistance induced by changing the capacitor polarity.)

### Post-lab Exercises:

#### ANALYSIS

Suppose that we modify the circuit of Fig. 2(a) so that there is a path for the capacitor to dissipate its voltage after the switch opens, as shown in Fig. 4. Based on the capacitor resistances you estimated in part (c) of the lab procedures, how large would the resistor  $R$  need to be in order for the capacitor resistance  $R_C$  to change the rate at which the capacitor voltage dissipates by about 10%? (E.g. how

<sup>16</sup> This makes modeling electrolytic capacitors whose voltage changes polarity tedious, to say the least.

large would  $R$  need to be before the measured time constant of the capacitor voltage decay changes by 10% when the resistance  $R_C$  is included?)

Hint: the resistances  $R$  and  $R_C$  are in parallel. They can be combined to a single equivalent resistance seen by the capacitor.

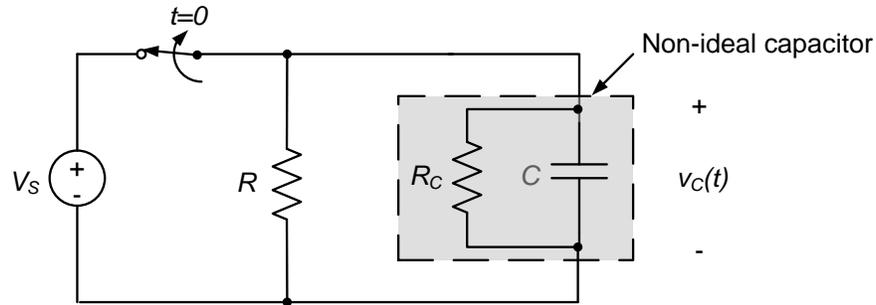


Figure 4. Circuit with external resistor connected across non-ideal capacitor.

## Real Analog Chapter 6: Lab Worksheets

### 6.3.2: Electrolytic Capacitor Leakage Currents (35 points total)

1. Attach to this worksheet an image of the oscilloscope window, showing the capacitor voltage, resulting from opening the switch in Fig. 2(a). (5 pts)
2. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuit's operation when the electrolytic capacitor is connected with the correct polarity. (5 pts)

**TA Initials:** \_\_\_\_\_

3. Attach to this worksheet an image of the oscilloscope window, showing the capacitor voltage, resulting from opening the switch in Fig. 2(b). In the space below, provide your estimate of the time constant of the circuit. (5 pts)
4. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuit's operation when the electrolytic capacitor is connected in reversed polarity. (5 pts)

**TA Initials:** \_\_\_\_\_

5. In the space below, provide a table giving your estimated time constants and the calculated capacitor resistance for both capacitor polarities. Comment on the differences between the two cases, including a percent change in resistance. (8 pts)
6. In the space below, provide your estimate of the resistance required in the circuit of Fig. 4 which changes the time constant of the capacitor by 10%. (7 pts)

## Real Analog Chapter 6: Lab Projects

### 6.4.1: Inductor Voltage-current Relations

In this assignment, we will measure the relationship between the voltage difference across a capacitor and the current passing through it. We will apply several types of time-varying signals to a series combination of a resistor and a capacitor. The voltage difference across the resistor, in conjunction with Ohm's law, will provide an estimate of the current through the capacitor. This current can be related to the voltage difference across the capacitor.

The results of our voltage-current measurements will be compared to analytical expectations.

#### Before beginning this lab, you should be able to:

- State voltage-current relationships for inductors in both differential and integral form
- Apply the inductor voltage-current relations to calculate a inductor's voltage from its current and vice-versa
- Use the Analog Discovery 2's arbitrary waveform generator and oscilloscope to apply and measure time-varying waveforms (Lab 6.2.1)

#### After completing this lab, you should be able to:

- Use the Analog Discovery 2 oscilloscope's math function to calculate the current through a known resistor from the measured voltage difference.
- Export data acquired by the Analog Discovery 2 to files for post-processing by other programs
- Verify a inductor's voltage-current relations using measured data

#### This lab exercise requires:

- Analog Discovery 2 module
- Digilent Analog Parts Kit

#### Symbol Key:

##### DEMO

Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.

##### ANALYSIS

Analysis; include principle results of analysis in laboratory report.

##### SIM

Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.

##### DATA

Record data in your lab notebook.

#### General Discussion:

We will use the circuit of Fig. 1 in this lab assignment. Both the voltage difference across the inductor and the resistor ( $v_L(t)$  and  $v_R(t)$ ) will be measured. From this data, we can readily compare the voltage across the inductor with the current through the inductor. Since the voltage across the resistor is measured, we can readily infer the current through the resistor via Ohm's law:

$$i_R(t) = \frac{v_R(t)}{R} \quad \text{Eq. 1}$$

The resistor and inductor are in series, so the current through the inductor is the same as the current through the resistor, so:

$$i_L(t) = \frac{v_R(t)}{R} \quad \text{Eq. 2}$$

Since we are also measuring the voltage difference across the inductor,  $v_L(t)$ , we can readily compare these parameters with our expectations based on our mathematical models of the capacitor voltage-current relationships.

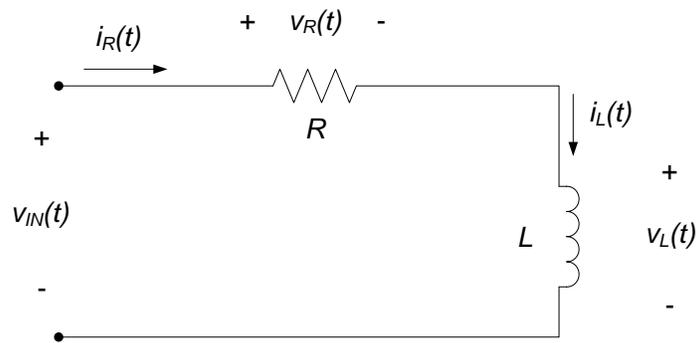


Figure 1. Series RL circuit.

### Pre-lab:

In this lab, we will apply sinusoidal signals to the inductor of Fig. 1. Mathematically, the form of the inductor current will be:

$$i_L(t) = A \cos(2\pi ft) \quad \text{Eq. 3}$$

Where  $A$  is the amplitude of the sinusoid (in volts) and  $f$  is the frequency (in Hz). The waveform is shown graphically in Fig. 2. For the circuit of Fig. 1, use the inductor voltage-current relations to calculate the inductor voltage resulting from application of the voltage of equation (3). Your results may be dependent up on the parameters  $A$ ,  $f$ , and  $L$ .

#### ANALYSIS

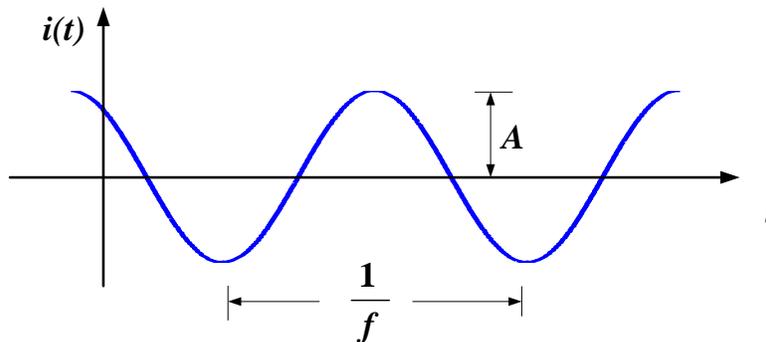


Figure 2. Basic waveform used in this lab.

### Lab Procedures:

Construct the circuit of Fig. 1 with  $L = 1\text{mH}$  and  $R = 100\Omega$ . Use channel 1 of your oscilloscope to measure the resistor voltage difference, and channel 2 of your oscilloscope to measure the inductor voltage difference. Use channel 1 of your waveform generator (W1) to apply the voltage  $v_{in}(t)$  in Fig. 1. Set up a math channel to calculate the current through the inductor per equation (2) in the pre-

lab<sup>17</sup>. Set the oscilloscope measurements to provide at least the amplitude of each of the three displayed waveforms.

**DATA**

1. Apply a sinusoidal input voltage with frequency = 1kHz, amplitude = 2V, and offset = 0V to the circuit of Fig. 1. Use your oscilloscope to display the data listed above (waveforms corresponding to C1, C2, and M1; measurement window displaying amplitudes of C1, C2, and M1). Export the data in the oscilloscope time window to a .csv file for later processing.

**DATA**

2. Apply a sinusoidal input voltage with frequency = 2 kHz, amplitude = 2V, and offset = 0V to the circuit of Fig. 1. Use your oscilloscope to display the data listed above (waveforms corresponding to C1, C2, and M1; measurement window displaying amplitudes of C1, C2, and M1). Export the data in the oscilloscope time window to a .csv file for later processing.

**DEMO**

3. Demonstrate operation of your circuit to the Teaching Assistant. Have the TA initial the appropriate page(s) of your lab notebook and the lab checklist.

**Post-lab Exercises:****ANALYSIS**

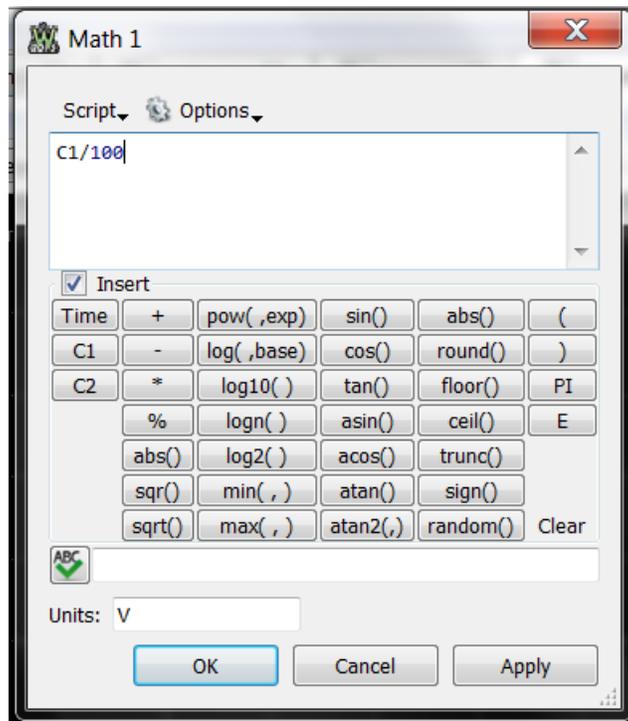
Import the data acquired in the lab procedures into your favorite numerical analysis software (e.g. Excel, Matlab, Octave, etc.). Use the software and the results of your pre-lab analysis to calculate the expected inductor voltage waveforms corresponding to the inductor current waveforms you measured in the lab procedures. Use the software to overlay plots of the expected and measured inductor voltages for each of the cases tested in the lab procedures. Comment briefly on the agreement between the measured and expected inductor voltages for each of the cases. In your comments, be sure to include a quantitative comparison (including percent difference) between the expected and measured amplitudes of the inductor voltages.

**ANALYSIS****Appendix A: Math channel to calculate current from resistor's voltage**

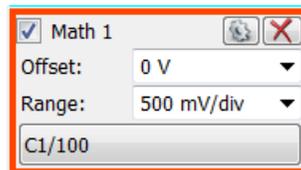
The Analog Discovery 2 provides capabilities for performing mathematical operations on the displayed waveforms and displaying the result. Essentially, there are two basic “types” of mathematical operations which can be performed: “Simple” and “Custom”. The simple math operations consist of addition, subtraction, or multiplication of the two channels. The custom operations are much more wide-ranging. In order to determine the resistor current, we want to divide the resistor voltage by a constant (the resistance value), so we will create a custom math channel. To do this, follow the steps below:

1. Click on “Add Channel” and choose “Custom” from the resulting drop-down menu.
2. A custom math function window will open, as shown below. Type the desired math function (typically a function of the scope channels, C1 and C2) in the text box in this window or use the buttons in the window to create the function. We are using channel 1 (C1) to measure the resistor voltage. The current through the resistor is simply the resistor voltage divided by the resistance value (100Ω), so our function is: C1/100, also shown in the Figure below. Click “OK” to display the function in the main window.

<sup>17</sup> Detailed instructions for doing this are provided in Appendix A.



3. The properties of the math channel display can be adjusted using the channel's control box, just as any with any other channel displayed by the scope. A typical control box is shown below:



4. The units of our math channel are amperes. It is nice to have the displayed units agree with the actual units of the measurement. To change the units, click on the gear icon on the control box. Volts, will typically be the default unit; if you want the vertical axis in amps, click the drop-down arrow next to "units:" and select **A** from the resulting menu<sup>18</sup>.

<sup>18</sup> Choices of units are volts (V), amps (A), and watts (W).



## Real Analog Chapter 6: Lab Projects

### 6.4.2: Non-Ideal Inductor Effects

Voltage-current relationships for ideal inductors do not always adequately explain measured inductor voltages and currents. In this assignment, we will measure inductor voltages and currents which do not agree well with idealized inductor relationships we have employed earlier. After recognizing the differences between the measured data and our mathematical models, we will use a non-ideal inductor model to better approximate the physical behavior of the inductor.

#### Before beginning this lab, you should be able to:

- Apply the inductor voltage-current relations to calculate an inductor's voltage from its current and vice-versa
- Use the Analog Discovery 2's arbitrary waveform generator and oscilloscope to apply and measure time-varying waveforms (Lab 6.2.1)
- Use a math channel on the Analog Discovery 2 to determine the current through a resistor from the voltage drop across the resistor (Labs 6.3.1, 6.4.1)

#### After completing this lab, you should be able to:

- Model a non-ideal inductor as an ideal inductor in series with a resistor
- Identify some effects of non-ideal inductors from measured data

#### This lab exercise requires:

- Analog Discovery 2 module
- Digilent Analog Parts Kit
- Digital multimeter (DMM)

#### Symbol Key:

- DEMO** Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
- ANALYSIS** Analysis; include principle results of analysis in laboratory report.
- SIM** Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
- DATA** Record data in your lab notebook.

#### General Discussion:

The basic circuit we will use in this assignment is the same series resistor-inductor circuit used in lab 6.4.1. This circuit is repeated in Fig. 1 below, for convenience. As in lab 6.4.1, the inductor voltage can be inferred from the voltage across the resistor and the resistance, as shown below:

$$i_L(t) = \frac{v_R(t)}{R} \quad \text{Eq. 1}$$

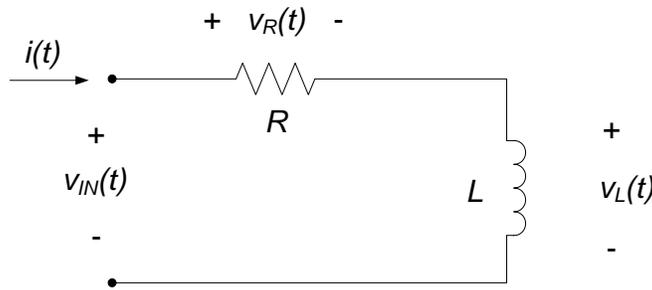


Figure 1. Series RL circuit.

In this assignment, we will apply triangular and square waveforms to the circuit of Fig. 1 above. The measured inductor voltage and current resulting from these waveforms will not agree well with the idealized inductor voltage-current relationships we have used to date. In order to interpret the data we acquire in this assignment, we will employ a non-idealized model of the inductor consisting of a series combination of an ideal inductor and a resistor as shown in Fig. 2.

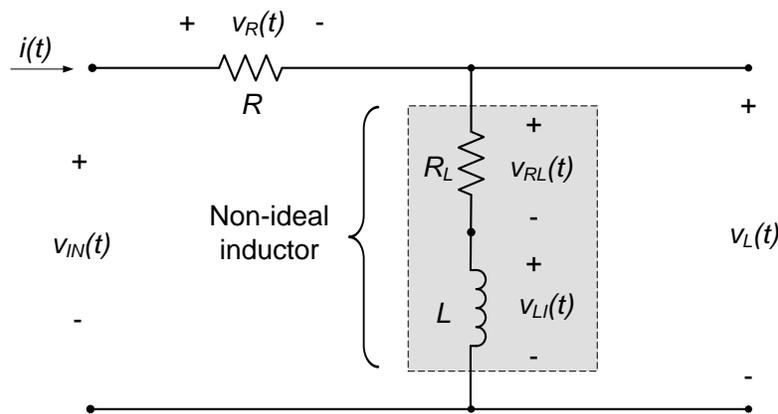


Figure 2. Circuit of Figure 1, with non-ideal model of inductor.

### Pre-lab:

In this lab, we will apply triangular and square signals to the circuit of Fig. 1<sup>19</sup>. The waveforms we will use are shown graphically in Fig. 3. In both cases,  $A$  is the amplitude of the signal (in amperes) and  $f$  is the frequency of the signal (in Hz).

<sup>19</sup> Please keep in mind that Figs. 1 and 2 are actually schematics of the same physical circuit. The difference between the two circuits is simply in the way the inductor is modeled mathematically – in Fig. 1 the inductor is modeled as being ideal, while in Fig. 2, the inductor is modeled as including a resistance. In Fig. 2, we are simply accounting for the fact that any physical inductor will dissipate energy.

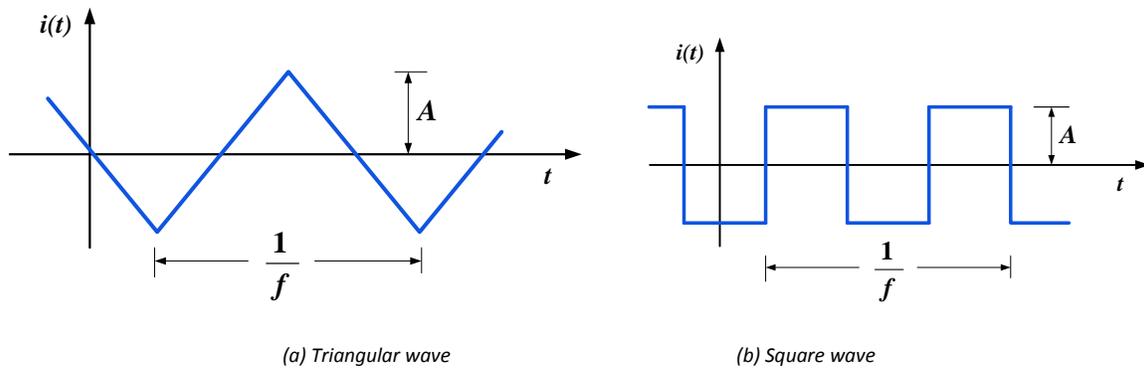


Figure 3. Basic waveforms used in this lab.

For the circuit of Fig. 1, use the inductor voltage-current relations to overlay sketches of the inductor voltage and the inductor current ( $v_L(t)$  and  $i(t)$ ) if the inductor current is:

ANALYSIS

a. A triangular wave,  $i(t)$ , with frequency ( $f$ ) and amplitude ( $A$ ) as shown in Fig. 2(a)

ANALYSIS

b. A square wave,  $i(t)$ , with frequency ( $f$ ) and amplitude ( $A$ ) as shown in Fig. 2(b).

Label your sketch to show the amplitude and period of the inductor current for both of the above cases. Your results may be dependent on the parameters  $A$ ,  $f$ , and  $L$ . Be sure that your sketches of voltage and current share the same time axis!

### Lab Procedures:

1. **Triangular waveform:** Construct the circuit of Fig. 1 with  $L = 1\text{mH}$  and  $R = 100\Omega$ . Use channel 1 of your oscilloscope to measure the resistor voltage difference, and channel 2 of your oscilloscope to measure the voltage across the inductor. Use channel 1 of your waveform generator (W1) to apply the voltage  $v_m(t)$  in Fig. 1. Set up a math channel to calculate the current through the capacitor per equation (2) in the pre-lab. Set the oscilloscope measurements to provide at least the amplitude of each of the three displayed waveforms.

DATA

i. Apply a triangular input voltage with frequency = 1kHz, amplitude = 1V, and offset = 0V to the circuit of Fig. 1. Use your oscilloscope to display the data listed above (waveforms corresponding to C1, C2, and M1; measurement window displaying amplitudes of C1, C2, and M1). Export the image of the oscilloscope window, showing the waveforms and their measured amplitudes.

DEMO

ii. Demonstrate operation of your circuit to the Teaching Assistant. Have the TA initial the appropriate page(s) of your lab notebook and the lab checklist.

DATA

iii. If the experiment goes according to plan, the measured voltage across the inductor will look something like the waveform shown in Fig. 4. This is not (hopefully) what your sketch of your expected response of the inductor voltage that you created in part (a) of the pre-lab looks like (since you were assuming that the inductor was ideal, per the assumptions of Fig. 1). We need to revisit our expectations as to how inductors actually behave!

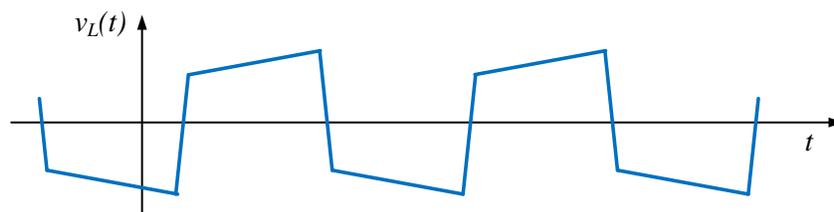


Figure 4. Likely shape of the measured inductor voltage waveform.

- iv. Now we will examine our measured inductor voltage response in the light of the non-ideal inductor model shown in Fig. 2. In Fig. 2, the measured inductor voltage is the sum of the “ideal” inductor voltage,  $v_{LI}(t)$ , and the voltage difference induced by the inductor’s internal resistance,  $v_{RL}(t)$ . We can still assume that the measured current ( $i(t)$ , in Figs. 1 and 2) is the current through both elements of our non-ideal inductor. Using your digital multimeter as an ohmmeter, measure the resistance of your inductor. Based on your measured waveform of the inductor current from part (b), sketch the voltages internal to the non-ideal inductor,  $v_{LI}(t)$  and  $v_{RL}(t)$ . Sketch the sum of these contributions and compare the result to the measured inductor voltage from part (b) above. Briefly discuss the ideal, non-ideal, and measured inductor voltages.

**DATA****ANALYSIS**

2. Apply a square wave input voltage with frequency = 100 Hz, amplitude = 2V, and offset = 0V to the circuit of Fig. 1. Repeat the process of part 1 of the experimental procedures to generate:

**DATA**

- v. The measured waveforms and amplitudes (as an image of the oscilloscope window).
- vi. A sketch of the expected inductor voltage using an ideal inductor model from the pre-lab. (Since the current is a piecewise constant value, the expected inductor voltage should be a constant, except when the current transitions from one value to another – during this transition, the inductor voltage should go to infinity.)
- vii. A sketch of your expected inductor voltage, based on the non-ideal inductor model of Fig. 3 and your measured inductor current. Include a brief discussion comparing the ideal, non-ideal, and measured inductor voltage differences.

**ANALYSIS**

## Real Analog Chapter 6: Lab Worksheets

### 6.4.2: Non-ideal Inductor Effects (45 points total)

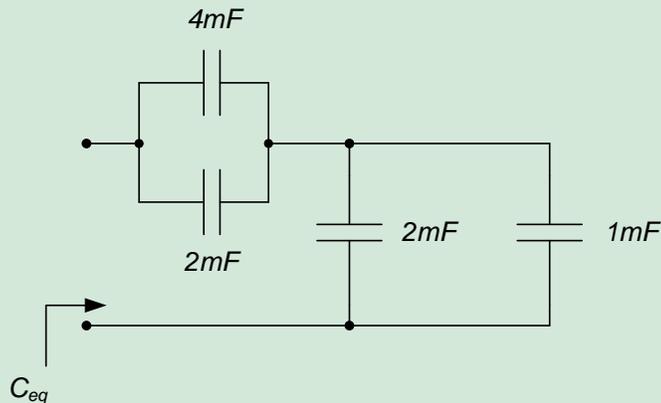
1. Attach to this worksheet your sketches of the inductor voltage and current resulting from the triangular and square waves of Fig. 3. (5 pts)
2. Attach to this worksheet an image of the oscilloscope window, showing the resistor voltage, the inductor voltage, and the inductor current waveforms and the measured amplitudes of these waveforms resulting from the 1kHz triangular input. (8 pts)
3. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuit's operation for sinusoidal inputs. (5 pts)

**TA Initials:** \_\_\_\_\_

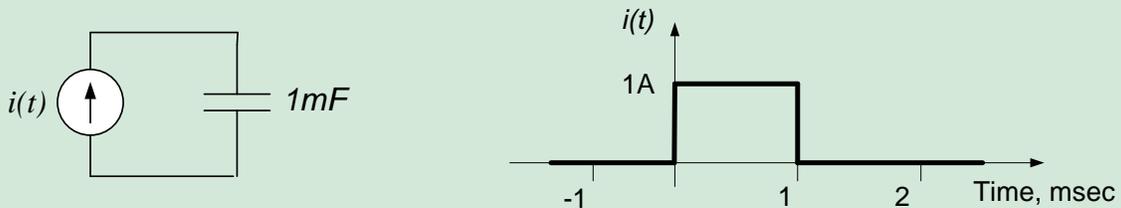
4. In the space below, provide the measured resistance of the inductor. (2 pts)
5. In the space below, sketch the expected voltages  $v_{Ll}(t)$  and  $v_{RL}(t)$  as indicated in Fig. 2 (based on your resistance measurement of 4 above). Also sketch the sum of these contributions. Briefly discuss the relationships between this latter sketch and your measured inductor voltage waveform. (5 pts)
6. Attach to this worksheet an image of the oscilloscope window, showing the resistor voltage, the inductor voltage, and the inductor current waveforms and the measured amplitudes of these waveforms resulting from the 100 Hz square wave input. (8 pts)
7. In the space below, sketch the expected inductor voltage based on the non-ideal inductor model and your measured inductor current. (7 pts)
8. In the space below, briefly compare the ideal, non-ideal, and measured inductor voltages. (5 pts)

## Real Analog Chapter 6: Homework

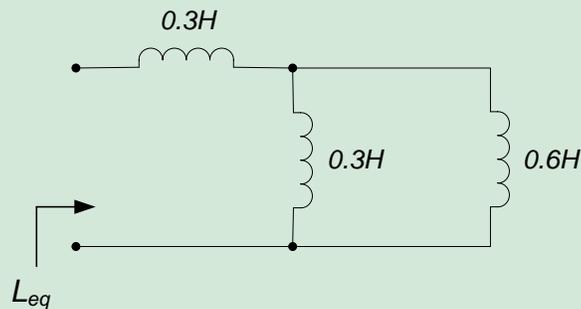
6.1 Determine the equivalent capacitance of the circuit below.



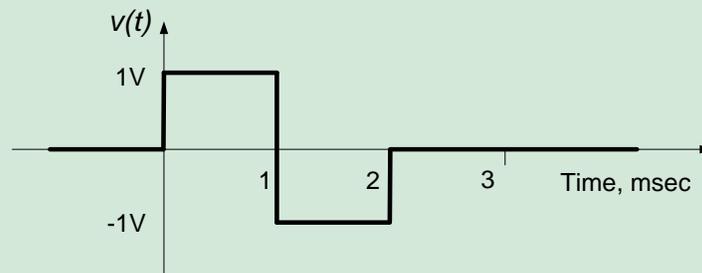
6.2 The capacitor in the circuit to the left below is initially uncharged. A 1A current is applied to the capacitor for 1msec, as shown to the right below. Sketch the voltage across the capacitor.



6.3 Determine the equivalent inductance of the circuit below.



6.4 The voltage across a 1mH inductor is as shown below. Sketch the current through the inductor. Assume that the inductor is relaxed at time  $t = 0$ .



## Chapter 7: First Order Circuits

### 7 Introduction and Chapter Objectives

First order systems are, by definition, systems whose input-output relationship is a *first order* differential equation. A first order differential equation contains a *first order derivative* but no derivative higher than first order – the order of a differential equation is the order of the highest order derivative present in the equation.

First order systems contain a *single energy storage element*. In general, the order of the input-output differential equation will be the same as the number of independent energy storage elements in the system. Independent energy storage cannot be combined with other energy storage elements to form a single equivalent energy storage element. For example, we previously learned that two capacitors in parallel can be modeled as a single equivalent capacitor – therefore, a parallel combination of two capacitors forms a single independent energy storage element.

First order systems are an extremely important class of systems. Many practical systems are first order; for example, the mass-damper system and the mass heating examples from section 6.1 are both first order systems. Higher order systems can often be approximated as first order systems to a reasonable degree of accuracy if they have a *dominant first order mode*. (System modes will be discussed later in this text.) Understanding first order systems and their responses is an important aspect to design and analysis of systems in general.

First order electrical circuits are a special class of first order systems – they are first order systems which are composed of electrical components. Since the electrical components which store energy are capacitors and inductors, first order circuits will contain either one (equivalent) capacitor or one (equivalent) inductor.

In this textbook, we are really interested only in the analysis of electrical circuits, so the bulk of this chapter, of course, deals with analysis of first order electrical circuits. However, at this stage of your studies, you probably do not have an intuitive grasp of the mechanisms of energy storage in electrical circuits and the response of electrical circuits with energy storage elements. Therefore, this chapter begins in Section 7.1 with a general discussion of the response of first order systems, using a sliding mass as an example within which to frame the basic concepts. This provides a relatively gentle introduction to the nomenclature and mathematics which will be used throughout this chapter, in the context of an example for which the student should have some physical insight. Section 7.1 can be omitted, however, without loss of clarity of the remaining sections. Sections 7.2 and 7.3 present the natural response of RC and RL circuits, respectively (RC circuits have a capacitor as an energy storage element, while RL circuits contain an inductor). The natural response of a system corresponds to the system response to some initial condition, with no forcing function provided to the system. In section 7.4, we present the force response of first order circuits, and in section 7.5 we examine the response of first order circuits to a specific forcing function – a step input.

#### After completing this chapter, you should be able to:

- Write the general form of the differential equation governing a first order system
- State, in physical terms, the significance of a differential equation's homogeneous and particular solutions
- Define, from memory, the relationships between a system's unforced response, zero-input response, natural response, and the homogeneous solution to the differential equation governing the system
- Define, from memory, the relationships between a system's forced response, zero-state response, and the particular solution to the differential equation governing the system
- Determine the time constant of a first order system from the differential equation governing the system
- Write mathematical expressions from memory, giving the form of the natural and step responses of a first order system

- Sketch the natural response of a first order system from the differential equation governing the system and the system's initial condition
- Sketch the step response of a first order system from the differential equation governing the system and the amplitude of the input step function
- Write the differential equation governing RC and RL circuits
- Determine the time constant of RC and RL circuits from their governing differential equations
- Determine the time constant of RC and RL circuits directly from the circuits themselves
- Determine initial conditions on arbitrary RC and RL circuits
- Write from memory the form of the natural responses of RC and RL circuits
- Determine the natural response of RC and RL circuits, given the governing differential equation and initial conditions
- Write the form of the differential equations governing forced first order electrical circuits
- Determine the time constant of a forced electrical circuit from the governing differential equation
- Write differential equations governing passive and active first order circuits
- Determine the differential equation governing the step response of a first order electrical circuit
- Write the form of the particular solution of a first order differential equation, to a step input
- Write the form of the step response of a first order electrical circuit
- Determine the final conditions (steady-state response) of a first order electrical circuit, to a step input
- Define DC gain for a circuit and relate it to the steady-state response to a step input
- Determine the step response of a first order electrical circuit from the governing differential equation, the initial conditions, and the final conditions

## 7.1 Introduction to First Order Systems

In this section, we introduce some basic nomenclature relative to first order system responses and illustrate these terms in the context of an example for which the reader may have an intuitive understanding: a mass sliding on a surface. This example, though not directly relevant to the study of electrical circuits, is intended to allow the reader to develop some physical insight into the terminology and concepts relative to the solution of first order differential equations. The concepts and results obtained with this example are then generalized to apply to any arbitrary first order system. These results are used in later sections to provide insight in the analysis of electrical circuits, for which the reader may not yet have an intuitive understanding.

Before discussing first-order electrical systems specifically, we will introduce the response of general first order systems. A general first order system is governed by a differential equation of the form:

$$a_1 \frac{dy(t)}{dt} + a_0 y(t) = f(t), \quad t > 0 \quad \text{Eq. 7.1}$$

Where  $f(t)$  is the (known) input to the system and  $y(t)$  is the response of the system.  $a_1$  and  $a_0$  are constants specific to the system being analyzed. We assume in equation (7.1) that the input function is applied only for times  $t > t_0$ . Thus, from equation (7.1), we can only determine the response of the system for times  $t > t_0$ .

In order to find the solution to equation (7.1), we require knowledge of the system's *initial condition*:

$$y(t = t_0) = y_0 \quad \text{Eq. 7.2}$$

The initial condition,  $y_0$ , defines the state of the system at time  $t = t_0$ . Since equation (7.1) describes a system which stores energy, the effect of the initial condition is to provide information as to the amount of energy stored in the system at time  $t = t_0$ .

The system described by equations (7.1) and (7.2) can be illustrated in block diagram form as shown in Fig. 7.1. The output of the system depends upon the initial condition,  $y_0$ , and the input function  $f(t)$ . The initial condition provides information relative to the energy stored in the system prior to application of the input function. The input function provides information relative to the energy being applied to the system from external sources. The input-output equation describes how the system transfers the energy initially present in the system and the energy added to the system to the system output.

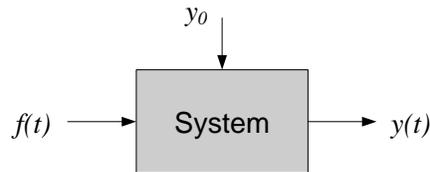


Figure 7.1. System block diagram.

The solution to equation (7.1) consists of two parts – the *homogeneous solution*,  $y_h(t)$ , and the particular solution,  $y_p(t)$ , as shown below:

$$y(t) = y_h(t) + y_p(t) \quad \text{Eq. 7.3}$$

The homogeneous solution is due to the properties of the system and the initial conditions applied to the system; it describes the response of the system if no input is applied to the system, so  $f(t)=0$ . The homogeneous solution is sometimes called the systems *natural response*, the *unforced response*, or the *zero input response*. Since all physical systems dissipate energy (according to the second law of Thermodynamics) the homogeneous solution must die out with time; thus,  $y_h(t) \rightarrow 0$  as  $t \rightarrow \infty$ .

The particular solution describes the systems response to the particular forcing function applied to the system; the form of the particular solution is dictated by the form of the forcing function applied to the system. The particular solution is also called the *forced response* or the *zero state response*.

Since we are concerned only with linear systems, superposition principles are applicable, and the overall system response is the sum of the homogeneous and particular solutions. Thus, equation (7.3) provides the system's overall response to both initial conditions and the particular forcing function being applied to the system.

The previous concepts are rather abstract, so we provide below an example of the application of the above concepts to a system for whose response the students should have some intuitive expectations. This example is intended to provide some physical insight into the concepts presented above prior to applying these concepts to electrical systems.

### 7.1.1 Mass-damper System Example

As an example of a system which includes energy storage elements we revisit the mass-damper system of section 6.1. The system under consideration is shown in Fig. 7.2. The applied force  $F(t)$  pushes the mass to the right. The mass's velocity is  $v(t)$ . The mass slides on a surface with sliding coefficient of friction  $b$ , which induces a force which opposes the mass's motion. The mass will have some initial velocity:

$$v(t = 0) = v_0 \quad \text{Eq. 7.4}$$

Consistent with section 6.1, we consider the applied force to be the input to our system and the mass's velocity to be the output. Figure 7.3 illustrates the system input-output relationship and initial conditions in block diagram form.

The governing equation for the system was determined in section 4.1 to be the first order differential equation:

$$m \frac{dv(t)}{dt} + bv(t) = F(t) \quad \text{Eq. 7.5}$$

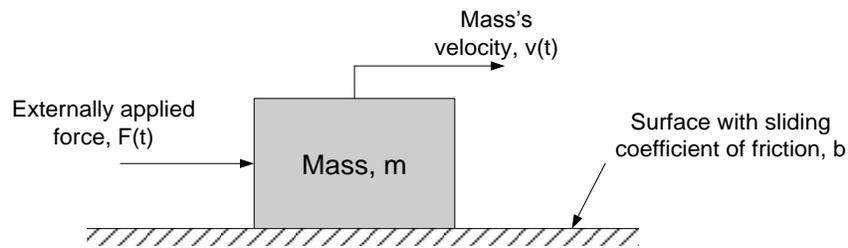


Figure 7.2. Sliding mass on surface with friction coefficient,  $b$ .

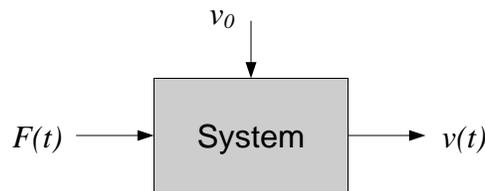


Figure 7.3. Block diagram of system shown in Figure 1.

We consider two cases of specific forcing functions in the following cases. In the first case, the forcing function is zero, and we determine the system's natural response or the homogeneous solution to equation (7.5) above. In the second case, the forcing function is a constant nonzero force applied to the mass with zero initial velocity.

### Case i: Natural (Homogeneous) Response

Let us consider first the case in which the mass has some initial velocity but no external force is applied to the mass. Intuitively, we expect that the velocity of the mass will decrease until the mass comes to rest. In this example, we will determine the solution of the differential equation (7.5) and compare this solution with our expectations.

With no applied forcing function, the differential equation governing the system is:

$$m \frac{dv(t)}{dt} + bv(t) = 0 \quad \text{Eq. 7.6}$$

The initial condition is given by equation (7.4) above, repeated here for convenience:

$$v(t = 0) = v_0$$

Equation (7.6) is a homogeneous differential equation, since there is no forcing function applied to the system. Thus, the particular solution in this case is  $y_p(t)=0$  and our overall solution is simply the homogeneous solution,  $y(t)=y_h(t)$ .

To solve the above differential equation, we rearrange equation (7.6) to give:

$$\frac{m}{b} \cdot \frac{dv(t)}{dt} = -v(t) \quad \text{Eq. 7.7}$$

Separating variables in equation (7.7) results in:

$$\frac{dv(t)}{v} = -\frac{b}{m} dt \quad \text{Eq. 7.8}$$

Incorporating dummy variables of integration and integrating both sides of (7.8) gives:

$$\int_{v_0}^{v(t)} \frac{d\xi(t)}{\xi} = -\frac{b}{m} \int_0^t d\zeta$$

Which evaluates to:

$$\ln(v) \Big|_{v_0}^{v(t)} = -\frac{b}{m} t \Rightarrow \ln[v(t)] - \ln[v_0] = -\frac{b}{m} t \Rightarrow \ln \left[ \frac{v(t)}{v_0} \right] = -\frac{b}{m} t$$

Taking the exponent of both sides of the above provides our final result:

$$v(t) = v_0 e^{-\frac{bt}{m}} \quad \text{Eq. 7.9}$$

A plot of the response given in equation (7.9) is shown in Fig. 7.4. This plot matches our previous expectations, the velocity of the mass at time  $t=0$  is  $v_0$  and the velocity decreases exponentially until the mass is (essentially) at rest. Referring to section 6.2, we see that the response of equation (7.9) can be written in terms of a time constant as:

$$v(t) = v_0 e^{-\frac{t}{\tau}}$$

Where the time constant  $\tau = \frac{m}{b}$ . This result also agrees with our intuition: as the friction coefficient decreases, the time constant increases and the mass comes to rest more slowly. Likewise, increasing the mass causes the time constant to increase – a larger mass will tend to “coast” for a longer time.

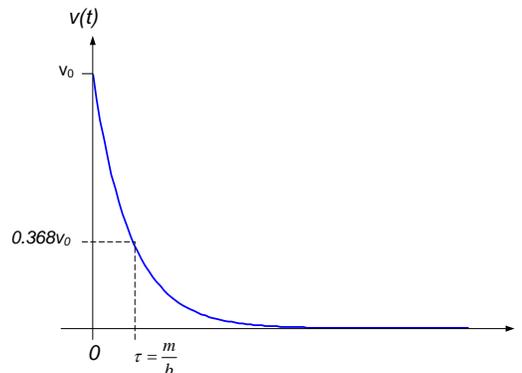


Figure 7.4. Homogeneous response of mass-damper system.

**Note:** The velocity of the mass tells us how much kinetic energy is being stored by the system. The initial condition provides the energy initially stored in the system. The calculated response describes how this energy is dissipated through the sliding friction. No energy is added to the system in this case, since the external applied force is zero.

### Case ii: Response to Step Input

We will now consider the case in which the mass is initially at rest, and a constant force is applied to the mass at time  $t=0$ . Intuitively, we expect the velocity of the mass to increase to some final value; the final velocity of the mass corresponds to the condition in which the frictional force is equal and opposite to the applied force (recall that in our model, the frictional force is proportional to velocity – as the velocity increases, the frictional force opposing the motion also increases). We now solve the governing differential equation for this system and compare the results to our expectations.

The differential equation governing the system, valid for  $t > 0$ , and initial condition, providing the energy in the system at  $t = 0$ , are:

$$m \frac{dv(t)}{dt} + bv(t) = F$$

$$v(t = 0) = 0 \quad \text{Eq. 7.10}$$

Where  $F$  is the magnitude of the (constant) applied force. Note that since  $F$  is constant, and only applied for times  $t > 0$ , we have a step input with magnitude  $F$ . We want to solve the above differential equation for  $t > 0$ ; since the input forcing function can be represented as a step function, this resulting solution is called the step response of the system.

For this case, we have both a nonzero forcing function and an initial condition to consider. Thus, we must determine both the homogeneous and particular solutions and superimpose the result per equation (7.3) above.

The homogeneous solution is determined from:

$$m \frac{dv_h(t)}{dt} + bv_h(t) = 0 \quad \text{Eq. 7.11}$$

Where  $v_h(t)$  is the homogeneous solution. This equation has been solved as case  $i$ ; the form of the solution is:

$$v_h(t) = K_1 e^{\frac{-bt}{m}} \quad \text{Eq. 7.12}$$

Where  $K_1$  is (in this case) an unknown constant which will be determined from our initial conditions.

The particular solution is determined from:

$$m \frac{dv_p(t)}{dt} + bv_p(t) = F \quad \text{Eq. 7.13}$$

Where  $v_p(t)$  is the particular solution to the differential equation in equation (7.10). Since the right-hand side of equation (7.13) is constant for  $t > 0$ , the left-hand side of the equation must also be constant for  $t > 0$  and  $v_p(t)$  must be constant for  $t > 0$ . If  $v_p(t)$  is constant,  $\frac{dv_p(t)}{dt}$  is zero and equation (7.13) simplifies to:

$$bv_p(t) = F$$

So that:

$$v_p(t) = \frac{F}{b} \quad \text{Eq. 7.14}$$

Superimposing equations (7.12) and (7.14), per the principle expressed in equation (7.3) results in:

$$v(t) = v_h(t) + v_p(t) = K_1 e^{\frac{-bt}{m}} + \frac{F}{b} \quad \text{Eq. 7.15}$$

We can now use our initial condition,  $v(t=0)=0$ , to determine the constant  $K_1$ . Evaluating equation (7.15) at  $t=0$  and applying the initial condition results in:

$$v(t = 0) = 0 = K_1 e^{\frac{-b \cdot 0}{m}} + \frac{F}{b} \quad \text{Eq. 7.16}$$

Since  $e^{-b \cdot 0} m = 1$ , equation (7.16) results in:

$$K_1 = -\frac{F}{b} \quad \text{Eq. 7.17}$$

Substituting equation (7.17) into equation (7.15) results in the overall solution:

$$v(t) = -\frac{F}{b} e^{-\frac{bt}{m}} + \frac{F}{b} = \frac{F}{b} \left(1 - e^{-\frac{bt}{m}}\right) \quad \text{Eq. 7.18}$$

If, as in case *i*, we define the time constant  $\tau = \frac{m}{b}$ , equation (7.13) can be expressed as:

$$v(t) = \frac{F}{b} \left(1 - e^{-\frac{t}{\tau}}\right) \quad \text{Eq. 7.19}$$

A plot of the system response is shown in Fig. 7.5. This plot matches our intuitive expectations: the initial velocity is zero; the applied force causes the mass to move. When the frictional and applied forces balance, the velocity of the mass becomes constant. The time constant is determined by the mass and the frictional coefficient; a larger mass results in a longer time constant – it takes longer to get a large mass to its final velocity than a small mass. The frictional coefficient also affects the system time constant; a smaller friction coefficient results in a longer time constant. This result seems counter-intuitive at first, since a smaller frictional coefficient should allow us to accelerate the mass more rapidly. However, the smaller frictional coefficient also results in a higher final velocity – since the time constant is defined by the time required to reach approximately 63.2% of the final velocity, the higher final velocity causes a longer time constant even though the mass is accelerating more rapidly. (If the damping coefficient is zero, the time constant goes to infinity. However, the final velocity also goes to infinity – it takes an infinite amount of time to get to 63.2% of an infinite velocity!)

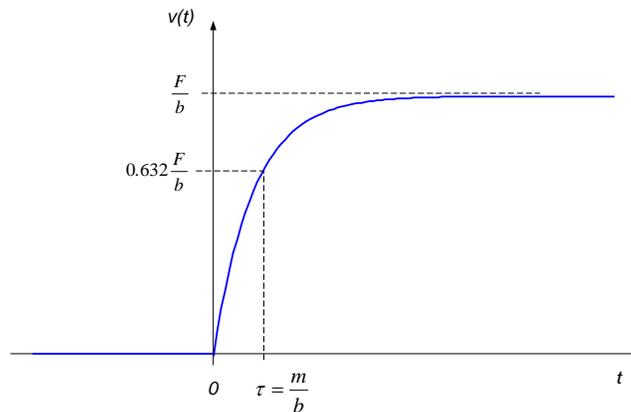


Figure 7.5. Step response of mass-damper system.

**Note:** The velocity of the mass again describes the energy stored by the system; in this case, the initial velocity is zero and the system has no energy before the force is applied. The applied force adds energy to the system by causing the mass to move. When the rate of energy addition by the applied force and energy dissipation by the friction balance, the velocity of the mass becomes constant and the energy stored in the system becomes constant.

## Summary

We use the results of the above examples to re-state some primary results in more general terms. It is seen above that the natural and step responses of first order systems are strongly influenced by the system time constant,  $\tau$ .

The original, general, differential equation – equation (7.1) above – can be re-written directly in terms of the system time constant. We do this by dividing equation (7.1) by the coefficient  $a_1$ . This results in:

$$\frac{dy(t)}{dt} + \frac{a_0}{a_1}y(t) = \frac{1}{a_1}f(t), \quad t > t_0 \quad \text{Eq. 7.20}$$

Defining  $\tau = \frac{a_1}{a_0}$ , equation (7.20) becomes:

$$\frac{dy(t)}{dt} + \frac{1}{\tau}y(t) = \frac{1}{a_1}f(t) \quad \text{Eq. 7.21}$$

The initial condition on equation (7.21) is as before:

$$y(t = t_0) = y_0 \quad \text{Eq. 7.22}$$

The cases of the system homogeneous response (or *natural* or *unforced* response) and step response are now stated more generally, for the system described by equations (7.21) and (7.22).

### 1. Homogeneous response

For the homogenous response  $f(t) = 0$ , and the system response is

$$y(t) = y_0 e^{-\frac{t}{\tau}} \text{ for } t \geq 0 \quad \text{Eq. 7.23}$$

The response is shown graphically in Fig. 7.6.

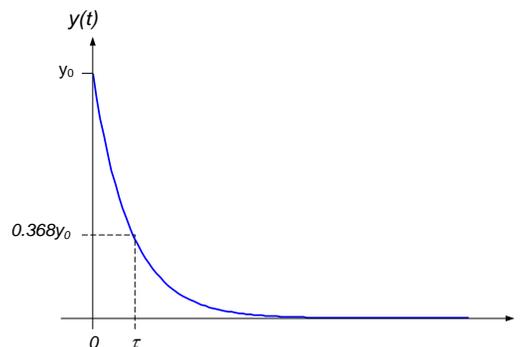


Figure 7.6. First-order system homogeneous response.

### 2. Step Response

For a step input of amplitude  $A$ ,  $f(t) = Au_0(t)$  where  $u_0(t)$  is the unit step function defined in section 6.1. Substituting this input function into equation (7.21):

$$\frac{dy(t)}{dt} + \frac{1}{\tau}y(t) = \frac{A}{a_1}, \quad t > 0 \quad \text{Eq. 7.24}$$

Using the approach of case 2. of our previous mass-damper system example, we determine the system response to be:

$$y(t) = \frac{A}{a_0} \left[ 1 - e^{-\frac{t}{\tau}} \right] \quad \text{Eq. 7.25}$$

This response is shown graphically in Fig. 7.7.

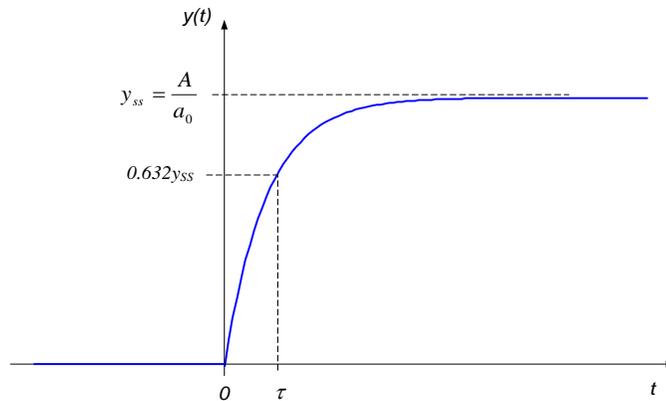


Figure 7.7. First order system response to step input with amplitude  $A$ .

## Section Summary

- A first order system is described by a first order differential equation. The order of the differential equation describing a system is the same as the number of independent energy storage elements in the system – a first order system has one independent energy storage element. (The number of energy dissipation elements is arbitrary, however.)
- The differential equation governing a first order system is of the form:

$$a_1 \frac{dy(t)}{dt} + a_0 y(t) = f(t)$$

Where  $y(t)$  is the system output,  $f(t)$  is the applied input to the system, and  $a_0$  and  $a_1$  are constants.

- The differential equation governing a first order system can also be written in the form:

$$\frac{dy(t)}{dt} + \frac{1}{\tau} y(t) = \frac{f(t)}{a_1}$$

Where  $y(t)$  is the system output,  $f(t)$  is the applied input to the system, and  $\tau$  is the system time constant.

- The system time constant is a primary parameter used to describe the response of first order systems.
- In this chapter, we considered two types of forcing functions: a zero-input case, in which  $f(t)=0$  and  $y(t=0)=y_0$ , and a step input case, in which  $f(t)=Au_0(t)$  and  $y(t=0)=0$ . For the zero-input case, the response is:

$$y(t) = y_0 e^{-\frac{t}{\tau}}, \quad t > 0$$

For the step input case, the so-called *step response* is:

$$y(t) = \frac{A}{a_0} [1 - e^{-t/\tau}], \quad t > 0$$

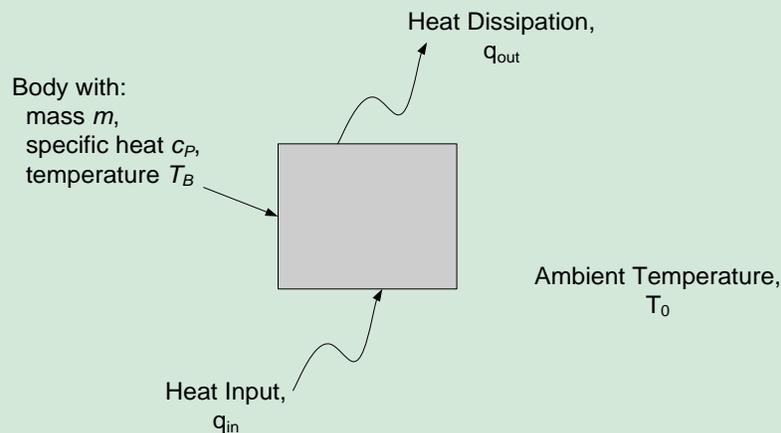
- The system response consists of two parts: a *homogeneous solution* and a *particular solution*. The response can also be considered to consist of a *transient response* and a *steady-state response*. The homogeneous solution and the transient response die out with time; they are due to a combination of the system characteristics and the initial conditions. The particular solution and the steady state response have the same form as the forcing function; they persist as  $t \rightarrow \infty$ . It can be seen from the above that, for the zero-input case, the steady state response is zero (since the forcing function is zero). The steady state

step response is  $\frac{A}{a_0}$ ; it is a constant value and is proportional to the magnitude of the input forcing function.

## 7.1 Exercises

- In Example 6.2, we examined a body which was subjected to external heating. The system is shown in the figure below. The mass of the body is  $m$ , the body material has a specific heat,  $c_p$ , and is at some temperature  $T_B$ . The surroundings are at an ambient temperature  $T_0$ . A heat input  $q_{in}$  is applied to the body, the heat dissipation between the body and its surroundings is  $q_{out}$ . It is common to assume that the heat dissipation  $q_{out}$  is proportional to the difference in temperature between the mass and its surrounds, so that  $q_{out}=h(T_B-T_0)$ . Incorporating this assumption into the governing equation for the system provided in Example 6.2, results in the following differential equation relating  $q_{in}$  and  $T_B$ :

$$mc_p \frac{d(T_B - T_0)}{dt} + h(T_B - T_0) = q_{in}$$



- If the body has some initial temperature  $T_i$ , and no heat is applied to the body (e.g.  $q_{in} = 0$ ),
  - What is the final temperature of the body?
  - What is the time constant (in terms of  $m$ ,  $c_p$ , and  $h$ )?
  - If the body mass is doubled, what is the effect on the time constant? Does this agree with your expectations based on your intuition?
  - Sketch the response of the body temperature ( $T_B$ ) vs. time. Label the initial condition and time constant on the sketch.
- If the body is initially at a temperature  $T_0$  (the same as the ambient surroundings) and a constant heat input  $q_{in}$  is applied starting at  $t = 0$ ,
  - What is the final temperature of the body (in terms of  $m$ ,  $c_p$ ,  $q_{in}$ , and  $h$ )?
  - What is the effect of doubling the heat input  $q_{in}$  on the final temperature? Does this agree with your intuition?
  - What is the effect of doubling the mass on the final temperature? Does this agree with your intuition?
  - What is the effect of doubling the mass on the time constant? Does this agree with your intuition?
  - Sketch the response  $T_B$  vs. time. Label the initial temperature, final temperature, and time constant on the sketch.

## 7.2 Natural Response of RC Circuits

In this section, we consider source-free circuits containing only resistors and a single capacitor – commonly referred to as RC circuits. Since these circuits contain only a single energy storage element – the capacitor – the

governing equations for the circuits will be first order differential equations. Since the circuits are source-free, no input is applied to the system and the governing differential equation will be homogeneous. Thus, in this section we will be examining the *natural response of RC circuits*.

To begin our investigation of the natural response of RC circuits, consider the simple resistor-capacitor combination shown in Figure 7.8. We assume that the capacitor is initially charged to some voltage,  $V_0$ , at time  $t=0$  (so that  $v(0)=V_0$ ). We want to determine the capacitor voltage,  $v(t)$ , for  $t>0$ .

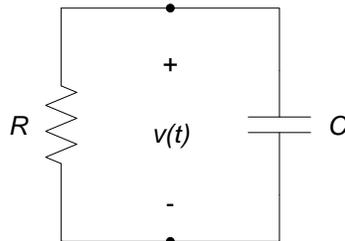


Figure 7.8. RC circuit;  $v(t=0) = V_0$ .

Applying Kirchhoff's current law at the positive terminal of the capacitor, as shown in Fig. 7.9, results in:

$$i_C(t) + i_R(t) = 0 \quad \text{Eq. 7.26}$$

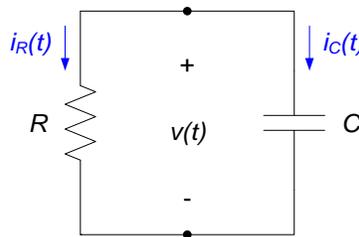


Figure 7.9. Currents in RC circuit.

Since  $i_R(t) = \frac{v(t)}{R}$  and  $i_C(t) = C \frac{dv(t)}{dt}$ , equation (7.26) can be written in terms of the capacitor voltage as:

$$C \frac{dv(t)}{dt} + \frac{v(t)}{R} = 0$$

Separation of variables results in:

$$\frac{dv}{v} = -\frac{1}{RC} dt$$

The integral of the above is:

$$\int_{V_0}^{v(t)} \frac{dv}{v} = -\frac{1}{RC} \int_0^t dt$$

Which evaluates to:

$$\ln[v(t)] - \ln[V_0] = -\frac{t}{RC}$$

Or

$$\ln \left[ \frac{v(t)}{V_0} \right] = -\frac{t}{RC}$$

Since  $e^{\ln(x)} = x$ , the above becomes:

$$v(t) = V_0 e^{-\frac{t}{RC}}$$

#### Alternate Approach to Solving the Above Differential Equation:

Since  $\frac{dv(t)}{dt} = -\frac{1}{RC} v(t)$  we see that the form of the voltage signal must not change as a result of differentiation.

Thus, assume that the voltage signal is of the form:

$$v(t) = K e^{-st}$$

Where  $K$  and  $s$  are unknown constants. If we substitute this into the original differential equation, we obtain:

$$-Ks e^{-st} = -\frac{K}{RC} e^{-st}$$

This is satisfied if we choose  $s = -\frac{1}{RC}$ . Employing our initial condition,  $v(0) = V_0$ , gives:

$$K e^{-\frac{0}{RC}} = K = V_0$$

Results in:

$$v(t) = V_0 e^{-\frac{t}{RC}}$$

As before.

The capacitor voltage response is shown graphically in Fig. 7.10. The voltage response is a decaying exponential with a time constant:

$$\tau = RC \tag{Eq. 7.27}$$

Thus, if we increase the resistance without changing capacitance, the circuit's time constant will increase. Likewise, increasing capacitance while maintaining the resistance constant will also increase the system's time constant. It is important to note that neither the resistance nor the capacitance alone specify the time constant, it is determined by the product of the two – if we simultaneously double the resistance and halve the capacitance, the system's time constant is unchanged.

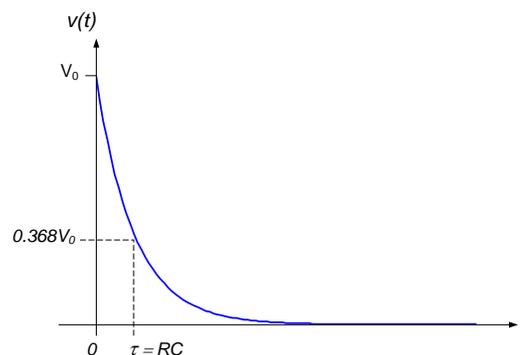


Figure 7.10. RC circuit natural response.

We can also obtain the above result by writing the governing differential equation directly in terms of the time constant. Previously, applying KCL to the circuit of Fig. 7.9, we obtained:

$$C \frac{dv(t)}{dt} + \frac{v(t)}{R} = 0$$

Which can be re-written (by dividing the equation by the capacitance,  $C$ , as:

$$\frac{dv(t)}{dt} + \frac{v(t)}{RC} = 0$$

Since the time constant for this circuit is  $\tau=RC$ , the above can be re-written as:

$$\frac{dv(t)}{dt} + \frac{1}{\tau} v(t) = 0 \quad \text{Eq. 7.28}$$

With initial condition:

$$v(0) = V_0 \quad \text{Eq. 7.29}$$

The solution to equation (7.28), subject to the initial condition of equation (7.29) is (per our results above):

$$v(t) = V_0 e^{-t/\tau} \text{ for } t \geq 0 \quad \text{Eq. 7.30}$$

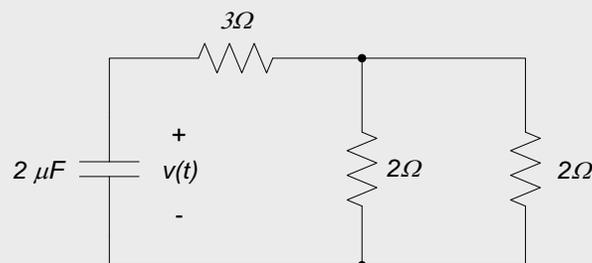
Where the time constant is defined per equation (7.27). The above approach matches our previous result. We will use the problem description provided by equations (7.28) and (7.29) and the solution of the form (7.30) most commonly in the subsequent examples. It should be emphasized, however, that the results are not dependent upon the solution approach – either of the other two approaches presented above yield the same conclusions.

## 7.2.1 Generalization to Multiple Resistors

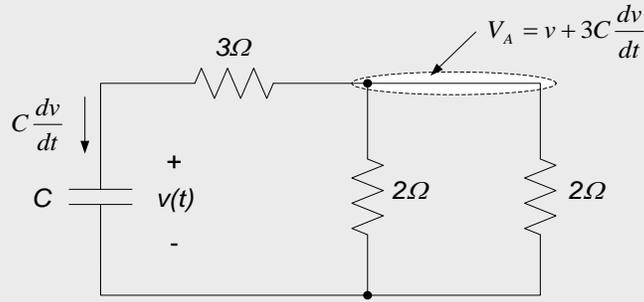
The resistance in the time constant of equation (7.27) can be more generally defined as the equivalent overall resistance of the circuit as seen by the capacitor. Thus, if we remove the capacitor from the circuit and create a Thevenin equivalent resistance as seen by the capacitor, the time constant will be the product of the capacitance and this equivalent resistance. We illustrate this point with an example.

### Example 7.1

Determine the voltage  $v(t)$  for the circuit below if  $v(0)=5V$ .



We will first solve the problem by writing the first order differential equation governing the system. To aid this process we re-draw the circuit as shown below, labeling the current through the capacitor, defining node “A”, and labeling the voltage at node “A”. For simplicity, we label the capacitor as having capacitance “ $C$ ” in the figure below.



The voltage at node A  $V_A = v + (3\Omega)C \frac{dv}{dt}$  is obtained by applying KVL around the outer loop of the circuit.

Applying KCL at node A results in:

$$C \frac{dv}{dt} + \frac{V_A}{2\Omega} + \frac{V_A}{2\Omega} = C \frac{dv}{dt} + V_A = 0$$

Which, when substituting  $V_A = v + 3C \frac{dv}{dt}$  results in:

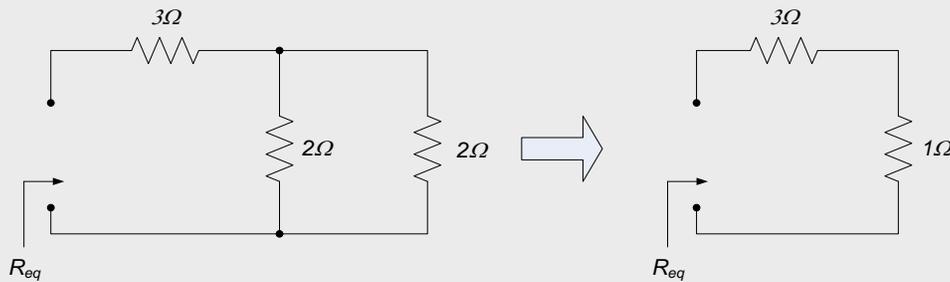
$$4C \frac{dv}{dt} + V_A = 0$$

This can be placed in the “standard form”,  $\frac{dv(t)}{dt} + \frac{1}{\tau v(t)} = 0$ , by dividing through by  $4C$ :

$$\frac{dv}{dt} + \frac{1}{4C} V_A = 0$$

Thus, the time constant  $\tau = 4C = 8 \times 10^{-6}$  seconds.

Removing the capacitor and using circuit reduction to determine an equivalent resistance results in the circuit shown to the left below. The parallel combination of two,  $2\Omega$  resistors results in an equivalent  $1\Omega$  resistor, as shown in the figure to the right below. The resulting series combinations simplifies to a single  $4\Omega$  resistance, so  $R_{eq}=4\Omega$ .



The circuit time constant can now be determined from  $R_{eq}$  and  $C$ . Thus:

$$\tau = R_{eq}C = (4\Omega)(2 \times 10^{-6}F) = 8 \times 10^{-6}\text{seconds}$$

Since the initial capacitor voltage is given to be  $5V$ ,  $v(t) = 5e^{\frac{-t}{8 \times 10^{-6}}} = 5e^{1.25 \times 10^5 t}V$ .

## 7.2.2 Determining Initial Conditions

The circuits we have considered so far in this chapter contain no sources, the circuits' initial conditions are given. In general, we will need to determine the initial conditions from a given source and/or switching operation. For example, a conceptual circuit showing how the initial condition for the circuit of Fig. 7.8 can be created is shown in Fig. 7.11(a). The switch in Fig. 7.11(a) has been closed for a long time; thus, just before the switch opens, the voltage across the capacitor is  $V_0$ . Opening the switch removes the source from the circuit of interest. Since the voltage across a capacitor cannot change suddenly, the capacitor still has voltage  $V_0$  immediately after the switch opens. (Mathematically, we say that  $v(t=0)=v(t=0^+)=V_0$ . Where time  $t=0$  is an infinitesimal time before the switch opens, and the time  $t=0^+$  is an infinitesimal time after the circuit opens.) Thus, for times  $t>0$ , the shaded portion of the circuit of Fig. 7.11(a) is identical to the circuit of Fig.7.11(b) from the viewpoint of the capacitor voltage. An example of the analysis of an RC circuit with an included source is provided below; note that the circuit being analyzed is still unforced - the source only provides an initial condition.

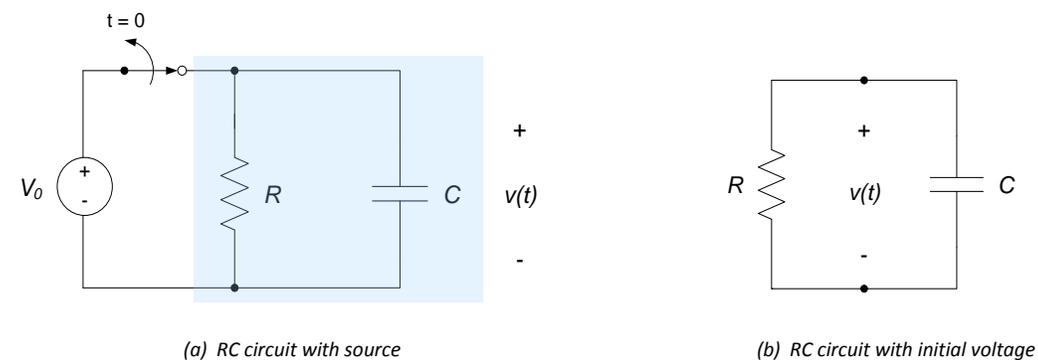
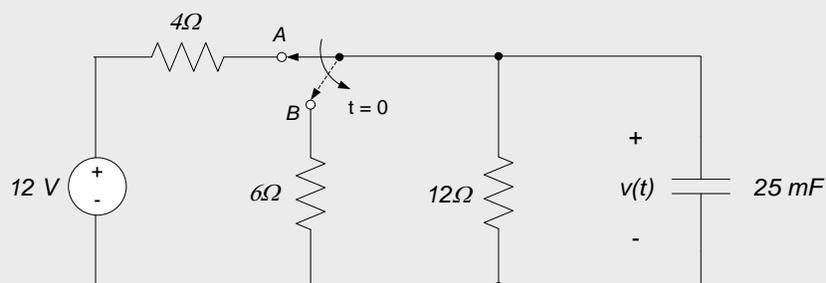


Figure 7.11. Capacitor energy storage.

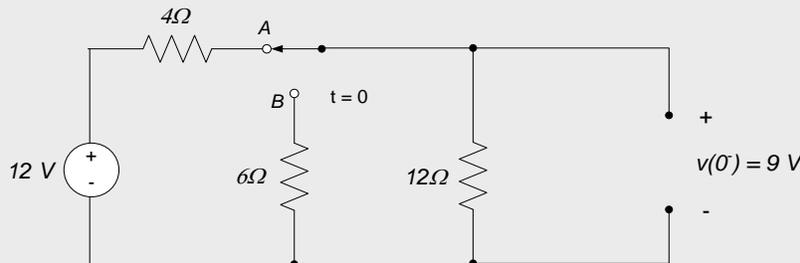
### Example 7.2: Switched Circuit Natural Response

Consider the circuit shown below. The switch is originally at position A; at time  $t=0$  seconds, the switch moves to position B in the circuit. We wish to determine the capacitor voltage,  $v(t)$  for  $t>0$ .



Before time  $t=0$ , we assume that the switch has been at position A for a long time – all transients have died off, and any voltages and currents in the circuit have become constant. Since the capacitor voltage-current relationship is  $i = C \frac{dv}{dt}$ , if all parameters are constant the capacitor current is zero and the capacitor looks like an open circuit. Replacing the capacitor with an open circuit, as shown in the figure below, allows us to determine the voltage across the capacitor before the switch moved to position B. It is fairly easy to see that the capacitor voltage is the same as the voltage across the  $12\Omega$  resistor. Since no current flows through the  $6\Omega$  resistor in the circuit, the voltage across the  $12\Omega$  resistor can be determined by a voltage division between the  $12\Omega$  resistor and the  $4\Omega$  resistor.

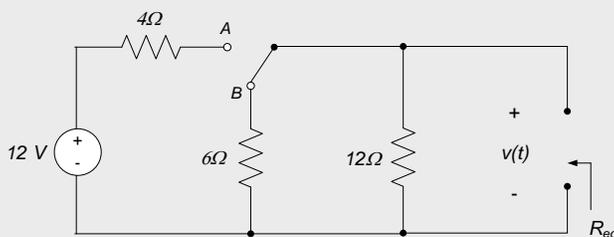
$$v(0^-) = 12V \left[ \frac{12\Omega}{12\Omega + 4\Omega} \right] = 9V$$



Since a capacitor cannot change its voltage instantaneously, the capacitor voltage just after the switch moves to position B is  $v(0^+) = v(0^-) = 9V$ , which gives us our initial condition on the capacitor voltage.

The system time constant can be determined from the capacitance and the equivalent resistance seen by the capacitor. The equivalent resistance can be determined by looking into the capacitor terminals with the switch at position B (recall that we are solving the differential equation for  $t > 0$ ). The appropriate circuit is shown below. The equivalent resistance consists of a parallel combination of the  $12\Omega$  and  $6\Omega$  resistors (note that the  $4\Omega$  resistor and the voltage source are no longer relevant to the problem – they are isolated from the capacitor after the switch changes to position B). The equivalent resistance is thus:

$$R_{eq} = \frac{(6\Omega)(12\Omega)}{6\Omega + 12\Omega} = 4\Omega$$



The system time constant is, therefore,  $\tau = R_{eq}C = (4\Omega)(25 \times 10^{-3}F) = 100 \times 10^{-3}$  seconds or 0.1 seconds. Appropriate substitution of the initial condition and time constant into equation (7.30) gives:

$$v(t) = 9e^{\frac{-t}{0.1}} = 9e^{-10t}V$$

## Section Summary

- The natural response of an RC circuit describes the capacitor voltage in a circuit consisting only of resistors and a single equivalent capacitance. The circuit is source-free; the response is entirely due to energy initially stored in the capacitor.
- The differential equation for an unforced RC circuit is of the form:

$$\frac{dv(t)}{dt} + \frac{1}{R_{eq}C}v(t) = 0$$

Where  $R_{eq}$  is the equivalent resistance “seen” by the capacitor.

- The RC circuit natural response is of the form:

$$v(t) = V_0 e^{-\frac{t}{\tau}} \text{ for } t \geq 0$$

Where  $V_0$  is the initial voltage across the capacitor and  $\tau$  is the circuit time constant.

- The time constant for any first order system can be determined from the differential equation governing the system. If the governing differential equation is written in the form:

$$\frac{dv(t)}{dt} + \frac{1}{\tau}v(t) = 0$$

The time constant  $\tau$  can be determined by inspection. Thus, by comparison with the above differential equation for RC circuits,  $\tau = R_{eq}C$ .

- Alternatively, for the special case of an RC circuit, the time constant can also be determined by:

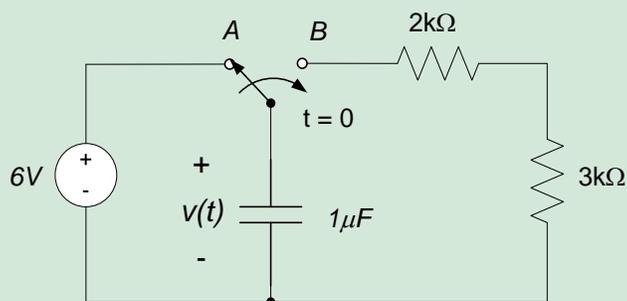
$$\tau = R_{eq}C$$

Where  $R^{eq}$  is the equivalent resistance “seen” by the capacitor and  $C$  is the capacitance in the circuit. (Notice, that to use this relation, we must accurately identify the circuit as a first order RC circuit before proceeding.)

- The capacitor properties can be useful in determining initial conditions for an RC circuit:
  - Capacitors behave like open-circuits when all circuit parameters are constant, and
  - Capacitor voltages cannot change instantaneously

## 7.2 Exercises

- The switch in the circuit below moves from position A to position B at time  $t = 0$ .
  - Write the differential equation governing  $v(t)$ ,  $t > 0$ .
  - Determine the time constant of the circuit from the differential equation of part a.
  - Use the capacitance and the equivalent resistance seen by the capacitor to check your answer to part b.
  - Determine the initial condition on the capacitor voltage,  $v(t=0^+)$
  - Determine  $v(t)$ ,  $t > 0$ .



## 7.3 Natural Response of RL Circuits

In this section, we consider source-free circuits containing only resistors and a single inductor – commonly referred to as RL circuits. Like RC circuits, these circuits contain only a single energy storage element – the inductor – and the governing equations for the circuits will be first order differential equations. Since the circuits are source-free, no input is applied to the system and the governing differential equation will be homogeneous; the response of the circuit is due entirely to any energy initially stored in the inductor. We will thus be concerned with the *natural response* of RL circuits.

We will base our discussion of RL circuit natural responses on the series resistor-inductor circuit shown in Figure 7.12. We assume that the inductor has some initial current,  $I_0$ , flowing through it at time  $t=0$  (so that  $i(0)=I_0$ ). We will determine the inductor current,  $i(t)$ , for  $t>0$ .

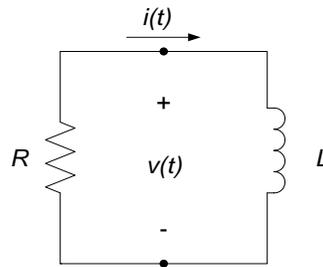


Figure 7.12. RL circuit with initial condition  $i(t=0) = I_0$ .

Since the inductor's voltage drop and current are related by  $v(t) = L \frac{di(t)}{dt}$ , application of Kirchoff's voltage law around the single loop in the circuit results in:

$$L \frac{di(t)}{dt} + Ri(t) = 0 \quad \text{Eq. 7.31}$$

In this chapter, we will solve this differential equation using the “alternate approach” presented in section 7.2 for capacitors. This approach consists of assuming a form of the solution, based on the form of the differential equation being solved. The assumed solution will contain unknown constants; these constants will be determined by plugging the assumed solution into the original differential and forcing the solution to satisfy the original differential equation and initial conditions. Since the differential equation is linear and has constant coefficients, the solution to the differential equation is unique – thus, if we can find any solution, we have found the only solution. This approach is an extremely common differential equation solution method, we will use it regularly in subsequent chapters.

The form of equation (7.31) indicates that  $i(t)$  must be a function which does not change form when it is differentiated ( $L \frac{di(t)}{dt}$  must cancel out  $Ri(t)$ ). The only function with this property is an exponential function. Thus, we assume that the current is of the form:

$$i(t) = Ke^{-st} \quad \text{Eq. 7.32}$$

Where  $K$  and  $s$  are unknown constants. Substituting equation (7.32) into equation (7.31) results in:

$$L(-Kse^{-st}) + R(Ke^{-st}) = 0$$

The above simplifies to:

$$(R - Ls)Ke^{-st} = 0$$

Which is satisfied if  $s = \frac{R}{L}$  or  $Ke^{-st} = 0$ . Choosing  $Ke^{-st} = 0$  results in the trivial solution  $i(t)=0$ , which will not, in general, satisfy the initial condition on the circuit. By the process of elimination, we choose  $s = \frac{R}{L}$  and the form of our solution becomes:

$$i(t) = Ke^{-\frac{tR}{L}} \quad \text{Eq. 7.33}$$

The unknown constant  $K$  is determined by applying the initial condition,  $i(0)=I_0$ . Evaluating equation (7.33) at time  $t=0$ , and equating the result to the given initial condition, we obtain:

$$i(0) = Ke^{\frac{-0 \cdot R}{L}} = K = I_0$$

Thus,  $K = I_0$ , and the solution to the differential equation is:

$$i(t) = I_0 e^{\frac{-tR}{L}} = I_0 e^{-\frac{t}{\tau}} \quad \text{Eq. 7.34}$$

And the circuit time constant is:

$$\tau = \frac{L}{R} \quad \text{Eq. 7.35}$$

Equation (7.35) indicates that increasing  $L$  or decreasing  $R$  causes the time constant to increase. Conversely, decreasing  $L$  or increasing  $R$  decreases the time constant. A plot of the response of equation (7.34) is shown in Fig. 7.13.

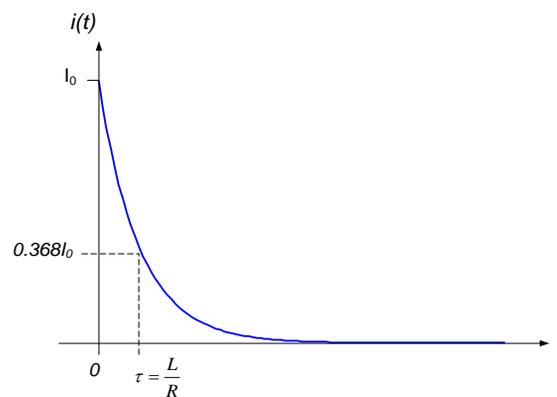


Figure 7.13. RL circuit natural response.

### 7.3.1 Generalization to Multiple Resistors

As with the RC circuit, the resistance in the time constant of equation (7.35) can be more generally defined as the equivalent overall resistance of the circuit as seen by the inductor. Thus, if we remove the inductor from the circuit and create a Thevenin equivalent resistance as seen by the inductor, the time constant will be the quotient of the inductance and the equivalent resistance.

### 7.3.2 Determining Initial Conditions

Though the initial condition is given in the above example, in general we will need to determine the initial condition from the application of some source and/or switching condition. For example, the circuit of Fig. 7.14 can be used to generate the initial condition of the example circuit above. In the circuit of Fig. 7.14, we assume that the switch has been closed for a long time and all circuit voltages and currents have become constant. When all circuit operating conditions are constant, the inductor behaves like a short circuit and all the current applied by the current source goes through the inductor and the inductor current is  $I_0$ . Since the current through an inductor cannot change suddenly, the inductor still has current  $I_0$  immediately after the switch opens.

Mathematically,  $i(t=0^-) = i(t=0^+) = I_0$ , where time  $t=0^-$  is an infinitesimal time before the switch opens, and the time  $t=0^+$  is an infinitesimal time after the circuit opens. Thus, for times  $t > 0$ , the shaded portion of the circuit of Fig. 7.14 is identical to the circuit of Fig. 7.12 from the viewpoint of the inductor current.

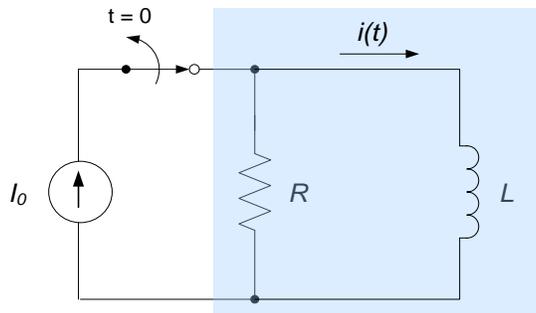
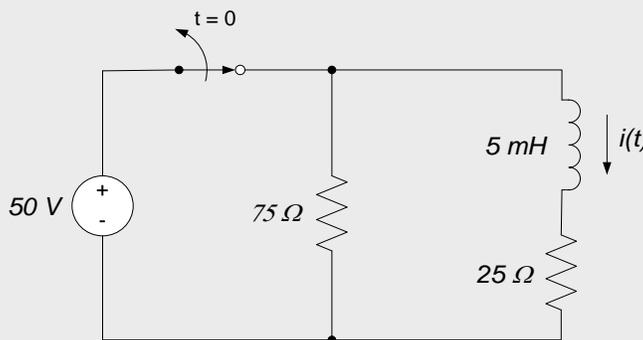


Figure 7.14. Circuit to realize the initial condition of the circuit of Figure 7.12.

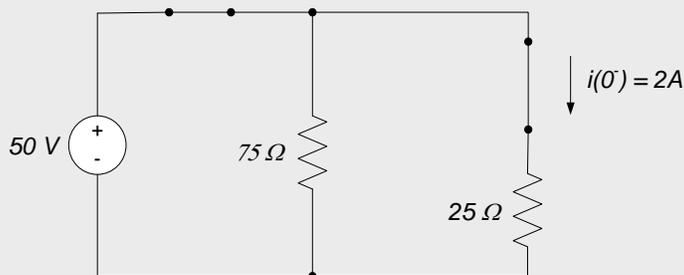
An example is provided below to illustrate the points made in this chapter.

### Example 7.3: Switched RL Circuit Natural Response

Consider the circuit shown below. The switch has been closed for a long time; at time  $t=0$  seconds, the switch suddenly opens. Determine the inductor current,  $i(t)$  for  $t>0$ .

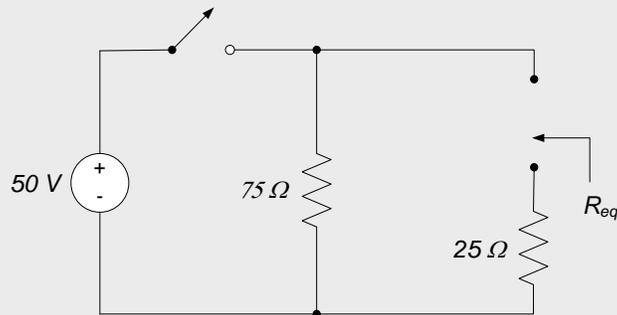


Since we are told that the switch has been closed for a long time, we assume that all voltages and currents in the circuit are constant before we open the switch. Therefore, before the switch is open the inductor behaves like a short circuit and the inductor current at time  $t=0^-$  can be determined by analyzing the circuit below:



From the above circuit, the inductor current just before the switch is opened is, from Ohm's law,  $i(0^-) = \frac{50V}{25\Omega} = 2A$ . Since an inductor cannot change its current suddenly, the current immediately after the circuit opens is  $i(0^+) = i(0^-) = 2A$ . This provides our initial condition on the inductor current.

The time constant of the circuit is determined from the inductance and the equivalent resistance of the circuit seen by the inductor, after the switch opens. The equivalent resistance can be determined by analyzing the circuit shown below.



From the circuit above it can be seen that, to the inductor, the 25Ω and 75Ω resistors are in series. Thus,  $R_{eq} = 25\Omega + 75\Omega = 100\Omega$ . The system time constant is, therefore:

$$\tau = \frac{L}{R_{eq}} = \frac{5 \times 10^{-3} H}{100\Omega} = 5 \times 10^{-5} \text{ seconds}$$

Substitution of the initial condition and time constant into equation (4) gives:

$$i(t) = 2e^{\frac{-t}{5 \times 10^{-5}}} = 2e^{-20,000t} A$$

## Section Summary

- The natural response of an RL circuit describes the inductor current in a circuit consisting only of resistors and a single equivalent inductance. The circuit is source-free; the response is entirely due to energy initially stored in the inductor.
- The differential equation for an unforced RL circuit is of the form:

$$\frac{di(t)}{dt} + \frac{R_{eq}}{L} i(t) = 0$$

Where  $R_{eq}$  is the equivalent resistance “seen” by the inductor.

- The RL circuit natural response is of the form

$$i(t) = I_0 e^{\frac{-t}{\tau}}, \text{ for } t \geq 0$$

Where  $I_0$  is the initial voltage across the capacitor and  $\tau$  is the circuit time constant.

- The time constant for a first order system can be determined from the differential equation governing the system. If the governing differential equation is written in the form:

$$\frac{di(t)}{dt} + \frac{1}{\tau} i(t) = 0$$

The time constant  $\tau$  can be determined by inspection. Thus, by comparison with the above differential equation for RL circuits,  $\tau = \frac{L}{R_{eq}}$ .

- Alternatively, for the special case of an RL circuit, the time constant can also be determined by:

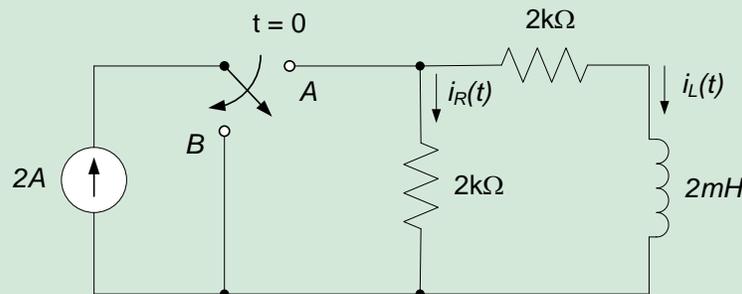
$$\tau = \frac{L}{R_{eq}}$$

Where  $R_{eq}$  is the equivalent resistance “seen” by the inductor and  $L$  is the inductance in the circuit. (Notice, that to use this relation, we must accurately identify the circuit as a first order RL circuit before proceeding.)

- The inductor properties can be useful in determining initial conditions for an RL circuit:
  - Inductors behave like short-circuits when all circuit parameters are constant, and
  - Inductors currents cannot change instantaneously

### 7.3 Exercises

1. The switch in the circuit below moves from position A to position B at time  $t = 0$ .
  - a. Write the differential equation governing  $i_L(t)$ ,  $t > 0$ .
  - b. Determine the time constant of the circuit from the differential equation of part a.
  - c. Use the inductance and the equivalent resistance seen by the inductor to check your answer to part b.
  - d. Determine the initial condition on the inductor current,  $i_L(t=0^+)$
  - e. Determine  $i_L(t)$ ,  $t > 0$ .
  - f. Determine the resistor current  $i_R(t)$  just before and just after the switch moves. (e.g. determine  $i_R(t=0^-)$  and  $i_R(t=0^+)$ .) Is the resistor current continuous with time? Is the current through the other resistor continuous with time?



### 7.4 Forced Response of First Order Circuits

In sections 7.2 and 7.3, we were concerned with the natural response of electrical circuits containing a single energy storage element. For the natural response, any sources in the circuit were isolated from the circuit prior to determining the circuit response, so that the circuit being analyzed contained no sources. Thus, the circuit response of interest was entirely due to the energy initially stored in the circuit’s capacitors or inductors. In these cases, all voltages and currents in the circuit die out with time.

In this section, we consider the case in which voltage or current sources are present in the first order circuit being analyzed. In this case, we must concern ourselves not only with the initial conditions in the circuit, but also with any *driving or forcing* functions applied to the circuit. The response of a system in the presence of an external input such as a voltage or current source is commonly called the *forced response* of the system. A primary difference between the natural response and the forced response of a system is that, although the natural response of a system always decays to zero, the forced response has no such restriction. In fact, the forced response of the system will take the same form as the forcing function, as time goes to infinity.

The differential equations governing the forced response of first order circuits are still, as implied, first order – thus, the circuits presented here will contain only a single energy storage element. Figure 7.15 shows two examples of forced first order circuits; Fig. 7.15(a) is a forced RC circuit and Fig. 7.15(b) is a forced RL circuit. The voltage sources  $v_s(t)$  in Fig. 7.15 provide an arbitrary input voltage which can vary as a function of time. Without loss of generality, we will concern ourselves only with determining the forced response of the voltages across capacitors or the currents through inductors. In spite of the simplicity of the circuits shown in Fig. 7.15, their



We notice now that the  $\frac{1}{RC}$  term in equation (7.37) corresponds to  $\frac{1}{\tau}$ , where  $\tau$  is the time constant of an RC circuit. Likewise, we note that the RL term in equation (7.39) corresponds to  $\frac{1}{\tau}$ , where  $\tau$  is the time constant of an RL circuit. Thus, both equation (7.37) and equation (7.39) are of the form:

$$\frac{dy(t)}{dt} + \frac{1}{\tau}y(t) = u(t) \quad \text{Eq. 7.40}$$

Where  $\tau$  is the time constant of the system,  $u(t)$  is the input to the system, and  $y(t)$  is the desired system parameter (a voltage across a capacitor or a current through an inductor, for example). Equation (7.40) can be solved, given knowledge of the initial conditions on  $y(t)$ ,  $y(0)=y_0$ . A block diagram of the system described by equation (7.40) is shown in Fig. 7.16.

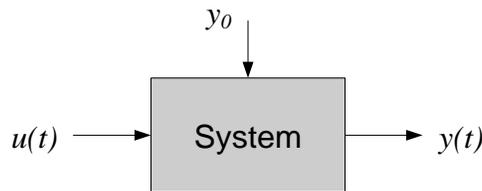


Figure 7.16. Block diagram of general forced first order system.

The solution to any forced differential equation can be considered to be formed of two parts: the *homogeneous solution* or *natural response* (which characterizes the portion of the response due to the system's time constant and initial conditions) and the *particular solution* or *forced response* (which characterizes the system's response to the forcing function  $u(t)$  after the natural response has died out). Thus, the system response  $y(t)$  in equation (7.40) and Fig. 7.16 can be expressed as:

$$y(t) = y_h(t) + y_p(t) \quad \text{Eq. 7.41}$$

Where  $y_h(t)$  is the homogeneous solution and  $y_p(t)$  is the particular solution.

We will not attempt to analytically determine the solution of equation (7.40) for the general case of an arbitrary forcing function  $u(t)$ ; instead, we will focus on specific types of inputs. Inputs of primary interest to us will consist of:

- Constant (step) input functions
- Sinusoidal input functions

The study of circuit responses to step functions is provided in Section 7.5. Sinusoidal input functions are discussed in later chapters.

### 7.4.1 Generalization to Multiple Resistors

As in the first order circuit natural response, the resistance in the time constant of equation (7.37) can be more generally defined as the equivalent overall resistance of the circuit as seen by the energy storage element. This conclusion follows directly from Thévenin's theorem. The circuits of Fig. 7.15 consist of energy storage elements (a capacitor and an inductor) which are connected to a circuit which can be considered to be the Thévenin equivalent circuit of a more complex circuit. Thus, the resistance  $R$  in the circuits of Fig. 7.15 can be the equivalent (Thévenin) resistance of an arbitrary linear circuit to which an energy storage element is connected. Thus, the resistances in the governing differential equations (7.37) and (7.39) can be considered to be Thévenin equivalent resistances. These equations thus can be written as:

$$\frac{dv_c(t)}{dt} + \frac{1}{R_{eq}C} v_c(t) = \frac{1}{R_{eq}C} v_s(t) \quad \text{Eq. 7.42}$$

For an RC circuit, and :

$$\frac{di_L(t)}{dt} + \frac{R_{eq}}{L} i_L(t) = \frac{1}{L} v_s(t) \quad \text{Eq. 7.43}$$

For an RL circuit, where  $R_{eq}$  is the equivalent (Thévenin) resistance seen by the energy storage element. This leads to the conclusion that the time constants for first order forced circuits can be written in terms of the Thevenin resistance seen by the energy storage element. The appropriate relationships are:

$$\tau = R_{eq}C \quad \text{Eq. 7.44}$$

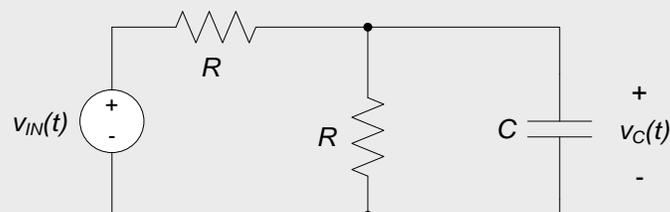
For RC circuits, and:

$$\tau = \frac{L}{R_{eq}} \quad \text{Eq. 7.45}$$

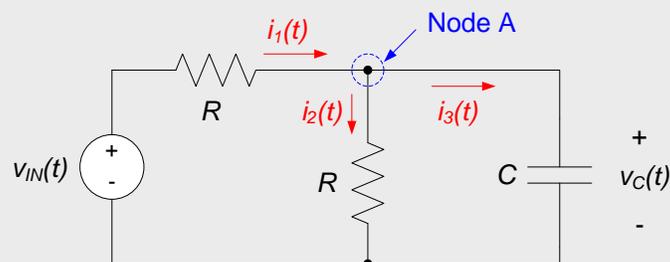
For RL circuits. Note that this conclusion is consistent with our previous results for unforced RC and RL circuits. We conclude this section with several examples in which we determine the differential equations governing first order electrical circuits. Note that we make no attempt to solve these differential equations – in fact, we cannot solve the differential equations, since we have not specified what the forcing functions are in the circuits below.

### Example 7.4

Determine the differential equation relating  $v_{in}(t)$  and  $v_c(t)$  in the circuit below.



We will apply KCL at node "A", as indicated in the figure below, to begin our analysis.



Thus,

$$i_1(t) = i_2(t) + i_3(t)$$

Using the voltage-current relations to write these currents in terms of voltages results in:

$$\frac{v_{in}(t) - v_c(t)}{R} = \frac{v_c(t)}{R} + C \frac{dv_c(t)}{dt}$$

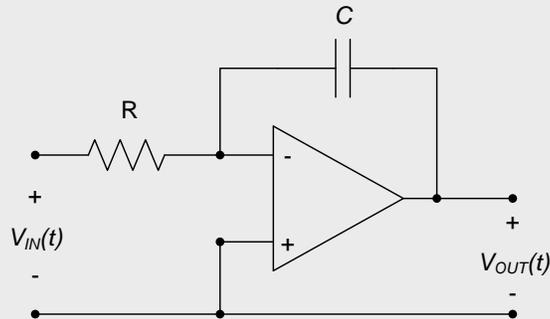
After a little algebra, the above results can be written in our standard first order circuit form as:

$$\frac{dv_c(t)}{dt} + \frac{2}{RC}v_c(t) = \frac{1}{RC}v_{in}(t)$$

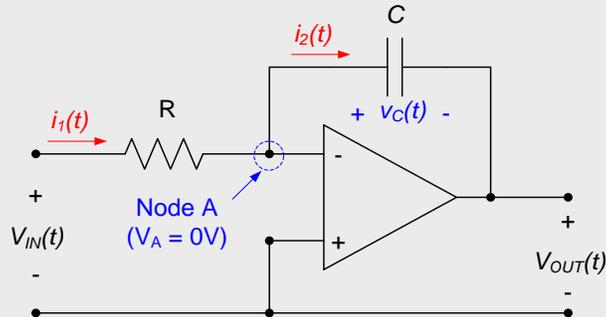
And the time constant of the circuit is  $\tau = \frac{RC}{2}$ .

### Example 7.5

Determine the differential equation relating  $V_{in}(t)$  and  $V_{out}(t)$  in the circuit below.



Consistent with our approach of defining variables as voltages across capacitors and currents through inductors, we define the capacitor voltage as  $v_c(t)$ , as shown in the figure below. Also in the figure below, node A is defined and the rules governing ideal op-amps are used to identify the node voltage  $V_A = 0V$ .



Applying KCL at node A in the circuit above gives:

$$i_1(t) = i_2(t)$$

The currents can be written in terms of the voltages in the circuit to provide:

$$\frac{V_{in}(t) - 0}{R} = C \frac{dv_c(t)}{dt}$$

The capacitor voltage can be written in terms of  $V_{OUT}$  (using KVL) as:

$$V_{OUT}(t) = -v_c(t)$$

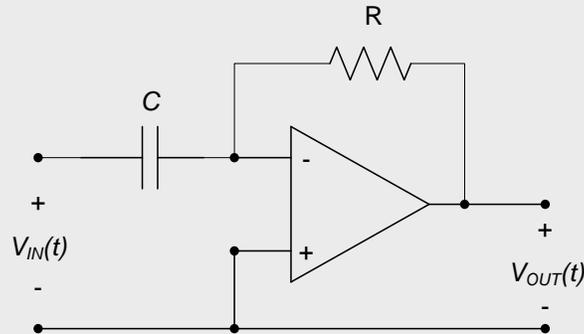
Thus, the governing differential equation for this circuit can be written as:

$$V_{OUT} = -\frac{1}{RC} \int V_{in}(t) dt$$

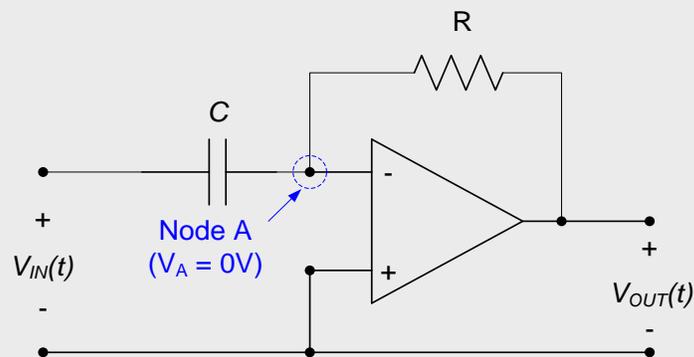
Note that this circuit is performing an integration.

### Example 7.6

Determine the differential equation relating  $V_{in}(t)$  and  $V_{out}(t)$  in the circuit below.



As in example 7.5, we define node A is defined and use the rules governing ideal op-amps to identify the node voltage  $V_A=0V$ , as shown in the figure below.



Writing KCL at node A directly in terms of the node voltages results in:

$$C \frac{dV_{in}(t)}{dt} = -\frac{V_{OUT}}{R}$$

So that:

$$V_{OUT} = -RC \frac{dV_{in}(t)}{dt}$$

And the output voltage is proportional to the derivative of the input voltage.

## 7.4.2 Cross-checking Results

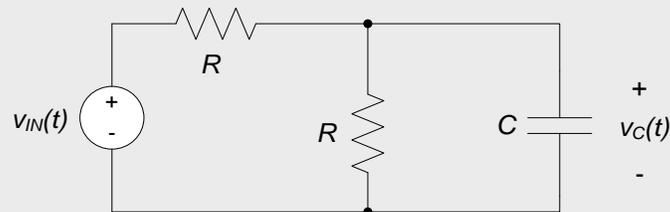
The above examples revolve entirely around determining the governing differential equation for the circuit. The actual circuit response depends upon the governing differential equation, the initial conditions, and the specific forcing function being applied to the circuit. In the above examples, the circuit time constants were inferred from the differential equation coefficients governing the forced response just as they were when we determined the natural response.

It is always desirable to check one's results in as many ways as possible. With this in mind, we would like to check to see if the differential equation we have written for a given electrical circuit makes sense before solving the equation for a specific forcing function. For first order circuits, at least, we can do this by determining a time constant directly from the circuit itself and comparing this time constant with the time constant inferred from the governing differential equation. The time constant of any first order forced circuit can be obtained by calculating the Thévenin resistance seen by the energy storage element and using equations (7.44) and (7.45) to provide the time constant. We will now revisit example 7.4 using this approach to validate the differential equation we previously determined for this circuit.

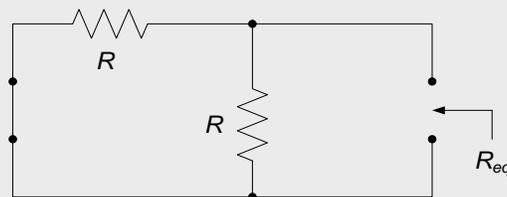
### Example 7.7

Check the time constant for the circuit of Example 7.4 by calculating the equivalent resistance seen by the capacitor.

The circuit of example 7.4 is shown below for reference.



We can determine the equivalent resistance seen by the capacitor by replacing the voltage source with a short circuit and looking at the resistance seen across the capacitor's terminals, as shown below:



The resistors are now in parallel, so that the equivalent resistance is  $R/2$ . The time constant is then  $\tau = R_{eq}C = \frac{RC}{2}$ , which agrees with the result of example 7.4

### Section Summary

- The forced response of a first order circuit describes the response of the circuit in the presence of (in general) both a non-zero initial condition and an arbitrary time-varying input function.
- The differential equation describing the forced response of a first order circuit is of the form:

$$\frac{dy(t)}{dt} + \frac{1}{\tau}y(t) = f(t)$$

Where  $f(t)$  is the forcing function applied to the circuit. The time constant,  $\tau$ , of the circuit is readily obtained from the differential equation when it is written in the above form.

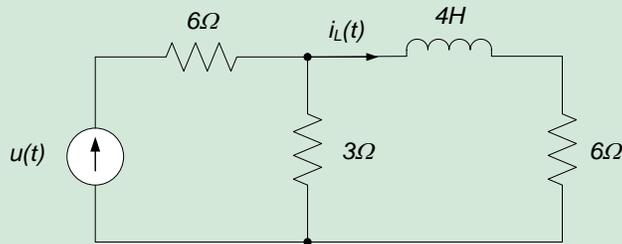
- The time constant for a first order forced system can also be determined directly from the circuit itself. The process is to determine the Thévenin resistance,  $R_{eq}$ , seen by the energy storage element and use

that resistance in the appropriate time constant formula as introduced in sections 7.2 and 7.3. For an RL circuit, the time constant is  $\tau = \frac{L}{R_{eq}}$ , while for an RC circuit the time constant is  $\tau=R_{eq}C$ .

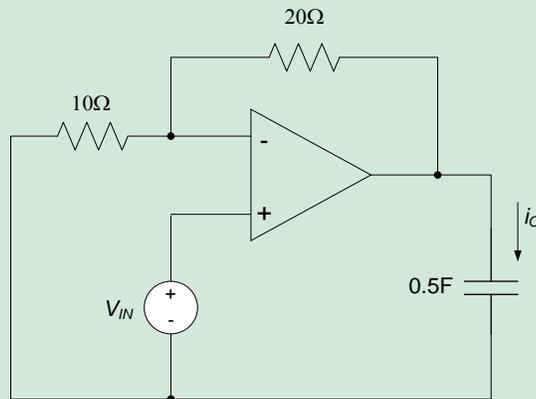
- Although the time constant can be determined from either the governing differential equation or the circuit itself, it is strongly recommended that both approaches be used and the results compared with one another to provide a cross-check of your analysis.

### 7.4 Exercises

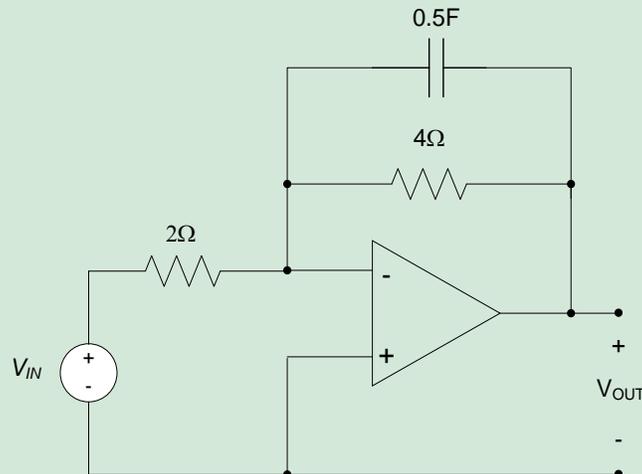
1. For the circuit below, write the differential equation governing  $i_L(t)$ . The input is the current source,  $u(t)$ .



2. Determine the differential equation governing  $i_C(t)$  in the circuit below:



3. Determine the differential equation governing  $V_{out}(t)$  in the circuit below.



## 7.5 Step Response of First Order Circuits

In section 7.4 we introduced the concept of the response of a first order circuit to an arbitrary forcing function. We will not attempt to solve this problem for an arbitrary forcing function; we will instead restrict our attention to specific forcing functions. In this section, we address the case in which the input consists of the sudden application of a constant voltage or current to a circuit; this type of input is typically modeled as a step function. The response of a system to this type of input, in the absence of any initial conditions, is called the *step response* of the system.

Figure 7.17 shows a conceptual circuit which applies a step input to an RC circuit, and an actual switched circuit which may be used to implement this forcing function.

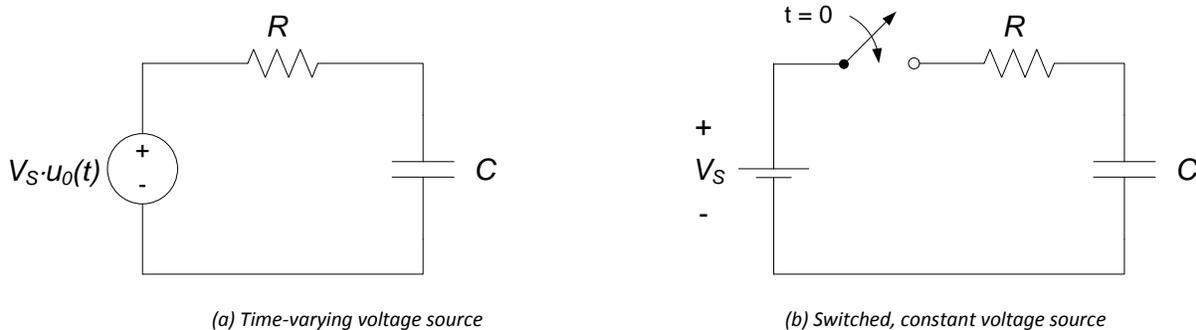


Figure 7.17. Circuits to provide step input to RC circuit.

In section 7.3, we saw that the differential equation governing the forced response of a first order circuit is of the form:

$$\frac{dy(t)}{dt} + \frac{1}{\tau}y(t) = u(t) \quad \text{Eq. 7.46}$$

where  $\tau$  is the time constant of the system,  $u(t)$  is the input to the system, and  $y(t)$  is the desired system parameter (for example, the voltage across a capacitor or the current through an inductor). In the case of a step input to the system, the input  $u(t)$  to the system is a constant,

$$u(t) = Au_0(t) \quad \text{Eq. 7.47}$$

Where  $u_0(t)$  is the unit step function.

We also saw, in section 7.3, that the solution of equation (7.46) can be written as the superposition of a homogeneous solution and the particular solution,

$$y(t) = y_h(t) + y_p(t) \quad \text{Eq. 7.48}$$

The *homogeneous solution*,  $y_h(t)$ , characterizes the portion of the response due to the system's time constant and initial conditions while the *particular solution*,  $y_p(t)$ , characterizes the system's response to the forcing function  $u(t)$  after the natural response has died out.

The homogeneous response is the solution to the homogeneous differential equation:

$$\frac{dy(t)}{dt} + \frac{1}{\tau}y(t) = 0 \quad \text{Eq. 7.49}$$

In Section 7.1, we showed that the form of the solution of the homogeneous response is:

$$y_h(t) = K_1 e^{-\frac{t}{\tau}} \quad \text{Eq. 7.50}$$

However, now we cannot determine  $K_1$  directly at this point. Any conditions we can generate from the circuit with which to determine unknown coefficients apply to the entire forced solution, not the homogeneous or particular solution individually.

Our next step, therefore, is to determine the particular solution by substituting the input of equation (7.47) into the differential equation (7.46) and solving for  $y_p(t)$ . The appropriate differential equation is, therefore:

$$\frac{dy_p(t)}{dt} + \frac{1}{\tau}y_p(t) = Au_0(t) \quad \text{Eq. 7.51}$$

The particular solution is appropriate after the homogeneous solution dies out. Thus, we evaluate equation (7.51) as  $t \rightarrow \infty$ . The right-hand side of equation (7.51) is a constant value as  $t \rightarrow \infty$ , since:

$$Au_0(t) = \begin{cases} 0, & t < 0 \\ A, & t > 0 \end{cases}$$

If the right-hand side of equation (7.51) is a constant, then the left-hand side of equation (6) must be a constant and the individual terms in the left-hand side of equation (7.51) must be constants. It follows, then, that  $y_p(t)$  is a constant and that  $\frac{dy_p}{dt} = 0$ . Therefore, as  $t \rightarrow \infty$ , equation (7.51) becomes:

$$\frac{1}{\tau}y_p(t) = A \Rightarrow y_p(t) = K_2 \quad \text{Eq. 7.52}$$

The overall solution, then, from equations (7.48), (7.50), and (7.52) is:

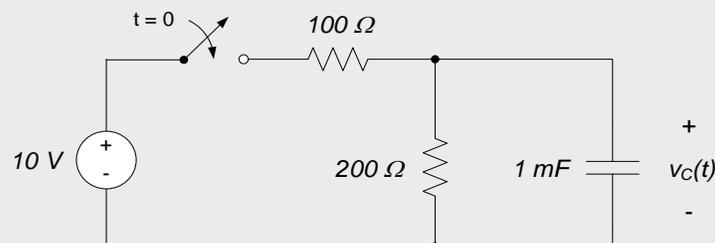
$$y(t) = K_1 e^{-\frac{t}{\tau}} + K_2 \quad \text{Eq. 7.53}$$

The unknown constants  $K_1$  and  $K_2$  are typically determined from evaluating the circuit's behavior for  $t=0$  and  $t \rightarrow \infty$ .

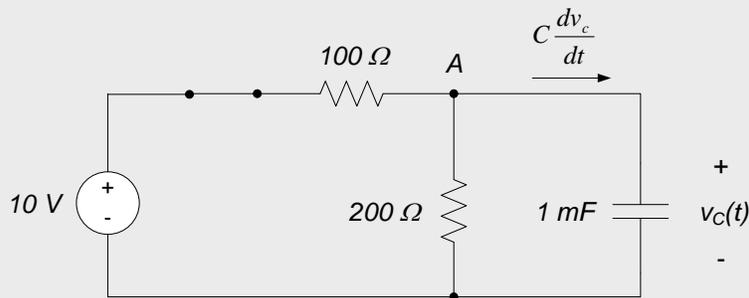
We illustrate the overall solution process with three examples below. The first two examples are of passive first order circuit responses; the third example is of an active first order circuit.

### Example 7.8

Determine the voltage across the capacitor,  $v_C(t)$  for  $t > 0$  in the circuit below. The switch has been open for a long time, and the initial voltage across the capacitor is zero.



When the switch is closed, the circuit is as shown below. The capacitor current has been labeled for later convenience. Note that the direction of the capacitor current is set to agree with the polarity of the capacitor voltage, according to the passive sign convention.



KCL at node “A” of the circuit shown above results in:

$$\frac{10 - v_c}{100\Omega} = \frac{v_c}{200\Omega} + 1 \times 10^{-3} \frac{dv_c}{dt}$$

Placing this in the form of equation (7.46) results in:

$$\frac{dv_c}{dt} + 15v_c = 100$$

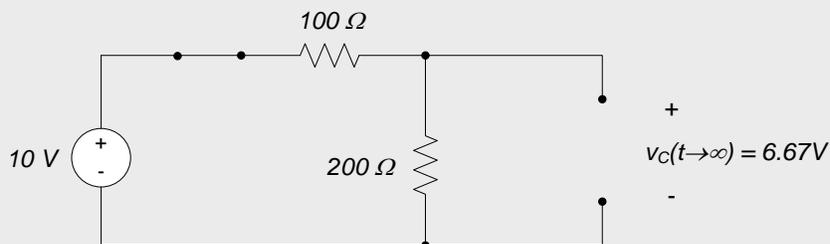
Thus, the time constant  $\tau = \frac{1}{15}$  seconds. From equation (7.53) above, the form of the solution is:

$$v_c(t) = K_1 e^{-\frac{t}{\tau}} + K_2$$

We now apply the given initial condition,  $v_c(0)=0$  to get:

$$0 = K_1 e^{-\frac{t}{\tau}} + K_2 \Rightarrow K_1 + K_2 = 0 \tag{*}$$

Another condition for determining the unknown constants is the capacitor voltage as  $t \rightarrow \infty$ . As  $t \rightarrow \infty$ , for a constant input, the capacitor becomes an open circuit. Thus, the circuit above becomes:



And the capacitor voltage can be determined from voltage division to be:

$$v_c(t \rightarrow \infty) = 10V \frac{200\Omega}{100\Omega} + 200\Omega = 6.67V$$

Substituting this result into the expression for the capacitor voltage results in:

$$6.67V = K_1 e^{-\frac{\infty}{\tau}} + K_2 \Rightarrow K_2 = 6.67V \tag{**}$$

Equations (\*) and (\*\*) provide two equations in two unknowns. Solving these results in:

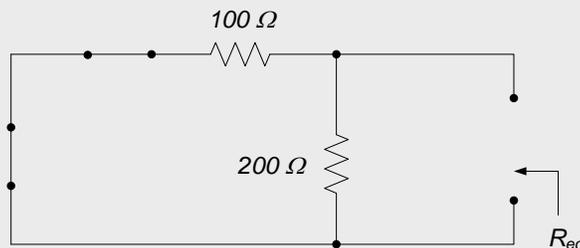
$$K_1 = -6.67V$$

$$K_2 = 6.67V$$

And the capacitor voltage becomes:

$$v_c(t) = 6.67(1 - e^{-15t})$$

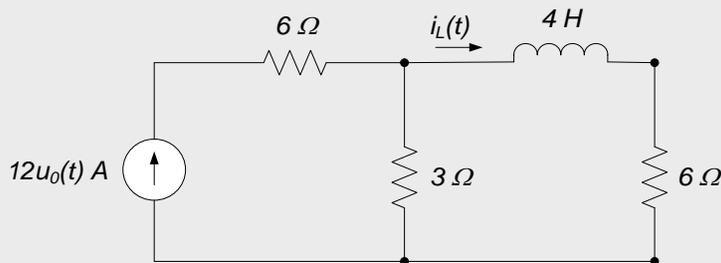
The value for  $\tau$  can be checked by determining the equivalent resistance seen by the capacitor. To do this, we kill the sources and look into the capacitor terminals. The appropriate circuit is shown below.



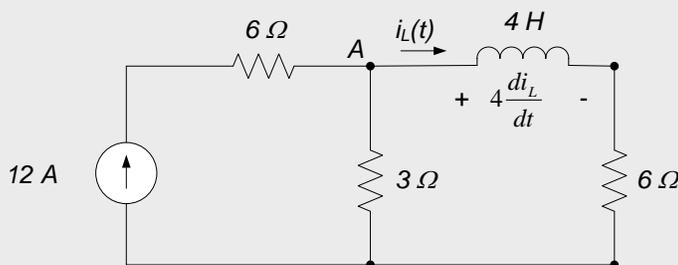
The equivalent resistance is  $R_{eq} = \frac{(100\Omega)(200\Omega)}{100\Omega+200\Omega} = 66.67\Omega$  and  $\tau = R_{eq}C = (66.67\Omega)(1mH) = \frac{1}{15}$  seconds, which checks our previous result.

### Example 7.9

Determine the current through the inductor,  $i_L(t)$ , in the circuit below. No energy is stored in the circuit prior to  $t=0$  seconds. The applied current input consists of a 12A step input applied at  $t=0$ .



The circuit is shown below for  $t>0$ , with the inductor current labeled for ease of reference.



KVL around the rightmost loop in the circuit results in the following voltage across the 3Ω resistor:

$$v_{3\Omega} = 4 \frac{di_L}{dt} + 6i_L$$

Employing this result and applying KCL at node A results in:

$$-12 = i_L + \frac{1}{3\Omega} \left( 4 \frac{di_L}{dt} + 6i_L \right)$$

Placing the above equation in the form of equation (7.46) results in:

$$\frac{di_L}{dt} + \frac{1}{4}i_L = 9$$

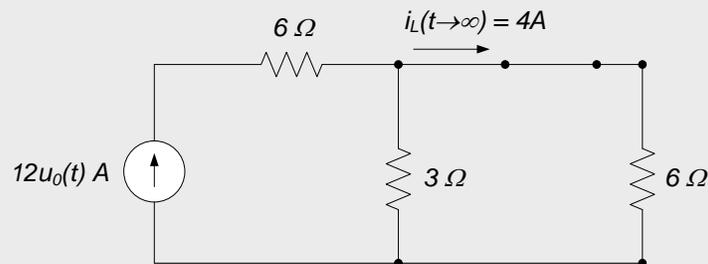
From this, we see that the circuit time constant is  $\tau=49$  and the form of  $i_L(t)$  is, from equation (7.53):

$$i_L(t) = K_1 e^{\frac{-4t}{9}} + K_2$$

From the given initial condition:

$$i_L(0) = K_1 + K_2 = 0$$

As  $t \rightarrow \infty$ , the inductor becomes a short circuit and the above circuit becomes:



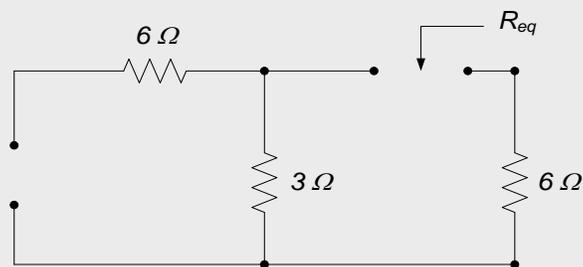
Current division allows us to determine that  $i_L(t \rightarrow \infty) = 12A \frac{3\Omega}{3\Omega+6\Omega} = 4A$ . Substituting this into equation governing  $i_L(t)$ , we obtain:

$$i_L(t \rightarrow \infty) = 4A = K_1 e^{-\infty} + K_2 \Rightarrow K_2 = 4A$$

Thus, the current through the inductor is:

$$i_L(t) = 4 \left( 1 - e^{\frac{-4t}{9}} \right)$$

Note that, once again, we can check our value for the time constant by killing any sources and determining the equivalent resistance seen by the inductor. The appropriate circuit is shown below:

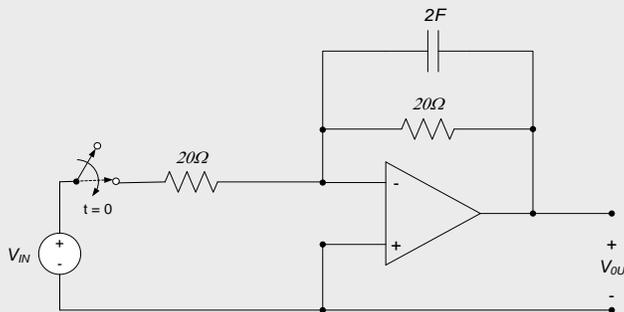


From the above circuit, the equivalent resistance is simply a series combination of the 3Ω and 6Ω resistors.

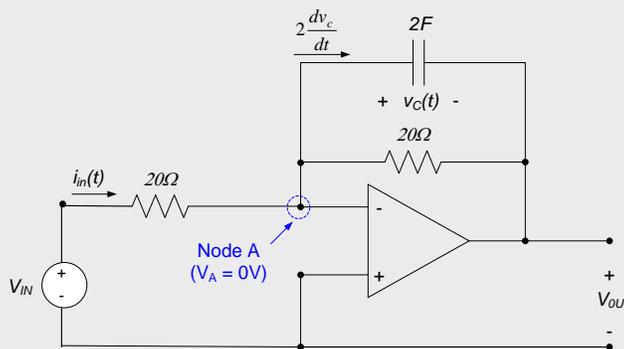
Thus,  $R_{eq}=9\Omega$ . For an RL circuit, the time constant  $\tau = \frac{L}{R} = \frac{4}{9}$  seconds. This agrees with the previous result we obtained from examining the form of the governing differential equation.

### Example 7.10

The switch in the circuit below closes at time  $t=0$ . Find  $V_{OUT}(t)$ , for  $t>0$ . The capacitor has no energy stored in it prior to  $t=0$ .



For time  $t>0$ , the switch is closed and the circuit is as shown on the figure below. Labeling node A on this circuit as shown, we can determine from the ideal operational amplifier rules that  $V_A=0V$ .



Applying KCL at node A results in:

$$\frac{V_{in} - 0}{20\Omega} = \frac{v_C(t)}{20\Omega} + 2 \frac{dv_C(t)}{dt}$$

Since  $v_C(t) = 0 - V_{OUT}(t)$ , the above equation can be written as:

$$\frac{V_{IN}}{20\Omega} = \frac{0 - V_{OUT}(t)}{20\Omega} - 2 \frac{dV_{OUT}(t)}{dt}$$

Placing this in the form of equation (7.46) results in:

$$\frac{dV_{OUT}(t)}{dt} + 40V_{OUT}(t) = -V_{IN}$$

Thus, the time constant  $\tau=40$  sec. From equation (7.53) above, the form of the solution is:

$$V_{OUT}(t) = K_1 e^{\frac{-t}{\tau}} + K_2$$

We now apply the given initial condition,  $v_C(0)=0$  to get:

$$0 = K_1 e^{\frac{-0}{\tau}} + K_2 \Rightarrow K_1 + K_2 = 0 \tag{*}$$

The output voltage as  $t \rightarrow \infty$  can be determined from by open-circuiting the capacitor and analyzing the resulting circuit. The circuit with the capacitor open-circuited is simply an inverting voltage amplifier with a gain of one, so:

$$V_{OUT}(t \rightarrow \infty) = -V_{IN}$$

Substituting this result into the expression for the output voltage results in:

$$-V_{IN} = K_1 e^{\frac{-\infty}{\tau}} + K_2 \Rightarrow K_2 = -V_{IN} \quad (**)$$

Equations (\*) and (\*\*) provide two equations in two unknowns. Solving these results in:

$$K_1 = V_{IN}$$

$$K_2 = -V_{IN}$$

And the output voltage is:

$$V_{OUT}(t) = -V_{IN} \left( 1 - e^{\frac{-t}{\tau}} \right)$$

## 7.5.1 Cross-checking Results

As in section 7.4, the examples above emphasize writing the governing differential equation for the circuit and determining the time constant from this differential equation. We also noted in section 7.4 that cross-checking results is crucial to producing reliable analyses, and that the time constant can also be determined directly from the circuit by calculating an equivalent resistance seen by the energy storage element. This cross-check was performed in exercises 7.8 and 7.9 above, as it should be.

For the case of a constant (or step) forcing function, we can use the final value of the solution as an additional cross-check of our results. In the above examples, we determined the final value of the response,  $y(t \rightarrow \infty)$ , based on the circuit behavior with capacitors replaced by open circuits and inductors replaced by short circuits. We can also determine the final value directly from the differential equation itself by examining the response of the differential equation as  $t \rightarrow \infty$ . This value is called the *steady-state* response of the circuit.

The general differential equation governing the step response of a first order circuit is given by equation (7.51) above. If we examine the solution to this differential equation as  $t \rightarrow \infty$ , we obtain only the particular solution,  $y_p(t)$ . (The homogeneous solution must go to zero as  $t \rightarrow \infty$ , leaving only the particular solution.) Thus, as  $t \rightarrow \infty$  equation (7.51) becomes:

$$\frac{dy_p(t)}{dt} + \frac{1}{\tau} y_p(t) = A \quad \text{Eq. 7.54}$$

Since the particular solution has the same form as the forcing function, and the forcing function is a constant, the derivative of the particular solution with respect to time is zero, and equation (7.54) becomes:

$$\frac{1}{\tau} y_p = A$$

And the steady state response,  $y_{ss}$ , is  $y_{ss} = A \cdot \tau$ . This value must agree with the final value obtained by short-circuiting inductors, open circuiting capacitors, and determining the final value from the circuit itself.

We apply this cross-check to the circuits of examples 7.8 and 7.9 below.

### Example 7.11

In example 7.8, we determined (directly from the circuit behavior) that the final value of the capacitor voltage was:

$$v_C(t \rightarrow \infty) = 6.67V$$

And that the governing differential equation for the circuit was:

$$\frac{dv_C}{dt} + 15v_C = 100$$

We wish to check our final value of capacitor voltage relative to the differential equation behavior.

We can determine the final value of capacitor voltage by assuming that the voltage in the differential equation is constant, and setting its derivative to zero. Thus, the steady-state capacitor voltage can be determined from:

$$15v_{ss} = 100$$

So that  $v_{ss} = \frac{100}{15} = 6.67V$  and agrees with the final value obtained by replacing the capacitor with an open circuit, as was done in example 7.8.

### Example 7.12

In example 7.9, we determined (directly from the circuit behavior) that the final value of the inductor current was:

$$i_L(t \rightarrow \infty) = 4A$$

and that the governing differential equation for the circuit was:

$$\frac{di_L}{dt} + \frac{9}{4}i_L = 9$$

So that  $i_{ss} = 9 \cdot \frac{4}{9} = 4A$  which agrees with the final value obtained by replacing the inductor with a short circuit, as was done in example 7.9.

## 7.5.2 DC Gain

The steady-state response of a circuit to a step input provides an important parameter which is often used to characterize the circuit's behavior. This parameter is called the DC gain, and is essentially the steady state response, normalized by the magnitude of the input step function. Mathematically, if the amplitude of the input step is  $A$ , the DC gain is given by:

$$DC \text{ gain} = \frac{v_{ss}}{A} \tag{Eq. 7.56}$$

So the DC gain is simply the ratio of the output amplitude to the input amplitude, as  $t \rightarrow \infty$ . This parameter is of comparable importance to the characterization of first order circuits as the time constant. If we know the time constant and the DC gain of the circuit, we can immediately sketch the response of the circuit to any step input.

### Example 7.13

Determine the DC gain for the circuit of example 7.8.

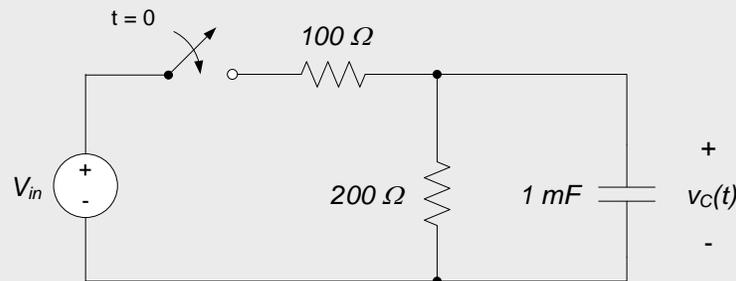
In example 7.8, the input voltage amplitude was 10V. The steady state output, the capacitor voltage, had an amplitude of:

$$v_C(t \rightarrow \infty) = 6.67V$$

Thus, the DC gain is simply the ratio of the input magnitude to the (steady-state) output magnitude:

$$DC \text{ gain} = \frac{6.67V}{10V} = 0.67$$

The DC gain can also be determined from the governing differential equation. This is probably easiest to do if we replace the original 10V source with an arbitrary voltage,  $V_{in}$ , as shown below.



Re-deriving the governing differential equation, as was done in example 7.8, results in:

$$\frac{dv_C}{dt} + 15v_C = 10V_{IN}$$

If we are only concerned with the steady-state response, the derivative term can be set to zero and:

$$15v_{SS} = 10V_{IN}$$

So that  $\frac{v_{SS}}{V_{in}} = \frac{10}{15} = 0.667$

Which agrees with the DC gain determined by calculating the steady state response to a specific input voltage and taking the same ratio.

## Section Summary

- The step response of a first order circuit describes the response of a first order circuit to an applied step function. Typically, the term “step response” implies that all initial conditions in the circuit are zero, but this is not a requirement for application of any of the concepts presented in this section
- The differential equation describing the forced response of a first order circuit is of the form:

$$\frac{dy(t)}{dt} + \frac{1}{\tau}y(t) = Au_0(t)$$

Where  $u_0(t)$  is the unit step function defined in section 6.2. The time constant,  $\tau$ , of the circuit is readily obtained from the differential equation when it is written in the above form.

- The form of the step response of a first order system is:

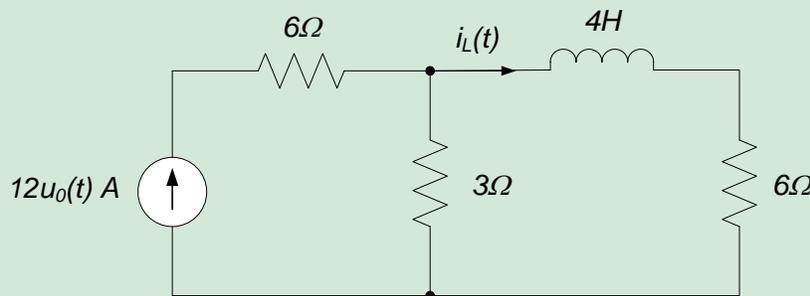
$$y(t) = K_1 e^{-\frac{t}{\tau}} + K_2$$

- The time constant in the solution above can be determined from either the governing differential equation or directly from the circuit itself.

- The unknown constants in the response are determined from initial,  $y(0^+)$ , and final,  $y(t \rightarrow \infty)$ , conditions. The initial conditions must be determined from the circuit itself. The final conditions can be determined from either the circuit itself or from the governing differential equation.
- Although both the time constant and the final value of the response can be determined from either the governing differential equation or the circuit itself, it is strongly recommended that both approaches be used and the results compared with one another to provide a cross-check of your analysis.
- The DC gain of a circuit provides the ratio of the output amplitude to the input amplitude, as  $t \rightarrow \infty$ , if the input is a constant value. The DC gain and the time constant are often used to characterize the response of a first order circuit.

## 7.5 Exercises

- The initial current in the circuit below is zero. (e.g.  $i_L(t = 0^-) = 0A$ .)
  - Write the differential equation governing  $i_L(t)$ .
  - From your result in part 1, determine the time constant of the circuit.
  - Write the form of the current  $i_L(t)$ .
  - Use conditions at  $t=0$  and  $t \rightarrow \infty$  to determine the unknown constants in the expression for  $i_L(t)$  in part 3.
  - Determine the equivalent resistance seen by the inductor to check your answer from part b.

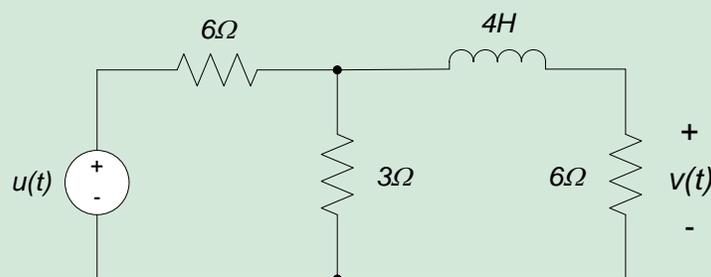


- An input voltage input,  $v(t)$ , is applied to a first order electrical circuit. The differential equation governing the resulting current,  $i(t)$ , through an inductor is determined to be:

$$2 \frac{di(t)}{dt} + 3i(t) = 5v(t)$$

What is the DC gain of the circuit? What are the units of DC gain for this circuit? What is the time constant of the circuit?

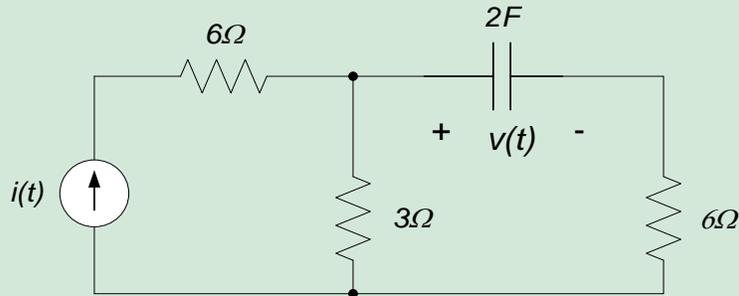
- What is the DC gain of the circuit below?  $u(t)$  is the voltage input to the circuit and  $v(t)$  is the response. What are the units of the DC gain for this circuit?



- The differential equation governing the circuit shown below is determined to be:

$$\frac{dv(t)}{dt} + \frac{1}{9}v(t) = i(t)$$

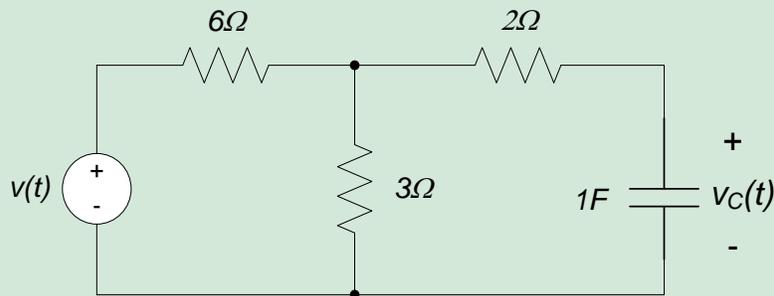
Where the current  $i(t)$  is the input to the circuit and the voltage  $v(t)$  is the circuit response. Without re-deriving the differential equation governing the circuit, do you feel that the given differential equation above accurately describes the circuit response?



5. The differential equation governing the circuit shown below is determined to be:

$$3 \frac{dv_C(t)}{dt} + v_C(t) = v(t)$$

Where the voltage  $v(t)$  is the input to the circuit and the capacitor voltage  $v_C(t)$  is the circuit response. Without re-deriving the differential equation governing the circuit, do you feel that the given differential equation above accurately describes the circuit response? Justify your answer.



## Real Analog Chapter 7: Lab Projects

### 7.2.1: Passive RC Circuit Natural Response

In this lab assignment, we will examine the natural response of a simple RC circuit. We will use both a manual switching operation and a square wave voltage source to create our circuit's natural response. We will see that the method used to create the response affects the circuit being measured.

#### Before beginning this lab, you should be able to:

- Determine the time constant of exponential functions
- Determine the natural response of passive RC circuits
- Correctly implement an electrolytic capacitor (Lab 6.3.2)
- Use the Analog Discovery 2 waveform generator to apply a time-varying voltage input to an electrical circuit (Lab 6.2.1)
- Use the Analog Discovery 2 oscilloscope to measure and display time-varying waveforms

#### After completing this lab, you should be able to:

- Use a manual switching operation to create the natural response of a first order circuit
- Use the trigger on the Analog Discovery 2 oscilloscope to acquire a signal
- Be able to explain in your own words the difference between continuous and single-sequence data acquisition
- Use the Analog Discovery 2 waveform generator to create the natural response of a first order circuit.
- Measure the initial condition and time constant of a first order circuit natural response

#### This lab exercise requires:

- Analog Discovery 2 module
- Digilent Analog Parts Kit

#### Symbol Key:

- DEMO** Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
- ANALYSIS** Analysis; include principle results of analysis in laboratory report.
- SIM** Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
- DATA** Record data in your lab notebook.

#### General Discussion:

The basic RC circuit being used in this assignment is shown in Fig. 1. We will be interested primarily in the measured vs. expected behavior of the capacitor voltage,  $v_c(t)$ . Initially, the voltage applied to the RC circuit is 5V. We obtain the natural response of the RC circuit by changing the applied voltage to 0V instantaneously at time  $t = 0$ . The natural response of the capacitor voltage is  $v_c(t)$ ,  $t > 0$ .

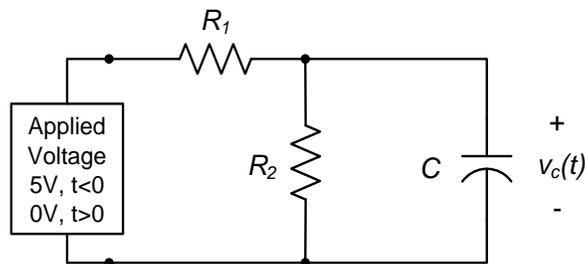
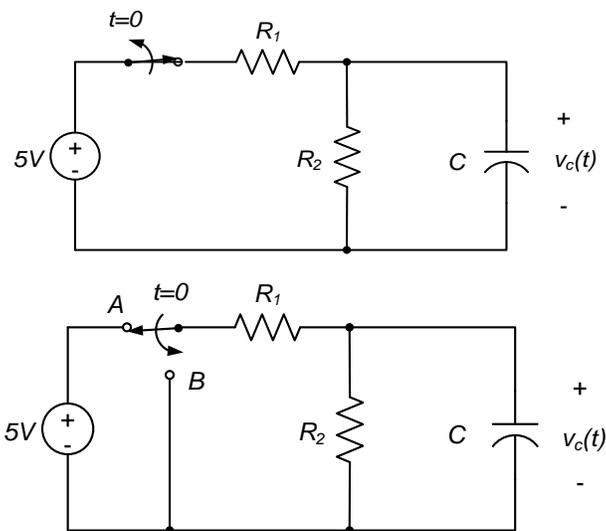


Figure 1. Basic RC circuit.

The way in which we reduce the applied voltage in Fig. 1 from 5V to 0V can have an effect on the circuit's natural response. In this lab assignment, we will use two different approaches to the switching process involved in changing the applied voltage:

- We will use a voltage source to apply the initial 5V, and physically open a switch to reduce the applied voltage to 0V. This will result in the circuit as shown in Fig. 2(a). Notice that in Fig. 2(a), the voltage source is replaced by an open circuit.
- A voltage source will be used to apply the 5V source, as above. However, in order to reduce the applied voltage to 0V, we will simply turn off the voltage source. This approach will result in the circuit shown in Fig. 2(b). Notice that in Fig. 2(b), the voltage source is replaced by a short circuit.



(a) Voltage change from switching

(b) Voltage change by turning "off" source.

Figure 2. Models of physical approaches to reducing applied voltage.

### Pre-lab:

#### ANALYSIS

Estimate the initial capacitor voltage,  $v_c(t < 0)$ , and the time constant for the circuits of Figs. 2(a) and 2(b). Your solutions may be functions of  $R_1$ ,  $R_2$ , and  $C$ .

### Lab Procedures:

#### DATA

- Construct the circuit shown in Fig. 1, using  $R_1 = 1\text{k}\Omega$ ,  $R_2 = 2.2\text{k}\Omega$ , and  $C = 22\mu\text{F}$ . (As always, measure the actual resistance value; measure the capacitance value if you have the appropriated instrument – some DMMs have a capacitance meter – otherwise, assume that the nominal capacitance value is correct.)

## DATA

- i. Use the Analog Discovery 2 oscilloscope to measure the capacitor voltage  $v_c(t)$  and use V+ on the Analog Discovery 2 to apply a 5V source to the circuit. While acquiring data with the oscilloscope, quickly disconnect the power supply from your circuit. Record an image of the oscilloscope window, showing the response  $v_c(t)$  of the capacitor voltage after the power is disconnected. The data acquisition process can be difficult unless you use the oscilloscope's trigger to acquire a single sequence of the data. Brief instructions for doing this are in Appendix A of this lab assignment, more detailed instructions are provided in the on-line tutorials on Digilent's website.

## DEMO

- ii. Demonstrate operation of your circuit to a teaching assistant and have them initial your lab notebook and the lab checklist.

## ANALYSIS

- iii. Estimate the time constant of the circuit from your measured data. Compare this result with your expectations based on your pre-lab analysis and the measured values of  $R_1$ ,  $R_2$ , and  $C$ . Calculate a percent difference between the expected and measured time constants. Comment briefly on your results.

- (b) Still using the circuit shown in Fig. 1, (with  $R_1=1k\Omega$ ,  $R_2=2.2k\Omega$ , and  $C = 22\mu F$  as in part a), use a square wave with an amplitude of 2.5V and an offset of 2.5V to create a square wave that oscillates between 0V and 5V. We will be using this square wave to implement a transition between 5V and 0V; use a very low frequency, 1Hz or so<sup>20</sup>.

## DATA

- i. Record an image of the oscilloscope window, showing the response  $v_c(t)$  of the capacitor voltage after the applied voltage goes to zero. Again, it is suggested that you use a trigger to assist in the data acquisition. The trigger settings from part (a) should also be appropriate for this section, but you may want to acquire the data differently. This is presented in Appendix B of this assignment.

## DEMO

- ii. Demonstrate operation of your circuit to a teaching assistant and have them initial your lab notebook and the lab checklist.

## ANALYSIS

- iii. Estimate the time constant of the circuit from your measured data. Compare this result with your expectations based on your pre-lab analysis and the measured values of  $R_1$ ,  $R_2$ , and  $C$ . Calculate a percent difference between the expected and measured time constants. Comment briefly on your results.

## Appendix A: Triggering and Single Acquisition

The *trigger* essentially defines where on the horizontal axis “zero” time occurs. The trigger point is commonly set by a particular feature on the waveform being measured. The basic trigger controls on the oscilloscope toolbar are shown in Fig. A1. These controls allow you to choose the trigger mode, the source, the condition, and the trigger level. Additional trigger controls are available by clicking on the **Green Arrow** on the oscilloscope menu bar.



Figure A1: Basic trigger controls.

Options for the primary trigger controls consist of the following:

<sup>20</sup> Since we really just want to turn “off” the voltage once after charging the capacitor, we want our square wave to be “on” and “off” for long times relative the time required for the circuit to respond. Typically, a “long” time is considered to be at least five times the time constant of the circuit. You can use this fact, along with your calculated time constant based on the pre-lab, to choose a square wave frequency yourself if you want.

- *Trigger mode*: basic options are **Normal**, **Auto**, or **None**. For this lab, we will use **Normal**.
- *Source*: Choose the channel which controls the trigger. A waveform feature on this channel will determine zero time.
- *Cond and Level*: These options specify the waveform feature used to set the trigger. *Cond* specifies a condition on the trigger – this is either **Rising** or **Falling**. If Rising is chosen, the trigger will set when the signal is increasing; Falling results in the trigger being set when the signal is decreasing. *Level* sets a voltage level for the trigger. In Fig. A2, example settings for this lab are shown. In this example, the trigger is set to activate when the signal first reaches 1V and is decreasing<sup>21</sup> (**Falling**). Fig. 2 shows that zero time on the horizontal axis corresponds to this condition on the wave form.

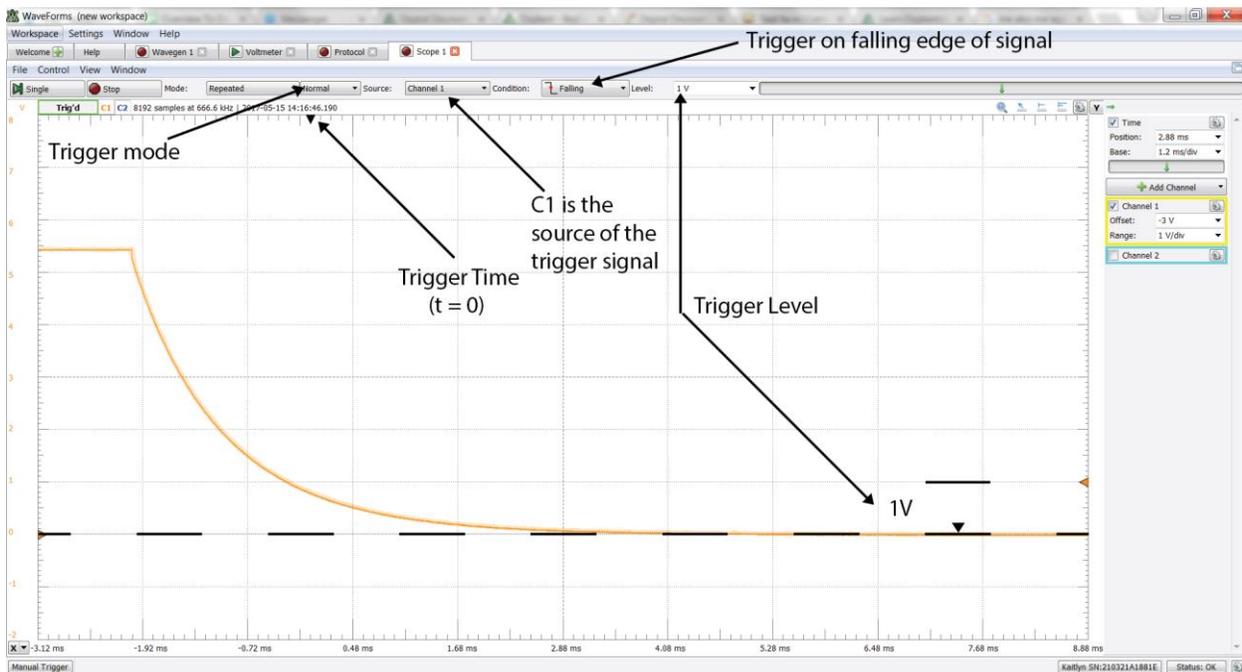


Figure A2. Example trigger settings and resulting waveform.

After you have set up the trigger, you can acquire the data. For part (a) of the lab procedures, we will generate a single natural response – once this response is generated, we want to display it in the oscilloscope window and freeze it there. We do not want to continue to display data after the response has decayed to zero. To generate a single screen of data and then stop acquiring additional data, click on the “Single” button on the scope instrument.

It will be worth your time, at this stage, to spend some time playing around with the trigger controls. Especially try changing the trigger point and the trigger level, as discussed below:

- Notice that the trigger point is denoted in the plot window by the black inverted triangle at the top of the plot window. The position of the trigger can be set by the *Pos* value in the time axis setting box, or by clicking on the trigger indicator (the black inverted triangle) with your left mouse button and dragging the trigger point to the desired position. Try it and observe its effect on the display.
- If the trigger source is one of the oscilloscope channels, the trigger level is shown on the plot window by the trigger level indicator – this is a triangular symbol of the same color as the trigger source channel on the right side of the plot window. For example, if the trigger source is channel 1, the indicator will be an orange triangle. The trigger level can be adjusted by clicking with your left mouse button on the trigger

<sup>21</sup> Since the natural response will decrease, it would be useless to try to trigger on a rising condition. This will only cause the scope to trigger when – and if – we turn the power back on.

level indicator and dragging the indicator to the desired location or by changing the trigger level in the drop-down box.

## Appendix B: Continuous Acquisition

In part (b) of these procedures, we are using the waveform generator to apply voltage to the circuit. The waveform generator continuously applies the selected waveform to your circuit, so the input to the circuit is a series of on/off steps which lasts as long as the waveform generator is running. In this case, we can use single-sequence acquisition as we did in part (a) and display the result of one arbitrary sample of turning power to the circuit off. However, it is more typical to use continuous acquisition of the waveform.

At small time scales on the oscilloscope, the oscilloscope is essentially taking a series of “frames” of data and successively displaying them in the main scope window<sup>22</sup>. If the waveform is repetitive, triggering allows us to assign a “zero time” to a particular feature on the signal. That feature gets placed on the same point on the plot window every time the oscilloscope screen updates; if the signal repeats itself based on this feature, the oscilloscope will display the same section of the signal every time the screen updates, making the signal appear to be unchanging.

To continuously acquire data for part (b) of this lab, set your trigger point to be as you desire. (The same settings as you used in part (a) should work, though you may feel like modifying them.) Apply power to your circuit using the waveform generator and then click on “Run”. The data should be displayed in the window. The waveform displayed should appear to be unchanging, but it is actually being updated at rapid intervals – the successive waveforms simply lie directly on top of one another.

---

<sup>22</sup> At time scales larger than about 100ms/div, the data scrolls from left to right across the window – things are happening slowly enough so that it makes sense to watch the signal evolve. If we try this approach at small time scales, the data scrolls by too quickly to be useful.

## Real Analog Chapter 7: Lab Worksheets

### 7.2.1: Passive RC Circuit Natural Response (35 points total)

1. In the space below, provide (from your pre-lab results) the initial capacitor voltages and time constants for the circuits of Fig. 2. (3 pts)
  
2. Provide below a schematic of the circuit you implemented, including actual resistance and capacitance values used in your circuit. (2 pts)
  
3. Attach to this worksheet an image of the oscilloscope window, showing the capacitor voltage response for the circuit of Fig. 2(a), in which  $V_+$  is used as the voltage source. Briefly discuss differences between the measured data and your estimates from the pre-lab (as always, this should include a percent difference between the values). (5 pts)
  
4. In the space below, provide your estimate of the circuit's time constant from the measured response data. Briefly discuss differences between the measured data and your estimates from the pre-lab (as always, this should include a percent difference between the values). (5 pts)
  
5. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuits' operation. (5 pts)

TA Initials: \_\_\_\_\_

6. Attach to this worksheet an image of the oscilloscope window, showing the capacitor voltage response for the circuit of Fig. 2(b), in which the waveform generator is used as the voltage source. Briefly discuss differences between the measured data and your estimates from the pre-lab (as always, this should include a percent difference between the values). (5 pts)

7. In the space below, provide your estimate of the circuit's time constant from the measured response data. Briefly discuss differences between the measured data and your estimates from the pre-lab (as always, this should include a percent difference between the values). (5 pts)

8. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuits' operation when using a square wave. (5 pts)

TA Initials: \_\_\_\_\_

## Real Analog Chapter 7: Lab Projects

### 7.3.1: Passive RL Circuit Natural Response

In this lab assignment, we will examine the natural response of a simple RL circuit. We will use both a manual switching operation and a square wave voltage source to create our circuit's natural response. We will see that the method used to create the response affects the circuit being measured.

#### Before beginning this lab, you should be able to:

- Determine the time constant of exponential functions
- Determine the natural response of passive RL circuits
- Use the Analog Discovery 2 waveform generator to apply a time-varying voltage input to an electrical circuit (Lab 6.2.1)
- Use the Analog Discovery 2 oscilloscope to measure and display time-varying waveforms
- Create a math channel on the Analog Discovery 2 scope (Labs 6.3.1, 6.4.1)

#### After completing this lab, you should be able to:

- Use a manual switching operation to create the natural response of a first order circuit
- Use the trigger on the Analog Discovery 2 oscilloscope to acquire a signal
- Be able to explain in your own words the difference between continuous and single-sequence data acquisition
- Use the Analog Discovery 2 waveform generator to create the natural response of a first order circuit.
- Measure the initial condition and time constant of a first order circuit natural response

#### This lab exercise requires:

- Analog Discovery 2 module
- Digilent Analog Parts Kit

#### Symbol Key:

- DEMO** Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
- ANALYSIS** Analysis; include principle results of analysis in laboratory report.
- SIM** Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
- DATA** Record data in your lab notebook.

#### General Discussion:

The basic RL circuit being used in this assignment is shown in Fig. 1. We will be interested primarily in the measured vs. expected behavior of the inductor current,  $i_L(t)$ . Initially, the voltage applied to the RL circuit is 5V. We obtain the natural response of the circuit by changing the applied voltage to 0V instantaneously at time  $t = 0$ . The natural response of the inductor current is  $i_L(t)$ ,  $t > 0$ .

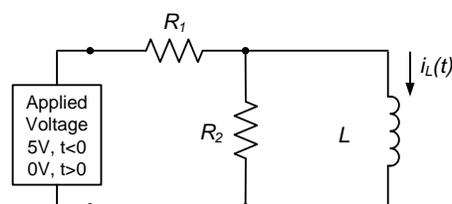


Figure 1. Basic RL circuit.

The way in which we reduce the applied voltage in Fig. 1 from 5V to 0V can have an effect on the circuit's natural response. In this lab assignment, we will use two different approaches to the switching process involved in changing the applied voltage:

- We will use a voltage source to apply the initial 5V, and physically open a switch to reduce the applied voltage to 0V. This will result in the circuit as shown in Fig. 2(a). Notice that in Fig. 2(a), the voltage source is replaced by an open circuit.
- A voltage source will be used to apply the 5V source, as above. However, in order to reduce the applied voltage to 0V, we will simply turn off the voltage source. This approach will result in the circuit shown in Fig. 2(b). Notice that in Fig. 2(b), the voltage source is replaced by a short circuit.

The implementation of the circuit of Fig. 1 is further complicated by the fact that we cannot directly measure a time-varying current; oscilloscopes will only measure voltages. Thus, we must infer the inductor current from parameters that we can actually measure. KCL tells us that the current through the inductor is the difference between the current through the resistor  $R_1$  and the current through the resistor  $R_2$ . The currents through the resistors  $R_1$  and  $R_2$  can be determined from the voltages across and Ohm's law. Thus, the inductor current can be determined from the voltages across the resistors as follows:

$$i_L(t) = \frac{v_{R1}(t)}{R_1} - \frac{v_{R2}(t)}{R_2} \quad \text{Eq. 1}$$

where the voltages  $v_{R1}(t)$  and  $v_{R2}(t)$  are as shown in Figs. 2. An oscilloscope can be used to measure these resistor voltages, and a math channel can be created to implement equation (1) and display the inductor current as a function of time.

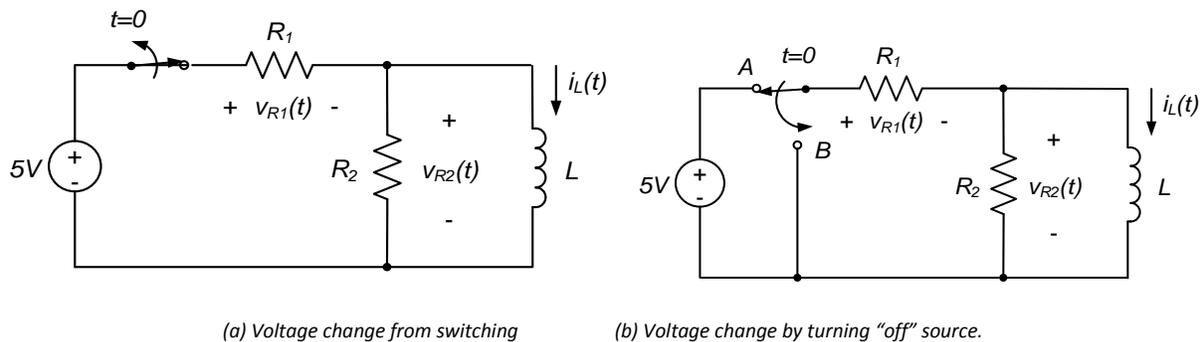


Figure 2. Models of physical approaches to reducing applied voltage.

### Pre-lab:

#### ANALYSIS

Estimate the initial inductor current,  $i_L(t < 0)$ , and the time constant for the circuits of Figs. 2(a) and 2(b). Your solutions may be functions of  $R_1$ ,  $R_2$ , and  $L$ .

### Lab Procedures:

#### DATA

- Construct the circuit shown in Fig. 1, using  $R_1=100\Omega$ ,  $R_2=47\Omega$ , and  $L = 1\text{mH}$ . (As always, measure the actual resistance values; you may assume that the nominal inductance value is correct.)

- Use the oscilloscope to measure the resistor voltages  $v_{R1}(t)$  and  $v_{R2}(t)$ . Set up a math channel according to equation (1) above to display the inductor current,  $i_L(t)$ . Apply power to the circuit by turning on the V+ supply. While acquiring data with the oscilloscope, quickly disconnect the power supply from your circuit. Record an image of the oscilloscope window, showing the response  $i_L(t)$  of the inductor current after the power is disconnected. The data acquisition process can be difficult unless you use the oscilloscope's trigger to acquire a single sequence of the data. Brief instructions for doing this are in Appendix A of this lab

#### DATA

assignment, more detailed instructions are provided in the on-line tutorials on Diligent's website.

**DEMO**

- ii. Demonstrate operation of your circuit to a teaching assistant and have them initial your lab notebook and the lab checklist.

**ANALYSIS**

- iii. Estimate the time constant of the circuit from your measured data. Compare this result with your expectations based on your pre-lab analysis and the measured values of  $R_1$ , and  $R_2$ . Calculate a percent difference between the expected and measured time constants. Comment briefly on your results.
- b. Construct the circuit of Fig. 2(b), still using  $R_1=1k\Omega$ ,  $R_2=2.2k\Omega$ , and  $L = 1mH$  as in part a). This circuit is essentially the same as the circuit of part (a), except that the waveform generator is used to provide power to the circuit. Set up the waveform generator to apply a square wave with an amplitude of 2.5V and an offset of 2.5V to your circuit. This square wave is used to implement the transition between 5V and 0V; use a very low frequency, 1Hz or so<sup>23</sup>.

**DATA**

- i. As in part a above, use the oscilloscope to measure the resistor voltages  $v_{R1}(t)$  and  $v_{R2}(t)$ . Set up a math channel according to equation (1) above to display the inductor current,  $i_L(t)$ . Apply power to the circuit by turning on the waveform generator. Record an image of the oscilloscope window, showing the natural response  $i_L(t)$  of the inductor current. Again, it is suggested that you use a trigger to assist in the data acquisition. The trigger settings from part (a) should also be appropriate for this section, but you may want to acquire the data differently. This is presented in Appendix B of this assignment.

**DEMO**

- ii. Demonstrate operation of your circuit to a teaching assistant and have them initial your lab notebook and the lab checklist.

**ANALYSIS**

- iii. Estimate the time constant of the circuit from your measured data. Compare this result with your expectations based on your pre-lab analysis and the measured values of  $R_1$  and  $R_2$ . Calculate a percent difference between the expected and measured time constants. Comment briefly on your results.

## Appendix A: Triggering and Single Acquisition

The *trigger* essentially defines where on the horizontal axis “zero” time occurs. The trigger point is commonly set by a particular feature on the waveform being measured. The basic trigger controls on the oscilloscope toolbar are shown in Fig. A1. These controls allow you to choose the trigger mode, the source, the condition, and the trigger level. Additional trigger controls are available by clicking on the **View** option on the oscilloscope menu bar and selecting the **Advanced Trigger** option.

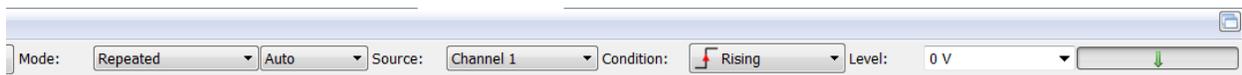


Figure A1: Basic trigger controls.

Options for the primary trigger controls consist of the following:

- *Trigger mode*: basic options are **Normal**, **Auto**, or **None**. For this lab, we will use **Normal**.

<sup>23</sup> Since we really just want to turn “off” the voltage once after charging the inductor, we want our square wave to be “on” and “off” for long times relative the time required for the circuit to respond. Typically, a “long” time is considered to be at least five times the time constant of the circuit. You can use this fact, along with your calculated time constant based on the pre-lab, to choose a square wave frequency yourself if you want.

- *Source*: Choose the channel which controls the trigger. A wave form feature on this channel will determine zero time.
- *Cond* and *Level*: These options specify the waveform feature used to set the trigger. *Cond* specifies a condition on the trigger – this is either **Rising** or **Falling**. If Rising is chosen, the trigger will set when the signal is increasing; Falling results in the trigger being set when the signal is decreasing. *Level* sets a voltage level for the trigger. In Fig. A2, example settings for this lab are shown. In this example, the trigger is set to activate when the signal first reaches 1V and is decreasing<sup>24</sup> (**Falling**). Fig. A2 shows that zero time on the horizontal axis corresponds to this condition on the wave form.

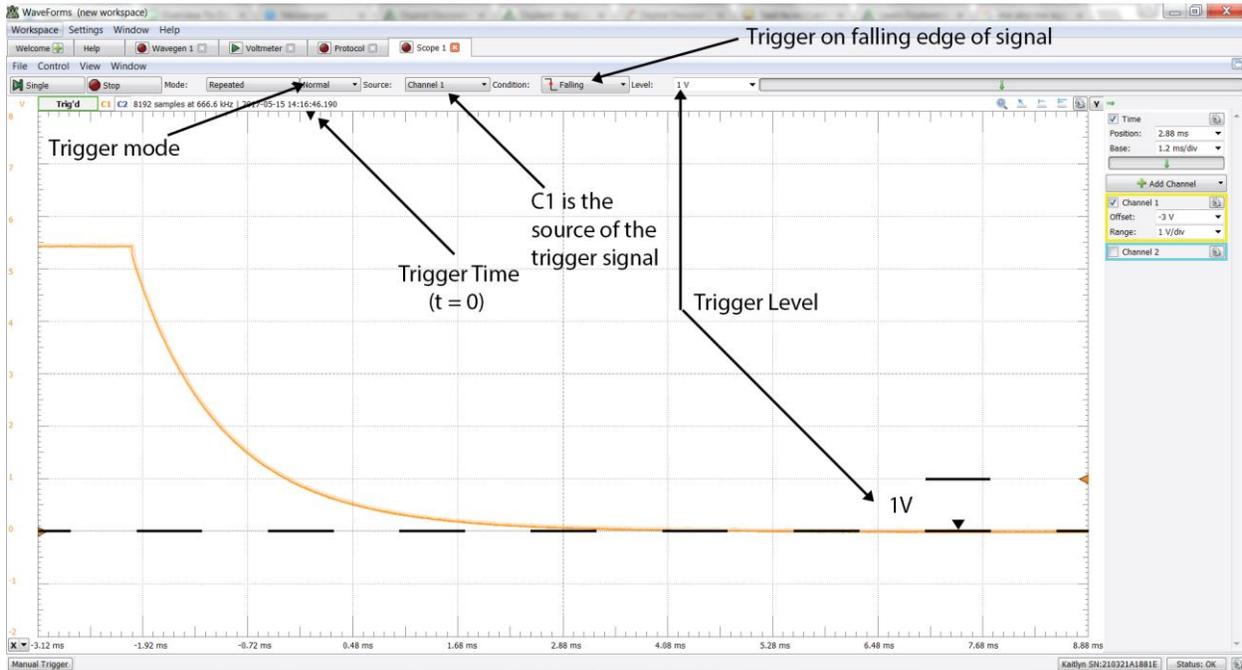


Figure A2. Example trigger settings and resulting waveform.

After you have set up the trigger, you can acquire the data. For part (a) of the lab procedures, we will generate a single natural response – once this response is generated, we want to display it in the oscilloscope window and freeze it there. We do not want to continue to display data after the response has decayed to zero. To generate a single screen of data and then stop acquiring additional data, click on the “Single” button on the scope instrument.

It will be worth your time, at this stage, to spend some time playing around with the trigger controls. Especially try changing the trigger point and the trigger level, as discussed below:

- Notice that the trigger point is denoted in the plot window by the black inverted triangle at the top of the plot window. The position of the trigger can be set by the *Position* value in the time axis setting box, or by clicking on the trigger indicator with your left mouse button and dragging the trigger point to the desired position. Try it and observe its effect on the display.
- If the trigger source is one of the oscilloscope channels, the trigger level is shown on the plot window by the trigger level indicator – this is a triangular symbol of the same color as the trigger source channel on the right side of the plot window. For example, if the trigger source is channel 1, the indicator will be a blue triangle. The trigger level can be adjusted by clicking with your left mouse button on the trigger level indicator and dragging the indicator to the desired location or by changing the trigger level in the drop-down box.

<sup>24</sup> Since the natural response will decrease, it would be useless to try to trigger on a rising condition. This will only cause the scope to trigger when – and if – we turn the power back on.

## Appendix B: Continuous Acquisition

In part (b) of these procedures, we are using the waveform generator to apply voltage to the circuit. The waveform generator continuously applies the selected waveform to your circuit, so the input to the circuit is a series of on/off steps which lasts as long as the waveform generator is running. In this case, we can use single-sequence acquisition as we did in part (a) and display the result of one arbitrary sample of turning power to the circuit off. However, it is more typical to use continuous acquisition of the waveform.

At small time scales on the oscilloscope, the oscilloscope is essentially taking a series of “frames” of data and successively displaying them in the main scope window<sup>25</sup>. If the waveform is repetitive, triggering allows us to assign a “zero time” to a particular feature on the signal. That feature gets placed on the same point on the plot window every time the oscilloscope screen updates; if the signal repeats itself based on this feature, the oscilloscope will display the same section of the signal every time the screen updates, making the signal appear to be unchanging.

To continuously acquire data for part (b) of this lab, set your trigger point to be as you desire. (The same settings as you used in part (a) should work, though you may feel like modifying them.) Apply power to your circuit using the waveform generator and then click on “Run”. The data should be displayed in the window. The waveform displayed should appear to be unchanging, but it is actually being updated at rapid intervals – the successive waveforms simply lie directly on top of one another.

---

<sup>25</sup> At time scales larger than about 100ms/div, the data scrolls from left to right across the window – things are happening slowly enough so that it makes sense to watch the signal evolve. If we try this approach at small time scales, the data scrolls by too quickly to be useful.

## Real Analog Chapter 7: Lab Worksheets

### 7.3.1: Passive RL Circuit Natural Response (35 points total)

1. In the space below, provide (from your pre-lab results) the initial inductor current and time constants for the circuits of Figure 2. (3 pts)
  
2. Provide below a schematic of the circuit you implemented, including actual resistance values used in your circuit. (2 pts)
  
3. Attach to this worksheet an image of the oscilloscope window, showing the inductor voltage, the resistor voltage, and the inductor current for the circuit of Figure 2(a), in which V+ is used as the voltage source. Briefly discuss differences between the measured data and your estimates from the pre-lab (as always, this should include a percent difference between the values). (5 pts)
  
4. In the space below, provide your estimate of the circuit's time constant from the measured response data. Briefly discuss differences between the measured data and your estimates from the pre-lab (as always, this should include a percent difference between the values). (5 pts)
  
5. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuits' operation. (5 pts)

**TA Initials:** \_\_\_\_\_

6. Attach to this worksheet an image of the oscilloscope window, showing the inductor voltage, the resistor voltage, and the inductor current for the circuit of Figure 2(b), in which the waveform generator is used as the voltage source. Briefly discuss differences between the measured data and your estimates from the pre-lab (as always, this should include a percent difference between the values). (5 pts)

7. In the space below, provide your estimate of the circuit's time constant from the measured response data. Briefly discuss differences between the measured data and your estimates from the pre-lab (as always, this should include a percent difference between the values). (5 pts)

8. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuits' operation when using a square wave. (5 pts)

TA Initials: \_\_\_\_\_

## Real Analog Chapter 7: Lab Projects

### 7.4.1: Inverting Differentiator

In this lab assignment, we will examine the forced response of a circuit which performs a differentiation – that is, the circuit output is the derivative with respect to time of the input to the circuit. We will apply sinusoids of various frequencies to the circuit and compare the output with our expectations based on analysis.

#### Before beginning this lab, you should be able to:

- Determine the input-output relationship for first order circuits
- Use the Analog Discovery 2 waveform generator to apply a time-varying voltage input to an electrical circuit (Lab 6.2.1)
- Use the Analog Discovery 2 oscilloscope to measure and display time-varying waveforms

#### After completing this lab, you should be able to:

- Measure the forced response of a circuit which performs a differentiation process

#### This lab exercise requires:

- Analog Discovery 2 module
- Digilent Analog Parts Kit

#### Symbol Key:



Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.



Analysis; include principle results of analysis in laboratory report.



Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.



Record data in your lab notebook.

#### General Discussion:

The circuit we will be concerned with in this assignment is shown in Fig. 1. The output of the circuit,  $V_{OUT}(t)$ , is proportional to the inverse of the derivative of the input voltage,  $V_{IN}(t)$ . The constant of proportionality is determined by the values of the resistance and capacitance in the circuit. In this assignment, we will analytically estimate the relationship between the circuit's input and output and compare this with measured input and output signals.

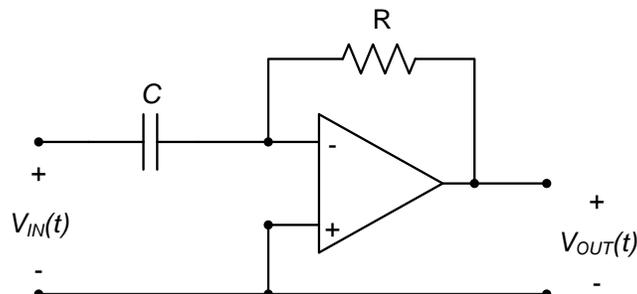


Figure 1. Differentiating circuit.

## Pre-lab:

## ANALYSIS

Determine the circuit output  $V_{OUT}(t)$  as a function of the circuit input,  $V_{IN}(t)$ . (This is the circuit's input-output relationship.) Your relationship will be a function of the resistance,  $R$ , and the capacitance,  $C$ .

## ANALYSIS

If the input function is a sinusoid of the form

$$V_{IN}(t) = A\cos(\omega t)$$

determine the output function. (Your solution will be a function of  $R$ ,  $C$ ,  $A$ , and  $\omega$ .)

**Note:**

The argument in the cosine function has units of radians. Thus,  $\omega$  has units of radians/second. This is different from the units of Hertz (cycles/second) that are used by most waveform generators. The conversion between Hz and radians/second is:

$$\omega = 2\pi f$$

where the frequency  $f$  has units of Hz. The above conversion is essential, if you want to compare calculations with data.

## Lab Procedures:

## DATA

- a. Construct the circuit shown in Fig. 1, using  $R=1.5k\Omega$  and  $C = 100nF$ . Use  $V+$  and  $V-$  as the positive and negative power supplies of the operational amplifier. Use the oscilloscope to measure both the input and output voltages  $V_{IN}(t)$  and  $V_{OUT}(t)$ . Set the oscilloscope measurements to provide at least the amplitude of each of these waveforms. (As always, measure the actual resistance value; measure the capacitance value if you have the appropriated instrument – some DMMs have a capacitance meter – otherwise, assume that the nominal capacitance value is correct.)

## DATA

- i. Apply a sinusoidal input voltage with frequency = 1kHz, amplitude = 1V, and offset = 0V to the circuit of Fig. 1. Use your oscilloscope to display the data listed above (waveforms corresponding to  $V_{IN}(t)$  and  $V_{OUT}(t)$ ; measurement window displaying amplitudes of  $V_{IN}(t)$  and  $V_{OUT}(t)$ ). Record the image of the oscilloscope window, showing the waveforms and their measured amplitudes.

## DATA

- ii. Apply a sinusoidal input voltage with frequency = 2kHz, amplitude = 1V, and offset = 0V to the circuit of Fig. 1. Use your oscilloscope to display the data listed above (waveforms corresponding to  $V_{IN}(t)$  and  $V_{OUT}(t)$ ; measurement window displaying amplitudes of  $V_{IN}(t)$  and  $V_{OUT}(t)$ ). Record the image of the oscilloscope window, showing the waveforms and their measured amplitudes.

## DATA

- iii. Apply a sinusoidal input voltage with frequency = 500Hz, amplitude = 1V, and offset = 0V to the circuit of Fig. 1. Use your oscilloscope to display the data listed above (waveforms corresponding to  $V_{IN}(t)$  and  $V_{OUT}(t)$ ; measurement window displaying amplitudes of  $V_{IN}(t)$  and  $V_{OUT}(t)$ ). Record the image of the oscilloscope window, showing the waveforms and their measured amplitudes.

## DEMO

- iv. Demonstrate operation of your circuit to a teaching assistant and have them initial your lab notebook and the lab checklist.

## Post-lab Exercises:

## ANALYSIS

For the three cases in the lab procedures (1kHz sinusoid, 2kHz sinusoid, 500Hz sinusoid), calculate the expected output voltage based on your pre-lab analysis. Make a table comparing listing the measured and expected amplitudes of the output voltage, and the percent difference between the measured and expected amplitudes.

**ANALYSIS**

Comment briefly on the overall shapes of the expected vs. measured waveforms. Is the phase difference (the time delay between the input and output sinusoids) of the measured voltages consistent with what you would expect from your pre-lab analysis?

## Real Analog Chapter 7: Lab Worksheets

### 7.4.1: Passive RL Circuit Natural Response (30 points total)

1. In the space below, provide (from your pre-lab results) the input-output relation for the circuit of Figure 1 and the expected output of the circuit resulting from the given sinusoidal forcing function. (3 pts)
2. Provide below a schematic of the circuit you implemented, including actual resistance (and, if possible, capacitance) values used in your circuit. (2 pts)
3. Attach to this worksheet an image of the oscilloscope window, showing waveforms corresponding to  $V_{IN}(t)$  and  $V_{OUT}(t)$  and a measurement window displaying amplitudes of  $V_{IN}(t)$  and  $V_{OUT}(t)$  for the 1kHz sinusoidal input. (5 pts)
4. Attach to this worksheet an image of the oscilloscope window, showing waveforms corresponding to  $V_{IN}(t)$  and  $V_{OUT}(t)$  and a measurement window displaying amplitudes of  $V_{IN}(t)$  and  $V_{OUT}(t)$  for the 2kHz sinusoidal input. (5 pts)
5. Attach to this worksheet an image of the oscilloscope window, showing waveforms corresponding to  $V_{IN}(t)$  and  $V_{OUT}(t)$  and a measurement window displaying amplitudes of  $V_{IN}(t)$  and  $V_{OUT}(t)$  for the 500Hz sinusoidal input. (5 pts)
6. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuits' operation. (5 pts)

TA Initials: \_\_\_\_\_

7. In the space below, provide a table showing the expected output voltage amplitudes, the measured voltage output amplitudes, and a percent difference between the two for each of the above test cases (500Hz, 1kHz, and 2kHz frequencies). Briefly discuss differences between the measured and expected values. (5 pts)

## Real Analog Chapter 7: Lab Projects

### 7.5.1: Passive RC Circuit Step Response

In this lab assignment, we will examine the step response of a simple RC circuit. We will use a square wave voltage source to emulate a step input to the system. An oscilloscope will be used to monitor both the applied input voltage and the response voltage from the circuit. Both the voltage across the capacitor and the voltage across the resistor will be measured and the waveforms across both inputs compared. We will also examine the effects of loading a passive RC circuit in this lab assignment.

#### Before beginning this lab, you should be able to:

- Determine the time constant of exponential functions
- State voltage-current relationships for inductors and capacitors
- Determine the natural response of passive first order (RL, RC) circuits
- Determine the step response of passive first order (RL, RC)

#### After completing this lab, you should be able to:

- Use a function generator to apply a square wave voltage input to an electrical circuit.
- Measure the time constant and steady-state response of first order passive electrical circuits
- State the potential effects of loading on a passive RC circuit

#### This lab exercise requires:

- Analog Discovery 2 module
- Digiilent Analog Parts Kit
- Digital multimeter (optional)

#### Symbol Key:

- |   |   |
|---|---|
|  | Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable. |
|  | Analysis; include principle results of analysis in laboratory report.   |
|  | Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.                      |
|  | Record data in your lab notebook.   |

#### General Discussion:

This lab assignment will be concerned with the simple series RC circuit shown in Fig. 1. We will be interested primarily in the measured vs. expected behavior of the capacitor voltage, but a secondary goal of this assignment is to compare the voltage differences across the capacitor and resistor. Thus, we will measure both the voltage differences  $v_R(t)$  and  $v_C(t)$  as shown in Fig. 1<sup>26</sup>.

<sup>26</sup> The oscilloscope instrument on the Analog Discovery 2 will allow you to take “double-sided” or “differential” voltage measurements – this allows you to directly measure the voltage difference across any component in the same way you measure a voltage difference using a DMM. Thus, we can measure the voltage differences in Figure 1 exactly as they are indicated on that figure. Many oscilloscopes, however, make single-sided measurements, in which voltage differences are all measured with respect to a “common” or ground node. In order to measure

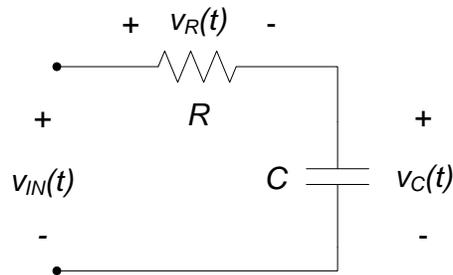


Figure 1. RC circuit being tested.

**Pre-lab:****ANALYSIS**

Estimate the time constant for the circuit shown in Fig. 1 if  $R=470\ \Omega$ , and  $C = 1\ \mu\text{F}$ . Also determine the steady state response of  $v_C(t)$  if  $v_{IN}$  is a step input with amplitude 5V. (e.g.  $v_{IN} = 5u_0(t)$  V.) Note: you do not need to write or solve a differential equation for either of the circuits in order to do this.

**Lab Procedures:**

- a. Construct the circuit shown in Fig. 1, using  $R=470\ \Omega$  and  $C = 1\ \mu\text{F}$ . (As always, measure the actual resistance value; measure the capacitance value if you have the appropriated instrument – some DMMs have a capacitance meter – otherwise, assume that the nominal capacitance value is correct.)

**DATA**

- i. Apply a 4V peak-to-peak square wave input with period = 10 ms (frequency = 100Hz) as shown in Fig. 2 to the circuit<sup>27</sup>. Display both  $v_{IN}(t)$  and  $v_{OUT}(t)$  on your oscilloscope window. Record the image of the oscilloscope window, showing the waveforms and save the signals as a data file for later plotting<sup>28</sup>.

**DATA****ANALYSIS**

- ii. Calculate the time constant and steady state response of the output voltage. Compare your results with your expectations based on the pre-lab analysis and comment on any differences.

**DEMO**

- iii. Demonstrate operation of your circuit to a teaching assistant and have them initial your lab notebook and the lab checklist.

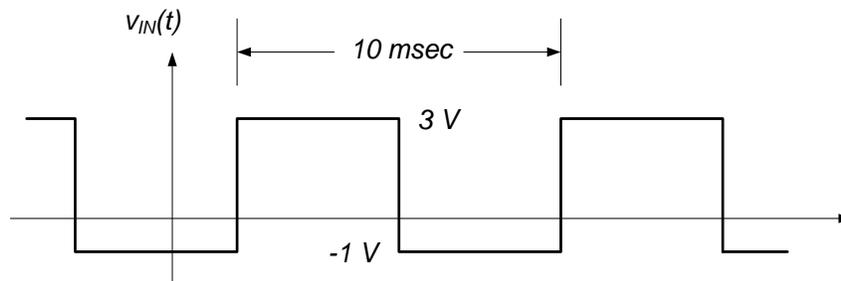


Figure 2. Input voltage signal.

the voltage across the resistor in Figure 1 using a single-sided oscilloscope, we would typically measure  $v_{in}(t)$  and  $v_C(t)$  and then using a math channel on the oscilloscope to take the difference between the two to get the voltage across the resistor

<sup>27</sup> In the terminology used in the Analog Discovery 2 waveform generators, we would say that the waveform of Figure 1 has an amplitude (zero-to-peak) of 2V and an average value (or offset) 1V.

<sup>28</sup> The “Export” button on the oscilloscope toolbar allows you to save measured data as a .csv file.

- b. RC circuits are often used for signal conditioning. The conditioned signal will, in general, be applied to a load in order to be useful.

**DATA**

- i. Apply a load to the RC circuit of Fig. 1 by constructing the circuit shown in Fig. 3. Use  $R = R_L = 470 \Omega$ , and  $C = 1 \mu\text{F}$ . Measure the output voltage across the load resistor, for the input voltage shown in Fig. 2. Record the image of the oscilloscope window, showing the waveforms and their measured amplitudes, and save the signals as a data file for later plotting.

**ANALYSIS**

- ii. Determine a time constant and steady state response from your measured data. How do these parameters compare with those of the unloaded circuit of part (a)? Do the loaded circuit parameters agree with your expectations? (Hint: calculate an equivalent resistance seen by the capacitor.)

**DEMO**

- iii. Demonstrate operation of your circuit to a teaching assistant and have them initial your lab notebook and the lab checklist.

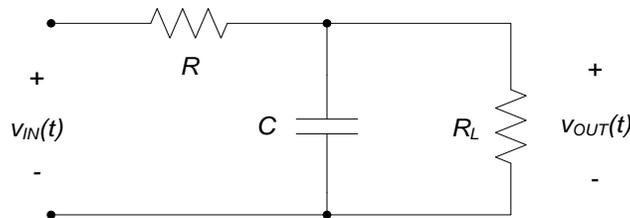


Figure 3. Loaded passive RC circuit.

### Post-lab Exercises:

**ANALYSIS**

- a. Import the oscilloscope data you acquired in part (a) of the lab procedures into Excel, Matlab, or any similar software package which provides basic mathematics and plotting capabilities. Plot the resistor and capacitor voltages. Use your software to sum the capacitor and resistor voltages. Plot the result and compare it to the input waveform you applied to the circuit. Comment on differences or similarities.

**ANALYSIS**

- b. Import the voltage  $v_{OUT}(t)$  you acquired in part (b) of the lab procedures. Create a plot displaying this data overlaid with the capacitor voltage of the unloaded circuit acquired in part(a) of the lab procedures. Comment on the differences between the two, including qualitative comparisons of the time constants and steady-state responses.

## Real Analog Chapter 7: Lab Worksheets

### 7.5.1: Passive RC Circuit Step Response (40 points total)

1. In the space below, provide (from your pre-lab results) the time constant for the circuit shown in Fig. 1 if  $R=470\ \Omega$ ,  $C = 1\ \mu\text{F}$ . Also provide your estimate steady state response of  $v_C(t)$  if  $v_{IN}$  is a step input with amplitude 5V. (3 pts)
  
2. Provide below a schematic of the circuit you implemented, including actual resistance and capacitance values used in your circuit. (2 pts)
  
3. Attach to this worksheet an image of the oscilloscope window, showing the input and output voltages. In the space below, provide your estimate of the time constant of the circuit. Briefly discuss differences between the measured data and your estimates from the pre-lab (as always, this should include a percent difference between the values). (5 pts)
  
4. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuits' operation. (5 pts)

TA Initials: \_\_\_\_\_

5. In the space below, provide a circuit schematic for the "loaded" circuit with measured resistor values (both the RC circuit resistance and the load resistance). (3 pts)



## Real Analog Chapter 7: Lab Projects

### 7.5.2: Passive RL Circuit Step Response

In this lab assignment, we will examine the step response of a simple RL circuit. We will use a square wave voltage source to emulate a step input to the system. An oscilloscope will be used to monitor both the applied input voltage and the response voltage from the circuit. Both the voltage across the inductor and the voltage across the resistor will be measured and the waveforms across both inputs compared.

#### Before beginning this lab, you should be able to:

- Determine the time constant of exponential functions
- State voltage-current relationships for inductors and capacitors
- Determine the natural response of passive first order (RL, RC) circuits
- Determine the step response of passive first order (RL, RC)

#### After completing this lab, you should be able to:

- Use a function generator to apply a square wave voltage input to an electrical circuit.
- Measure the time constant and steady-state response of first order passive electrical circuits
- State the potential effects of loading on a passive RL circuit

#### This lab exercise requires:

- Analog Discovery 2 module
- Digiilent Analog Parts Kit
- Digital multimeter (optional)

#### Symbol Key:

- DEMO** Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
- ANALYSIS** Analysis; include principle results of analysis in laboratory report.
- SIM** Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
- DATA** Record data in your lab notebook.

#### General Discussion:

This lab assignment will be concerned with the simple series RL circuit shown in Fig. 1. We will be interested primarily in the measured vs. expected behavior of the resistor voltage, but a secondary goal of this assignment is to compare the voltage differences across the inductor and resistor. Thus, we will measure both the voltage differences  $v_R(t)$  and  $v_L(t)$  as shown in Fig. 1<sup>29</sup>.

<sup>29</sup> The oscilloscope instrument on the Analog Discovery 2 will allow you to take “double-sided” or “differential” voltage measurements – this allows you to directly measure the voltage difference across any component in the same way you measure a voltage difference using a DMM. Thus, we can measure the voltage differences in Figure 1 exactly as they are indicated on that figure. Many oscilloscopes, however, make single-sided measurements, in which voltage differences are all measured with respect to a “common” or ground node. In order to measure the voltage across the inductor in Fig. 1 using a single-sided oscilloscope, we would typically measure  $v_{in}(t)$  and  $v_R(t)$  and then using a math channel on the oscilloscope to take the difference between the two to get the voltage across the resistor

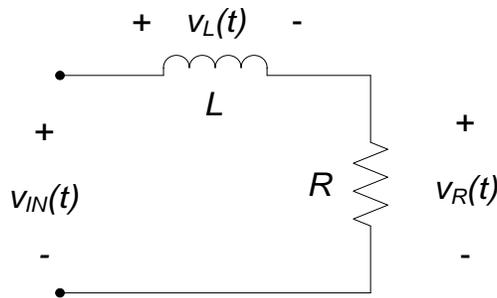


Figure 1. RL circuit being tested.

**Pre-lab:****ANALYSIS**

Estimate the time constant for the circuit shown in Figs. 1 if  $R=200\ \Omega$  and  $L = 1\text{mH}$ . Also determine the steady state response of  $v_C(t)$  if  $v_{IN}$  is a step input with amplitude 2V. (e.g.  $v_{IN} = 2u_0(t)$  V.) Note: you do not need to write or solve a differential equation for either of the circuits in order to do this.

**Lab Procedures:****DATA**

- a. Construct the circuit shown in Fig. 1, using  $R=200\ \Omega$  and  $L = 1\text{mH}$ . (As always, measure the actual resistance value; assume that the nominal inductance value of the inductor in your parts kit is correct.)

- i. Use a square wave input to the circuit to emulate a step function. The input step should have an amplitude which goes from zero to 2V (as shown in Fig. 2) and have a period which is long enough to allow the circuit to reach steady-state<sup>30</sup>. Display both  $v_{IN}(t)$  and  $v_{OUT}(t)$  on your oscilloscope window. Record the image of the oscilloscope window, showing the waveforms and save the signals as a data file for later plotting<sup>31</sup>.

**DATA****ANALYSIS**

- ii. Calculate the time constant and steady state response of the resistor voltage. Compare your results with your expectations based on the pre-lab analysis and comment on any differences.

**DEMO**

- iii. Demonstrate operation of your circuit to a teaching assistant and have them initial your lab notebook and the lab checklist.

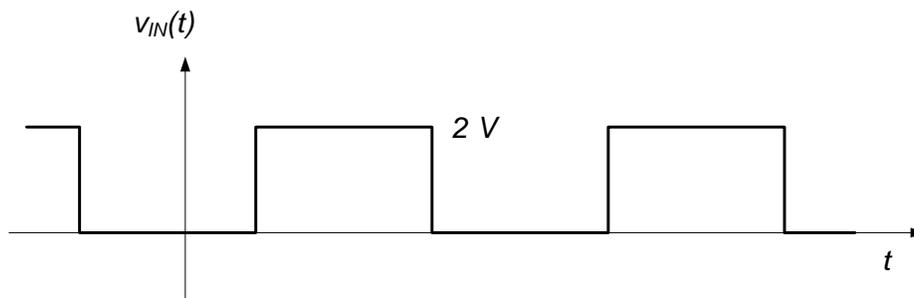


Figure 2. Input voltage signal.

- b. First order circuits are often used for signal conditioning. The conditioned signal will, in general, be applied to a load in order to be useful.

<sup>30</sup> It is generally assumed that a first order circuit has reached steady state after a time corresponding to approximately five time constants.

<sup>31</sup> The "Export" button on the oscilloscope toolbar allow you to save measured data as a .csv file.

## DATA

- i. Apply a load to the RL circuit of Fig. 1 by constructing the circuit shown in Fig. 3. Use  $R = R_L = 200 \Omega$ , and  $L = 1\text{mH}$ . Measure the output voltage across the load resistor, for the input voltage shown in Fig. 2. Sketch the input and output voltage signals in your lab notebook, and save the signals as a data file for later plotting.

## ANALYSIS

- ii. Determine a time constant and steady state response from your measured data. How do these parameters compare with those of the unloaded circuit of part (a)? Do the loaded circuit parameters agree with your expectations? (Hint: calculate an equivalent resistance seen by the capacitor.)

## DEMO

- iii. Demonstrate operation of your circuit to a teaching assistant and have them initial your lab notebook and the lab checklist.

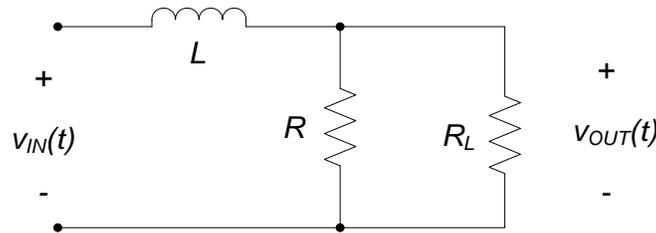


Figure 3. Loaded passive RL circuit.

## Post-lab Exercises:

## ANALYSIS

- a. Import the oscilloscope data you acquired in part (a) of the lab procedures into Excel, Matlab, or any similar software package which provides basic mathematics and plotting capabilities. Plot the resistor and inductor voltages. Use your software to sum the inductor and resistor voltages. Plot the result and compare it to the input waveform you applied to the circuit. Comment on differences or similarities.

## ANALYSIS

- b. Import the voltage  $v_{OUT}(t)$  you acquired in part (b) of the lab procedures. Create a plot displaying this data overlaid with the capacitor voltage of the unloaded circuit acquired in part(a) of the lab procedures. Comment on the differences between the two, including qualitative comparisons of the time constants and steady-state responses.

## Real Analog Chapter 7: Lab Worksheets

### 7.5.2: Passive RL Circuit Step Response (40 points total)

1. In the space below, provide (from your pre-lab results) the time constant for the circuit shown in Fig. 1 if  $R=200\ \Omega$  and  $L = 1\text{mH}$ . Also provide your estimate steady state response of  $v_R(t)$  if  $v_{IN}$  is a step input with amplitude 5V. (3 pts)
2. Provide below a schematic of the circuit you implemented, including actual resistance value used in your circuit. (2 pts)
3. Attach to this worksheet an image of the oscilloscope window, showing the input voltage, the inductor voltage, and the resistor voltage. In the space below, provide your estimate of the time constant of the circuit. Briefly discuss differences between the measured data and your estimates from the pre-lab (as always, this should include a percent difference between the values). (5 pts)
4. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuits' operation. (5 pts)

**TA Initials:** \_\_\_\_\_

5. In the space below, provide a circuit schematic for the "loaded" circuit with measured resistor values (both the RL circuit resistance and the load resistance). (3 pts)



## Real Analog Chapter 7: Lab Projects

### 7.5.3: Active RC Circuit Step Response

In lab assignments 7.5.1 and 7.5.2, we examined the response of passive first order circuits. These circuits can be very useful, for example, in signal conditioning. However, passive first order circuits have similar drawbacks to passive resistive circuits. One major problem is that the addition of a load to the circuit can significantly modify the circuit's behavior, which may necessitate a re-design of the circuit anytime a different load is applied to the circuit. Another drawback is the inability to amplify any input signal – the energy out of a passive circuit cannot exceed the energy provided to the circuit.

Active circuits can resolve these issues. Active circuits, since the power they supply comes from external sources, are somewhat immune to loading effects. The external sources of an active circuit also allow these circuits to amplify input signals – the output from these circuits can contain considerably more energy than is being provided by the input signal. In this lab assignment, we construct an active RC circuit and note that loading of the circuit – unlike loading of the circuit of lab 7.5.1 – does not significantly affect the circuit's behavior.

#### Before beginning this lab, you should be able to:

- Analyze, design, and build operational amplifier-based circuits
- State voltage-current relationships for inductors and capacitors
- Determine the natural and step responses of active first order circuits
- State the potential effects of loading on passive first order circuits (Labs 7.5.1, 7.5.2)

#### After completing this lab, you should be able to:

- Use a function generator to apply a square wave voltage input to an electrical circuit.
- Measure the time constant and steady-state response of first order active electrical circuits

#### This lab exercise requires:

- Analog Discovery 2 module
- Digilent Analog Parts Kit
- Digital multimeter (optional)

#### Symbol Key:

- |   |   |
|---|---|
|  | Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable. |
|  | Analysis; include principle results of analysis in laboratory report.   |
|  | Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.                      |
|  | Record data in your lab notebook.   |

#### General Discussion:

In lab assignment 7.5.1, we (hopefully) noted that loading passive circuits can have a significant effect on their response. Active circuits tend to be less susceptible to loading effects. In this part of the assignment, we construct an active RC circuit with the same time constant as the circuit of lab 7.5.1. We then apply the load of lab 7.5.1 to this active circuit and observe the response.

The circuit shown in Fig. 1 has an input-output relationship:

$$\frac{dV_{OUT}(t)}{dt} + \frac{1}{RC}V_{OUT}(t) = -\frac{1}{RC}V_{IN}(t)$$

which, with the exception of a sign change, is the same as that of the circuit of Fig. 1(a) of Lab 7.5.1. Thus, the response of the circuit of Fig. 1 will have the same time constant and steady state gain as the circuit of lab 7.5.1, but will be of opposite sign.

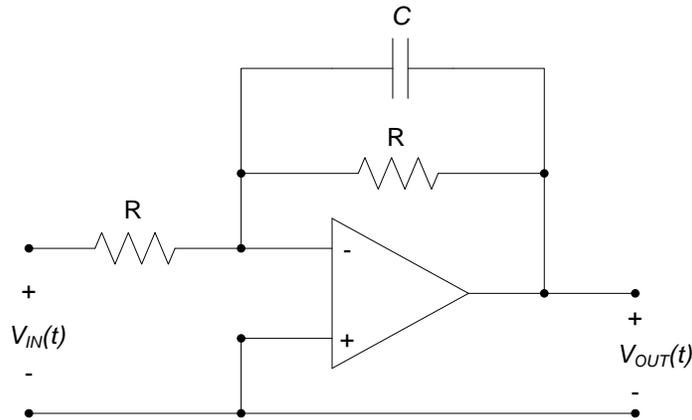


Figure 1. (Inverting) active RC circuit.

### Pre-lab:

None

### Lab Procedures:

#### DATA

- Construct the circuit shown in Fig. 1, using  $R=470\ \Omega$ , and  $C = 1\ \mu\text{F}$  (As always, measure the actual resistance and capacitance values (if possible) and record them in your lab notebook.)
  - Apply a 4V peak-to-peak square wave input with period = 10 ms (frequency = 100Hz) as shown in Fig. 2 to the circuit. (Make sure the supply voltage ranges applied to the operational amplifier are adequate to provide the full range of output voltage.) Measure both  $V_{IN}(t)$  and  $V_{OUT}(t)$ ; record an image of the oscilloscope window, showing the waveforms.

#### DATA

#### ANALYSIS

- Calculate the time constant and steady state output voltage. Compare your results with your results from lab 7.5.1 and comment on any differences.

#### DEMO

- Demonstrate operation of your circuit to a teaching assistant and have them initial your lab notebook and the lab checklist.

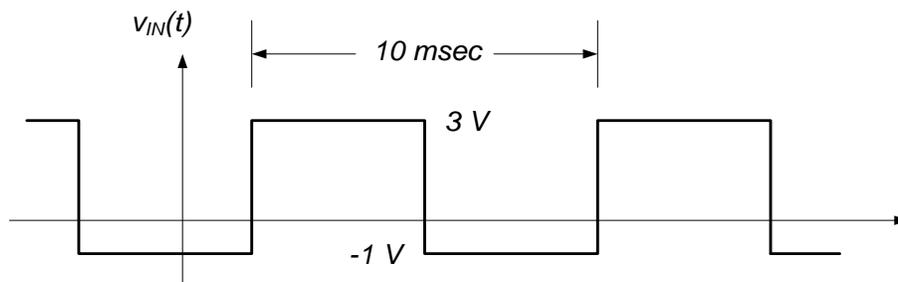


Figure 2. Input voltage signal.

- Gradually increase the frequency of the input square wave and note the output voltage response.

## DATA

- i. Tabulate the peak-to-peak input and output voltages for (at least) frequencies of 300Hz, 500Hz, 1000Hz, and 2000Hz. Comment on the trends between the peak-to-peak input and output voltages and the input frequency. Comment on the reasons for this behavior. (Hint: as the input function changes faster than the circuit time constant, the capacitor in the circuit does not have time to fully charge or discharge before the input voltage changes.)

## DEMO

- ii. Demonstrate operation of your circuit to a teaching assistant and have them initial your lab notebook and the lab checklist.

- c. Apply a load to the RC circuit of Fig. 1 by constructing the circuit shown in Fig. 2. Use  $R = R_L = 470 \Omega$ .

## DATA

- i. Measure the  $V_{IN}$  and  $V_{OUT}$ , for the input voltage shown in Fig. 2. Record an image of the oscilloscope window, showing the waveforms.

## ANALYSIS

- ii. Determine a time constant and steady state response from your measured data. How do these parameters compare with those of the unloaded circuit of part (a) of Lab 7.5.1? How do they agree with the loaded passive circuit of part (c) of Lab 7.5.1?

## DEMO

- iii. Demonstrate operation of your circuit to a teaching assistant and have them initial your lab notebook and the lab checklist.

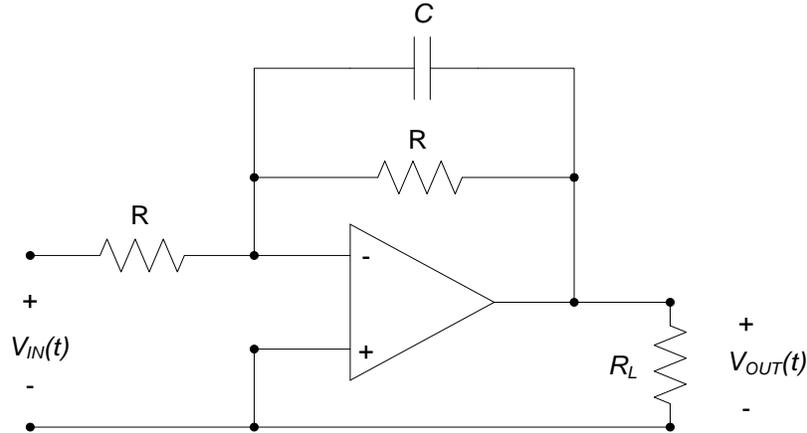


Figure 3. Loaded active RC circuit.



- b. Discuss your observations relative to peak-to-peak input and output voltage amplitudes vs. frequency and comment on possible reasons for these trends. (5 pts)

- c. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuits' operation. (5 pts)

**TA Initials:** \_\_\_\_\_

3. Loaded circuit response (20 pts)

- a. Provide below a schematic of the loaded active RC circuit with measured resistance and (if possible) capacitance values. (3 pts)

- b. Attach, to this worksheet, the image of your oscilloscope time window, showing the measured input and output data. (4 pts)

- c. Provide your estimates of the active RC circuit time constant and steady-state response. (3 pts)

- d. Compare the response of the loaded active RC circuit with the unloaded active RC circuit of part (a). Also compare the response of the loaded active RC circuit with the unloaded passive circuit of Lab 7.5.1. Discuss your observations. (5 pts)

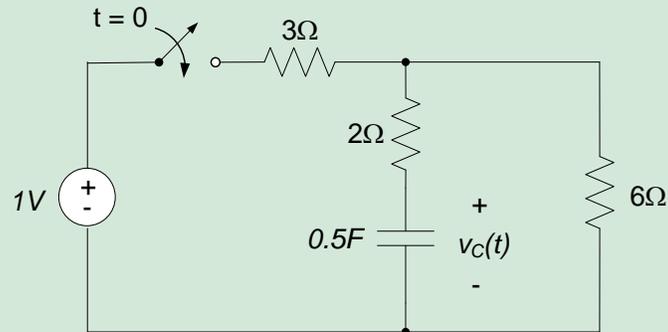
4. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuits' operation. (5 pts)

**TA Initials:** \_\_\_\_\_

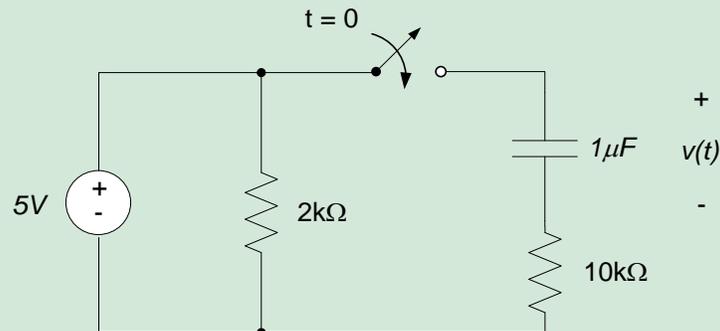
## Real Analog Chapter 7: Homework

7.1 For the circuit below, determine

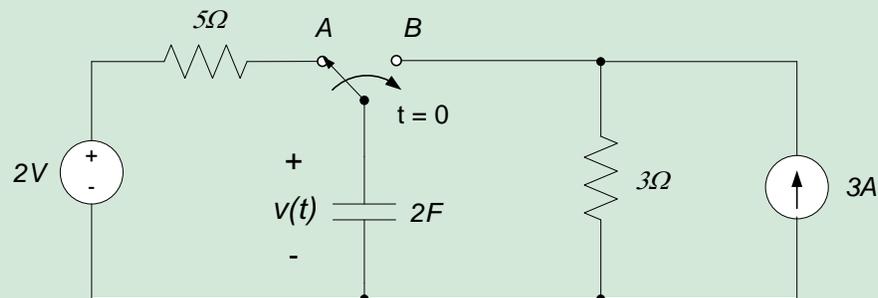
- $v_C(t)$ ,  $t > 0$
- $v_C(t)$ ,  $t > 0$  if the capacitance is  $1F$
- $v_C(t)$ ,  $t > 0$  if the capacitance is  $0.25F$



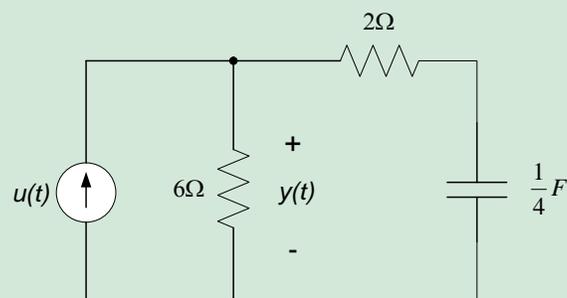
7.2 Find  $v(t)$ ,  $t > 0$ , in the circuit below.



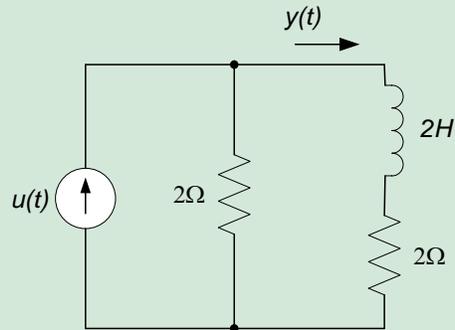
7.3 For the circuit shown, the switch moves from position A to position B at time  $t = 0$ . Find  $v(t)$ ,  $t > 0$ .



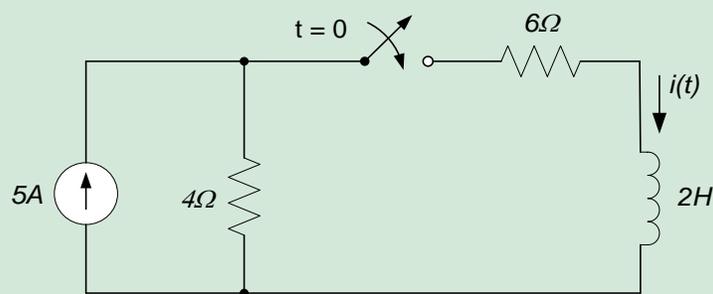
7.4 For the circuit shown, the input is the current source  $u(t) = 2u_0(t)A$  and the output is the voltage across the  $6\Omega$  resistor,  $y(t)$ . Find  $y(t)$ ,  $t > 0$ .



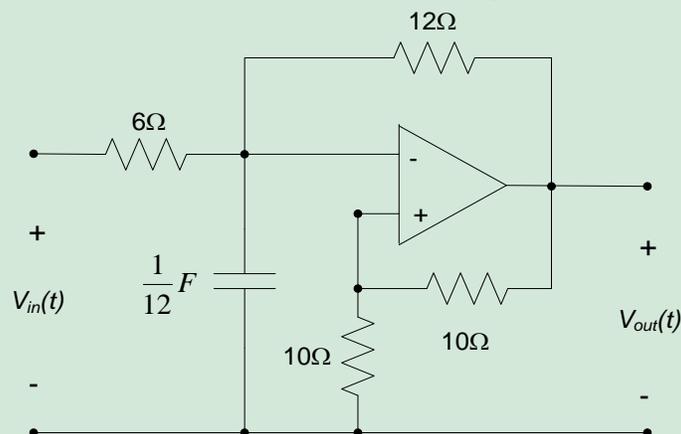
- 7.5 For the circuit shown, the input is the current source  $u(t)$  and the output is the current through the inductor,  $y(t)$ .
- Determine the differential equation relating  $u(t)$  and  $y(t)$ .
  - If  $u(t) = 3u_o(t)$ , determine  $y(t)$ ,  $t > 0$ .
  - Does your answer to part (b) agree with your expectations as to the circuit's physical behavior as  $t \rightarrow \infty$ ? Why or why not?



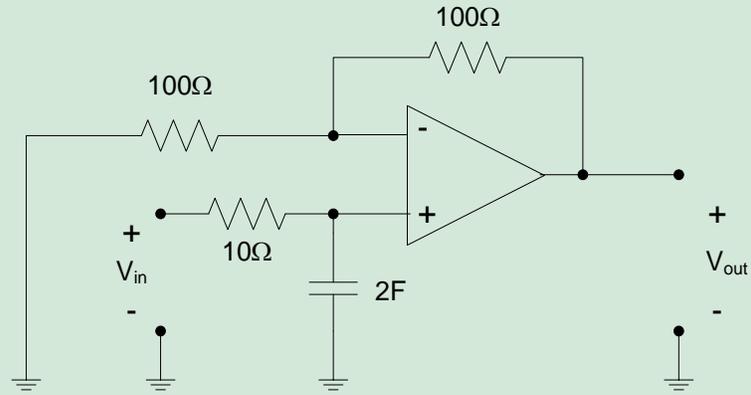
- 7.6 For the circuit shown, the switch closes at time  $t = 0$ .
- Write the differential equation governing  $i(t)$ ,  $t > 0$ .
  - Determine initial ( $t = 0$ ) and final ( $t \rightarrow \infty$ ) conditions on the current  $i(t)$ . You may assume that no energy is stored in the inductor before  $t = 0$ .
  - Find  $i(t)$ ,  $t > 0$ .



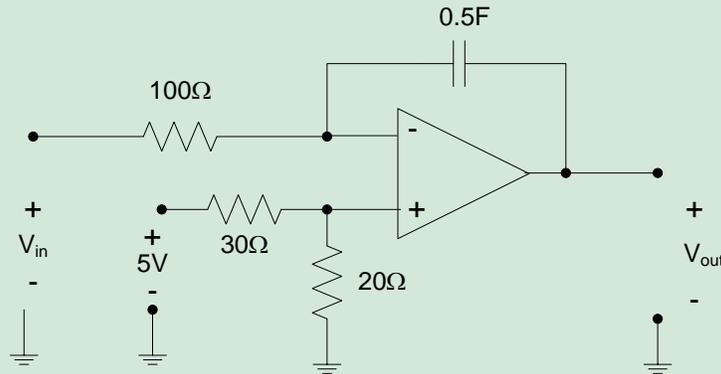
- 7.7 For the circuit below, determine the differential equation relating  $V_{out}(t)$  and  $V_{in}(t)$ .



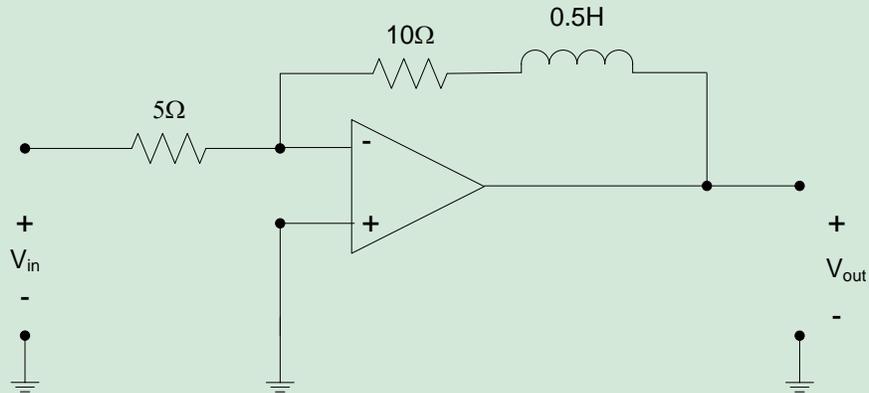
- 7.8 Determine the differential equations relating  $V_{out}$  and  $V_{in}$  for the circuit below.



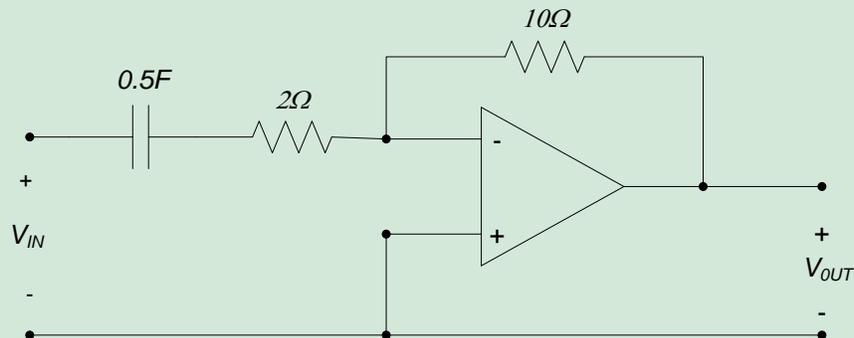
7.9 Determine the differential equations relating  $V_{out}$  and  $V_{in}$  for the circuit below.



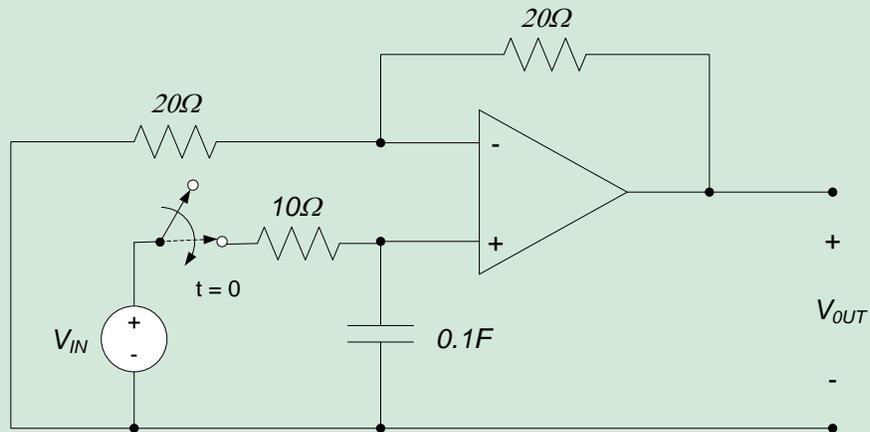
7.10 Determine the differential equations relating  $V_{out}$  and  $V_{in}$  for the circuit below.



7.11 Determine the differential equation relating  $V_{IN}(t)$  and  $V_{OUT}(t)$  for the circuit below.

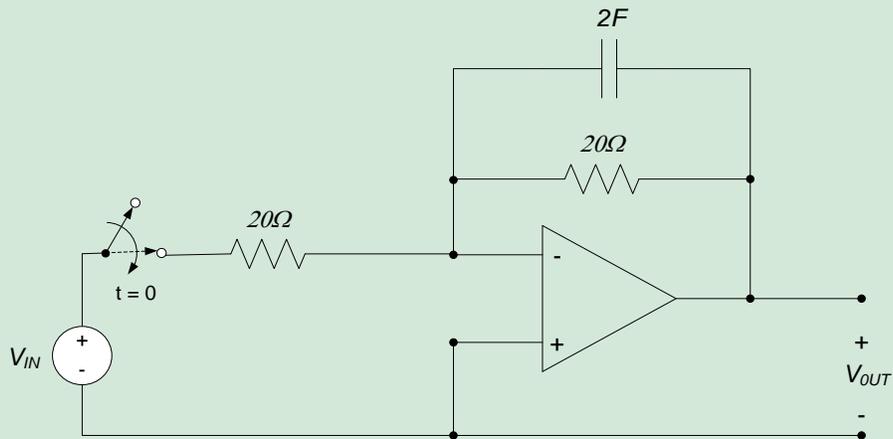


7.12 Find  $V_{OUT}(t)$ ,  $t > 0$ , in the circuit below.



7.13 For the circuit below, determine

- the differential equation governing  $V_{OUT}(t)$ ,  $t > 0$
- $V_{OUT}(t)$ ,  $t > 0$ .
- Does your solution for part (b) agree with your expectations, based on the circuit's behavior as  $t \rightarrow \infty$ ?



## Chapter 8: Second Order Circuits

### 8 Introduction and Chapter Objectives

Second order systems are, by definition, systems whose input-output relationship is a *second order* differential equation. A second order differential equation contains a *second order derivative* but no derivative higher than second order. Second order systems contain *two independent energy storage elements*, per our comments in Chapter 7 pertaining to the relationship between the number of energy storage elements in a system and the system order

Second order systems, like first order systems, are an extremely important class of systems. In previous chapters, we saw that the natural response of first order systems decays exponentially with time – the natural response decays monotonically to zero. The natural response of second order systems can, however, oscillate with time – we will see that a second order systems response can contain sinusoidal components. The motion of a pendulum, for example, can be modeled by a second order system. These oscillations are due to the transfer of energy between the two energy storage mechanisms; a pendulum, for example, oscillates because of the cyclic exchange of potential and kinetic energy of the mass.

Since the natural response of second-order systems can oscillate with time, their response can be fundamentally different than the response of first order systems. In the introduction to chapter 7, we noted that it is common to approximate higher-order systems as first order systems (at the time, we said that such a system has dominant first order modes). This approximation is not possible if the natural response of the higher order system oscillates. However, it may be possible to approximate the response of such a system as a second order system. Systems which behave approximately as second order systems have what are called *dominant second order modes*. In fact, the natural response of any higher order system can be considered in terms of the responses of multiple first and/or second order systems<sup>32</sup>. This is why an understanding of first and second order system responses is so crucial to the engineer – these responses provide the building blocks for understanding the responses of all linear systems.

In this textbook, of course, we are interested in the response of electrical circuits. Thus, we begin this chapter with a presentation of two simple second order electrical circuits: the series RLC and parallel RLC circuits. In section 8.1, we derive the governing equations for these circuits and use the results to write the general form of the differential equation governing second order systems. This equation is in terms of two very important parameters: the system *natural frequency* and the system *damping ratio*. The homogeneous solution of this general equation is determined in sections 8.2 and 8.4. In section 8.2, we develop the form of the solution (in terms of the natural frequency and damping ratio). Since the response of second order systems contains complex exponential functions, we provide some material (in section 8.3) relative to complex exponentials and sinusoidal signals. This material will provide us the necessary background to allow us to determine the natural response. Section 8.3 is optional for readers who are comfortable with complex exponential and sinusoidal signals. The overall natural response is developed in section 8.4, using the solution form presented in section 8.2 and the background material on complex exponentials in section 8.3.

#### After completing this chapter, you should be able to:

- Write differential equations governing second order circuits
- Define damping ratio and natural frequency from the coefficients of a second order differential equation

---

<sup>32</sup> In fact, in this chapter we will see that the responses of some second order systems can be interpreted in terms of two first order system responses.

- Express the form of the natural response of an arbitrary second order system in terms of complex exponentials, the damping ratio, and the natural frequency
- Summarize the behavior of the complex exponentials in the system natural response for the damping ratio ranges below:
  - Damping ratio greater than one
  - Damping ratio less than one
  - Damping ratio equal to one
- Write complex numbers in terms of complex exponentials
- Express sinusoidal signals in terms of complex exponentials
- Classify *overdamped*, *underdamped*, and *critically damped* systems according to their damping ratio
- Identify the expected shape of the natural response of over-, under-, and critically damped systems
- State from memory the definition of an underdamped second order system's overshoot, rise time, and steady-state response
- Use the coefficients of a second order system's governing equation to estimate the system's overshoot, rise time, and steady-state response

## 8.1 Introduction to Second Order Systems

We will develop our discussion of second order systems in the context of two electrical circuits examples.

### Example 8.1: Series RLC Circuit

Consider the circuit shown in Fig. 8.1 below, consisting of a resistor, a capacitor, and an inductor (this type of circuit is commonly called an RLC circuit). The circuit contains two energy storage elements: an inductor and a capacitor. The energy storage elements are independent, since there is no way to combine them to form a single equivalent energy storage element. Thus, we expect the governing equation for the circuit to be a second order differential equation. We will develop equations governing both the capacitor voltage,  $v_C(t)$  and the inductor current,  $i_L(t)$  as indicated in Fig. 8.1.

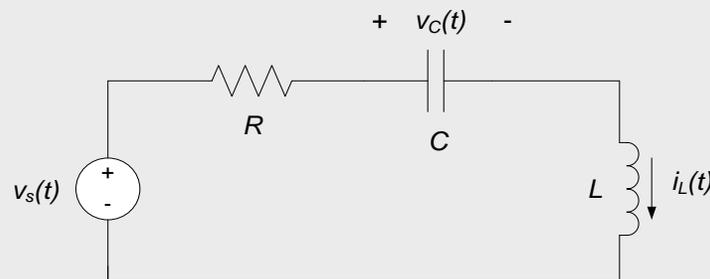


Figure 8.1. Series RLC circuit.

In order to determine the governing equations for  $v_C(t)$  and  $i_L(t)$  we will attempt to write two first-order differential equations for the system and then combine these equations to obtain the desired second order differential equation. To facilitate this process, the circuit of Fig. 8.1 is repeated in Fig. 8.2 with the node and loop we will use labeled. Note that we also label the current through the capacitor in terms of the capacitor voltage and the voltage across the inductor in terms of the inductor current.

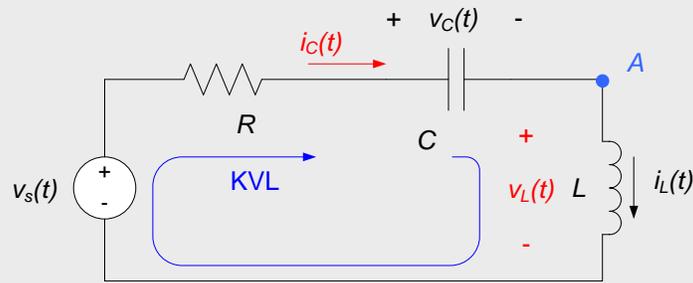


Figure 8.2. Series RLC circuit with node and loop defined.

The voltage-current relationships for inductors and capacitors indicate that, in Fig. 8.2,  $i_C(t) = C \frac{dv_C(t)}{dt}$  and  $V_L(t) = L \frac{di_L(t)}{dt}$ . Using the latter of these relations, KVL around the indicated loop in Fig. 8.2 provides:

$$v_s(t) = Ri_L(t) + v_C(t) + L \frac{di_L}{dt} \quad \text{Eq. 8.1}$$

KCL at node A, along with the voltage-current relation for the capacitor, indicates that:

$$C \frac{dv_C(t)}{dt} = i_L(t) \quad \text{Eq. 8.2}$$

We can determine the equation governing the capacitor voltage by differentiating equation (8.2) with respect to time to obtain an expression for the derivative of the inductor current:

$$C \frac{d^2v_C(t)}{dt^2} = \frac{di_L(t)}{dt} \quad \text{Eq. 8.3}$$

Substituting equations (8.2) and (8.3) into equation (8.1) results in:

$$v_s(t) = RC \frac{dv_C(t)}{dt} + v_C(t) + LC \frac{d^2v_C(t)}{dt^2}$$

Rearranging this slightly results in:

$$\frac{d^2v_C(t)}{dt^2} + \frac{R}{L} \frac{dv_C(t)}{dt} + \frac{1}{LC} v_C(t) = \frac{1}{LC} v_s(t) \quad \text{Eq. 8.4}$$

To determine the relationship governing the inductor current, we can again use equation (8.2) to write the capacitor voltage as:

$$v_C(t) = \frac{1}{C} \int_0^t i_L(t) dt \quad \text{Eq. 8.5}$$

Where we assume that the voltage across the capacitor at time  $t=0$  is zero; e.g.  $v_C(0)=0$ .

Substituting equation (8.5) into equation (8.1) results in the integro-differential equation:

$$v_s(t) = Ri_L(t) + \frac{1}{C} \int_0^t i_L(t) dt + L \frac{di_L}{dt}$$

In general, we prefer not to work with a mixture of derivatives and integrals in the same equation, so we differentiate the above to obtain our final expression for  $i_L(t)$ :

$$\frac{d^2i_L(t)}{dt^2} + \frac{R}{L} \frac{di_L(t)}{dt} + \frac{1}{LC} i_L(t) = \frac{1}{L} \frac{dv_s(t)}{dt} \quad \text{Eq. 8.6}$$

**Important Tip:** Equations (8.1) and (8.2) consist of two coupled first order differential equations in two unknowns:  $i_L(t)$  and  $v_C(t)$ . This set of differential equations completely describes the behavior of the circuit – if we are given appropriate initial conditions and the input function  $v_S(t)$  they can be solved to determine the inductor currents and capacitor voltages. Once the capacitor voltage and inductor current are known, the energy in the system is completely defined and we can determine any other desired circuit parameters. Any manipulations of equations (8.1) and (8.2) we performed subsequently do not fundamentally increase the information we have about the circuit – we were simply rearranging equations (8.1) and (8.2) to create a single equation with the desired unknown.

### Example 8.2: Parallel RLC Circuit

Our second exemplary circuit is the parallel combination of a resistor, capacitor, and inductor shown in Figure 8.3. The circuit is, for relatively obvious reasons, called a parallel RLC circuit. The forcing function to the circuit is provided by a current source,  $i_S(t)$ . The circuit of Fig. 8.3, like that of Fig. 8.2, contains two independent energy storage elements –we expect the governing equations for the circuit to be second order differential equations. We will again develop equations governing both the capacitor voltage,  $v_C(t)$  and the inductor current,  $i_L(t)$  as indicated in Fig. 8.3.

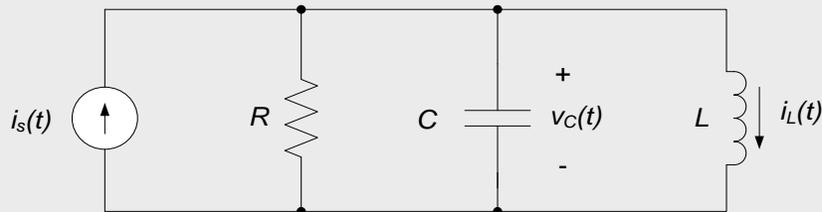


Figure 8.3. Parallel RLC circuit.

Consistent with our approach for the series RLC circuit, we will write first order differential equations using the variables  $v_C(t)$  and  $i_L(t)$  and subsequently combine these equations to eliminate the undesired unknown. Figure 8.4 shows the node and loop we will use to generate these equations. Figure 8.4 also shows the current  $i_C(t)$  through the capacitor in terms of the capacitor voltage and the voltage across the inductor in terms of the inductor current.

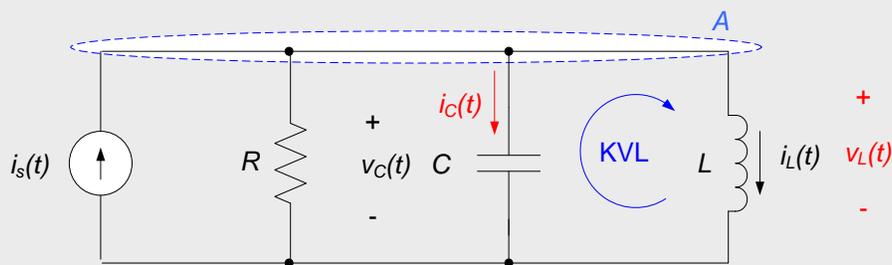


Figure 8.4. Parallel RLC circuit with node and loop defined.

KVL around the indicated loop provides:

$$L \frac{di_L(t)}{dt} = v_C(t) \quad \text{Eq. 8.7}$$

KCL at node A provides:

$$i_S(t) = \frac{v_C(t)}{R} + i_L(t) + C \frac{dv_C(t)}{dt} \quad \text{Eq. 8.8}$$

As in example 8.1, equations (8.7) and (8.8) completely describe the circuit's response. However, to gain additional insight into the individual parameters  $v_C(t)$  and  $i_L(t)$ , we rearrange these equations into second order differential equations in a single dependent variable. For example, we can differentiate equation (8.7) to obtain:

$$L \frac{d^2 i_L(t)}{dt^2} = \frac{dv_C(t)}{dt} \quad \text{Eq. 8.9}$$

Equations (8.7) and (8.9) can be substituted into equation (8.8) to obtain a second order differential equation in the variable  $i_L(t)$ . After some manipulation, the resulting equation is:

$$\frac{d^2 i_L(t)}{dt^2} + \frac{1}{RC} \frac{di_L(t)}{dt} + \frac{1}{LC} i_L(t) = \frac{1}{LC} i_S(t) \quad \text{Eq. 8.10}$$

Likewise, we can integrate equation (8.7) and use the result to write equation (8.8) in terms of the capacitor voltage:

$$\frac{d^2 v_C(t)}{dt^2} + \frac{1}{RC} \frac{dv_C(t)}{dt} + \frac{1}{LC} v_C(t) = \frac{1}{C} \frac{dv_S(t)}{dt} \quad \text{Eq. 8.11}$$

The important thing to note about the above examples is that equations (8.4), (8.6), (8.10), and (8.11) can all be written in the form:

$$\frac{d^2 y(t)}{dt^2} + 2\zeta\omega_n \frac{dy(t)}{dt} + \omega_n^2 y(t) = f(t) \quad \text{Eq. 8.12}$$

Where  $y(t)$  is the system parameter of interest (for example, a voltage or current in an electrical circuit),  $\omega_n$  is the undamped natural frequency and  $\zeta$  is the damping ratio; the physical significance of these parameters will be presented later in this series of chapters. The point being made here is that the governing equation for any second order system can be written in the form of equation (8.12); thus, we will focus on this format for our discussion of the solution of second order differential equations.

## Section Summary

- Second order systems have two independent energy storage elements. These circuits are governed by second order differential equations.
- Unlike first order circuits, the natural response of second order circuits can oscillate. This oscillation is due to energy exchanges between the two energy storage elements (inductors and/or capacitors, in electrical circuits). The oscillations will die out with time due to energy dissipation elements (resistors, in electrical circuits).
- The general differential equation governing second order circuits is of the form:

$$\frac{d^2 y(t)}{dt^2} + 2\zeta\omega_n \frac{dy(t)}{dt} + \omega_n^2 y(t) = d(t)$$

where  $y(t)$  is a voltage or current of interest in the circuit.

- In the equation above,  $\omega_n$  is called the *undamped natural frequency* and  $\zeta$  is called the damping ratio. These parameters (along with the DC gain of the circuit, as presented in section 7.5) govern the shape of the circuit natural response.

## 8.1 Exercises

1. The differential equation governing a circuit's natural response is:

$$\frac{d^2y(t)}{dt^2} + 32\frac{dy(t)}{dt} + 64y(t) = 0$$

Where  $y(t)$  is the circuit response. What are:

- The circuit's natural frequency and,
- The circuit's damping ratio

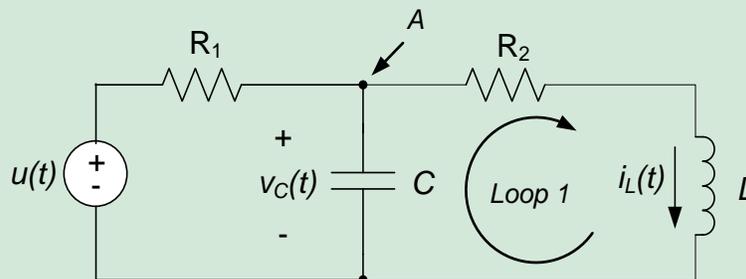
2. The differential equation governing a circuit's natural response is:

$$\frac{d^2y(t)}{dt^2} + 8\frac{dy(t)}{dt} + 64y(t) = 0$$

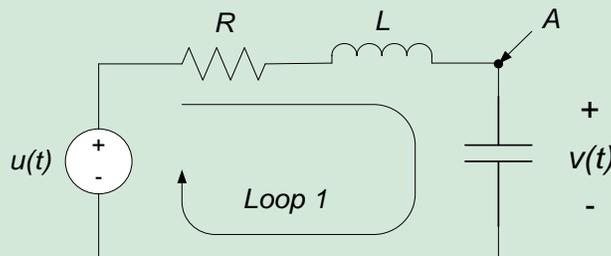
Where  $y(t)$  is the circuit response. What are:

- The circuit's natural frequency and,
- The circuit's damping ratio

3. For the circuit below, determine the differential equation for  $i_L(t), t > 0$ . (Hint: write KCL at node A and KVL around loop 1 to get two equations in two unknowns,  $v_C(t)$  and  $i_L(t)$ . Then combine the equations to eliminate  $v_C(t)$ .)



4. For the circuit shown below, apply KCL at node A and KVL around loop 1 to write two first order differential equations in two unknowns: the current through the inductor and the voltage across the capacitor. Combine these equations to write a single second order differential equation in the voltage  $v(t)$ .



## 8.2 Second Order System Natural Response, Part 1

In section 8.1, we determined that the the differential equation governing a second-order system could be written in the form:

$$\frac{d^2y(t)}{dt^2} + 2\zeta\omega_n \frac{dy(t)}{dt} + \omega_n^2 y(t) = f(t)$$

Where  $y(t)$  is any system parameter of interest (for example, a voltage or current in an electrical circuit),  $\omega_n$  and  $\zeta$  are the *undamped natural frequency* and the *damping ratio* of the system, respectively, and  $f(t)$  is a forcing function applied to the system. In general,  $f(t)$  is an arbitrary function of the physical input to the system. (The physical input to the system can be, for example, a voltage or current source;  $f(t)$  is a function of these power sources. In section 8.1, we saw examples in which  $f(t)$  was proportional to an applied voltage or current or proportional to the derivative of an applied voltage or current.)

In this chapter, we will develop the homogeneous solution to the above second order differential equation. For the homogeneous case, the forcing function  $f(t)=0$ .

In this chapter, we develop the homogeneous solution to the differential equation provided in equation (8.12) of section 8.1. The appropriate differential equation to be solved is thus:

$$\frac{d^2y_h(t)}{dt^2} + 2\zeta\omega_n \frac{dy_h(t)}{dt} + \omega_n^2 y_h(t) = 0 \quad \text{Eq. 8.13}$$

In equation (8.13),  $y_h(t)$  is the solution to the homogeneous, or unforced differential equation given by equation (8.13). A second order differential equation requires two initial conditions in order to solve it; we will take our initial conditions to be the value of the function  $y(t)$  at  $t=0$  and the derivative of the function  $y(t)$  at  $t=0$ . We will state our initial conditions as:

$$\begin{aligned} y(t = 0) &= y_0 \\ \left. \frac{dy(t)}{dt} \right|_{t=0} &= y'_0 \end{aligned} \quad \text{Eq. 8.14}$$

Our approach to the solution of equation (8.13) will be consistent with our previous approach to the solution of first order homogeneous differential equations: we will assume the form of the differential equation (8.13) plug this assumed solution into equation (8.13) and then use our initial conditions to determine any unknown constants in the solution.

Examination of equation (8.13) leads us to conclude that the solution  $y_h(t)$  of equation (8.13) must be a function whose form does not change upon differentiation. Thus, we assume (consistent with our approach in section 7.3) that the solution to equation (8.13) will be of the form:

$$y_h(t) = Ke^{st} \quad \text{Eq. 8.15}$$

Substituting equation (8.15) into equation (8.13) results in:

$$(Ks^2)e^{st} + 2\zeta\omega_n(Ks)e^{st} + K\omega_n^2e^{st} = 0$$

The above can be simplified to:

$$[s^2 + 2\zeta\omega_n s + \omega_n^2]Ke^{st} = 0$$

The solutions to the above equation are  $Ke^{st} = 0$  and  $s^2 + 2\zeta\omega_n s + \omega_n^2 = 0$ . The first of these results in the trivial solution,  $K=0$ , which in general will not allow us to satisfy our initial conditions. Thus, in our solution given by equation (8.15), we choose  $s$  according to:

$$s^2 + 2\zeta\omega_n s + \omega_n^2 = 0 \quad \text{Eq. 8.16}$$

Since equation (8.16) is quadratic, values of  $s$  which satisfy it are given by:

$$s = \frac{-2\zeta\omega_n \pm \sqrt{(2\zeta\omega_n)^2 - 4(2\zeta\omega_n)\omega_n^2}}{2}$$

After simplification, this provides:

$$s = -\zeta\omega_n \pm \omega_n\sqrt{\omega^2 - 1} \quad \text{Eq. 8.17}$$

Equations (8.15) and (8.17) indicate that there are two possible solutions to equation (8.13). Since the original differential equation is linear, we know that superposition is valid and our overall solution can be a linear combination of the two solutions provided by equations (8.15) and (8.17). Thus, we take our overall solution to be of the form:

$$y_h(t) = K_1 e^{s_1 t} + K_2 e^{s_2 t}$$

Where  $s_1$  and  $s_2$  are provided by equation (8.17) so that:

$$y_h(t) = K_1 e^{(-\zeta\omega_n + \omega_n\sqrt{\zeta^2 - 1})t} + K_2 e^{-(\omega_n\sqrt{\zeta^2 - 1})t}$$

Which can be re-written as:

$$y_h(t) = e^{-\zeta\omega_n t} \left[ K_1 e^{(\omega_n\sqrt{\zeta^2 - 1})t} + K_2 e^{-(\omega_n\sqrt{\zeta^2 - 1})t} \right] \quad \text{Eq. 8.18}$$

The initial conditions, given by equations (8.14) can be used to determine the unknown constants,  $K_1$  and  $K_2$ .

Let us briefly examine the form of equation (8.18) before providing examples of the homogeneous solution for specific circuit-related examples. We do this by examining individual terms in equation (8.18):

- In equation (8.18), the term  $e^{-\zeta\omega_n t}$  is an exponential function of the form discussed in section 6.2. Thus, we know that this term corresponds to a decaying exponential, as long as the term  $\zeta\omega_n$  is positive.
- There are three possible forms which the term  $e^{\pm(\omega_n\sqrt{\zeta^2 - 1})t}$  can take:
  - If  $\zeta > 1$ , the terms  $e^{\pm(\omega_n\sqrt{\zeta^2 - 1})t}$  are either growing or decaying exponentials of the form discussed in section 6.2 (if  $\zeta > 1$   $e^{(\omega_n\sqrt{\zeta^2 - 1})t}$  grows exponentially with time and  $e^{-(\omega_n\sqrt{\zeta^2 - 1})t}$  decays exponentially with time).
  - If  $\zeta = 1$ , the terms  $e^{\pm(\omega_n\sqrt{\zeta^2 - 1})t}$  are constant and equal to one ( $e^{\pm(0)t} = 1$ ).
  - If  $\zeta < 1$ , the terms  $e^{\pm(\omega_n\sqrt{\zeta^2 - 1})t}$  are *complex exponentials*. (The term  $\sqrt{\zeta^2 - 1} = j\sqrt{1 - \zeta^2}$ , where  $j = \sqrt{-1}$ . Thus, the term  $e^{\pm(\omega_n\sqrt{\zeta^2 - 1})t} = e^{\pm(j\omega_n\sqrt{1 - \zeta^2})t}$  and we have an exponential raised to an imaginary power.

Before examining the above results in more detail and performing some physical, circuit-related examples, we present some material in section 8.3 relative to complex exponentials and sinusoidal signals. This material will provide us a context within which we can place our solution of equation (8.18). Section 8.3 is optional for readers who are comfortable with complex exponential and sinusoidal signals.

## Section Summary

- The form of the natural response of a general second order system is:

$$y_h(t) = e^{-\zeta\omega_n t} \left[ K_1 e^{(\omega_n\sqrt{\zeta^2 - 1})t} + K_2 e^{-(\omega_n\sqrt{\zeta^2 - 1})t} \right]$$

Where  $K_1$  and  $K_2$  are constants which depend upon the system initial conditions.

- The form of the  $e^{\pm(\omega_n\sqrt{\zeta^2-1})t}$  terms in the system response depends strongly upon the damping ratio. If  $\zeta < 1$ , these terms become growing or decaying exponentials. If  $\zeta < 1$ , these terms are complex exponentials, and the solution will have sinusoidal components. The relationship between complex exponentials is presented in more depth in section 8.3. If  $\zeta = 1$ , the  $e^{\pm(\omega_n\sqrt{\zeta^2-1})t}$  are simply one.
- The  $e^{-\zeta\omega_n t}$  term in the natural response causes the overall solution to decay as time increases. Thus, the natural response goes to zero as  $t \rightarrow \infty$ .

## 8.3 Sinusoidal Signals and Complex Exponentials

Sinusoidal signals and complex exponentials are extremely important to any engineer who is concerned with determining the dynamic response of a system. Electrical circuits, in particular, are often characterized by their response to sinusoidal inputs.

This chapter provides some background relative to these signals.

### 8.3.1 Sinusoidal Signals

Sinusoidal signals are represented in terms of sine and/or cosine functions. In general, we will represent sinusoids as cosine functions. Our general expression for a sinusoidal signal is:

$$v(t) = V_p \cos(\omega t + \theta) \quad \text{Eq. 8.19}$$

Where  $V_p$  is the zero-to-peak amplitude of the sinusoid,  $\omega$  is the radian frequency of the sinusoid (we will always use radians/second as the units of  $\omega$ ) and  $\theta$  is the phase angle of the sinusoid (in units of either radians or degrees are used for phase angle – recall that  $2\pi$  radians =  $360^\circ$ ). A representative plot of a sinusoidal signal is provided in Fig. 8.5. In Fig. 8.5, the frequency of the sinusoid is indicated as a period of the signal (the period is defined as the shortest time interval at which the signal repeats itself). The radian frequency of a sinusoid is related to the period by:

$$\omega = \frac{2\pi}{T} \quad \text{Eq. 8.20}$$

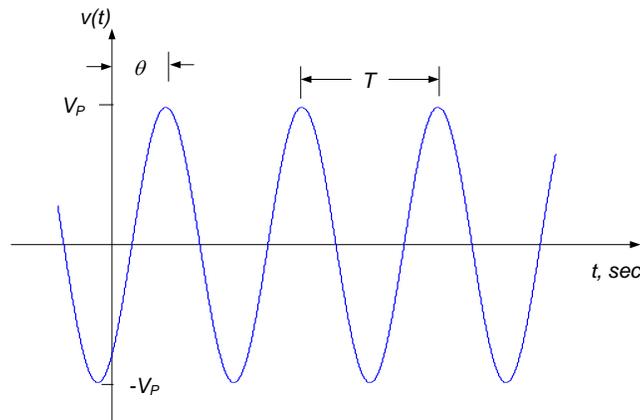


Figure 8.5. Arbitrary sinusoidal signal.

**Note:** Complex exponential signals have both real and imaginary parts; when we introduce complex exponentials later in this chapter, we will see that the cosine function is the real part of a complex exponential signal. Complex exponentials make dynamic systems analysis relatively simple – thus, we often analyze a signals response in terms

of complex exponentials. Since any measurable quantity is real-valued, taking the real part of the analytical result based on complex exponentials will result in a cosine function. Thus, cosines become a natural way to express signals which vary sinusoidally.

The frequency of a sinusoidal signal is alternately expressed in units of Hertz (abbreviated Hz). A Hertz is the number of cycles which the sinusoid goes through in one second. Thus, Hertz correspond to cycles/second. The frequency of a signal in Hertz is related to the period of the signal by:

$$f = \frac{1}{T} \quad \text{Eq. 8.21}$$

Radian frequencies relate to frequencies in Hertz by:

$$f = \frac{2\pi}{\omega} \Leftrightarrow \omega = 2\pi f \quad \text{Eq. 8.22}$$

Although frequencies of signals are often expressed in Hertz, it is not a unit which lends itself to calculations. Thus, all our calculations will be performed in radian frequency – if given a frequency in Hertz, it should be converted to radians/second before any calculations are performed based on this frequency.

### 8.3.2 Complex Exponentials

In our presentation of complex exponentials, we first provide a brief review of complex numbers. A complex number contains both real and imaginary parts. Thus, we may write a complex number  $A$  as:

$$A = a + jb \quad \text{Eq. 8.23}$$

Where:

$$j = \sqrt{-1} \quad \text{Eq. 8.24}$$

The complex number  $A$  can be represented on orthogonal axes representing the real and imaginary part of the number, as shown in Fig. 8.6. (In Fig. 8.6, we have taken the liberty of representing  $A$  as a vector, although it is really just a number.) We can also represent the complex number in polar coordinates, also shown in Fig. 8.6. The polar coordinates consist of a magnitude  $|A|$  and phase angle  $\theta_A$ , defined as:

$$|A| = \sqrt{a^2 + b^2} \quad \text{Eq. 8.25}$$

$$\theta_A = \tan^{-1} \left( \frac{b}{a} \right) \quad \text{Eq. 8.26}$$

Notice that the phase angle is defined counterclockwise from the positive real axis. Conversely, we can determine the rectangular coordinates from the polar coordinates from:

$$a = \text{Re}\{A\} = |A|\cos(\theta_A) \quad \text{Eq. 8.27}$$

$$b = \text{Im}\{A\} = |A|\sin(\theta_A) \quad \text{Eq. 8.28}$$

Where the notation  $\text{Re}\{A\}$  and  $\text{Im}\{A\}$  denote the real part of  $A$  and the imaginary part of  $A$ , respectively.

The polar coordinates of a complex number  $A$  are often represented in the form:

$$A = |A|\angle\theta_A \quad \text{Eq. 8.29}$$

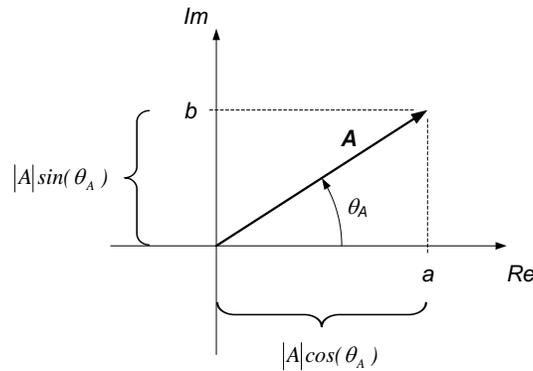


Figure 8.6. Representation of a complex number in rectangular and polar coordinates.

An alternate method of representing complex numbers in polar coordinates employs complex exponential notation. Without proof, we claim that:

$$e^{j\theta} = 1\angle\theta \quad \text{Eq. 8.30}$$

Thus,  $e^{j\theta}$  is a complex number with magnitude 1 and phase angle  $\theta$ . From Fig. 8.6, it is easy to see that this definition of the complex exponential agrees with Euler's equation:

$$e^{\pm j\theta} = \cos \theta \pm j \sin \theta \quad \text{Eq. 8.31}$$

With the definition of equation (8.30), we can define any arbitrary complex number in terms of complex exponentials. For example, our previous complex number  $A$  can be represented as:

$$A = |A|e^{j\theta_A} \quad \text{Eq. 8.32}$$

We can generalize our definition of the complex exponential to time-varying signals. If we define a time varying signal  $e^{j\omega t}$ , we can use equation (8.31) to write:

$$e^{\pm j\omega t} = \cos \omega t \pm j \sin \omega t \quad \text{Eq. 8.33}$$

The signal  $e^{j\omega t}$  can be visualized as a unit vector rotating around the origin in the complex plane; the tip of the vector scribes a unit circle with its center at the origin of the complex plane. This is illustrated in Fig. 8.7. The vector rotates at a rate defined by the quantity  $\omega$ —the vector makes one complete revolution every  $\frac{2\pi}{\omega}$  seconds.

The projection of this rotating vector on the real axis traces out the signal  $\cos \omega t$ , as shown in Fig. 8.7, while the projection of the rotating vector on the imaginary axis traces out the signal  $\sin \omega t$ , also shown in Fig. 8.7.

Thus, we interpret the complex exponential function  $e^{j\omega t}$  as an alternate “type” of sinusoidal signal. The real part of this function is  $\cos \omega t$  while the imaginary part of this function is  $\sin \omega t$ .

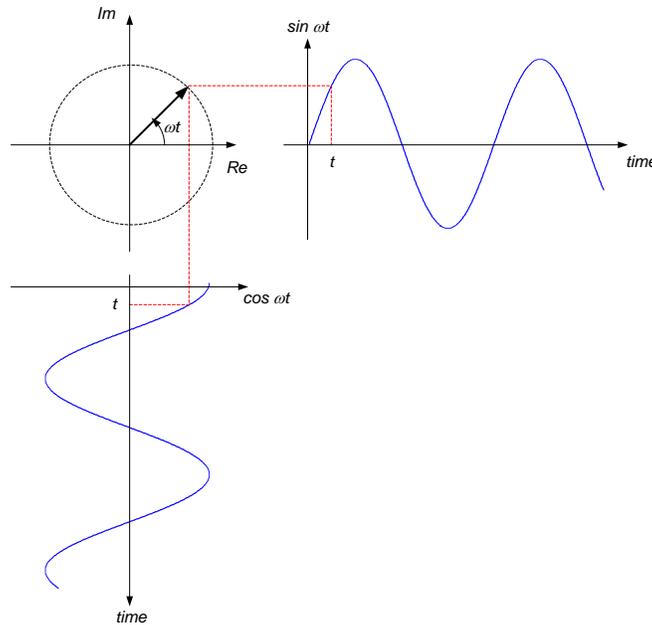


Figure 8.7. Illustration of  $e^{j\omega t}$ .

## Section Summary

- We will represent sinusoidal signal in terms of cosine functions. The general form of our sinusoidal signals is:  $v(t) = V_p \cos(\omega t + \theta)$ .
- Sinusoidal signals can also be represented as complex exponentials. The relationship is an extension of Euler's equation, and is:

$$e^{\pm j\omega t} = \cos \omega t \pm j \sin \omega t$$

1. By extension, cosine signals can be represented in terms of complex exponentials as:

$$\cos(\omega t) = \frac{e^{j\omega t} + e^{-j\omega t}}{2}$$

2. The above general sinusoidal signal can be expressed as the real part of a complex exponential:

$$V_p \cos(\omega t + \theta) = \text{Re}\{V_p e^{j(\omega t + \theta)}\}$$

## 8.2 Exercises

1. A complex valued signal  $v(t)$  is given by  $v(t) = e^{j(\pi t - 30^\circ)}$ . What is the real part of the signal?
2. Represent the signal  $v(t) = \cos(300t)$  in terms of complex exponentials.
3. Represent the complex number  $3 - j3$  in polar coordinates.
4. Represent the complex number  $e^{j\pi}$  in rectangular coordinates.
5. Represent the complex number  $e^{-\frac{j\pi}{2}}$  in rectangular coordinates.

## 8.4 Second Order System Natural Response, Part 2

In section 8.2, we developed the form of the solution of the differential equation governing the natural response of second order systems. The form of the solution contained so-called complex exponentials; the background

material relative to these signals was provided in section 8.3. We are thus now in a position to re-examine and interpret the solution presented in section 8.2

In section 8.2, we noted that the form of the natural response of a second order system was strongly dependent upon the damping ratio,  $\zeta$ . If the damping ratio was greater than one, all terms in the response decay exponentially, but if the damping ratio was between zero and one some terms in the response became complex exponentials – in section 8.3, we saw that this corresponded to an oscillating signal. Thus, depending upon the value of damping ratio, the response could decay exponentially or oscillate. In this chapter, we will quantify and formalize these results. This section concludes with an extended example of a second order system natural response.

In section 8.1, the differential equation governing the natural response of a second order system was written as:

$$\frac{d^2 y_h(t)}{dt^2} + 2\zeta\omega_n \frac{dy_h(t)}{dt} + \omega_n^2 y_h(t) = 0 \quad \text{Eq. 8.34}$$

Where  $y(t)$  is any system parameter of interest,  $\omega_n$  is the *undamped natural frequency* and  $\zeta$  is the *damping ratio*. The initial conditions are the value of the function  $y(t)$  at  $t=0$  and the derivative of the function  $y(t)$  at  $t=0$ :

$$y(t = 0) = y_0$$

$$\left. \frac{dy(t)}{dt} \right|_{t=0} = y'_0 \quad \text{Eq. 8.35}$$

In section 8.2, we wrote the solution to equation (8.34) in the form:

$$y_h(t) = e^{-\zeta\omega_n t} \left[ K_1 e^{(\omega_n\sqrt{\zeta^2-1})t} + K_2 e^{-(\omega_n\sqrt{\zeta^2-1})t} \right] \quad \text{Eq. 8.36}$$

Where  $K_1$  and  $K_2$  are unknown coefficients which can be determined by application of the initial conditions provided in equation (8.35). The form of the solution of equation (8.36) will fall into one of three categories, depending on the value of damping ratio. The three possible cases are:

1. If  $\zeta > 1$ , all terms in the solution will be either growing or decaying exponentials and the solution will decay exponentially with time. If the damping ratio is large, this decay rate can be very slow. A system with  $\zeta > 1$  is said to be *overdamped*.
2. If  $\zeta < 1$ , the terms  $e^{\pm(\omega_n\sqrt{\zeta^2-1})t}$  are *complex exponentials*. Thus, have terms in our solution which are exponentials raised to an imaginary power and the solution can oscillate. A system with  $\zeta < 1$  is said to be *underdamped*.
3.  $\zeta = 1$ ; the form of the solution in this case is approximately that of case 1 above, in that the solution will decay exponentially. However, in this case, the response decay rate will be faster than the response of any overdamped system with the same natural frequency. Systems with  $\zeta = 1$  are said to be *critically damped*.

Details of the responses for each of the above three cases are provided in the subsections below.

## 1. Overdamped System

For an overdamped system,  $\zeta > 1$ , and equation (8.36) becomes as shown in equation (8.37).

$$y_h(t) = e^{-\zeta\omega_n t} \left[ \frac{\dot{y}_0 + (\zeta + \sqrt{\zeta^2 - 1})\omega_n y_0}{2\omega_n\sqrt{\zeta^2 - 1}} e^{\omega_n t\sqrt{\zeta^2 - 1}} + \frac{\dot{y}_0 - (\zeta - \sqrt{\zeta^2 - 1})\omega_n y_0}{2\omega_n\sqrt{\zeta^2 - 1}} e^{-\omega_n t\sqrt{\zeta^2 - 1}} \right] \quad \text{Eq. 8.37}$$

In equation (8.37), the  $e^{-\zeta\omega_n t}$  term is a decaying exponential with time constant  $\frac{1}{\zeta\omega_n}$ . The  $e^{\omega_n t\sqrt{\zeta^2-1}}$  is a growing exponential with time constant  $\frac{1}{\omega_n\sqrt{\zeta^2-1}}$ . Thus, the overall system response is a sum of two decaying exponential signals, one which is proportional to  $e^{-\zeta\omega_n t} \cdot e^{\omega_n t\sqrt{\zeta^2-1}}$  and the other which is proportional to  $e^{-\zeta\omega_n t} \cdot e^{-\omega_n t\sqrt{\zeta^2-1}}$ .

The term  $e^{-\zeta\omega_n t} \cdot e^{\omega_n t\sqrt{\zeta^2-1}}$  is the product of two exponentials: one which grows with time, and the other which decays with time. The decaying exponential time constant,  $\frac{1}{\zeta\omega_n}$ , is smaller than the growing exponential time constant,  $\frac{1}{\omega_n\sqrt{\zeta^2-1}}$ . Thus, the product of the two will decay with time, though the decay rate may be very slow.

(Note that in the limit as  $\zeta \rightarrow \infty$ ,  $\frac{1}{\zeta\omega_n} \approx \frac{1}{\omega_n\sqrt{\zeta^2-1}}$ , the two time constants are nearly identical, and this term becomes constant with time.)

The term  $e^{\zeta\omega_n t} \cdot e^{-\omega_n t\sqrt{\zeta^2-1}}$  is the product of two decaying exponentials; this term will, in general, decay quickly relative to the  $e^{\zeta\omega_n t} \cdot e^{-\omega_n t\sqrt{\zeta^2-1}}$  term.

An example of the response of an overdamped system is shown in Fig. 8.8, for various values of damping ratio. The two system time constants are readily observable in this example. Note that as the damping ratio increases, the overall time required for the system response to decay to zero increases. The response of overdamped systems cannot oscillate; however, the response can change sign once (e.g. the function is allowed one zero-crossing).

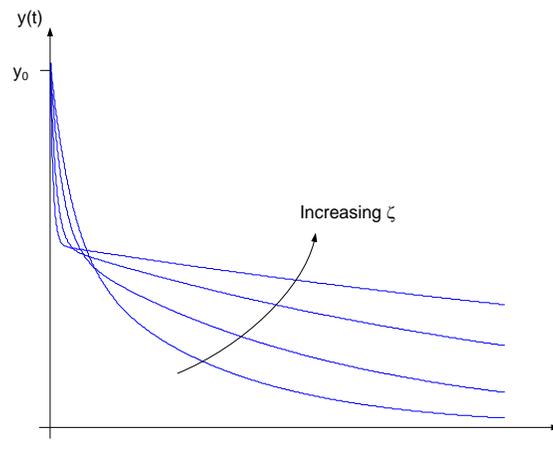


Figure 8.8. Overdamped system response.

## 2. Underdamped System

For an underdamped system,  $\zeta < 1$ , and equation (8.36) becomes as shown in equation (8.38).

$$y_h(t) = e^{-\zeta\omega_n t} \left[ \frac{\dot{y}_0 + \zeta\omega_n y_0}{\omega_n\sqrt{1-\zeta^2}} \sin(\omega_n t\sqrt{1-\zeta^2}) + y_0 \cos(\omega_n t\sqrt{1-\zeta^2}) \right] \quad \text{Eq. 8.38}$$

The solution is a decaying sinusoid. The decay rate is set by the term  $e^{-\zeta\omega_n t}$ , while the oscillation frequency of the sinusoid is  $\omega_n\sqrt{1-\zeta^2}$ . The oscillation frequency seen in the natural response is thus not identically the natural frequency of the system; it is also influenced by the damping ratio. This leads to the definition of the *damped natural frequency*:

$$\omega_d = \omega_n\sqrt{1-\zeta^2} \quad \text{Eq. 8.39}$$

Oscillations seen in the system response will have radian frequency  $\omega_d$ ; thus, the period of the oscillations is  $\frac{2\pi}{\omega_d}$ .

Example responses for underdamped systems are shown in Fig. 8.9; the responses shown are all for the same natural frequency and initial conditions – only the damping ratio varies. Note that smaller damping ratios result in slower decay rates for the response, oscillations persist for longer and are more pronounced for smaller damping ratios.

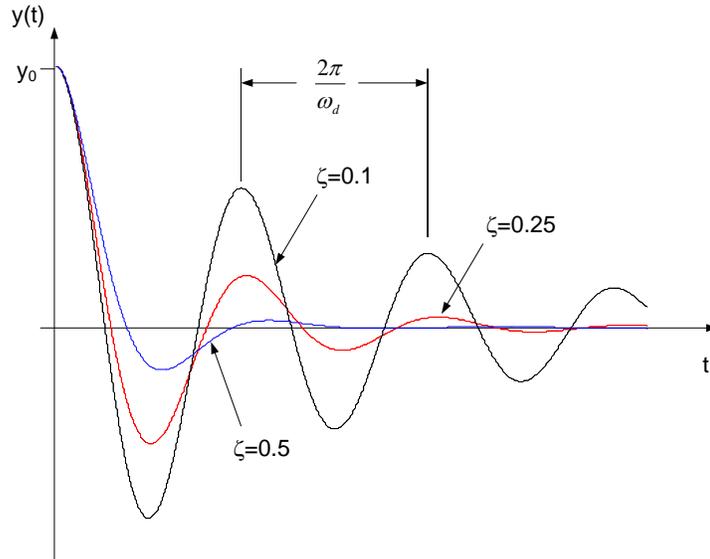


Figure 8.9. Underdamped system response.

### 3. Critically Damped System

For a critically damped system,  $\zeta = 1$ , and equation (8.36) becomes as shown in equation (8.40).

$$y_h(t) = e^{-\zeta\omega_n t} [y_0 + (\dot{y}_0 + \omega_n y_0 t)] \tag{Eq. 8.40}$$

The critically damped system response does not oscillate although, as with the overdamped case, one zero crossing of the function is allowed. The importance of the critically damped system response is that, for a particular natural frequency, it has the shortest decay time without oscillation of any system. An example response of a critically damped system is shown in Fig. 8.10.

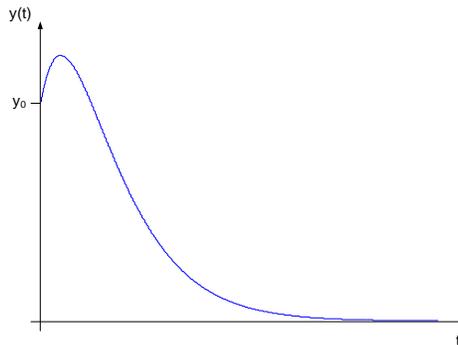
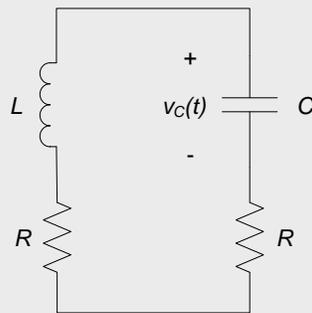


Figure 8.10. Critically damped system response.

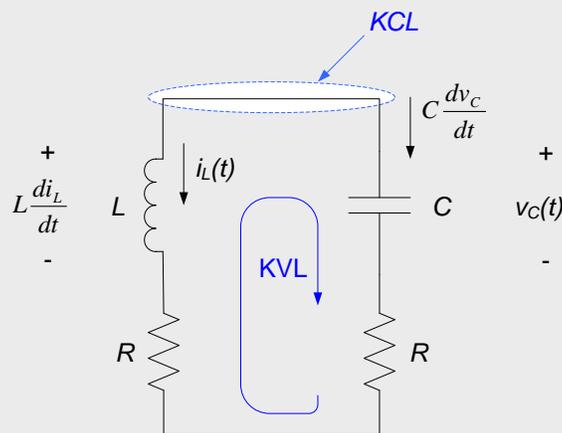
### Example 8.3

For the circuit shown below:

1. Write the differential equation for  $v_C(t)$ .
2. If  $L=1H$ ,  $R=200\Omega$ , and  $C=1 \times 10^{-6}F$ , find the undamped natural frequency, the damping ratio, and the damped natural frequency.
3. For the conditions in part (2), is the system underdamped, overdamped, or critically damped?
4. For the values of  $L$  and  $C$  in part (2), determine the value of  $R$  that makes the system critically damped.
5. If  $v_C(0)=1V$  and  $i_L(0)=0.01A$ , what are the appropriate initial conditions to solve the differential equation determined in part (1)?



- a) As usual, we define the voltage across the capacitor and the current through the inductor as our variables and write KVL and KCL in terms of these variables. The figure below shows these variables, along with the associated currents through capacitors and voltages across inductors.



KCL at the indicated node results in:

$$i_L(t) + C \frac{dv_C(t)}{dt} = 0 \quad \text{Eq. 8.41}$$

KVL around the indicated loop provides:

$$L \frac{di_L(t)}{dt} + 2Ri_L(t) = v_C(t) \quad \text{Eq. 8.42}$$

The above two equations can be combined to obtain an equation for  $v_C(t)$ . To do this, we use the first equation to obtain:

$$i_L(t) = -C \frac{dv_C(t)}{dt} \quad \text{Eq. 8.43}$$

Differentiating equation (8.43) provides:

$$\frac{di_L(t)}{dt} = -C \frac{d^2v_C(t)}{dt^2} \quad \text{Eq. 8.44}$$

Substituting equations (8.43) and (8.44) into equation (8.42) results in:

$$-LC \frac{d^2v_C(t)}{dt^2} - 2RC \frac{dv_C(t)}{dt} = v_C(t)$$

Dividing the above by LC and grouping terms gives our final result:

$$\frac{d^2v_C(t)}{dt^2} + \frac{2R}{L} \frac{dv_C(t)}{dt} + \frac{1}{LC} v_C(t) = 0 \quad \text{Eq. 8.45}$$

b) Equation (5) is of the form:

$$\frac{d^2y(t)}{dt^2} + 2\zeta\omega_n \frac{dy(t)}{dt} + \omega_n^2 y(t) = 0 \quad \text{Eq. 8.46}$$

Equating coefficients in equations (8.45) and (8.46) and substituting  $L=1H$ ,  $R=200\Omega$ , and  $C=1 \times 10^{-6}F$  results in:

$$2\zeta\omega_n = \frac{2R}{L} = 400 \quad \text{Eq. 8.47}$$

$$\omega_n^2 = \frac{1}{LC} = 1 \times 10^6 \quad \text{Eq. 8.38}$$

Solving equation (8) for the natural frequency results in  $\omega_n=1000$  rad/sec. Substituting this result into equation (8.47) and solving for the damping ratio gives  $\zeta=0.2$ . The damped natural frequency is:

$$\omega_d = \omega_n \sqrt{1 - \zeta^2} = 979.8 \text{ rad/sec}$$

- c) The damping ratio determined in part (b) is  $\zeta=0.2$ ; since this is less than one, the system is underdamped.  
 d) In order for the system to be critically damped, the damping ratio  $\zeta = 1$ . From equation (8.47) with  $\zeta = 1$ , we obtain:

$$2\zeta\omega_n = \frac{1R}{L} \Rightarrow 2(1)(1000) = \frac{2R}{1H} \Rightarrow R = 1000\Omega$$

- e) Initial condition on  $v_C(t)$  are  $v_C(0)$  and  $\frac{dv_C(t)}{dt}|_{t=0}$ . We are given  $v_C(-)=1V$  in the problem statement, but we need to determine  $\frac{dv_C(t)}{dt}|_{t=0}$ ; the current through the inductor,  $i_L(0)$  can be used to determine this. The current through the inductor is related to the capacitor voltage via equation (8.43) above:

$$i_L(t) = -C \frac{dv_C(t)}{dt}$$

So at time  $t=0$ ,

$$i_L(0) = -(1 \times 10^{-6}F) \frac{dv_C(t)}{dt}|_{t=0} = 0.01A$$

Solving for  $\frac{dv_C(t)}{dt}|_{t=0}$ :

$$\left. \frac{dv_C(t)}{dt} \right|_{t=0} = \frac{1}{C} i_L(0) = -\frac{0.01A}{1 \times 10^{-6}F} = -10,000 V/sec$$

We conclude this example with plots of the system response for underdamped, critically damped, and overdamped conditions.

Figure 8.11 shows the response of the circuit described by the differential equation determined in part (a) above, for the circuit parameters provided in part (b), to the initial conditions of part (e). Thus, the governing differential equation is:

$$\frac{d^2v_C(t)}{dt^2} + \frac{2R}{L} \frac{dv_C(t)}{dt} + \frac{1}{LC} v_C(t) = 0$$

With  $L=1H$ ,  $R=200\Omega$ , and  $C=1 \times 10^{-6}F$ , the differential equation becomes:

$$\frac{d^2v_C(t)}{dt^2} + 400 \frac{dv_C(t)}{dt} + 1 \times 10^6 v_C(t) = 0$$

The initial conditions are, from part (e):

$$v_C(0) = 1V$$

$$\left. \frac{dv_C(t)}{dt} \right|_{t=0} = -10,000 V/sec$$

Using MATLAB to evaluate the differential equation results in Fig. 8.11. Fig. 8.11 agrees with our expectations based on the calculations of part (b). In part (b), we determined that the damping ratio  $\zeta=0.2$ , so that the system is underdamped – Fig. 8.11 exhibits the oscillations (multiple zero axis crossings) that we would expect from an underdamped system. Likewise, we determined in part (b) that the damped natural frequency of the system is approximately 980 rad/sec. The period of the oscillations we would expect to see in the response is therefore:

$$T = \frac{2\pi}{\omega_d} = 0.0064 \text{ seconds}$$

This value is consistent with the period of the oscillations seen in Fig. 8.11.

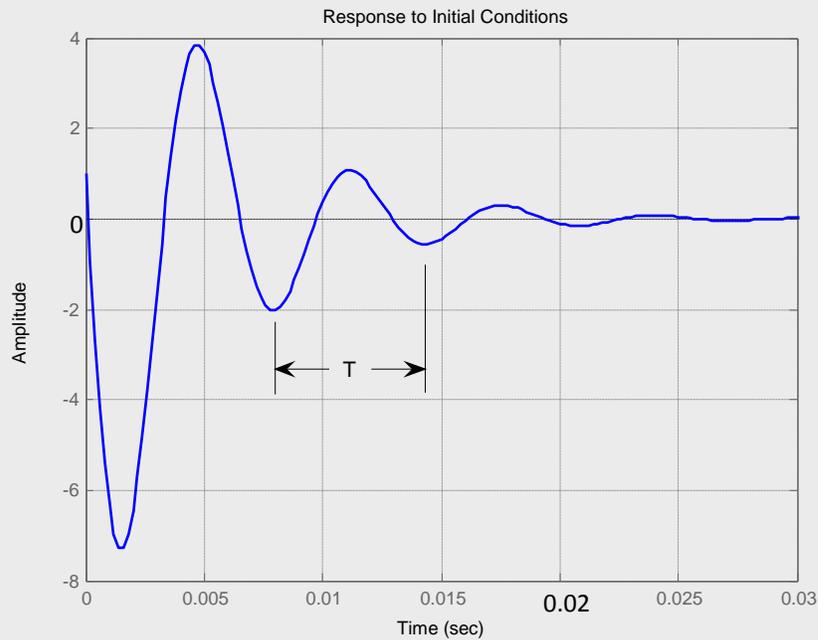


Figure 8.11. Underdamped response to initial conditions.

In part (d) above, we determined that the value of  $R$  resulting in a critically damped system is  $R=1000\Omega$ . Re-evaluating the above governing differential equation with this value for  $R$  results in:

$$\frac{d^2 v_C(t)}{dt^2} + 2000 \frac{dv_C(t)}{dt} + 1 \times 10^6 v_C(t) = 0$$

The initial conditions are as in the above example:

$$v_C(0) = 1V$$

$$\left. \frac{dv_C(t)}{dt} \right|_{t=0} = -10,000 V/sec$$

The resulting response is shown in Fig. 8.12. This plot also matches our expectations, though we have fewer quantitative results against which to compare it. The response does not oscillate (the response does have one and only one zero crossing, which is allowable for a critically damped or overdamped system. The response also appears to be composed of exponential signals, which is consistent with our expectations.

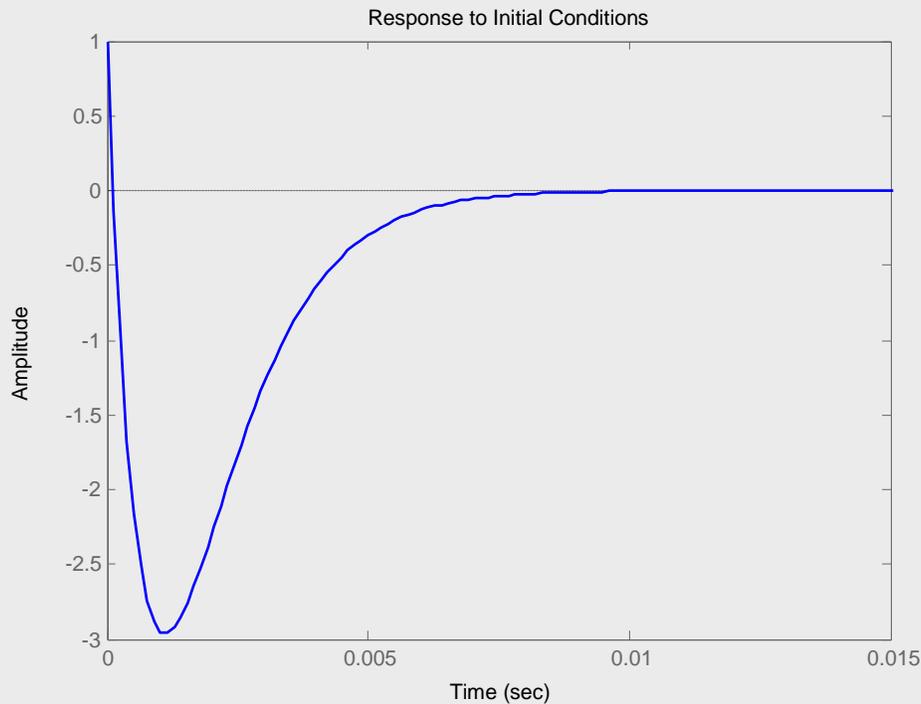


Figure 8.12. Critically damped system response.

In order to obtain a better understanding of critically damped vs. overdamped systems, we increase  $R$  to  $3000\Omega$ . The resulting damping ratio is  $\zeta=3$ ; increasing  $R$  above the critically damped value will result in an overdamped system since the damping ratio is proportional to  $R$ . We will expect the response shape to be somewhat like that shown in Fig. 8.12 (it will still be composed of decaying exponential functions) but the overdamped system should decay more slowly. This overdamped system response is shown in Fig. 8.13. This response agrees with our qualitative expectations – the response does not oscillate, and the decay time is longer than that shown in Fig. 8.12.

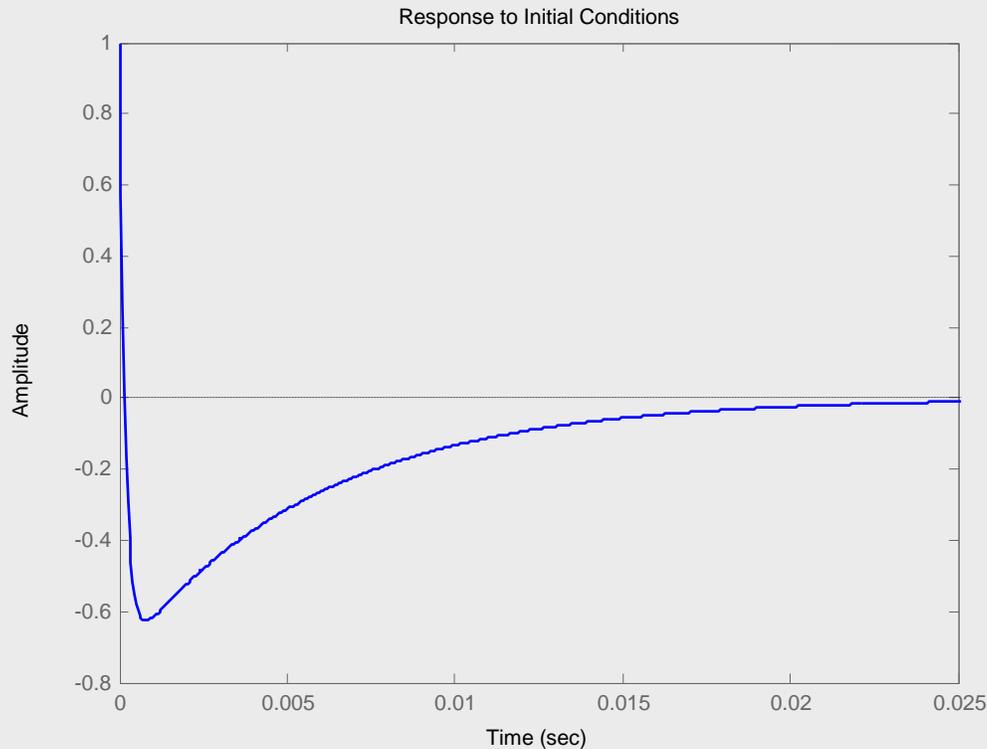


Figure 8.13. Overdamped system response.

## Section Summary

- It is common to categorize second order systems by their damping ratio. This also characterizes the shape of their natural response. The three categories are:
  - If  $\zeta > 1$ , the system is said to be *overdamped*. For this case, the response will decay exponentially with time with no oscillations. If the damping ratio is large, this decay rate can be very slow.
  - If  $\zeta < 1$ , the system is said to be *underdamped*. In this case, the natural response can oscillate. Increasing the damping ratio tends to reduce the amplitude of the oscillations, and cause the oscillations to die out more quickly.
  - $\zeta = 1$ , the solution is said to be critically damped. In this case, the response will not oscillate, and the decay rate of the response will be faster than the response of any overdamped system with the same natural frequency.

For an underdamped system, the oscillations observed in the response have a radian frequency  $\omega_d$  defined as:

$$\omega_d = \omega_n \sqrt{1 - \zeta^2}$$

$\omega_d$  is called the *damped natural frequency* of the system. The period of the oscillations in the natural frequency (the time between successive peaks) is  $\frac{2\pi}{\omega_d}$ .

## 8.3 Exercises

1. The differential equation governing a circuit with output  $y(t)$  is given by:

$$\frac{d^2 y(t)}{dt^2} + 6 \frac{dy(t)}{dt} + 144 y(t) = 0$$

What are the damping ratio and natural frequency of the circuit? Is the circuit under-, over-, or critically damped?

2. The differential equation governing a circuit's natural response is:

$$\frac{d^2y(t)}{dt^2} + 32\frac{dy(t)}{dt} + 64y(t) = 0$$

Where  $y(t)$  is the circuit response. What are:

- The circuit's natural frequency,
  - The circuit's damping ratio, and
  - The two time constants governing the circuit
3. The differential equation governing a circuit's natural response is:

$$\frac{d^2y(t)}{dt^2} + 8\frac{dy(t)}{dt} + 64y(t) = 0$$

Where  $y(t)$  is the circuit response. What are:

- The circuit's natural frequency,
- The circuit's damping ratio, and
- The circuit's damped natural frequency?

## 8.5 Second Order System Step Response

In this section, we address the case in which the input to a second order system consists of the sudden application of a constant voltage or current to the circuit; this type of input can be modeled as a step function. The response of a system to this type of input is called the *step response* of the system.

The material presented in this section will emphasize the development of qualitative relationship between the damping ratio and natural frequency of a system and the system's time-domain response. We will also see that we can quantitatively relate several specific response parameters to the system's damping ratio and natural frequency. This approach allows us to infer a great deal about the expected system response directly from the damping ratio and natural frequency of the system, without explicitly solving the differential equation governing the system. This approach is also useful in system design, since we can readily determine the damping ratio and natural frequency necessary to provide the desired response shape. Since the damping ratio and natural frequency are typically functions of resistances, capacitances, and inductances, we can readily design a system to produce the desired response.

In section 8.1, we wrote a general differential equation governing a second order system as:

$$\frac{d^2y(t)}{dt^2} + 2\zeta\omega_n\frac{dy(t)}{dt} + \omega_n^2y(t) = f(t) \quad \text{Eq. 8.49}$$

Where  $y(t)$  is any system parameter of interest (for example, a voltage or current in an electrical circuit),  $\omega_n$  and  $\zeta$  are the *undamped natural frequency* and the *damping ratio* of the system, respectively, and  $f(t)$  is a forcing function applied to the system.

In this chapter section, we restrict our attention to the specific case in which  $f(t)$  is a step function. Thus, the forcing function to the system can be written as:

$$f(t) = Au_0(t) = \begin{cases} 0, & t < 0 \\ A, & t > 0 \end{cases} \quad \text{Eq. 8.50}$$

Thus, the differential equation governing the system becomes:

$$\frac{d^2y(t)}{dt^2} + 2\zeta\omega_n \frac{dy(t)}{dt} + \omega_n^2 y(t) = Au_0(t) \quad \text{Eq. 8.51}$$

In addition to the above restriction on the forcing function, we will assume that the initial conditions are all zero (we sometimes say that the system is *initially relaxed*). Thus, for the second-order system above, our initial conditions will be:

$$\begin{aligned} y(t=0) &= 0 \\ \frac{dy(t)}{dt} \Big|_{t=0} &= 0 \end{aligned} \quad \text{Eq. 8.52}$$

Solving equation (8.51) with the initial conditions provided in equations (8.52) results in the *step response* of the system.

As in our discussion of forced first order system responses in section 8.1, we write the overall solution of the differential equation of equation (8.51) as the sum of a particular solution and a homogeneous solution. Thus:

$$y(t) = y_h(t) + y_p(t)$$

The homogeneous solution of second order differential equations has been discussed in sections 8.1 and 8.4 and will not be repeated here. The particular solution of the differential equation (8.51) can be obtained by examining the solution to the equation after the homogeneous solution has died out. Letting  $t \rightarrow \infty$  in equation (8.51) and noting that the forcing function is a constant as  $t \rightarrow \infty$  allows us to set  $\frac{d^2y(t \rightarrow \infty)}{dt^2} = \frac{dy(t \rightarrow \infty)}{dt} = 0$  and thus,

$$\omega_n^2 y_p(t) = A \Rightarrow y_p(t) = \frac{A}{\omega_n^2} \quad \text{Eq. 8.53}$$

Combining the particular and homogeneous solutions, assuming the system is underdamped ( $\zeta < 1$ ), and employing the initial conditions results in our final expression for the step response of an underdamped second order system:

$$y(t) = \frac{A}{\omega_n^2} \left\{ 1 - e^{-\zeta\omega_n t} \left[ \cos(\omega_d t) + \frac{\zeta}{\sqrt{1-\zeta^2}} \sin(\omega_d t) \right] \right\} \quad \text{Eq. 8.54}$$

Where  $\omega_d = \omega_n \sqrt{1 - \zeta^2}$  is the damped natural frequency of the system, previously defined in chapter 8.4.

It is common to interpret an underdamped second order system's response in terms of the damping ratio and the natural frequency, rather than direct evaluation of equation (8.54). Figure 8.14 shows a typical step response for an underdamped second order system. The system response overshoots to a maximum value  $y_p$  and has steady-state response  $y_{ss}$ . The maximum overshoot is generally normalized by the steady-state response and is presented in terms of a variable  $M_P$  defined as:

$$M_P = \frac{y_p - y_{ss}}{y_{ss}} \quad \text{Eq. 8.55}$$

$M_P$  is often presented as a percent, obtained by multiplying equation (8.55) by 100. Other parameters of interest in characterizing the step response are the period of any oscillations in the response ( $T$  in Fig. 8.14) and the rise time,  $t_r$ . The rise time is defined as the time required for the system response to go from 10% to 90% of the steady state response. The rise time is often used as an indication of how quickly a second order system responds.

The time domain parameters  $M_p$ ,  $t_r$ , and  $T$  are readily related to the parameters  $\xi$ ,  $\omega_n$ , and  $\omega_d$ . We provide the following relations here, without proof:

$$M_p = e^{\frac{-\pi\xi}{\sqrt{1-\xi^2}}} \quad \text{Eq. 8.56}$$

$$t_r \approx \frac{1.8}{\omega_n} \quad \text{Eq. 8.57}$$

Note: for small damping ratios, ( $\xi < 0.6$ ), equation (8.56) is often approximated as:

$$M_p \approx -\frac{\xi}{0.06} \quad \text{Eq. 8.59}$$

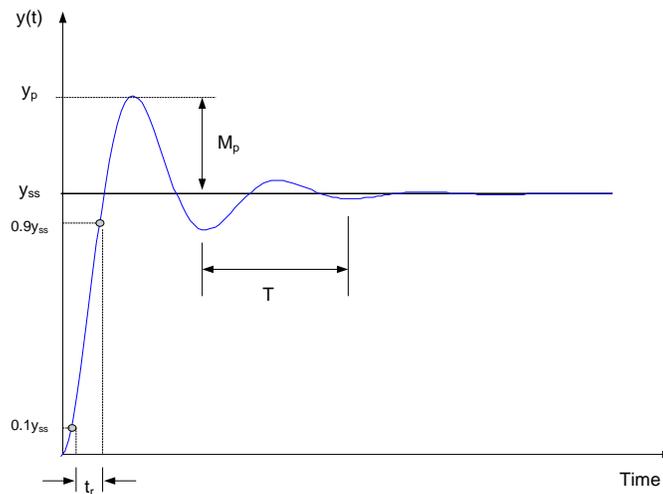


Figure 8.14. Underdamped second order system step response.

## Section Summary

- Typical parameters used to characterize the step response of an underdamped system are the rise time, the maximum overshoot, and the frequency of the oscillations in the response. These parameters are defined as follows:
  - The rise time,  $t_r$ , is the time required for the response to go from 10% to 90% of its steady-state response
  - The maximum overshoot provides the maximum value achieved by the response. The maximum overshoot is a normalized value, defined as:

$$M_p = \frac{y_p - y_{ss}}{y_{ss}}$$

Where  $y_p$  is the peak (or maximum) value of the response and  $y_{ss}$  is the steady state system response.  $M_p$  is often expressed as a percent, by multiplying the above quantity by 100.

- The radian frequency of the oscillations in the step response is given by the damped natural frequency:

$$\omega_d = \omega_n \sqrt{1 - \zeta^2}$$

The period of the oscillations in the response is then given by:

$$T = \frac{2\pi}{\omega_d}$$

- The rise time and maximum overshoot can be related to the damping ratio and natural frequency of the system. The appropriate relations are provided below:
  - $t_r \approx \frac{1.8}{\omega_n}$
  - $M_p = e^{\frac{-\pi\xi}{\sqrt{1-\xi^2}}}$  or  $M_p \approx \frac{\xi}{0.6}$  (for small damping ratios,  $\xi < 0.6$ )

The period of the oscillations can be used to cross-check the above results, since it depends upon both the damping ratio and natural frequency.

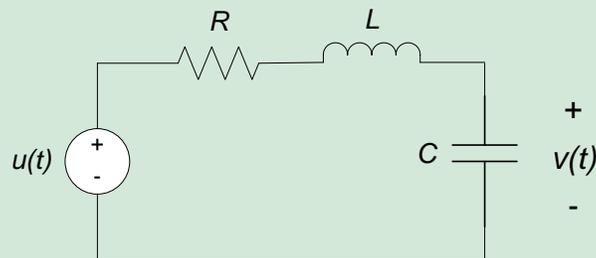
## 8.4 Exercises

1. The differential equation governing the circuit shown below is:

$$\frac{d^2v(t)}{dt^2} + 12\frac{dv(t)}{dt} + 400v(t) = 400u(t)$$

If  $u(t) = 2u_0(t)$ , what are:

- The circuit's natural frequency,
- The circuit's damping ratio,
- The percent overshoot ( $M_p$ ), the rise time ( $t_r$ ), and the steady-state response of the capacitor voltage
- What is the maximum value seen by the capacitor voltage?



## Real Analog Chapter 8: Lab Projects

### 8.5.1: Series RLC Circuit Step Response

This lab will emphasize modeling and testing of a series RLC second order circuit. This lab assignment will consist of two parts:

- In Part I of this assignment, the step response of a given circuit is analyzed and tested. The measured response of the circuit is compared with expectations based on the damping ratio and natural frequency of the circuit.
- Part II of this assignment consists of a simple design problem: the circuit of Part I is to be re-designed to make it critically damped, without changing either the natural frequency or the DC gain. Again, the circuit step response is measured and compared to expectations.

#### Before beginning this lab, you should be able to:

- Model practical inductors as an ideal inductor in series with a resistance (Lab 6.4.2)
- Determine natural frequency and damping ratio for second order circuits
- State the relationship between damping ratio and maximum overshoot for underdamped systems
- State the relationship between natural frequency and rise time for underdamped systems
- Use the Analog Discovery 2 to apply and measure time-varying waveforms

#### After completing this lab, you should be able to:

- Measure the step response of a second order electrical circuit
- Estimate the damping ratio and natural frequency of an underdamped second order circuit from measured step response data

#### This lab exercise requires:

- Analog Discovery 2 module
- Digiilent Analog Parts Kit

#### Symbol Key:

##### DEMO

Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.

##### ANALYSIS

Analysis; include principle results of analysis in laboratory report.

##### SIM

Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.

##### DATA

Record data in your lab notebook.

#### Second Order Series RLC Circuit:

The general differential equation governing a second order system is:

$$\frac{d^2y(t)}{dt^2} + 2\zeta\omega_n \frac{dy(t)}{dt} + \omega_n^2 y(t) = f(t) \quad \text{Eq. 1}$$

Where  $y(t)$  is any system parameter of interest (for example, a voltage or current in an electrical circuit),  $\omega_n$  and  $\zeta$  are the *undamped natural frequency* and the *damping ratio* of the system, respectively, and  $f(t)$  is a forcing function applied to the system. Overdamped systems have a damping ratio greater than one, underdamped systems have a damping ratio less than one, and critically damped systems have a damping ratio of exactly one.

## Pre-lab:

## ANALYSIS

a. Write the differential equation relating  $V_{out}$  and  $V_{in}$  for the system shown in Figure 1. Your solution will be a function of the circuit parameters  $R$ ,  $L$ , and  $C$ .

## ANALYSIS

b. From your differential equation, estimate the damping ratio, natural frequency, damped natural frequency, and DC gain of the circuit. These parameters will be functions of  $R$ ,  $L$ , and  $C$ .

## ANALYSIS

c. Estimate the rise time ( $t_r$ ), overshoot ( $M_p$ ), and frequency of any oscillations you would expect to see in the step response of the circuit. (Also as functions of  $R$ ,  $L$ , and  $C$ .)

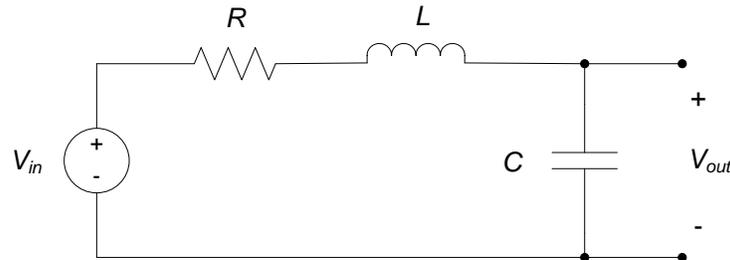


Figure 1. Series RLC circuit.

## Lab Procedures:

## DATA

Construct the circuit shown in Fig. 1, using  $C = 100\text{nF}$ ,  $L = 1\text{mH}$ , and  $R = 1.1\Omega$ . (As always, measure the actual resistance values and the capacitance value if you have a DMM which measures capacitance; you may assume that the nominal inductance value is correct. Also, use your DMM to measure the inductor resistance; the non-ideal inductor effects can have a strong effect on your results in this lab assignment.)

a. Use your arbitrary waveform generator to apply a 2V step input to the circuit, at a low enough frequency so that the circuit can reach steady-state between pulses<sup>33</sup>. Use your oscilloscope channels to measure both the input and output voltages ( $V_{out}$  and  $V_{in}$ ) for the circuit<sup>34</sup>. Record an image of the oscilloscope window, showing the signals  $V_{in}(t)$  and  $V_{out}(t)$ .

## DATA

## ANALYSIS

b. Estimate the rise time, overshoot, damping ratio, natural frequency, DC gain, and the period of any oscillations in the response and compare them to the values calculated in the pre-lab. Comment on any differences in your lab notebook. (Hint: the capacitor and inductor you use are non-ideal; the inductor, in particular, may have a non-negligible resistance. See Chapter 6.4 for a practical inductor model, including an inductor resistance.)

## DEMO

c. Demonstrate operation of your circuit to the TA and have them initial your lab notebook and the lab checklist.

## Critically Damped Series RLC Circuit:

We will now modify the circuit of Part I in order to make the circuit critically damped, without changing the natural frequency or DC gain of the circuit.

<sup>33</sup> Suggestion: A square wave with 1V amplitude and 1V offset can provide an input that goes from 0V to 2V. As long as the period of the square wave period is more than about 10msec, the system should have more than enough time to reach steady-state.

<sup>34</sup> The input voltage  $V_{in}$  provides a good source for a trigger. If you trigger off the rising edge of this signal, your zero time should correspond to the time at which the step input is applied.

**Pre-lab:****ANALYSIS**

For the circuit of Fig. 1, determine values of  $R$ ,  $L$ , and  $C$  necessary to make the circuit critically damped, with the same natural frequency of the circuit in Part I. The circuit should have a DC gain of one, as in Part I of this assignment.

**Lab Procedures:****DATA****DEMO**

- a. Construct the circuit you designed in the pre-lab and experimentally determine the step response of your circuit. Record an image of the oscilloscope window, showing the signals  $V_{in}(t)$  and  $V_{out}(t)$ . Comment on your results vs. your expectations for a critically damped circuit.
- b. Demonstrate operation of your circuit to the TA and have them initial your lab notebook and the lab checklist.

## Real Analog Chapter 8: Lab Worksheets

### 8.5.1 Part 1: Second Order Series RLC Circuit (35 points total)

1. In the space below, provide the differential equation governing the circuit. Attach, to this worksheet, your derivation of the differential equation. (6 pts)
2. Attach, to this worksheet, plots of the input step function you applied to the circuit and the resulting circuit step response. Annotate your plot to indicate the rise time, overshoot, and oscillation frequency. Provide the rise time, overshoot, and oscillation frequency in the space below. (9 pts)
3. Provide below your estimate of the damping ratio, natural frequency, DC gain, and damped natural frequency, as determined from the step response data. (6 pts)
4. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuits' operation. (7 pts)

**TA Initials:** \_\_\_\_\_

5. Discuss your results below. Be sure to include a comparison of appropriate parameters (e.g. damping ratio, natural frequency, damped natural frequency, rise time, steady state response) relative to the values expected as a result of your pre-lab analysis. (7 pts)



## Real Analog Chapter 8: Lab Projects

### 8.5.2: Parallel RLC Circuit Response

This lab will emphasize modeling and testing of a parallel second order circuit containing two resistors, a capacitor, and an inductor. In this assignment, the step response of the given circuit is analyzed and tested. The measured response of the circuit is compared with expectations based on the damping ratio and natural frequency of the circuit.

#### Before beginning this lab, you should be able to:

- Model practical inductors as an ideal inductor in series with a resistance (Lab 6.4.2)
- Determine natural frequency and damping ratio for second order circuits
- State the relationship between damping ratio and maximum overshoot for underdamped systems
- State the relationship between natural frequency and rise time for underdamped systems
- Use the Analog Discovery 2 to apply and measure time-varying waveforms

#### After completing this lab, you should be able to:

- Measure the step response of a second order electrical circuit
- Estimate the damping ratio and natural frequency of an underdamped second order circuit from measured step response data

#### This lab exercise requires:

- Analog Discovery 2 module
- Digilent Analog Parts Kit
- Digital multimeter (optional)

#### Symbol Key:



Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.



Analysis; include principle results of analysis in laboratory report.



Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.



Record data in your lab notebook.

#### General Discussion:

In this lab assignment, we will analyze the circuit of Fig. 1 to estimate the maximum overshoot, rise time, and DC gain of the circuit step response. The circuit will then be tested, these parameters measured, and the measurements compared with expectations from the analysis.

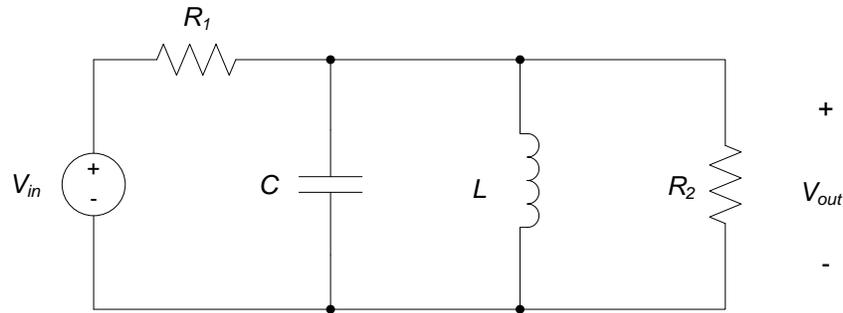


Figure 1. Circuit schematic.

## Pre-lab:

## ANALYSIS

- a. Write the differential equation relating  $V_{out}$  and  $V_{in}$  for the system shown in Fig. 1 in the form:

$$\frac{d^2 V_{out}(t)}{dt^2} + 2\zeta\omega_n \frac{dV_{out}(t)}{dt} + \omega_n^2 V_{out}(t) = K \cdot V_{in}(t)$$

## ANALYSIS

- b. From your differential equation, estimate the damping ratio and natural frequency of the circuit if  $R_1 = 47\Omega$ ,  $R_2 = 1\text{k}\Omega$ ,  $C = 10\mu\text{F}$ , and  $L = 1\text{mH}$ .

## ANALYSIS

- c. Estimate the rise time ( $t_r$ ), overshoot ( $M_p$ ), and frequency of any oscillations you would expect to see in the step response of the circuit.

## Lab Procedures:

## DATA

Construct the circuit shown in Fig. 1 using the component values provided in part (b) of the pre-lab. Record measured values of  $R_1$ ,  $R_2$ , and  $C$  (if your DMM has the ability to measure capacitance). Also measure the resistance of the inductor; you may assume that the nominal inductance of the inductor value is correct.

## DATA

- a. Use your arbitrary waveform generator to apply a 2V step input to the circuit, at a low enough frequency so that the circuit can reach steady-state between pulses. Use your oscilloscope to measure both the input and output voltages ( $V_{out}$  and  $V_{in}$ ) for the circuit.

## DEMO

- b. Demonstrate operation of your circuit to the TA and have them initial your lab notebook and the lab checklist.

## DATA

- c. Record an image of the oscilloscope window, showing the signals  $V_{in}(t)$  and  $V_{out}(t)$ . Save the data to a file for later plotting.

## ANALYSIS

- d. Estimate the damping ratio and natural frequency of the circuit from your data. Express the difference between the measured values and those expected from your pre-lab analysis as a percentage of the expected value. Comment on the measured response relative to your expectations from the pre-lab analysis.

## ANALYSIS

- e. Re-do the calculations of part (c) of the pre-lab, including the measured inductor resistance. Compare the results of this analysis to the measured damping ratio and natural frequency of the circuit. Express the differences as a percentage of the expected values. Comment on the differences between the measured response and the results of this analysis.

## Real Analog Chapter 8: Lab Worksheets

### 8.5.2: Parallel RLC Circuit Response (40 points total)

1. Provide below the differential equation governing the circuit. Attach your derivation of this differential equation to this worksheet. (8 pts)
2. Attach, to this worksheet, plots of the input step function you applied to the circuit and the resulting circuit step response. Annotate your plot to indicate the rise time, overshoot, and oscillation frequency. Provide the rise time, overshoot, and oscillation frequency in the space below. (7 pts)
3. Provide below your estimate of the damping ratio, DC gain, and natural frequency, as determined from the step response data. (6 pts)
4. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuits' operation. (9 pts)

**TA Initials:** \_\_\_\_\_

5. Compare your measured vs. expected parameters (e.g. damping ratio, natural frequency, damped natural frequency, rise time, steady state response). Where appropriate, express differences in terms of a percent of the expected value. Provide at least one reason why measured values might disagree with expectations based on your pre-lab analysis. (10 pts)

## Real Analog Chapter 8: Lab Projects

### 8.5.3: RLC Circuit Response

This lab will emphasize modeling and testing of a second order circuit containing two resistors, a capacitor, and an inductor. In this assignment, the step response of the given circuit is analyzed and tested. The measured response of the circuit is compared with expectations based on the damping ratio and natural frequency of the circuit.

#### Before beginning this lab, you should be able to:

- Model practical inductors as an ideal inductor in series with a resistance (Lab 6.4.2)
- Determine natural frequency and damping ratio for second order circuits
- State the relationship between damping ratio and maximum overshoot for underdamped systems
- State the relationship between natural frequency and rise time for underdamped systems
- Use the Analog Discovery 2 to apply and measure time-varying waveforms

#### After completing this lab, you should be able to:

- Measure the step response of a second order electrical circuit
- Estimate the damping ratio and natural frequency of an underdamped second order circuit from measured step response data

#### This lab exercise requires:

- Analog Discovery 2 module
- Digilent Analog Parts Kit
- Digital multimeter (optional)

#### Symbol Key:

-  Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
-  Analysis; include principle results of analysis in laboratory report.
-  Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
-  Record data in your lab notebook.

#### General Discussion:

In this lab assignment, we will analyze the circuit of Fig. 1 to estimate the maximum overshoot, rise time, and DC gain of the circuit step response. The circuit will then be tested, these parameters measured, and the measurements compared with expectations from the analysis.

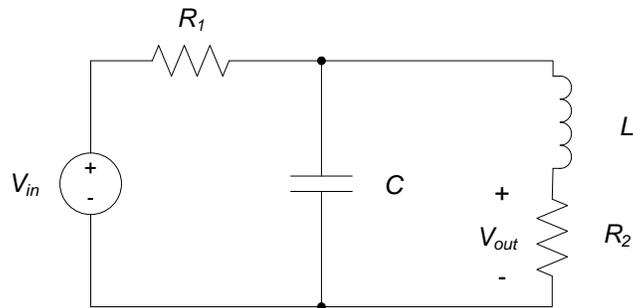


Figure 1. Circuit schematic.

## Pre-lab:

## ANALYSIS

- a. Write the differential equation relating  $V_{out}$  and  $V_{in}$  for the system shown in Figure 1 in the form:

$$\frac{d^2V_{out}(t)}{dt^2} + 2\zeta\omega_n \frac{dV_{out}(t)}{dt} + \omega_n^2 V_{out}(t) = K \cdot V_{in}(t)$$

## ANALYSIS

- b. From your differential equation, estimate the damping ratio and natural frequency of the circuit if  $R_1 = 47\Omega$ ,  $R_2 = 1.1\Omega$ ,  $C = 10\mu\text{F}$ , and  $L = 1\text{mH}$ .

## ANALYSIS

- c. Estimate the rise time ( $t_r$ ), overshoot ( $M_P$ ), and frequency of any oscillations you would expect to see in the step response of the circuit.

## Lab Procedures:

## DATA

Construct the circuit shown in Fig. 1 using the component values provided in part (b) of the pre-lab. Record measured values of  $R_1$ ,  $R_2$ , and  $C$  (if your DMM has the ability to measure capacitance). Also measure the resistance of the inductor; you may assume that the nominal inductance of the inductor value is correct.

## DATA

- a. Use your arbitrary waveform generator to apply a 2V step input to the circuit, at a low enough frequency so that the circuit can reach steady-state between pulses. Use your oscilloscope to measure both the input and output voltages ( $V_{out}$  and  $V_{in}$ ) for the circuit.

## DEMO

- b. Demonstrate operation of your circuit to the TA and have them initial your lab notebook and the lab checklist.

## DATA

- c. Record an image of the oscilloscope window, showing the signals  $V_{in}(t)$  and  $V_{out}(t)$ . Save the data to a file for later plotting.

## ANALYSIS

- d. Estimate the damping ratio and natural frequency of the circuit from your data. Express the difference between the measured values and those expected from your pre-lab analysis as a percentage of the expected value. Comment on the measured response relative to your expectations from the pre-lab analysis.

## ANALYSIS

- e. Re-do the calculations of part (c) of the pre-lab, including the measured inductor resistance. Compare the results of this analysis to the measured damping ratio and natural frequency of the circuit. Express the differences as a percentage of the expected values. Comment on the differences between the measured response and the results of this analysis.

## Real Analog Chapter 8: Lab Worksheets

### 8.5.3: RLC Circuit Response (40 points total)

1. Provide below the differential equation governing the circuit. Attach your derivation of this differential equation to this worksheet. (8 pts)
2. Attach, to this worksheet, plots of the input step function you applied to the circuit and the resulting circuit step response. Annotate your plot to indicate the rise time, overshoot, and oscillation frequency. Provide the rise time, overshoot, and oscillation frequency in the space below. (7 pts)
3. Provide below your estimate of the damping ratio, DC gain, and natural frequency, as determined from the step response data. (6 pts)
4. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuits' operation. (9 pts)

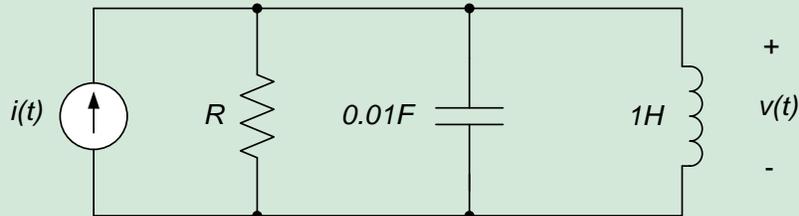
**TA Initials:** \_\_\_\_\_

5. Compare your measured vs. expected parameters (e.g. damping ratio, natural frequency, damped natural frequency, rise time, steady state response). Where appropriate, express differences in terms of a percent of the expected value. Provide at least one reason why measured values might disagree with expectations based on your pre-lab analysis. (10 pts)

## Real Analog Chapter 8: Homework

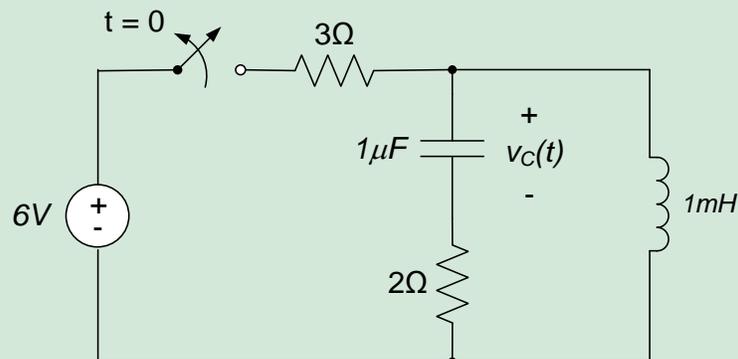
8.1 For the circuit below,

- Determine the second order differential equation governing  $v(t)$ .
- Find  $R$  so that the circuit is critically damped.



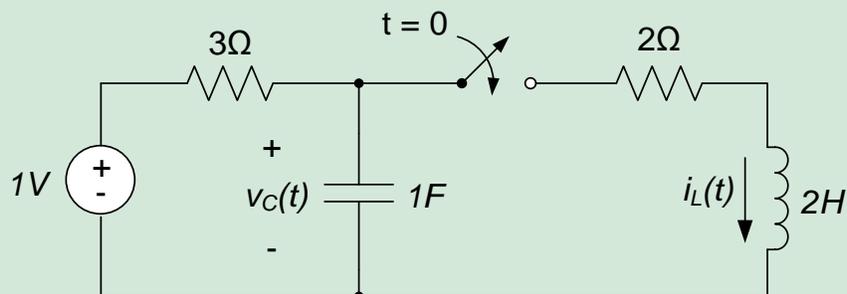
8.2 For the circuit below,

- Determine the differential equation that governs  $v_C(t)$ ,  $t > 0$ .
- Is the system under, over, or critically damped?



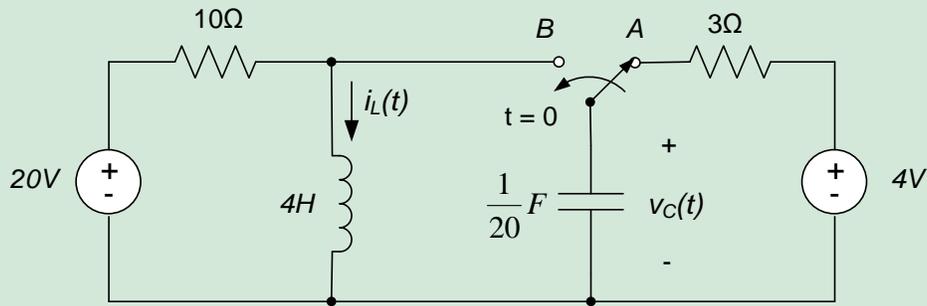
8.3 For the circuit below,

- Determine the differential equation for  $i_L(t)$ ,  $t > 0$ .
- Determine the initial ( $t=0+$ ) and final ( $t \rightarrow \infty$ ) conditions on  $v_C(t)$  and  $i_L(t)$ .

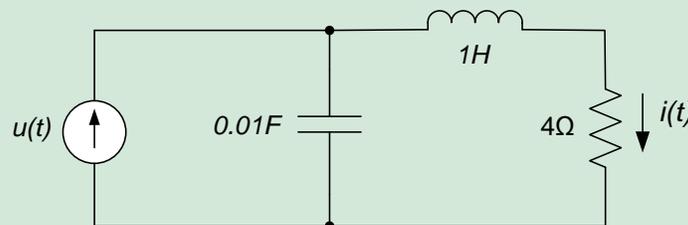


8.4 The switch moves from position A to position B at  $t=0$  seconds.

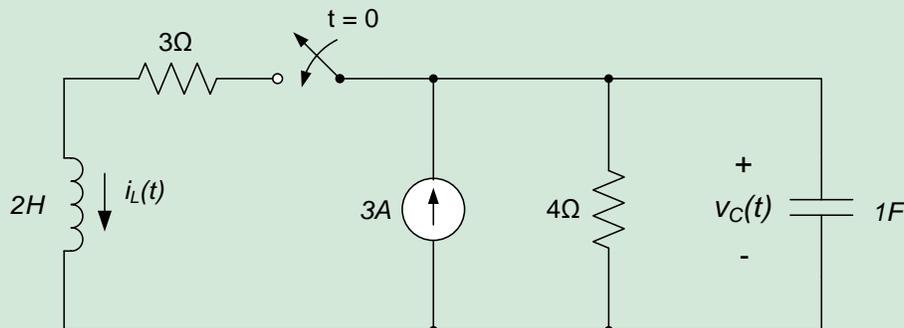
- Determine the differential equation that governs  $i_L(t)$ ,  $t > 0$ .
- Determine initial ( $t=0+$ ) and final ( $t \rightarrow \infty$ ) conditions on  $v_C(t)$  and  $i_L(t)$ .



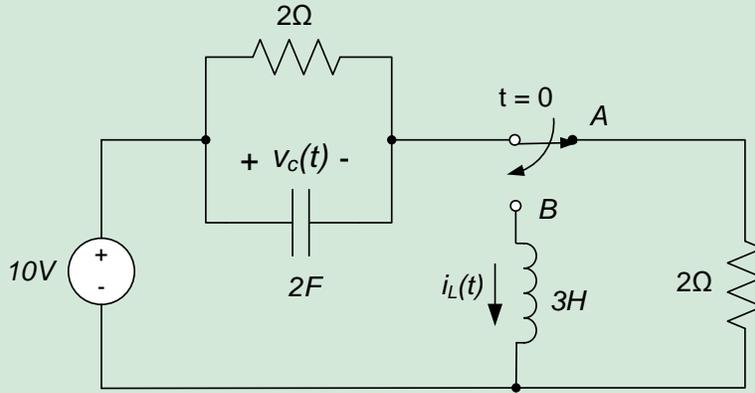
- 8.5 For the circuit below,
- Determine the differential equation governing  $i(t)$ .
  - Determine  $\zeta$  and  $\omega_n$
  - Find the maximum value of current through the resistor in response to a step input  $u(t) = 3u_0(t)$  A.



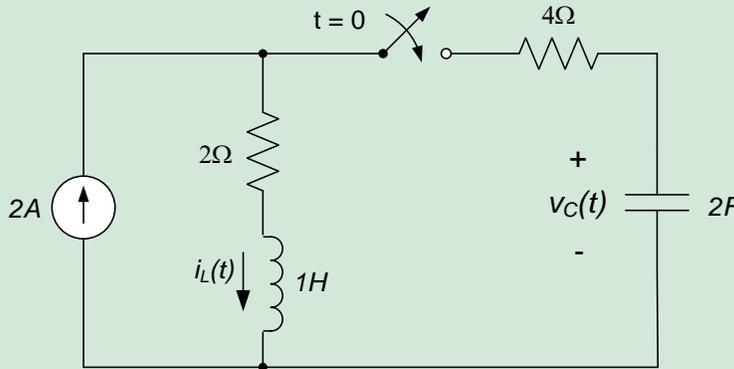
- 8.6 A current in a second order circuit is described by the differential equation
- $$\frac{d^2 i(t)}{dt^2} + 6 \frac{di(t)}{dt} + 100i(t) = 50u_0(t)$$
- Determine the damping ratio, undamped natural frequency, and damped natural frequency for the circuit.
  - Sketch the response of the circuit to a unit step input. Include numerical values for  $t_r$ , the maximum value of  $i(t)$ , and the steady-state value of  $i(t)$ .
- 8.7 For the circuit below, determine
- The differential equation for  $i_L(t)$ ,  $t > 0$ .
  - The initial ( $t=0^+$ ) and final ( $t \rightarrow \infty$ ) conditions on  $v_C(t)$  and  $i_L(t)$ .



- 8.8 The switch moves from position A to position B at  $t=0$  seconds.
- Determine the differential equation that governs  $v_C(t)$ ,  $t > 0$ .
  - Determine initial ( $t=0^+$ ) and final ( $t \rightarrow \infty$ ) conditions on  $v_C(t)$  and  $i_L(t)$ .



- 8.9 The switch moves from position A to position B at  $t=0$  seconds.
- Determine the differential equation that governs  $v_C(t)$ ,  $t>0$ .
  - Determine initial ( $t=0+$ ) and final ( $t \rightarrow \infty$ ) conditions on  $v_C(t)$  and  $i_L(t)$ .



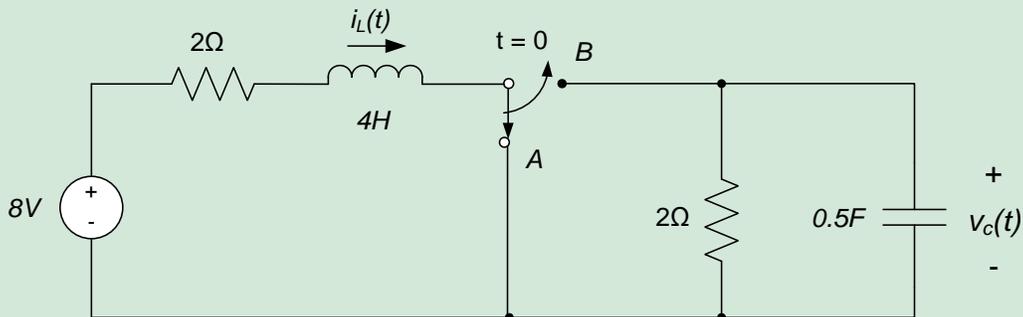
- 8.10 The differential equation governing a voltage  $v_{out}(t)$  in a circuit is:

$$2 \frac{d^2 v_{out}(t)}{dt^2} + 4 \frac{dv_{out}(t)}{dt} + 50 v_{out}(t) = 100 v_{in}(t)$$

Determine the maximum value of  $v_{out}(t)$  resulting from a step voltage input

$$v_{in}(t) = \begin{cases} 0, & t < 0 \\ 2V, & t > 0 \end{cases}$$

- 8.11 For the circuit below, determine
- The differential equation for  $i_L(t)$ ,  $t>0$
  - The initial ( $t=0+$ ) and final ( $t \rightarrow \infty$ ) conditions on  $v_C(t)$  and  $i_L(t)$ .



## Chapter 9: Introduction to State Variable Models

### 9 Introduction and Chapter Objectives

In our analysis approach of dynamic systems so far, we have defined variables which describe the energy in the circuit energy storage elements (voltages across capacitors and currents through inductors). We then used KVL and KCL to write differential equations describing the circuit, in terms of those variables. The resulting equations were then combined into a single differential equation governing the parameter in which we were interested (the *input-output equation* for the system); this equation was solved to determine the response of the circuit. This approach is useful for low-order systems, such as the first or second order systems we have examined so far, but it becomes cumbersome for higher-order systems. An alternate modeling approach, *state variable* (or *state space*) modeling, has a number of advantages over the approach we have been using to date, especially for higher-order systems.

In this chapter, we will provide a very brief introduction to the topic of state variable modeling<sup>35</sup>. The brief presentation provided here is intended simply to introduce the reader to the basic concepts of state variable models, since they are a natural – and relatively painless – extension of the analysis approach we have used in Chapters 7 and 8. Introduction to state variable models at this stage also allows the reader to perform *numerical simulations* of system responses. Numerical simulations are computer-generated solutions to the differential equations governing the system. Most numerical approaches to the solution of differential equations require the equations to be in state variable form<sup>36</sup>.

State variable models of dynamic systems consist of several first order differential equations, in several different variables. (These variables are called the *state variables*.) The state variables for a system must completely describe the energy contained in all the energy storage elements in the system, so natural choices of state variables for electrical circuits are the voltages across capacitors and the currents through inductors. If there are  $N$  state variables required to describe the circuit, the state variable model is created by applying KVL and KCL to obtain  $N$  first-order differential equations in these  $N$  variables. Please notice that this approach is exactly the approach we have used in Chapter 8 to determine the differential equation governing a second order circuit – we are simply eliminating the step in which the individual equations are combined into a single, higher-order differential equation in a single unknown. Since the mathematics associated with combining the individual equations can be tedious, state variable models are actually easier to create than input-output equations!

A brief description of state variable models is provided in section 9.1 of this chapter. An example of the state variable model for a third order system is provided to illustrate development of the model. Section 9.2 provides information relative to numerical simulation of a state variable model using MATLAB, and section 9.3 provides Octave syntax to perform the same processes<sup>37</sup>.

If desired, this chapter can be skipped without loss of continuity.

#### After completing this chapter, you should be able to:

- Define state variables for electrical circuits
- Write differential equations governing electrical circuits in state variable form
- Use MATLAB and/or Octave to simulate the impulse response of an electrical circuit

---

<sup>35</sup> We will provide a fairly in-depth presentation of state variable models later in this text (in Chapter 14), when we explicitly address modeling of higher-order systems.

<sup>36</sup> The differential equation solvers in MATLAB and Octave, for example, require the equations to be in state variable form. Circuit simulation software packages such as SPICE create the equations governing the circuit in state variable form before solving them

<sup>37</sup> Information relative to acquiring MATLAB and Octave are provided in the relevant sections.

- Use MATLAB and/or Octave to simulate the step response of an electrical circuit
- Use MATLAB and/or Octave to plot the state trajectory of an electrical circuit

## 9.1 Introduction to State Variable Models

### 9.1.1 Background and Introduction

As their name implies, state variable models are based on the concept of a system's state. The state of a system is the minimum amount of information necessary to completely characterize the system at some instant in time. More specifically, if we know the state at any time, and the input to the system for all subsequent times, we can determine the output of the system at any subsequent time<sup>38</sup>. It turns out that the system's state uniquely determines the energy in all the system's energy storage elements and vice-versa. If the energy in any of the energy storage elements changes, the system's state changes.

The *state variables* are the smallest set of variables which completely describe the state (or the energy storage) of the system. The choice of state variables is not unique, but one possible choice of state variable is those variables which describe the energy stored in all of the independent energy storage elements in the system. For example, in electrical circuits, inductors store energy as current and capacitors store energy as voltage. If we choose as state variables the currents in inductors and the voltages across the capacitors, we will have created a legitimate set of state variables for the circuit.

Since the state variables are independent, they can be visualized as a set of orthogonal axes defining a space. The space defined by the state variables is called *state space* of the system. If the system is described by  $N$  state variables, the state space will be  $N$ -dimensional. The state of the system at any given time can be visualized as a point in the state space.

In general, as the system responds to some input, the system's state will change over time. Since the state of the system is a point in state space, the change in the system's state can be visualized as tracing a path over time in the state space. This path is called the *state trajectory*.

### 9.1.2 Form of State Variable Models

State variable models, as mentioned previously, represent an  $N^{\text{th}}$  order system as  $N$  first order differential equations in  $N$  unknowns. (The unknowns are the state variables.) For linear, lumped-parameter, time invariant systems, these equations will take the form:

$$\begin{aligned}
 \dot{x}_1(t) &= a_{11}x_1(t) + a_{12}x_2(t) + \cdots + a_{1N}x_N(t) + b_1u(t) \\
 \dot{x}_2(t) &= a_{21}x_1(t) + a_{22}x_2(t) + \cdots + a_{2N}x_N(t) + b_2u(t) \\
 &\vdots \\
 \dot{x}_n &= a_{N1}x_1(t) + a_{N2}x_2(t) + \cdots + a_{NN}x_N(t) + b_Nu(t)
 \end{aligned}
 \tag{Eq. 9.1}$$

Where  $x_1(t), x_2(t), \dots, x_N(t)$  are the system and  $u(t)$  is the input to the system. The overdot notation denotes differentiation with respect to time;  $\dot{x}_k(t) = \frac{dx_k(t)}{dt}$ . (We will assume that no derivatives of the input are applied to the system – this is a special case which we will avoid in this introductory chapter.)

<sup>38</sup> Thus, the initial conditions of a system constitute the state of the system.

It turns out that, if all system states are known, we can determine any other parameter in the system. In fact, any other parameter in the system can be written as a linear combination of the states and the input. Thus, we can write the system output as:

$$y(t) = c_1x_1(t) + c_2x_2(t) \cdots + c_Nx_N(t) + du(t) \quad \text{Eq. 9.2}$$

Equations (9.1) and (9.2) are commonly written in matrix form as:

$$\underline{\dot{x}}(t) = A\underline{x}(t) + \underline{b}u(t) \quad \text{Eq. 9.3}$$

$$y(t) = \underline{c}\underline{x}(t) + du(t) \quad \text{Eq. 9.4}$$

In equation (9.3), the vector  $\underline{x}(t)$  is an  $N \times 1$  column vector containing the system state variables. The matrix  $A$  is a square  $N \times N$  matrix, and the vector  $\underline{b}$  is an  $N \times 1$  column vector. The vector  $\underline{\dot{x}}(t)$  is an  $N \times 1$  column vector containing the derivatives of the state variables as a function of time. In equation (9.4), the vector  $\underline{c}$  is a  $1 \times N$  row vector, and  $d$  is a scalar. Equation (9.3) provides the *state equations* for the system, and equation (9.4) is called the *output equation* of the system.

It is possible to define more than one output in a system. To define multiple outputs, we simply create a vector of outputs, each row of which is an equation of the form of equation (9.2). For example, if we define  $P$  outputs, the set of output equations becomes:

$$\begin{aligned} y_1 &= c_{11}x_1 + c_{12}x_2 + \cdots + c_{1N}x_N + d_1u \\ y_2 &= c_{21}x_1 + c_{22}x_2 + \cdots + c_{2N}x_N + d_2u \\ &\vdots \\ y_P &= c_{P1}x_1 + c_{P2}x_2 + \cdots + c_{PN}x_N + d_Pu \end{aligned} \quad \text{Eq. 9.5}$$

In the case of multiple outputs, we modify our matrix expression of equation (9.4) to:

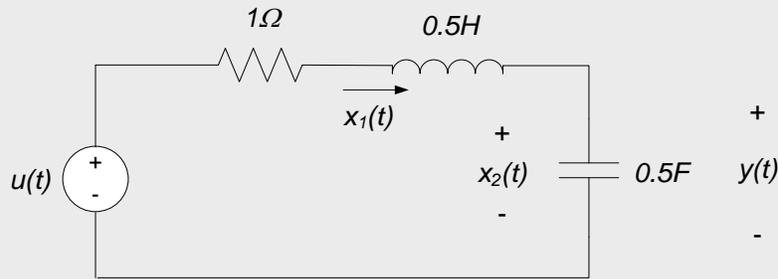
$$\underline{y}(t) = C\underline{x}(t) + \underline{d}u(t) \quad \text{Eq. 9.6}$$

Where  $\underline{y}(t)$  is a  $P \times 1$  column vector containing the outputs,  $C$  is a  $P \times N$  matrix, and  $\underline{d}$  is a  $P \times 1$  column vector.

Creation of the state variable model for an electrical circuit is probably best described by example. Examples of creation of state variable models for both second and third order circuits are provided in the examples below. Please notice that the creation of a third order state variable model is not significantly more difficult than creation of a second order state variable model. Creation of a third order input-output equation is, however, generally considerably more difficult than creation of a second order input-output model. (Try, for example, creating an input-output relation for the circuit of example 9.2 below.)

### Example 9.1: State Variable Model of Series RLC Circuit

A series RLC circuit is shown below. Appropriate state variables are the current through the inductor and the voltage across the capacitor, as shown.



Applying KVL around the circuit loop, we obtain:

$$u = x_1 + 0.5\dot{x}_1 + x_2$$

Applying KCL at the node between the inductor and capacitor results in:

$$x_1 = 0.5\dot{x}_2$$

Rearranging the above equations and placing them in matrix form results in:

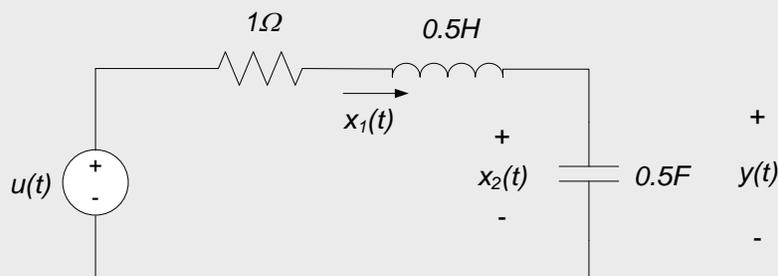
$$\begin{bmatrix} \dot{x}_1 \\ \dot{x}_2 \end{bmatrix} = \begin{bmatrix} -2 & -2 \\ 2 & 0 \end{bmatrix} \begin{bmatrix} x_1 \\ x_2 \end{bmatrix} + \begin{bmatrix} 2 \\ 0 \end{bmatrix} u(t)$$

Since the output  $y = x_2$ , the output equation is:

$$y = \begin{bmatrix} 0 & 1 \end{bmatrix} \begin{bmatrix} x_1 \\ x_2 \end{bmatrix} + 0 \cdot u$$

### Example 9.2: State Variable Model of Third Order Circuit

Consider the circuit shown below. The input to the system is the voltage  $u(t)$ , and the output variable is the voltage across the resistor,  $y(t)$ . There are three energy storage elements in the system (two inductors and a capacitor) so we will expect the system to be third order with three state variables. These state variables are chosen to be the currents through the inductors and the voltage difference across the capacitor; these are indicated on the figure below.



We write the state equations by applying KVL and KCL to the circuit. Since there are three state variables, three state equations must be written. Applying KVL around the leftmost loop results in:

$$u(t) = L_1\dot{x}_1(t) + x_3(t)$$

Applying KVL around the rightmost loop results in:

$$x_3(t) = L_2\dot{x}_2(t) + Rx_2(t)$$

Note that in the equation above, the voltage across the resistor is written as  $Rx_2$ , rather than  $y$ . This is consistent with the general state equation format which requires that the derivative of each state variable be written only in terms of the other state variables and the input. Our final state equation is obtained by applying KCL at the node interconnecting the two inductors and the capacitor:

$$x_1(t) = x_2(t) + C\dot{x}_3(t)$$

The above can be re-written in matrix form as:

$$\begin{bmatrix} \dot{x}_1(t) \\ \dot{x}_2(t) \\ \dot{x}_3(t) \end{bmatrix} = \begin{bmatrix} 0 & 0 & \frac{-1}{L_1} \\ 0 & -R & \frac{1}{L_2} \\ \frac{1}{C} & \frac{-1}{C} & 0 \end{bmatrix} \begin{bmatrix} x_1(t) \\ x_2(t) \\ x_3(t) \end{bmatrix} + \begin{bmatrix} \frac{1}{L_1} \\ 0 \\ 0 \end{bmatrix} u(t)$$

The above state equations allow us to determine any parameter of interest in the circuit. Our output variable is the voltage across the resistor,  $R$ . We can use Ohm's law to write the equation describing the desired output in terms of the state variable  $x_2$  to obtain:

$$y(t) = [0 \quad R \quad 0] \begin{bmatrix} x_1(t) \\ x_2(t) \\ x_3(t) \end{bmatrix} + 0 \cdot u(t)$$

## Section Summary

- The system's *state* completely describes the system. If we know the state of the system at some time  $t_0$ , and the input to the system for all times  $t \geq t_0$ , we can determine the output of the system for all times  $t \geq t_0$ . The state of the system must uniquely describe the energy stored in all energy storage elements in the system.
- The *state variables* are a set of system variables which describe the system state. A system's state variables are not unique – there are a variety of variables which can describe the energy in a system. However, the number of state variables must correspond to the number of independent energy storage elements in the system. Since inductors store energy in terms of current and capacitors store energy in terms of voltage, one possible choice of state variables is the voltages across capacitors and the currents through inductors.
- The *state equations* for the system are a set of  $N$  first order differential equations, in  $N$  state variables. If the system is linear and time invariant, the state equations can be written in matrix form as:

$$\dot{\underline{x}}(t) = \underline{A}\underline{x}(t) + \underline{b}u(t)$$

- The state equations are typically obtained by application of Kirchhoff's laws to the circuit.
- The system output at any time can be determined from the states and the input at that time. The *output equation* for a system is a linear combination of the states and the input which provide the desired output. In the case of a linear, time invariant system with a single output, the output equation can be written in matrix form as:

$$y(t) = \underline{c}\underline{x}(t) + du(t)$$

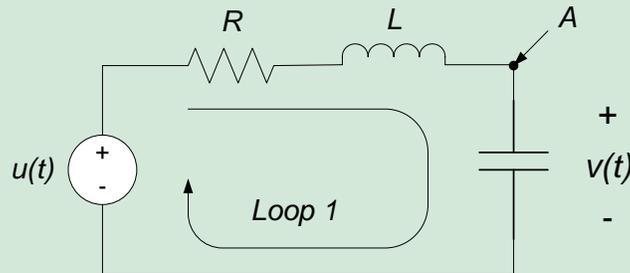
- If multiple outputs are desired, the above matrix form of the output equation can be generalized as:

$$\underline{y}(t) = \underline{C}\underline{x}(t) + \underline{d}u(t)$$

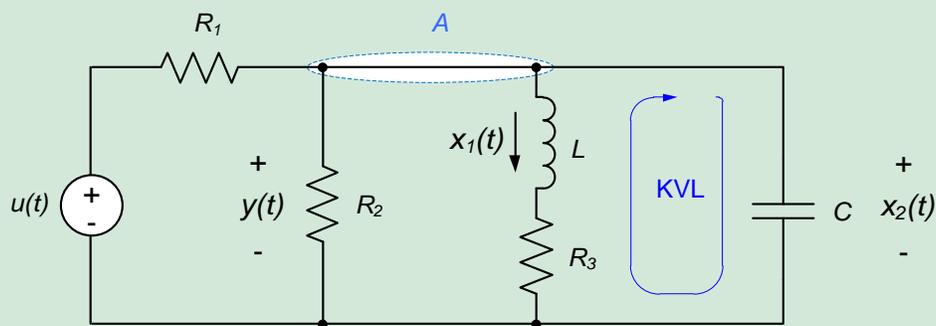
Where  $\underline{y}(t)$  is a column vector of the outputs,  $C$  is a matrix, and  $\underline{d}$  is a column vector.

## 9.1 Exercises

- For the circuit shown below, apply KCL at node A and KVL around loop 1 to write two first order differential equations in two unknowns: the current through the inductor and the voltage across the capacitor. Place the two equations in state variable format.



- Write a state variable model  $\{A,b,c,d\}$  for the circuit of problem 1 if the output is the voltage across the capacitor.
- Write a state variable model  $\{A,b,c,d\}$  for the circuit of problem 1 if the output is the voltage across the inductor.
- Write a state variable model  $\{A,b,c,d\}$  for the circuit of problem 1 if the output is the voltage across the resistor.
- Write a state variable model for the circuit below.  $u(t)$  is the input voltage, and  $y(t)$  is the output. (Hint: you might want to try applying KCL at node A and KVL around the indicated loop.)



## 9.2 Numerical Simulation of System Responses Using MATLAB

Analytical solutions of the state variable or input-output models is difficult or impossible for higher-order or nonlinear systems. Thus, numerical (or computer-based) solutions of these differential equations have become increasingly popular. This chapter provides a brief outline describing the use of some special-purpose MATLAB commands for simulating the response of linear, time invariant systems (these are systems which are governed by linear differential equations with constant coefficients).

Some MATLAB functions presented in this chapter are available in MATLAB's Control Systems Toolbox. The Control Systems Toolbox comes with the Student Edition of MATLAB. For more information about MATLAB products, see the MathWorks web site at <http://www.mathworks.com/>.

This section assumes some knowledge of basic MATLAB syntax. For those who are not familiar with MATLAB, a brief overview of the necessary topics is provided in Appendix A.1 of this textbook.

## 9.2.1 Basic Commands for Simulation of Linear, Time-invariant Systems

MATLAB's Control Systems Toolbox contains a number of special-purpose commands for simulating the response of linear, time-invariant systems. Among these commands are commands specific to determining the step response and natural response of systems; we will restrict our attention to these commands in this chapter. Later courses in your engineering curriculum will most likely present more general-purpose MATLAB commands.

To calculate system responses (e.g. to solve the differential equations of interest), we will use only MATLAB's **step** and **initial** commands in this chapter. The **step** command calculates a unit step response for the system, while the **initial** command calculates the natural response of a system to some set of initial conditions. We will also use the **ss** command to create state space model objects, to send to the **step** and **initial** commands. Basic syntax for these commands is provided below.

**step:** The command **[y,x,t]=step(sys)** returns the step response of the state variable model described by model object **sys**. The vector **y** contains the system output, the matrix **x** contains the states, and the vector **t** contains the time samples.

- The command **[y,x]=step(sys)** returns the step response as above, but calculated over the specified time vector **t**.
- The command **[y,t]=step(sys)** returns the system output as above, and the times at which the response is calculated, but not the state variables **x**.
- **step(sys)** with no left-hand arguments results in a plot of the step response output.

**initial:** Response of linear system to an initial condition. **[y,x,t]=initial(sys,x0)** returns the response of the system described by the model object **sys** to an initial condition contained in the vector **x0**. Variations on this command are similar to those provided above for the **step** command.

**ss:** Create a state space model object. **sys = ss(A, b, c, d)** returns an object named **sys** which provides a state space model corresponding to the matrices provided in **A, b, c,** and **d**.

### Example 9.3: Step Response of Series RLC Circuit

Determine and plot the response of the system of example 9.1 if:

$$u(t) = \begin{cases} 0V, & t < 0 \\ 2V, & t \geq 0 \end{cases}$$

And the circuit is initially relaxed (i.e. all voltages and currents in the system are initially zero). Also plot the state trajectory for this input. The state equations for the circuit of example 9.1 were previously determined to be:

$$\begin{bmatrix} \dot{x}_1 \\ \dot{x}_2 \end{bmatrix} = \begin{bmatrix} -2 & -2 \\ 2 & 0 \end{bmatrix} \begin{bmatrix} x_1 \\ x_2 \end{bmatrix} + \begin{bmatrix} 2 \\ 0 \end{bmatrix} u(t)$$

While the output equation is:

$$y = [0 \quad 1] \begin{bmatrix} x_1 \\ x_2 \end{bmatrix} + 0 \cdot u$$

Thus, the matrices describing the state space model are:

$$A = \begin{bmatrix} -2 & -2 \\ 2 & 0 \end{bmatrix}, \underline{b} = \begin{bmatrix} 2 \\ 0 \end{bmatrix}, \underline{c} = [0 \quad 1], d = 0$$

To simulate the response of the system we first need to input the state variable model. We begin by defining the  $A$ ,  $b$ ,  $c$ , and  $d$  matrices as follows:

```
>> A = [-2 -2; 2 0];
>> b = [2; 0];
>> c = [0 1];
>> d = 0;
```

The >> symbols denote the command prompt at MATLAB's command window; they are included here to emphasize MATLAB commands.

It is generally desirable to create a model object in MATLAB to represent the system model<sup>39</sup>. To create a state space system model, the command is `ss`. Arguments to the command are the above matrices; the output is the system model object. To create a model of our system, we type:

```
>> sys = ss(A, b, c, d);
```

Our workspace now contains a state space model object of our system named "sys".

We can simulate the response of this system to the desired input by using MATLAB's `step` command. The step command assumes that the system is initially relaxed and the input to the system is:

$$u(t) = \begin{cases} 0, & t < 0 \\ 1, & t \geq 0 \end{cases}$$

The input to our system is exactly twice this input, so we can simply scale our output by a factor of two. (This works because the system is linear – don't try this with a nonlinear system!) The appropriate commands are (note that we have to scale both the output and the states, since we will be plotting the output response and the state trajectory):

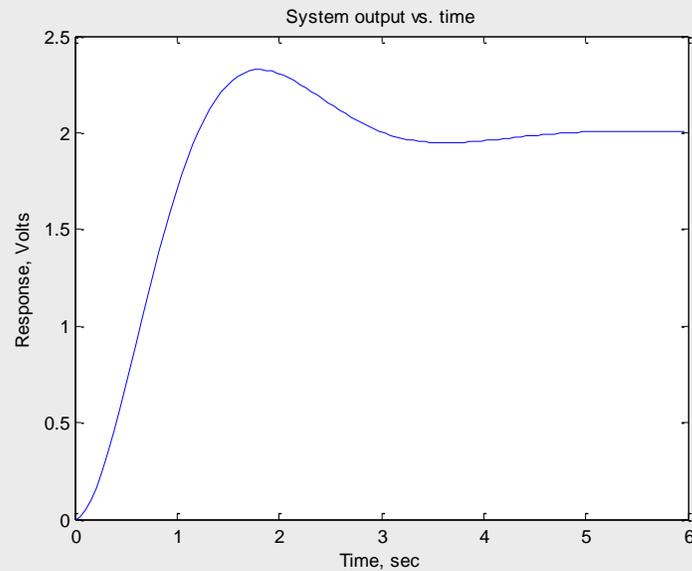
```
>> [y, x, t] = step(sys);
>> y = 2 * y;
>> x = 2 * x;
```

The final step is to plot the responses. Plotting the output response can be accomplished with the following commands:

```
>> figure
>> plot(t, y)
>> title('System output vs, time')
>> xlabel('Time, sec')
>> ylabel('Response, Volts')
```

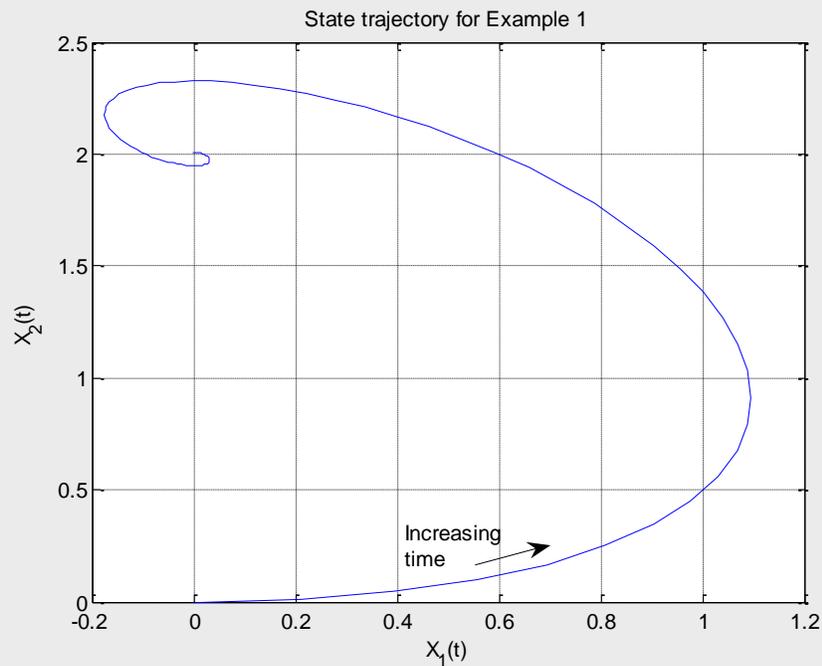
Which results in the figure below:

<sup>39</sup> This is not entirely necessary, since the step and initial commands will accept the A, b, c, and d matrices directly as arguments. It is, however, encouraged.



We can plot the state trajectory by plotting the second state vector,  $x_1(t)$ , vs. the first state vector,  $x_2(t)$ . MATLAB returns the first state vector as the first column of the  $x$  matrix, the second state vector as the second column of the  $x$  matrix, and so on. Thus, we can plot the state trajectory with the following commands:

```
>> plot(x(:,1),x(:,2))
>> grid
>> xlabel('X1(t)')
>> ylabel('X2(t)')
>> title('State trajectory for Example 1')
```



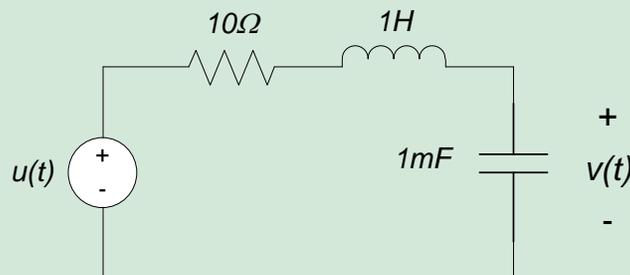
### Section Summary:

- If you have access to MATLAB's control system toolbox,

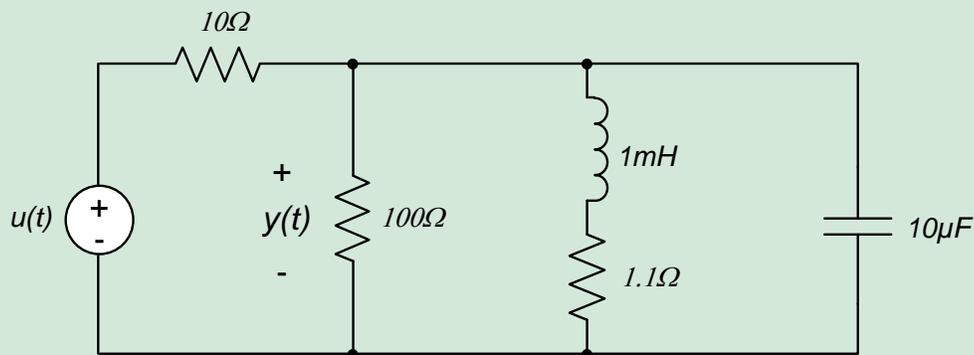
- A state space model object can be created with the `ss` function. Inputs to the function are the A, b, c, and d matrices of the state space model.
- The `step` command can be used to calculate the step response of the system.
- The `initial` command can be used to calculate the natural response of the system.

## 9.2 Exercises

1. Simulate the response  $y(t)$  of the circuit below, if  $u(t)$  is a unit step function.



2. Simulate the response  $y(t)$  of the circuit below, if  $u(t)$  is a unit step function.



## 9.3 Numerical Simulation of System Responses Using Octave

This chapter provides a brief outline describing the use of some special-purpose Octave commands for simulating the response of linear, time invariant systems (these are systems which are governed by linear differential equations with constant coefficients).

Octave is an open-source software with many of the same capabilities as MATLAB. Unlike MATLAB, however, Octave is available for free. For more information about Octave, see the website at [www.gnu.org](http://www.gnu.org). Octave syntax is very similar to MATLAB syntax, so the commands in this section are very similar to those provided in section 9.2.

This section assumes some knowledge of basic Octave syntax. For those who are not familiar with Octave, a brief overview of the necessary topics is provided in Appendix A.2 of this textbook.

### 9.3.1 Basic Commands for Simulation of Linear, Time-invariant Systems

Like MATLAB, Octave provides a number of special-purpose commands for simulating the response of linear, time-invariant systems. Among these commands are commands specific to determining the step response and natural response of systems; we will restrict our attention to these commands in this chapter.

To calculate system responses (e.g. to solve the differential equations of interest), we will use only Octave's **step** and **initial** commands in this chapter. The **step** command calculates a unit step response for the system, while the **initial** command calculates the natural response of a system to some set of initial conditions. We will also use the **ss** command to create state space model objects, to send to the **step** and **initial** commands. Basic syntax for these commands is provided below.

**step:** The command **[y,t]=step(sys)** returns the step response of the state variable model described by the model object **sys**. The vector **y** contains the system output, the matrix **x** contains the states, and the vector **t** contains the time samples.

- The command **[y]=step(sys,t)** returns the step response as above, but calculated over the specified time vector **t**.
- The command **[y,t]=step(sys)** returns the system output as above, and the times at which the response is calculated, but not the state variables **x**.
- **step(sys)** with no left-hand arguments results in a plot of the step response output.

**initial:** Response of linear system to an initial condition. **[y,x,t]=initial(sys,x0)** returns the response of the system described by the state space model  $\{A,b,c,d\}$  to an initial condition contained in the vector **x0**. Variations on this command are similar to those provided above for the **step** command.

**ss:** Create a state space model object. **sys = ss(A, b, c, d)** returns an object named **sys** which provides a state space model corresponding to the matrices provided in **A, b, c, and d**.

#### Example 9.4: Step Response of Series RLC Circuit

Determine and plot the response of the system of example 9.1 if:

$$u(t) = \begin{cases} 0V, & t < 0 \\ 2V, & t \geq 0 \end{cases}$$

and the circuit is initially relaxed (i.e. all voltages and currents in the system are initially zero). Also plot the state trajectory for this input. The state equations for the circuit of example 9.1 were previously determined to be:

$$\begin{bmatrix} \dot{x}_1 \\ \dot{x}_2 \end{bmatrix} = \begin{bmatrix} -2 & -2 \\ 2 & 0 \end{bmatrix} \begin{bmatrix} x_1 \\ x_2 \end{bmatrix} + \begin{bmatrix} 2 \\ 0 \end{bmatrix} u(t)$$

While the output equation is:

$$y = [0 \quad 1] \begin{bmatrix} x_1 \\ x_2 \end{bmatrix} + 0 \cdot u$$

Thus, the matrices describing the state space model are:

$$A = \begin{bmatrix} -2 & -2 \\ 2 & 0 \end{bmatrix}, b = \begin{bmatrix} 2 \\ 0 \end{bmatrix}, c = [0 \quad 1], d = 0$$

To simulate the response of the system we first need to input the state variable model. We begin by defining the A, b, c, and d matrices as follows:

```
> A = [-2 -2; 2 0];
> b = [2; 0];
> c = [0 1];
> d = 0;
```

The > symbols denote the command prompt at Octave's command window; they are included here to emphasize Octave commands.

Unlike MATLAB, Octave requires you to create a model object to represent the system model. To create a state space system model, the command is `ss`. Arguments to the command are the above matrices; the output is the system model object. To create a model of our system, we type:

```
> sys = ss(A, b, c, d);
```

Our workspace now contains a state space model object of our system named “sys”.

We can simulate the response of this system to the desired input by using Octave’s `step` command. The `step` command assumes that the system is initially relaxed and the input to the system is:

$$u(t) = \begin{cases} 0, & t < 0 \\ 1, & t \geq 0 \end{cases}$$

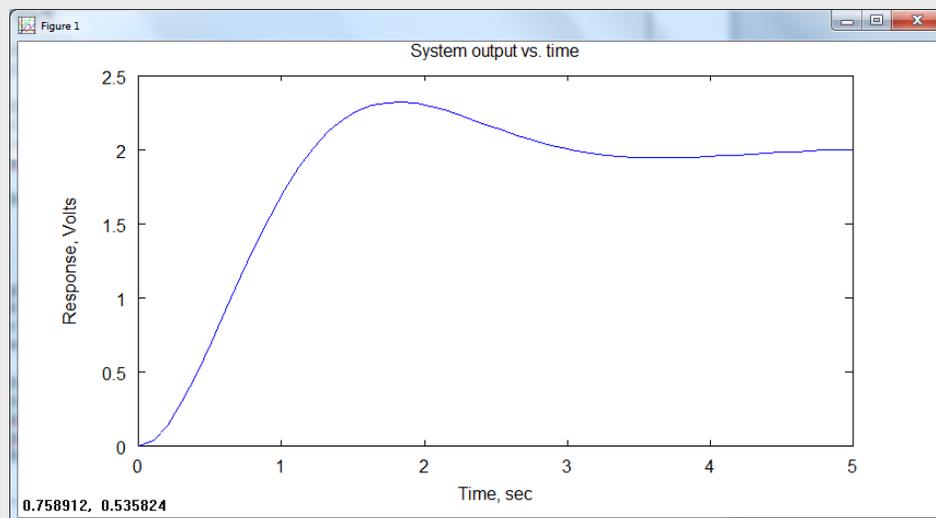
The input to our system is exactly twice this input, so we can simply scale our output by a factor of two. (This works because the system is linear – don’t try this with a nonlinear system!) The appropriate commands are (note that we have to scale both the output and the states, since we will be plotting the output response and the state trajectory):

```
> [y, t] = step(sys);
> y = 2 * y;
```

The final step is to plot the response. Plotting the output response can be accomplished with the following commands:

```
> plot(t, y)
> title('System output vs. time')
> xlabel('Time, sec')
> ylabel('Response, Volts')
```

which results in the figure below:



In order to plot the state trajectory using Octave, we must modify our state space model somewhat. Unlike MATLAB, the states themselves are not returned by Octave’s `step` command. However, we can alter our `c` and `d` matrices so that the output, `y`, contains both system states. We will set up our output equations as follows:

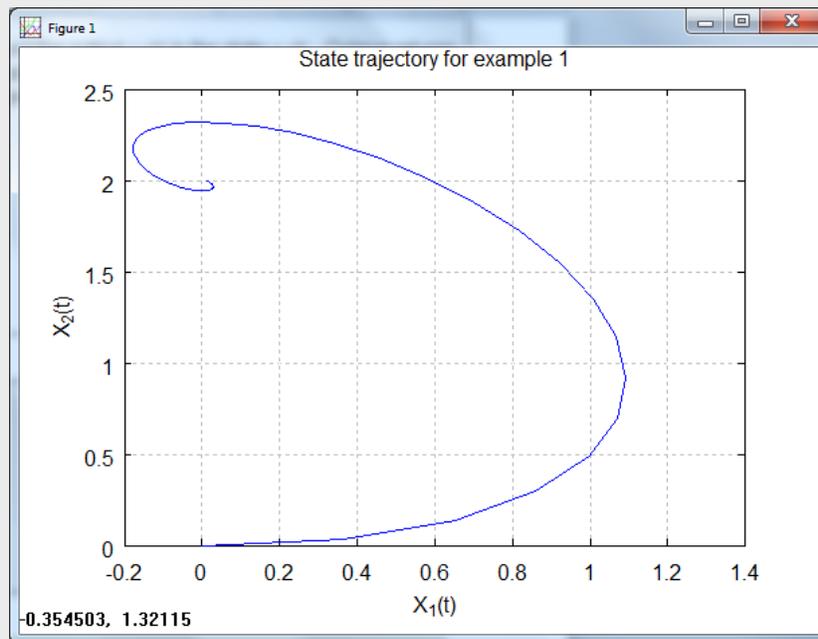
$$\begin{bmatrix} y_1(t) \\ y_2(t) \end{bmatrix} = \begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix} \begin{bmatrix} x_1 \\ x_2 \end{bmatrix} + \begin{bmatrix} 0 \\ 0 \end{bmatrix} u$$

This is implemented in Octave by re-setting the  $c$  and  $d$  matrices, re-calculating the model object, and re-calculating the system response with the following commands:

```
> c = [1 0; 0 1];
> d = [0; 0];
> sys = ss(A, b, c, d);
> [y, t] = step(sys);
> y = 2 * y;
```

Now, the output  $y_1(t)$  is simply the state  $x_1(t)$  and the output  $y_2(t)$  is the state  $x_2(t)$ . Octave returns the first state vector as the first row of the  $y$  matrix, and the second state vector as the second row of the  $y$  matrix. Thus, we can plot the state trajectory with the following commands:

```
> plot(y(1, :), y(:, 2))
> grid
> xlabel('X1(t)')
> ylabel('X2(t)')
> title('State trajectory for Example 1')
```

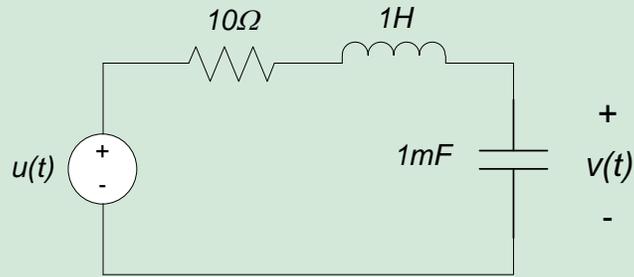


## Section Summary

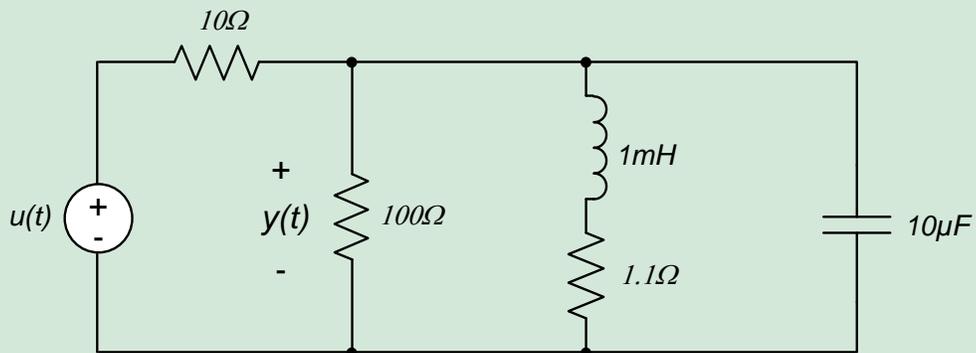
- Octave allows you to simulate linear, time invariant systems with the following commands:
  - A state space model object can be created with the **ss** function. Inputs to the function are the  $A$ ,  $b$ ,  $c$ , and  $d$  matrices of the state space model.
  - The **step** command can be used to calculate the step response of the system.
  - The **initial** command can be used to calculate the natural response of the system.

## 9.3 Exercises

1. Simulate the response  $y(t)$  of the circuit below, if  $u(t)$  is a unit step function.



2. Simulate the response  $y(t)$  of the circuit below, if  $u(t)$  is a unit step function.



## Real Analog Chapter 9: Lab Projects

### 9.3.1: State Variable Model of Series RLC Circuits

This lab will emphasize the use of the state variable approach for modeling electrical circuits.

In this assignment, we will create a state variable model of an electrical circuit and simulate the response of the circuit using MATLAB and/or Octave. We will then build and test the circuit. The circuit of interest in this lab assignment is a series RLC circuit; the circuit is actually the same circuit as in Part I of Lab 8.5.1. Unlike Lab 8.5.1, however, in this lab assignment we will analyze the circuit using state variable methods and use available computer software to simulate the circuit responses<sup>40</sup> and verify our analysis by measuring both of the circuit states. The experimentally-determined responses and state trajectory will be compared with the simulated responses.

#### Before beginning this lab, you should be able to:

- Identify appropriate state variables for electrical systems
- Determine an appropriate state variable model for an electrical system from a schematic of the circuit.
- Use MATLAB or Octave to simulate a state variable model's response

#### After completing this lab, you should be able to:

- Use state variable methods to model electrical systems
- Experimentally measure the response of a system's state variables to an arbitrary input
- Plot state trajectories for a system from experimental data and simulation results

#### This lab exercise requires:

- Analog Discovery 2 module
- Diligent Analog Parts Kit
- Digital multimeter (optional)

#### Symbol Key:

	Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
	Analysis; include principle results of analysis in laboratory report.
	Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
	Record data in your lab notebook.

#### General Discussion:

In this lab assignment, we will create a state variable model of the circuit shown in Fig. 1. We will simulate the response of the circuit to a step input and compare the simulated step response to the measured step response for the circuit.

<sup>40</sup> In Lab Project 8.5.1, we estimated a few important circuit response parameters (rise time, overshoot, steady-state response) based on the expected response of second order systems and some approximations. Simulating the circuit response using computer software provides the response as a function of time. This requires fewer assumptions and approximations than in Lab 8.5.1.

The circuit of Fig. 1 has two independent energy storage elements, and thus has two states. We will define states as the voltages across the capacitor and the current through the inductor. Our lab equipment does not readily allow for measurement of time-varying currents; therefore, we will use the measured voltage across the resistor and Ohm's law to infer the current through the inductor.

We will be measuring both system states in order to plot a *state trajectory* for the system. The state variables can be thought of as the axes of a coordinate system; the values of the state variables at any point in time uniquely define a point in that coordinate system. The coordinate system is called a *state space*, it corresponds to the space of all possible combinations of the state variables. Typically, when an input is applied to a system, the state variables will respond by changing from one value to another. The path in the state space through which the state variables pass is called the *trajectory*.

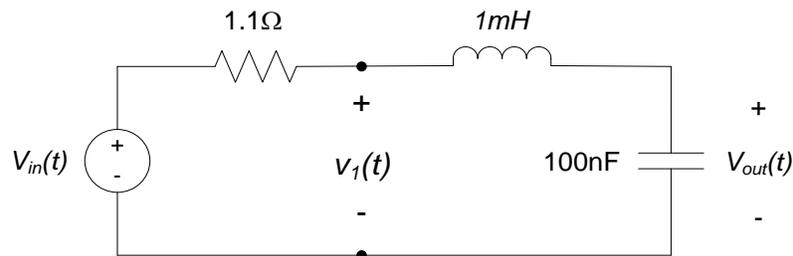


Figure 1. Circuit schematic.

### Pre-lab:

#### ANALYSIS

Determine a state variable model for the circuit shown in Fig. 1, if the input voltage  $V_{in}(t) = 2u_0(t)$  Volts, where  $u_0(t)$  is the unit step function. Use the voltage across the capacitor and the current through the inductor as your states. Your model should output both system states. Simulate the response of your system using MATLAB or Octave. Plot the responses of both system states vs. time and the state trajectory.

### Lab Procedures:

#### DATA

Construct the circuit shown in Fig. 1. Record actual values for the resistor and capacitor. (If your DMM does not have the capability of measuring capacitance, you may omit the capacitance measurement.)

#### DATA

a. Use a square wave input from your function generator to emulate a 3V step input to the system. Record the amplitude of the square wave you are applying to the system; you will need that in order to compare your data with your analysis from the pre-lab. Note: be sure that the frequency of the square wave is low enough to allow the system to reach a steady-state response.

#### DEMO

b. Demonstrate operation of your circuit to the TA and have them initial your lab notebook and the lab checklist.

#### DATA

c. Set up the oscilloscope to measure the voltage across the capacitor and the current through the inductor.

Oscilloscopes do not provide the capability to measure time-varying currents, so we will infer the current through the inductor from the voltage across the nominal  $1.1\Omega$  resistor Fig. 1. If we measure the voltage  $v_R(t)$ , we can calculate the inductor current as  $\frac{v_R(t)}{1.1\Omega}$ . To set up your oscilloscope to display the current through the inductor, do the following:

- i. Use, for example, channel 1 to measure the input voltage  $V_{in}(t)$  and channel 2 to measure the voltage,  $v_1(t)$ . The voltage  $v_R(t)$  is the difference between these two voltages.

- ii. Click on “Add Channel” and select **Custom**. A custom math channel control panel should appear.
- iii. Click on the button on the bottom of this window to set up the math expression; a math editor window should open. In the text box in this window, type the expression which determines current from the measured voltage on channel 2:  $(CH1-C2)/1.1$ . (Assuming that the resistance value of the resistor is  $1.1\Omega$ ; you should actually divide  $(C1-C2)$  by whatever the actual resistance value is that you used.)
- iv. Click “OK” to create the math channel.

**DATA**

- d. Record an image of the oscilloscope window, showing the natural responses of both states. Also record the responses of the system states to a file.

**DEMO**

- e. Demonstrate operation of your circuit to the TA and have them initial your lab notebook and the lab checklist.
- f. Plot the responses measured in part (d) using your favorite plotting software (Excel, Matlab, ...)
- g. Plot the state trajectory as measured in part (d).

**ANALYSIS**

- h. Compare the step responses and the state trajectory you measured to the expected response from the pre-lab. Discuss the comparison in your lab report.

**Post-lab Exercises:****ANALYSIS**

- a. Overlay the simulated output,  $y(t)$ , vs. time from your pre-lab simulations and the measured output response vs. time. Comment on the differences between the two.

**SIM**

- b. Re-do your simulations from the pre-lab using the actual component values. Also, include the resistance of the inductor in your simulation. (This resistance can be added directly to the  $1.1\Omega$  resistance, since the inductor and resistor are in series. No fundamental changes to your model are necessary.) Overlay plots of the simulation results (state responses vs. time and state trajectories) with the measured state responses and state trajectories. Comment on the differences between the simulation and the measured data.



ii. Simulated and measured state trajectories. (7 pts)

6. In the space below, provide a comparison between the measured and simulated step responses (e.g. a maximum difference between the measured and simulated responses) and discussion of measured vs. simulated step responses and state trajectories (include at least one possible source of any differences between the measured and simulated results). (7 pts)

## Real Analog Chapter 9: Lab Projects

### 9.3.2: Second Order Circuit Response

This lab will emphasize the use of the state variable approach for modeling electrical circuits.

In this assignment, we will create a state variable model of a second order electrical circuit and simulate the response of the circuit using MATLAB and/or Octave. We will build and test the circuit, and measure both of the circuit states. The experimentally-determined responses and state trajectory will be compared with the simulated responses.

#### Before beginning this lab, you should be able to:

- Identify appropriate state variables for electrical systems
- Determine an appropriate state variable model for an electrical system from a schematic of the circuit.
- Use MATLAB or Octave to simulate the response of a state variable model to an arbitrary input

#### After completing this lab, you should be able to:

- Use state variable methods to model electrical systems
- Experimentally measure the response of a system's state variables to an arbitrary input
- Plot state trajectories for a system from experimental data and simulation results

#### This lab exercise requires:

- Analog Discovery 2 module
- Digilent Analog Parts Kit
- Digital multimeter (optional)

#### Symbol Key:

- DEMO** Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
- ANALYSIS** Analysis; include principle results of analysis in laboratory report.
- SIM** Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
- DATA** Record data in your lab notebook.

#### General Discussion:

In this lab assignment, we will create a state variable model of the circuit shown in Fig. 1. We will simulate the response of the circuit to a step input and compare the simulated step response to the measured step response for the circuit.

The circuit of Fig. 1 has two independent energy storage elements, and thus has two states. We will define states as the voltages across the capacitor and the current through the inductor. Our lab equipment does not readily allow for measurement of time-varying currents; therefore, we will use the measured voltage across the resistor and Ohm's law to infer the current through the inductor.

We will be measuring both system states in order to plot a *state trajectory* for the system. The state variables can be thought of as the axes of a coordinate system; the values of the state variables at any point in time uniquely define a point in that coordinate system. The coordinate system is called a *state space*, it corresponds to the space

of all possible combinations of the state variables. Typically, when an input is applied to a system, the state variables will respond by changing from one value to another. The path in the state space through which the state variables pass is called the *trajectory*.

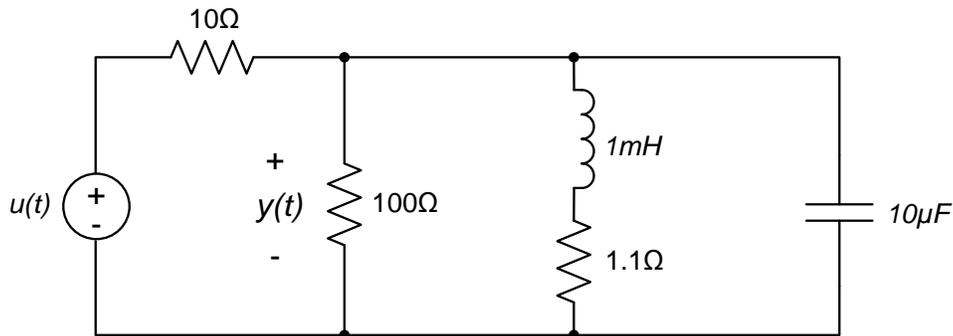


Figure 1. Circuit schematic.

### Pre-lab:

#### ANALYSIS

Determine a state variable model for the circuit shown in Fig. 1. Use the voltage across the capacitor and the current through the inductor as your states. Simulate the response of your system using MATLAB or Octave. Plot the responses of both system states vs. time and the state trajectory. Your model should output both system states. The input voltage  $V_{in}(t) = u_o(t)$  Volts, where  $u_o(t)$  is the unit step function.

### Lab Procedures:

#### DATA

Construct the circuit shown in Fig. 1. Record actual values for all resistors and the capacitor. (If your DMM does not have the capability of measuring capacitance, you may omit the capacitance measurement.)

#### DATA

a. Use a square wave input from your function generator to emulate a 1V step input to the system. Record the amplitude of the square wave you are applying to the system; you will need that in order to compare your data with your analysis from the pre-lab. Note: be sure that the frequency of the square wave is low enough to allow the system to reach a steady-state response.

#### DEMO

b. Demonstrate operation of your circuit to the TA and have them initial your lab notebook and the lab worksheet.

#### DATA

c. Set up the oscilloscope to measure the voltage across the capacitor and the current through the inductor. To do this, we will refer to the circuit schematic shown in Fig. 2 below. The capacitor voltage measurement is simple – simply use one of the oscilloscope channels (say, channel 1) to measure  $v_c(t)$  indicated on Fig. 2.

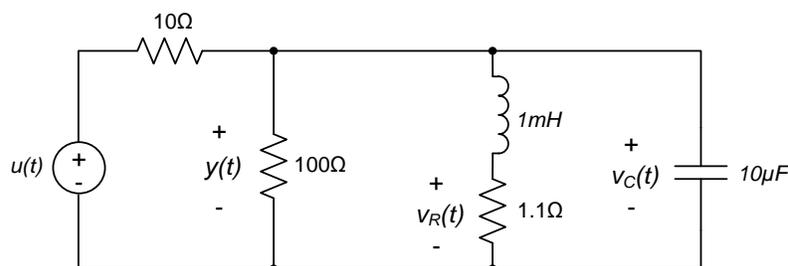


Figure 2. Schematic indicating measured voltages.

Oscilloscopes do not provide the capability to measure time-varying currents, so we will infer the current through the inductor from the voltage across the nominal  $1.1\Omega$  resistor Fig. 1. If we measure the voltage  $v_R(t)$ , we can calculate the inductor current as  $\frac{v_R(t)}{1.1\Omega}$ . To set up your oscilloscope to display the current through the inductor, do the following:

- i. Use, for example, channel 2 (CH2) to measure the input voltage  $v_R(t)$ .
- ii. Click on the “Add Channel” button on the oscilloscope toolbar, select **Custom**. A custom math channel control panel should appear.
- iii. Click on the button on the bottom of this window to set up the math expression; a math editor window should open. In the text box in this window, type the expression which determines current from the measured voltage on channel 2:  $C2/1.1$ . (Assuming that the resistance value of the resistor is  $1.1\Omega$ ; you should actually divide  $v_R(t)$  by whatever the actual resistance value is that you used.)
- iv. Click “OK” to create the math channel.

**DATA**

- d. Record an image of the oscilloscope window, showing the responses of both states. Also record the responses of the system states to a file<sup>41</sup>.

**DEMO**

- e. Demonstrate operation of your circuit to the TA and have them initial your lab notebook and the lab worksheet.
- f. Plot the responses measured in part (d) using your favorite plotting software (Matlab, Octave, Excel,...)
- g. Plot the state trajectory as measured in part (d).

**ANALYSIS**

- h. Compare the step responses and the state trajectory you measured to the expected response from the pre-lab. Discuss the comparison in your lab report.

**Post-lab Exercises:****ANALYSIS**

- a. Overlay the simulated output,  $y(t)$ , vs. time from your pre-lab simulations and the measured output response vs. time. Comment on the differences between the two.

**SIM**

- b. Re-do your simulations from the pre-lab using the actual component values. Also, include the resistance of the inductor in your simulation. (This resistance can be added directly to the  $1.1\Omega$  resistance, since the inductor and resistor are in series. No fundamental changes to your model are necessary.) Overlay plots of the simulation results (state responses vs. time and state trajectories) with the measured state responses and state trajectories. Comment on the differences between the simulation and the measured data.

<sup>41</sup> Hint: the “Export” button on the scope instrument can be used to export the data to either a .csv or .txt file. Most application software (e.g. MATLAB, Excel) allow you to import data in either of these formats.

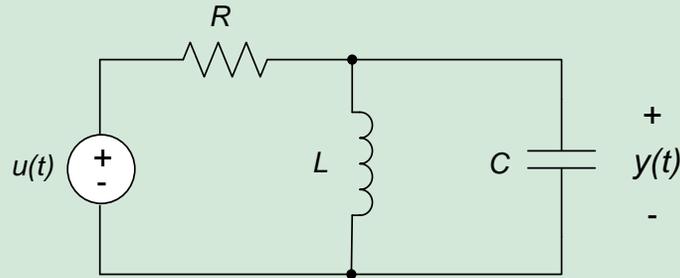


ii. Simulated and measured state trajectories. (7 pts)

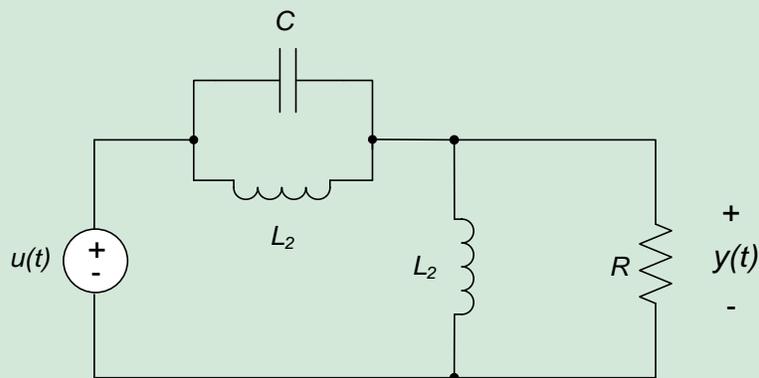
6. In the space below, provide a comparison between the measured and simulated step responses (e.g. a maximum difference between the measured and simulated responses) and discussion of measured vs. simulated step responses and state trajectories (include at least one possible source of any differences between the measured and simulated results). (7 pts)

## Real Analog Chapter 9: Homework

9.1 Determine a state variable model,  $\{A,b,c,d\}$  for the circuit shown.  $u(t)$  is the input and  $y(t)$  is the output.

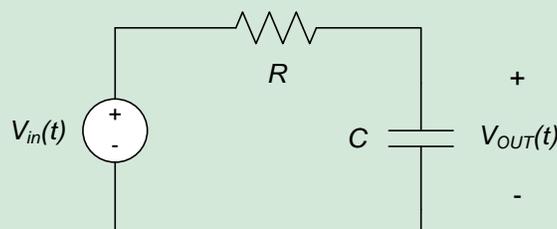


9.2 Determine a state variable model,  $\{A,b,c,d\}$  for the circuit shown.  $u(t)$  is the input and  $y(t)$  is the output.



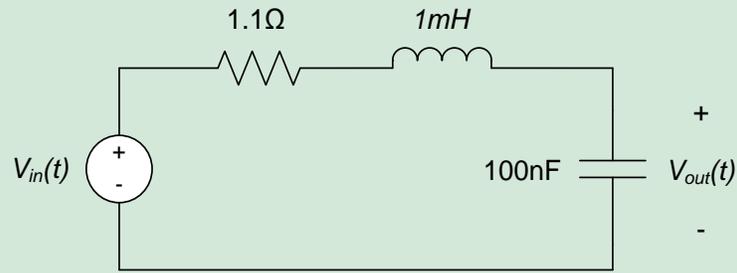
9.3 For the circuit below,

- Determine a state variable model,  $\{A,b,c,d\}$  if  $V_{in}(t)$  is the input and  $V_{out}(t)$  is the output.
- Simulate and plot (using MATLAB or Octave) the response  $V_{out}(t)$  of the circuit if  $R = 470 \Omega$ ,  $C = 1 \mu\text{F}$ , and the input  $V_{in}(t) = 4u_0(t)$ , where  $u_0(t)$  is the unit step function. From your plot, estimate the time constant of the circuit.
- Estimate the time constant of the circuit from the capacitance and equivalent resistance seen by the capacitor. Compare this value to the time constant estimated from the simulated response. Comment on any differences.



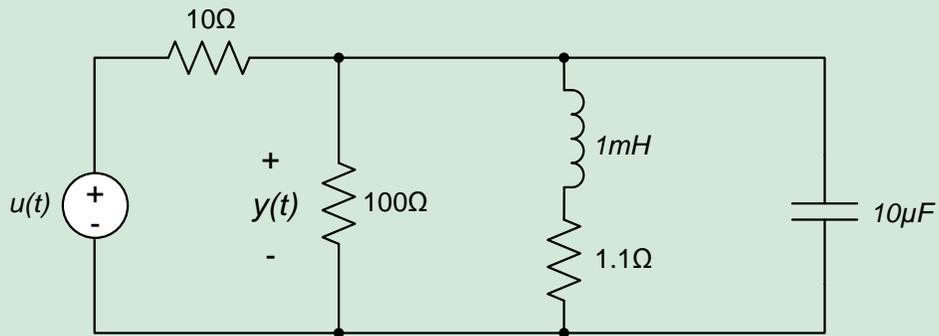
9.4 For the circuit below,

- Determine a state variable model,  $\{A,b,c,d\}$  if  $V_{in}(t)$  is the input and  $V_{out}(t)$  is the output.
- Simulate and plot (using MATLAB or Octave) the response  $V_{out}(t)$  of the circuit to an input  $V_{in}(t) = 3u_0(t)$ , where  $u_0(t)$  is the unit step function. From your plots, estimate the maximum overshoot and rise time of the response.
- Determine an input-output relationship for the circuit and estimate the natural frequency, damping ratio, and final value  $V_{out}(t \rightarrow \infty)$ . From these, estimate the maximum overshoot and rise time of the response. Compare these values to those measured from your simulation. Comment on any differences.



9.5 For the circuit below,

- a. Determine a state variable model,  $\{A, b, c, d\}$  if  $V_{in}(t)$  is the input and  $V_{out}(t)$  is the output.
- b. Simulate and plot (using MATLAB or Octave) the response of the circuit to an input  $V_{in}(t) = 2u_0(t)$ , where  $u_0(t)$  is the unit step function.



## Chapter 10: Steady-state Sinusoidal Analysis

### 10 Introduction and Chapter Objectives

We will now study dynamic systems which are subjected to sinusoidal forcing functions. Previously, in our analysis of dynamic systems, we determined both the *unforced response* (or *homogeneous solution*) and the *forced response* (or *particular solution*) to the given forcing function. In the next several chapters, however, we will restrict our attention to only the system's forced response to a sinusoidal input; this response is commonly called the sinusoidal steady-state system response. This analysis approach is useful if we are concerned primarily with the system's response after any initial conditions have died out, since we are ignoring any transient effects due to the system's natural response.

Restricting our attention to the steady-state sinusoidal response allows a considerable simplification in the system analysis: we can solve algebraic equations rather than differential equations. This advantage often more than compensates for the loss of information relative to the systems natural response. For example it is often the case that a sinusoidal input is applied for a very long time relative to the time required for the natural response to die out, so that the overall effects of the initial conditions are negligible.

Steady-state sinusoidal analysis methods are important for several reasons:

- Sinusoidal inputs are an extremely important category of forcing functions. In electrical engineering, for example, sinusoids are the dominant signal in the electrical power industry. The alternating current (or AC) signals used in power transmission are, in fact, so pervasive that many electrical engineers commonly refer to any sinusoidal signal as “AC”. Carrier signals used in communications systems are also sinusoidal in nature.
- The simplification associated with the analysis of steady state sinusoidal analysis is often so desirable that system responses to non-sinusoidal inputs are interpreted in terms of their sinusoidal steady-state response. This approach will be developed when we study Fourier series.
- System design requirements are often specified in terms of the desired steady-state sinusoidal response of the system.

In section 10.1 of this chapter, we qualitatively introduce the basic concepts relative to sinusoidal steady state analyses so that readers can get the “general idea” behind the analysis approach before addressing the mathematical details in later sections. Since we will be dealing exclusively with sinusoidal signals for the next few chapters, section 10.2 provides review material relative to sinusoidal signals and complex exponentials. Recall from chapter 8 that complex exponentials are a mathematically convenient way to represent sinusoidal signals. Most of the material in section 10.2 should be review, but the reader is strongly encouraged to study section 10.2 carefully -- we will be using sinusoids and complex exponentials extensively throughout the remainder of this text, and a complete understanding of the concepts and terminology is crucial. In section 10.3, we examine the forced response of electrical circuits to sinusoidal inputs; in this section, we analyze our circuits using differential equations and come to the important conclusion that steady-state response of a circuit to sinusoidal inputs is governed by algebraic equations. Section 10.4 takes advantage of this conclusion to perform steady-state sinusoidal analyses of electrical circuits without writing the governing differential equation for the circuit! Finally, in section 10.5, we characterize a system's response purely by its effect on a sinusoidal input. This concept will be used extensively throughout the remainder of this textbook.

**After completing this chapter, you should be able to:**

- State the relationship between the sinusoidal steady state system response and the forced response of a system

- For sinusoidal steady-state conditions, state the relationship between the frequencies of the input and output signals for a linear, time-invariant system
- State the two parameters used to characterize the sinusoidal steady-state response of a linear, time-invariant system
- Define periodic signals
- Define the amplitude, frequency, radian frequency, and phase of a sinusoidal signal
- Express sinusoidal signals in phasor form
- Perform frequency-domain analyses of electrical circuits
- Sketch phasor diagrams of a circuit's input and output
- State the definition of impedance and admittance
- State, from memory, the impedance relations for resistors, capacitors, and inductors
- Calculate impedances for resistors, capacitors, and inductors
- State how to use the following analysis approaches in the frequency domain:
  - KVL and KCL
  - Voltage and current dividers
  - Circuit reduction techniques
  - Nodal and mesh analysis
  - Superposition, especially when multiple frequencies are present
  - Thévenin's and Norton's theorems
- Determine the load impedance necessary to deliver maximum power to a load
- Define the frequency response of a system
- Define the magnitude response and phase response of a system
- Determine the magnitude and phase responses of a circuit

## 10.1 Introduction to Steady-state Sinusoidal Analysis

In this chapter, we will be almost exclusively concerned with sinusoidal signals, which can be written in the form:

$$f(t) = A \cos(\omega t + \theta) \quad \text{Eq. 10.1}$$

Where  $A$  is the *amplitude* of the sinusoid,  $\omega$  is the *angular frequency* (in radians/second) of the signal, and  $\theta$  is the *phase angle* (expressed in radians or degrees) of the signal.  $A$  provides the peak value of the sinusoid,  $\omega$  governs the rate of oscillation of the signal, and  $\theta$  affects the translation of the sinusoid in time. A typical sinusoidal signal is shown in Fig. 10.1.

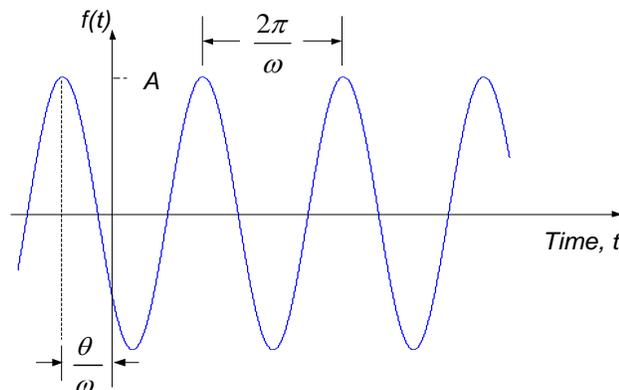


Figure 10.1. Sinusoidal signal.

If the sinusoidal signal of Fig. 10.1 is applied to a linear time invariant system, the response of the system will consist of the system's natural response (due to the initial conditions on the system) superimposed on the

system's forced response (the response due to the forcing function). As we have seen in previous chapters, the forced response has the same form as the forcing function. Thus, if the input is a constant value the forced response is constant, as we have seen in the case of the step response of a system. In the case of a sinusoidal input to a system, the forced response will consist of a sinusoid of the same frequency as the input sinusoid. Since the natural response of the system decays with time, the steady state response of a linear time invariant system to a sinusoidal input is a sinusoid, as shown in Fig. 10.2. The amplitude and phase of the output may be different than the input amplitude and phase, but both the input and output signals have the same frequency.

It is common to characterize a system by the ratio of the magnitudes of the input and output signals ( $\frac{B}{A}$  in Fig. 10.2) and the difference in phases between the input and output signals ( $\phi - \theta$  in Fig. 10.2) at a particular frequency. It is important to note that the ratio of magnitudes and difference in phases is dependent upon the frequency of the applied sinusoidal signal.

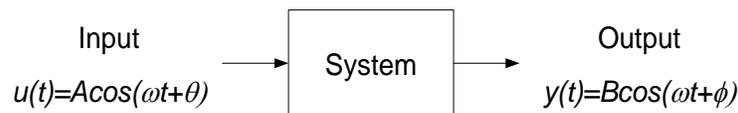


Figure 10.2. Sinusoidal steady-state input-output relation for a linear time invariant system.

### Example 10.1: Series RLC Circuit Response

Consider the series RLC circuit shown in Fig. 10.3 below. The input voltage to the circuit is given by:

$$v_s(t) = \begin{cases} 0, & t < 0 \\ \cos(5t), & t \geq 0 \end{cases}$$

Thus, the input is zero prior to  $t=0$ , and the sinusoidal input is suddenly “switched on” at time  $t=0$ . The input forcing function is shown in Fig. 10.4(a). The circuit is “relaxed” before the sinusoidal input is applied, so the circuit initial conditions are:

$$y(0^-) = \left. \frac{dy}{dt} \right|_{t=0^-} = 0$$

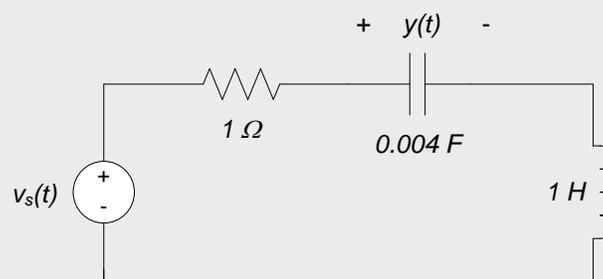
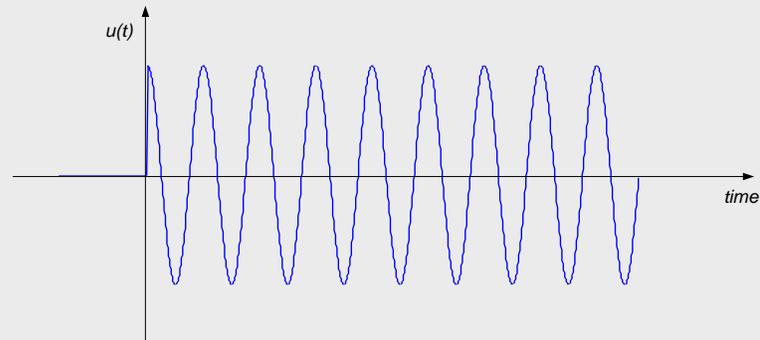


Figure 10.3. Series RLC circuit; output is voltage across capacitor.

This circuit has been analyzed previously in Chapter 8, and the derivation of the governing differential equation will not be repeated here. The full output response of the circuit is shown in Fig. 10.4(b). The natural response of the circuit is readily apparent in the initial portion of the response but these transients die out quickly, leaving only the sinusoidal steady-state response of the circuit. It is only this steady state response in which we will be interested for the next several modules. With knowledge of the frequency of the signals, we can define both the input and (steady-state) output by their amplitude and phase, and characterize the circuit by the ratio of the output-to-input amplitude and the difference in the phases of the output and input.



(a) Input signal



Steady-State Response

(b) Output signal.

Figure 10.4. Input and output signals for circuit of Figure 10.3.

## Section Summary

- Sinusoidal signals can be expressed mathematically in the form:

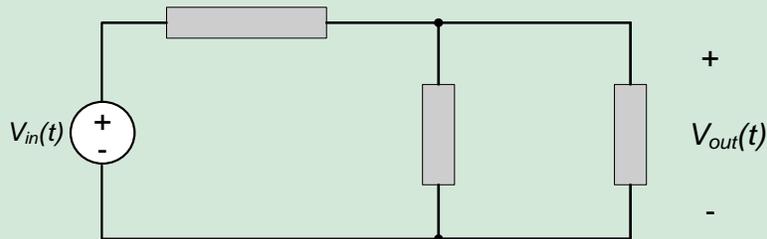
$$f(t) = A \cos(\omega t + \theta)$$

- In the above,  $A$  is the amplitude of the sinusoid, it describes the maximum and minimum values of the signal.
- In the above,  $\theta$  is the phase angle of the sinusoid, it describes the time shift of the sinusoid relative to a pure cosine.
- In the above,  $\omega$  is the radian frequency of the sinusoid. The sinusoid repeats itself at time intervals of  $\frac{2\pi}{\omega}$  seconds.
- A sinusoidal signal is completely described by its frequency, its amplitude, and its phase angle.
- The steady-state response of a linear, time-invariant system to a sinusoidal input is a sinusoid with the same frequency.
- Since the frequencies of the input and output are the same, the relationship between the input and output sinusoids is completely characterized by the relationships between:
  - The input and output amplitudes.

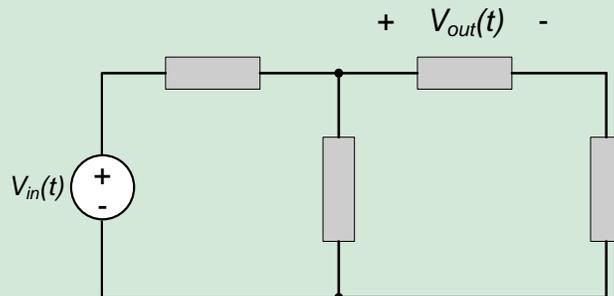
- The input and output phase angles.

## 10.1 Exercises

1. In the circuit below, all circuit elements are linear and time invariant. The input voltage  $V_{in}(t) = 10 \cos(2t + 40^\circ)$ . What is the radian frequency of the output voltage  $V_{out}(t)$ ?



2. In the circuit below, all circuit elements are linear and time invariant. The input voltage is  $V_{in}(t) = 10 \cos(2t + 40^\circ)$ . The output voltage is of the form  $V_{out}(t) = A \cos(\omega t + \phi^\circ)$ . If the ratio between the input and output,  $\left| \frac{V_{out}}{V_{in}} \right| = 0.5$  and the phase difference between the input and output is  $20^\circ$ , what are:
  - a. The radian frequency of the output,  $\omega$ ?
  - b. The amplitude of the output,  $A$ ?
  - c. The phase angle of the output,  $\phi$ ?



## 10.2 Sinusoidal Signals, Complex Exponentials, and Phasors

In this section, we will review properties of sinusoidal functions and complex exponentials. We will also introduce phasor notation, which will significantly simplify the sinusoidal steady-state analysis of systems, and provide terminology which will be used in subsequent sinusoidal steady-state related modules.

Much of the material presented here has been provided previously in Chapter 8; this material is, however, important enough to bear repetition. Likewise, a brief overview of complex arithmetic, which will be essential in using complex exponentials effectively, is provided at the end of this section. Readers who need to review complex arithmetic may find it useful to peruse this overview before reading the material in this section relating to complex exponentials and phasors.

### 10.2.1 Sinusoidal Signals

The sinusoidal signal shown in Fig. 10.5 is represented mathematically by:

$$f(t) = V_p \cos(\omega t) \quad \text{Eq. 10.2}$$

The *amplitude* or *peak value* of the function is  $V_p$ .  $V_p$  is the maximum value achieved by the function; the function itself is bounded by  $+V_p$  and  $-V_p$ , so that  $-V_p \leq f(t) \leq V_p$ . The *radian frequency* or *angular frequency* of the function is  $\omega$ ; the units of  $\omega$  are radians/second. The function is said to be *periodic*; *periodic* functions repeat themselves at regular intervals, so that:

$$f(t + nT) = f(t) \quad \text{Eq. 10.3}$$

Where  $n$  is any integer and  $T$  is the *period* of the signal. The sinusoidal waveform shown in Fig. 10.5 goes through one complete cycle or period in  $T$  seconds. Since the sinusoid of equation (10.2) repeats itself every  $2\pi$  radians, the period is related to the radian frequency of the sinusoid by:

$$\omega = \frac{2\pi}{T} \quad \text{Eq. 10.4}$$

It is common to define the frequency of the sinusoid in terms of the number of cycles of the waveform which occur in one second. In these terms, the frequency  $f$  of the function is:

$$f = \frac{1}{T} \quad \text{Eq. 10.5}$$

The units of  $f$  are cycles/second or Hertz (abbreviated Hz). The frequency and radian frequency are related by:

$$f = \frac{\omega}{2\pi} \quad \text{Eq. 10.6}$$

Or equivalently:

$$\omega = 2\pi f \quad \text{Eq. 10.7}$$

Regardless of whether the sinusoid's rate of oscillation is expressed as frequency or radian frequency, it is important to realize that the argument of the sinusoid in equation (10.2) must be expressed in radians. Thus, equation (10.2) can be expressed in terms of frequency in Hz as:

$$f(t) = \cos(2\pi f t) \quad \text{Eq. 10.8}$$

To avoid confusion in our mathematics, we will almost invariably write sinusoidal functions in terms of radian frequency as shown in equation (10.2), although Hz is generally taken as the standard unit for frequency (experimental apparatus, for example, commonly express frequency in Hz).

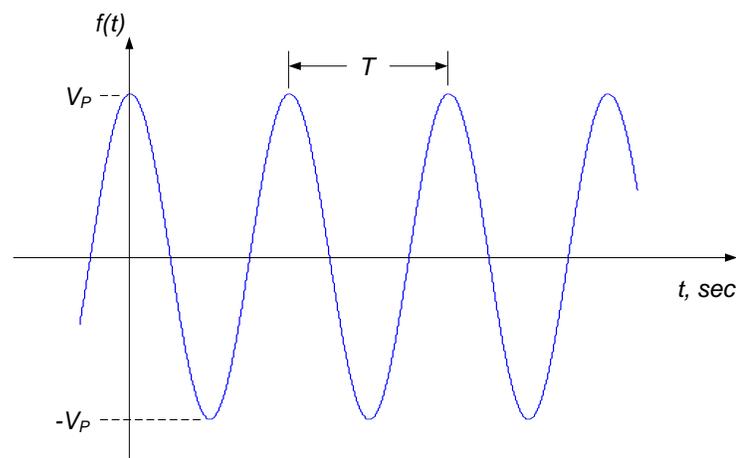


Figure 10.5. Pure cosine waveform.

A more general expression of a sinusoidal signal is:

$$v(t) = V_P \cos(\omega t + \theta) \quad \text{Eq. 10.9}$$

Where  $\theta$  is the *phase angle* or *phase* of the sinusoid. The phase angle simply translates the sinusoid along the time axis, as shown in Fig. 10.6. A positive phase angle shifts the signal left in time, while a negative phase angle shifts the signal right – this is consistent with our discussion of step functions in section 6.1, where it was noted that subtracting a value from the unit step argument resulted a time delay of the function. Thus, as shown in Figure 10.6, a positive phase angle causes the sinusoid to be shifted left by  $\theta/\omega$  seconds.

The units of phase angle should be radians, to be consistent with the units of  $\omega t$  in the argument of the cosine. It is typical, however, to express phase angle in degrees, with  $180^\circ$  corresponding to  $\pi$  radians. Thus, the conversion between radians and degrees can be expressed as:

$$\text{Number of degrees} = \frac{180}{\pi} \times \text{Number of radians}$$

For example, we will consider the two expressions below to be equivalent, though the expression on the right-hand side of the equal sign contains a mathematical inconsistency:

$$V_P \cos\left(\omega t + \frac{\pi}{2}\right) = V_P \cos(\omega t + 90^\circ)$$

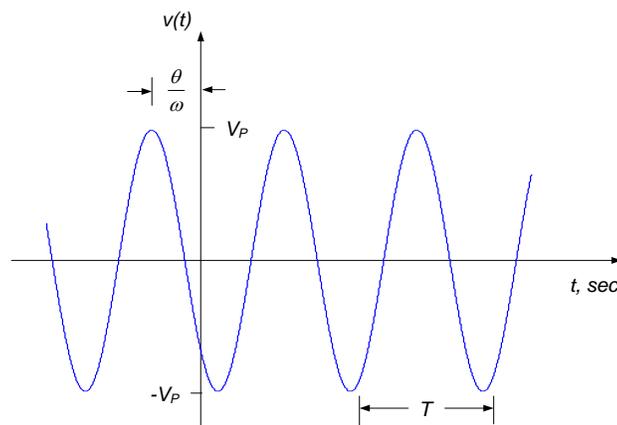


Figure 10.6. Cosine waveform with non-zero phase angle.

For convenience, we introduce the terms *leading* and *lagging* when referring to the sign on the phase angle,  $\theta$ . A sinusoidal signal  $v_1(t)$  is said to *lead* another sinusoid  $v_2(t)$  of the same frequency if the phase difference between the two is such that  $v_1(t)$  is shifted left in time relative to  $v_2(t)$ . Likewise,  $v_1(t)$  is said to *lag* another sinusoid  $v_2(t)$  of the same frequency if the phase difference between the two is such that  $v_1(t)$  is shifted right in time relative to  $v_2(t)$ . This terminology is described graphically in Fig. 10.7.

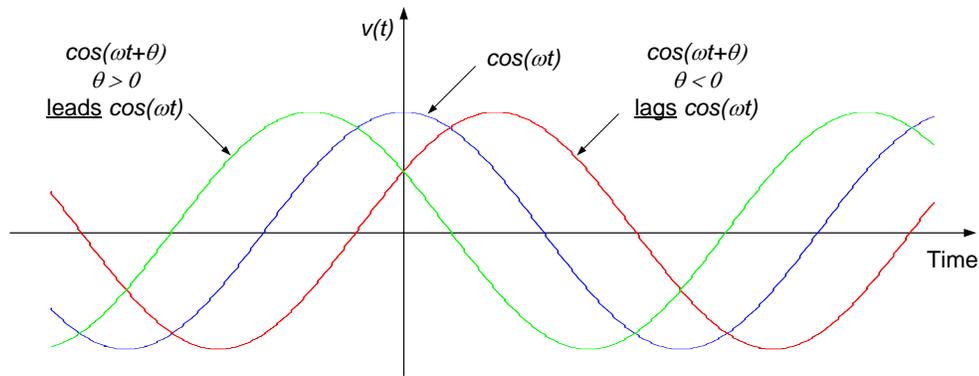


Figure 10.7. Leading and lagging sinusoids.

Finally, we note that the representation of sinusoidal signals as a phase shifted cosine function, as provided by equation (10.9), is completely general. If we are given a sinusoidal function in terms of a sine function, it can be readily converted to the form of equation (10.9) by subtracting a phase of  $\frac{\pi}{2}$  (or  $90^\circ$ ) from the argument, since:

$$\sin(\omega t) = \cos\left(\omega t - \frac{\pi}{2}\right)$$

Likewise, sign changes can be accounted for by a  $\pm\pi$  radian phase shift, since:

$$-\cos(\omega t) = \cos(\omega t \pm \pi)$$

Obviously, we could have chosen either a cosine or sine representation of a sinusoidal signal. We prefer the cosine representation, since a cosine is the real part of a complex exponential. In the next module, we will see that sinusoidal steady-state circuit analysis is simplified significantly by using complex exponentials to represent the sinusoidal functions. The cosine is the real part of a complex exponential (as we saw previously in chapter 8). Since all measurable signals are real valued, we take the real part of our complex exponential-based result as our physical response; this results in a solution of the form of equation (10.9).

Since representation of sinusoidal waveforms as complex exponentials will become important to us in circuit analysis, we devote the following subsection to a review of complex exponentials and their interpretation as sinusoidal signals.

## 10.2.2 Complex Exponentials and Phasors

Euler's identity can be used to represent complex numbers as complex exponentials:

$$e^{j\theta} = \cos \theta \pm j \sin \theta \quad \text{Eq. 10.10}$$

If we generalize equation (9) to time-varying signals of arbitrary magnitude, we can write:

$$V_p e^{\pm j(\omega t + \theta)} = V_p \cos(\omega t + \theta) \pm j V_p \sin(\omega t + \theta) \quad \text{Eq. 10.11}$$

So that:

$$V_p \cos(\omega t + \theta) = \text{Re}\{V_p e^{\pm j(\omega t + \theta)}\} \quad \text{Eq. 10.12}$$

And:

$$V_p \sin(\omega t + \theta) = \text{Im}\{V_p e^{\pm j(\omega t + \theta)}\} \quad \text{Eq. 10.13}$$

Where  $Re\{V_p e^{\pm j(\omega t + \theta)}\}$  and  $Im\{V_p e^{\pm j(\omega t + \theta)}\}$  denote the real part of  $V_p e^{\pm j(\omega t + \theta)}$  and the imaginary part of  $V_p e^{\pm j(\omega t + \theta)}$ , respectively. The complex exponential of equation (10.11) can also be written as:

$$V_p e^{\pm j(\omega t + \theta)} = V_p e^{j\theta} e^{j\omega t} \quad \text{Eq. 10.14}$$

The term  $V_p e^{j\theta}$  on the right-hand side of equation (10.14) is simply a complex number which provides the magnitude and phase information of the complex exponential of equation (10.11). From equation (10.12), this magnitude and phase can be used to express the magnitude and phase angle of a sinusoidal signal of the form given in equation (10.9).

The complex number in polar coordinates which provides the magnitude and phase angle of a time-varying complex exponential, as given in equation (10.14) is called a *phasor*. The phasor representing  $V_p \cos(\omega t + \theta)$  is defined as:

$$\underline{V} = V_p e^{j\theta} = V_p \angle \theta \quad \text{Eq. 10.15}$$

We will use a capital letter with an underscore to denote a phasor. Using bold typeface to represent phasors is more common; our notation is simply for consistency between lecture material and written material – boldface type is difficult to create on a whiteboard during lecture!

**Note:** The phasor representing a sinusoid does not provide information about the frequency of the sinusoid – frequency information must be kept track of separately.

### 10.2.3 Complex Arithmetic Review

Much the material in this section has been provided previously in section 8.3. It is repeated here to emphasize its importance and to expand slightly upon some crucial topics.

In our presentation of complex exponentials, we first provide a brief review of complex numbers. A complex number contains both real and imaginary parts. Thus, we may write a complex number  $\underline{A}$  as:

$$\underline{A} = a + jb \quad \text{Eq. 10.16}$$

Where:

$$j = \sqrt{-1} \quad \text{Eq. 10.17}$$

And the underscore denotes a complex number. The complex number  $\underline{A}$  can be represented on orthogonal axes representing the real and imaginary part of the number, as shown in Fig. 10.8. (In Figure 10.8, we have taken the liberty of representing  $\underline{A}$  as a vector, although it is really just a number.) We can also represent the complex number in polar coordinates, also shown in Figure 10.8. The polar coordinates consist of a magnitude  $|A|$  and phase angle  $\theta_A$ , defined as:

$$|A| = \sqrt{a^2 + b^2} \quad \text{Eq. 10.18}$$

$$\theta_A = \tan^{-1} \left( \frac{b}{a} \right) \quad \text{Eq. 10.19}$$

Notice that the phase angle is defined counterclockwise from the positive real axis. Conversely, we can determine the rectangular coordinates from the polar coordinates from:

$$a = Re\{\underline{A}\} = |A| \cos(\theta_A) \quad \text{Eq. 10.20}$$

$$b = Im\{\underline{A}\} = |A| \sin(\theta_A) \quad \text{Eq. 10.21}$$

Where the notation  $Re\{\underline{A}\}$  and  $Im\{\underline{A}\}$  denote the real part of  $\underline{A}$  and the imaginary part of  $\underline{A}$ , respectively.

The polar coordinates of a complex number of  $\underline{A}$  are often represented in the form:

$$\underline{A}|A|\angle\theta_A \quad \text{Eq. 10.22}$$

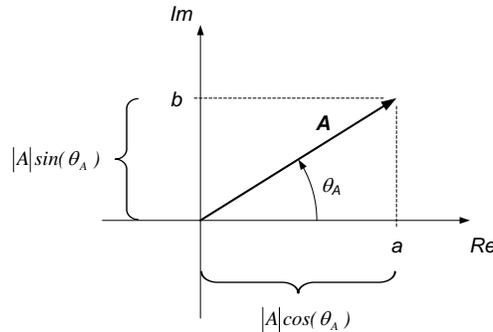


Figure 10.8. Representation of a complex number in rectangular and polar coordinates.

An alternate method of representing complex numbers in polar coordinates employs complex exponential notation. Without proof, we claim that:

$$e^{j\theta} = 1\angle\theta \quad \text{Eq. 10.23}$$

Thus,  $e^{j\theta}$  is a complex number with magnitude 1 and phase angle  $\theta$ . From Fig. 10.8, it is easy to see that this definition of the complex exponential agrees with Euler's equation:

$$e^{\pm j\theta} = \cos\theta \pm j \sin\theta \quad \text{Eq. 10.24}$$

With the definition of equation (10.23), we can define any arbitrary complex number in terms of complex numbers. For example, our previous complex number  $\underline{A}$  can be represented as:

$$\underline{A} = |A|e^{j\theta_A} \quad \text{Eq. 10.25}$$

We can generalize our definition of the complex exponential to time-varying signals. If we define a time varying signal  $e^{j\omega t}$ , we can use equation (10.24) to write:

$$e^{j\omega t} = \cos\omega t \pm j \sin\omega t \quad \text{Eq. 10.26}$$

The signal  $e^{j\omega t}$  can be visualized as a unit vector rotating around the origin in the complex plane; the tip of the vector scribes a unit circle with its center at the origin of the complex plane. This is illustrated in Fig. 10.9. The vector rotates at a rate defined by the quantity  $\omega$ —the vector makes one complete revolution every  $\frac{2\pi}{\omega}$  seconds.

The projection of this rotating vector on the real axis traces out the signal  $\cos\omega t$ , as shown in Fig. 10.7, while the projection of the rotating vector on the imaginary axis traces out the signal  $\sin\omega t$ , also shown in Fig. 10.9.

Thus, we interpret the complex exponential function  $e^{j\omega t}$  as an alternate “type” of sinusoidal signal. The real part of this function is  $\cos\omega t$  while the imaginary part of this function is  $\sin\omega t$ .

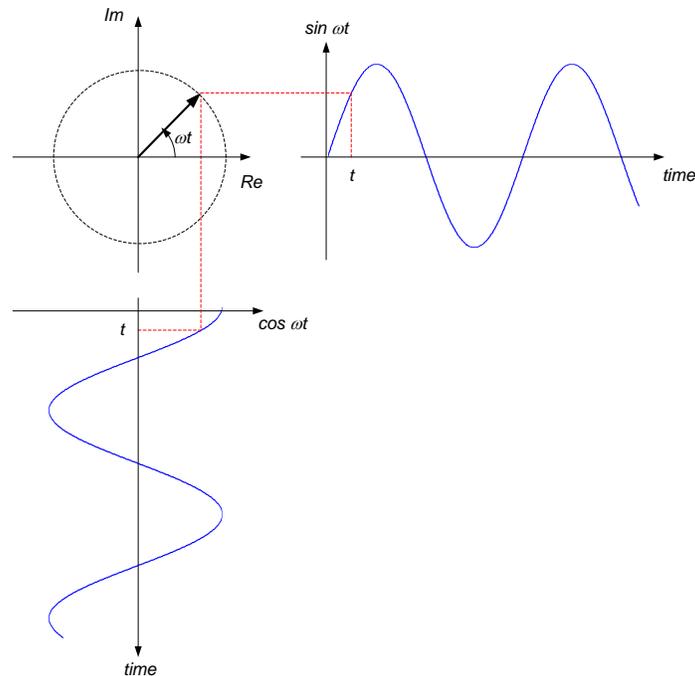


Figure 10.9. Illustration of  $e^{j\omega t}$ .

Addition and subtraction of complex numbers is most easily performed in rectangular coordinates. Given two complex numbers  $\underline{A}$  and  $\underline{B}$ , defined as:

$$\begin{aligned}\underline{A} &= a + jb \\ \underline{B} &= c + jd\end{aligned}$$

The sum and difference of the complex number can be determined by:

$$\underline{A} + \underline{B} = (a + c) + j(b + d)$$

And:

$$\underline{A} - \underline{B} = (a - c) + j(b - d)$$

Multiplication and division, on the other hand, are probably most easily performed using polar coordinates. If we define two complex numbers as:

$$\underline{A} = |A|e^{j\theta_A} = |A|\angle\theta_A$$

$$\underline{B} = |B|e^{j\theta_B} = |B|\angle\theta_B$$

The product and quotient can be determined by:

$$\underline{A} \cdot \underline{B} = |A|e^{j\theta_A} \cdot |B|e^{j\theta_B} = |A| \cdot |B|e^{j(\theta_A + \theta_B)} = |A| \cdot |B|\angle(\theta_A + \theta_B)$$

And:

$$\frac{\underline{A}}{\underline{B}} = \frac{|A|e^{j\theta_A}}{|B|e^{j\theta_B}} = \frac{|A|}{|B|}\angle(\theta_A - \theta_B)$$

The conjugate of a complex number, denoted by a  $*$ , is obtained by changing the sign on the imaginary part of the number. For example, if  $\underline{A} = a + jb = |A|e^{j\theta}$ , then:

$$\underline{A}^* = a - jb = |A|e^{-j\theta}$$

Conjugation does not affect the magnitude of the complex number, but it changes the sign on the phase angle. It is easy to show that:

$$\underline{A} \cdot \underline{A}^* = |A|^2$$

Several useful relationships between polar and rectangular coordinate representations of complex numbers are provided below. The reader is encouraged to prove any that are not self-evident.

$$\begin{aligned} j &= 1 \angle 90^\circ \\ -j &= 1 \angle -90^\circ \\ \frac{1}{j} &= -j = 1 \angle -90^\circ \\ 1 &= 1 \angle 0^\circ \\ -1 &= 1 \angle 180^\circ \end{aligned}$$

## Section Summary

- Periodic signals repeat themselves at a specific time interval. Sinusoidal signals are a special case of periodic signals.
- A sinusoidal signal can always be written in the form  $v(t) = V_p \cos(\omega t + \theta)$ .
- It is often convenient, when analyzing a system's steady-state response to sinusoidal inputs, to express sinusoidal signals in terms of complex exponentials. This is possible because of Euler's formula:

$$e^{\pm j\omega t} = \cos \omega t \pm j \sin \omega t$$

- From Euler's formula, a sinusoidal signal can be expressed as the real part of a complex exponential:

$$v(t) = V_p \cos(\omega t + \theta) = \text{Re}\{V_p e^{\pm j(\omega t + \theta)}\}$$

- The magnitude and phase angle of a complex exponential signal are conveniently expressed as a phasor:

$$\underline{V} = V_p e^{j\theta}$$

- Using phasor notation, the above complex exponential signal can be written as:

$$V_p e^{\pm j(\omega t + \theta)} = \underline{V} e^{j\omega t}$$

- Phasors can then be operated on arithmetically in the same way as any other complex number. However, when operating on phasors, keep in mind that you are dealing with the amplitude and phase angle of a sinusoidal signal.

## 10.2 Exercises

- Express the following complex numbers in rectangular form:

- 1.1.  $3e^{j45^\circ}$

- 1.2.  $5\sqrt{2}e^{j135^\circ}$

- 1.3.  $2.5e^{j90^\circ}$

- 1.4.  $6e^{j\pi}$

2. Express the following complex numbers in complex exponential form:
  - 2.1.  $2 - j2$
  - 2.2.  $-j3$
  - 2.3.  $6$
  - 2.4.  $3 + j$
  
3. Evaluate the following expressions. Express your results in complex exponential form.
  - 3.1.  $-\frac{j}{2(j+1)}$
  - 3.2.  $\frac{2-j2}{4+j4}$
  - 3.3.  $2e^{j45^\circ} \cdot \frac{2}{j+1}$
  - 3.4.  $j + \frac{2}{j}$
  
4. Represent the following sinusoids in phasor form:
  - 4.1.  $3 \cos(5t - 60^\circ)$
  - 4.2.  $-2 \cos(300t + 45^\circ)$
  - 4.3.  $\sin(6t)$
  - 4.4.  $7 \cos(3t)$
  
5. Write the signal representing the real part of the following complex exponentials:
  - 5.1.  $5\sqrt{2}e^{j(100t-45^\circ)}$
  - 5.2.  $3e^{j\pi}e^{j3t}$
  - 5.3.  $2e^{j(\pi t-30^\circ)} + 4e^{j(4t+20^\circ)}$

### 10.3 Sinusoidal Steady-state System Response

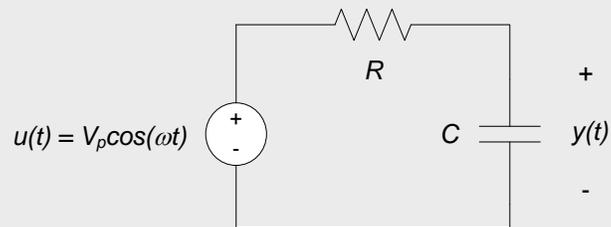
In this section, the concepts presented in sections 10.1 and 10.2 are used to determine the sinusoidal steady-state response of electrical circuits. We will develop sinusoidal steady-state circuit analysis in terms of examples, rather than attempting to develop a generalized approach à priori. The approach is straightforward, so that a general analysis approach can be inferred from the application of the method to several simple circuits.

The overall approach to introducing sinusoidal steady-state analysis techniques used in this section is as follow:

- We first determine the sinusoidal steady-state response of a simple RC circuit, by solving the differential equation governing the system. This results directly in a solution which is a function of time; it is a *time domain* analysis technique. The approach is mathematically tedious, even for the simple circuit being analyzed.
- We then re-analyze the same RC circuit using complex exponentials and phasors. This approach results in the transformation of the governing time domain differential equation into an algebraic equation which is a function of frequency. It is said to describe the circuit behavior in the *frequency domain*. The frequency domain equation governing the system is then solved using phasor techniques and the result transformed back to the time domain. This approach tends to be mathematically simpler than the direct solution of the differential equation in the time domain, though in later sections we will simplify the approach even further.
- Several other examples of sinusoidal steady-state circuit analysis are then performed using frequency domain techniques in order to demonstrate application of the approach to more complex circuits. It will be seen that, unlike time-domain analysis, the difficulty of the frequency domain analysis does not increase drastically as the circuit being analyzed becomes more complex.

### Example 10.2: RC Circuit Sinusoidal Steady-state Response via Time-domain Analysis

In the circuit below, the input voltage is  $u(t) = V_p \cos(\omega t)$  volts and the circuit response (or output) is the capacitor voltage,  $y(t)$ . We want to find the steady-state response (as  $t \rightarrow \infty$ ).



The differential equation governing the circuit is:

$$\frac{dy(t)}{dt} + \frac{1}{RC}y(t) = \frac{V_p}{RC \cos(\omega t)} \quad \text{Eq. 10.27}$$

Since we are concerned only with the steady-state response, there is no need to determine the homogeneous solution of the differential equation (or, equivalently, the natural response of the system) so we will not be concerned with the initial conditions on the system – their effect will have died out by the time we are interested in the response. Thus, we only need to determine the particular solution of the above differential equation (the forced response of the system). Since the input function is a sinusoid, the forced response must be sinusoidal, so we assume that the forced response  $y_f(t)$  has the form:

$$y_f(t) = A \cos(\omega t) + B \sin(\omega t) \quad \text{Eq. 10.28}$$

Substituting equation (10.28) into equation (10.27) results in:

$$-A\omega \sin(\omega t) + B\omega \cos(\omega t) + \frac{1}{RC} [A \cos(\omega t) + B \sin(\omega t)] = \frac{V_p}{RC} \cos(\omega t) \quad \text{Eq. 10.29}$$

Equating coefficients on the sine and cosine terms results in two equations in two unknowns:

$$-A\omega + \frac{B}{RC} = 0 \quad B\omega + \frac{A}{RC} = \frac{V_p}{RC} \quad \text{Eq. 10.30}$$

Solving equations (10.30) results in:

$$A = \frac{V_p}{1+(\omega RC)^2} \quad B = \frac{V_p \omega RC}{1+(\omega RC)^2} \quad \text{Eq. 10.31}$$

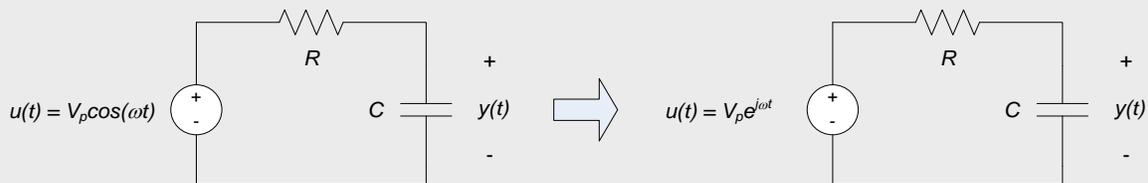
Substituting equations (10.31) into equation (10.28) and using the trigonometric identity  $A \cos(\omega t) + B \sin(\omega t) = \sqrt{A^2 + B^2} \cos\left[\omega t - \tan^{-1}\left(\frac{B}{A}\right)\right]$  results in (after some fairly tedious algebra):

$$y_f(t) = \frac{V_p}{\sqrt{1+(\omega RC)^2}} \cos[\omega t - \tan^{-1}(\omega RC)] \quad \text{Eq. 10.32}$$

**Note:** In all steps of the above analysis, the functions being used are functions of time. That is, for a particular value of  $\omega$ , the functions vary with time. The above analysis is being performed in the *time domain*.

### Example 10.3: RC Circuit Sinusoidal Steady-state Response via Frequency-domain Analysis

We now repeat Example 10.2, using phasor-based analysis techniques. The circuit being analyzed is shown in the figure to the left below for reference; the input voltage is  $u(t) = V_p \cos(\omega t)$  volts and the circuit response (or output) is the capacitor voltage,  $y(t)$ . We still want to find the steady-state response (as  $t \rightarrow \infty$ ). In this example, we replace the physical input,  $u(t) = V_p \cos(\omega t)$ , with a conceptual input based on a complex exponential as shown in the figure to the right below. The complex exponential input is chosen such that the real part of the complex input is equivalent to the physical input applied to the circuit. We will analyze the conceptual circuit with the complex valued input.



The differential equation governing the circuit above is the same as in example 10.2, but with the complex input:

$$\frac{dy(t)}{dt} + \frac{1}{RC}y(t) = \frac{V_P}{RC}e^{j\omega t} \quad \text{Eq. 10.33}$$

As in example 10.2, we now assume a form of the forced response. In this case, however, our solution will be assumed to be a complex exponential:

$$y(t) = |Y|e^{j(\omega t + \theta)} \quad \text{Eq. 10.34}$$

Which can be written in phasor form as:

$$y(t) = \underline{Y}e^{j\omega t} \quad \text{Eq. 10.35}$$

Where the phasor  $\underline{Y}$  is a complex number which can be expressed in either exponential or polar form:

$$y(t) = \underline{Y}e^{j\omega t} \quad \text{Eq. 10.36}$$

Substituting (10.35) into equation (10.33) and taking the appropriate derivative results in:

$$j\omega \underline{Y}e^{j\omega t} + \frac{1}{RC} \underline{Y}e^{j\omega t} = \frac{V_P}{RC} e^{j\omega t} \quad \text{Eq. 10.37}$$

We can divide equation (10.37) by  $e^{j\omega t}$  to obtain:

$$j\omega \underline{Y} + \frac{1}{RC} \underline{Y} = \frac{V_P}{RC} \quad \text{Eq. 10.38}$$

Equation (10.38) can be solved for  $\underline{Y}$ :

$$\left(j\omega + \frac{1}{RC}\right) \underline{Y} = \frac{V_P}{RC} \Rightarrow \underline{Y} = \frac{\frac{V_P}{RC}}{j\omega + \frac{1}{RC}} \quad \text{Eq. 10.39}$$

So that:

$$\underline{Y} = \frac{V_P}{1 + j\omega RC} \quad \text{Eq. 10.40}$$

The magnitude and phase of the output response can be determined from the phasor  $\underline{Y}$ :

$$|Y| = \frac{V_P}{\sqrt{1+(\omega RC)^2}}$$

$$\angle_Y = -\tan^{-1}(\omega RC) \quad \text{Eq. 10.41}$$

The complex exponential form of the system response is then, from equation (10.35):

$$y(t) = \frac{V_P}{\sqrt{1+(\omega RC)^2}} e^{j(\omega t - \tan^{-1}(\omega RC))} \quad \text{Eq. 10.42}$$

Since our physical input is the real part of the conceptual input, and since all circuit parameters are real valued, our physical output is the real part of equation (10.42) and the forced response is:

$$y_f(t) = \frac{V_P}{\sqrt{1+(\omega RC)^2}} \cos[\omega t - \tan^{-1}(\omega RC)] \quad \text{Eq. 10.43}$$

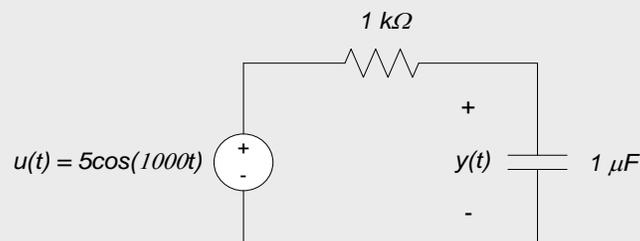
Which agrees with our result from the time-domain analysis of example 10.2.

#### Notes:

- The transition from equation (10.37) to equation (10.38) removed the time-dependence of our solution. The solution is now no longer a function of time! The solution includes the phasor representations of the input and output, as well as (generally) frequency. Thus, equation (10.38) is said to be in the *phasor domain* or, somewhat more commonly, the *frequency domain*. The analysis remains in the frequency domain until we reintroduce time in equation (10.43).
- Equations in the frequency domain are algebraic equations rather than differential equations. This is a significant advantage mathematically, especially for higher-order systems.
- Circuit components must have purely real values for the above process to work. We do not prove this, but merely make the claim that the process of taking the real part of the complex exponential form of the system response is not valid if circuit components (or any coefficients in the differential equation governing the system) are complex valued. Fortunately, this is not a strong restriction – complex values do not exist in the physical world.
- The complex exponential we use for our “conceptual” input,  $V_P e^{j\omega t}$ , is not physically realizable. That is, we cannot create this signal in the real world. It is a purely mathematical entity which we introduce solely for the purpose of simplifying the analysis. The complex form of the output response given by equation (10.42) is likewise not physically realizable.

### Example 10.4: Numerical Example and Phasor Diagrams

We now examine the circuit shown below. This circuit is simply the circuit of Example 10.3, with  $R = 1k\Omega$ ,  $C = 1\mu F$ ,  $V_P = 5V$ , and  $\omega = 1000$  rad/second.



In phasor form, the input is  $u(t) = \underline{U}e^{j1000t}$ , so that the phasor  $\underline{U}$  is  $\underline{U} = 5e^{j0^\circ} = 5\angle 0^\circ$ .

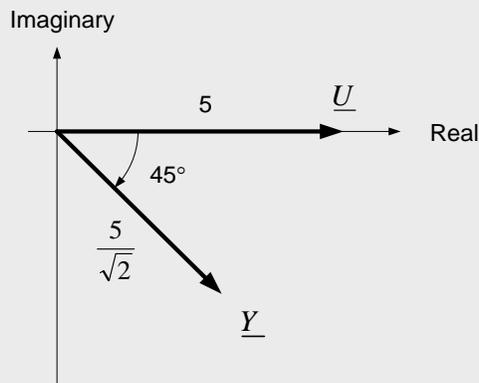
The phasor form of the output is given by equations (10.41):

$$|Y| = \frac{V_P}{\sqrt{1 + (\omega RC)^2}} = \frac{5}{\sqrt{1 + (1000 \cdot 1000 \cdot 1 \times 10^{-6})}} = \frac{5}{\sqrt{2}}$$

$$\angle \theta_T = -\tan^{-1}(\omega RC) = -\tan^{-1}(1000 \cdot 1000 \cdot 1 \times 10^{-6}) = -\frac{\pi}{4} = -45^\circ$$

And the phasor  $\underline{Y}$  can be written as  $\underline{Y} = \frac{5}{\sqrt{2}} e^{-j45^\circ} = \frac{5}{\sqrt{2}} \angle -45^\circ$

We can create a phasor diagram of the input phasor  $\underline{U}$  and the output phasor  $\underline{Y}$

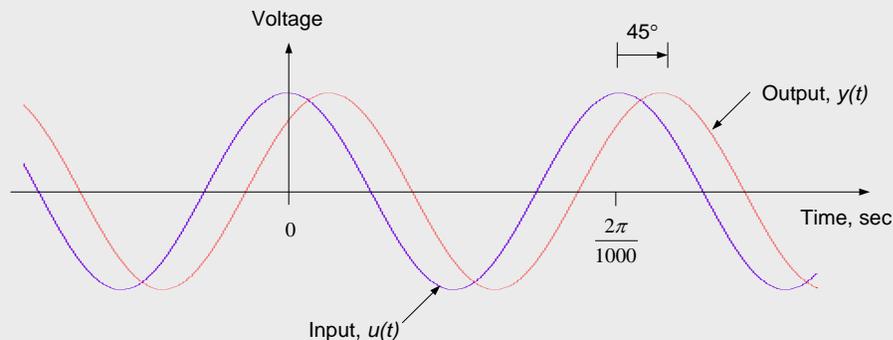


The phasor diagram shows the input and output phasors in the complex plane. The magnitudes of the phasors are typically labeled on the diagram, as is the phase difference between the two phasors. Note that since the phase difference between  $\underline{Y}$  and  $\underline{U}$  is negative, the output  $y(t)$  lags the input  $u(t)$ .

The time-domain form of the output is:

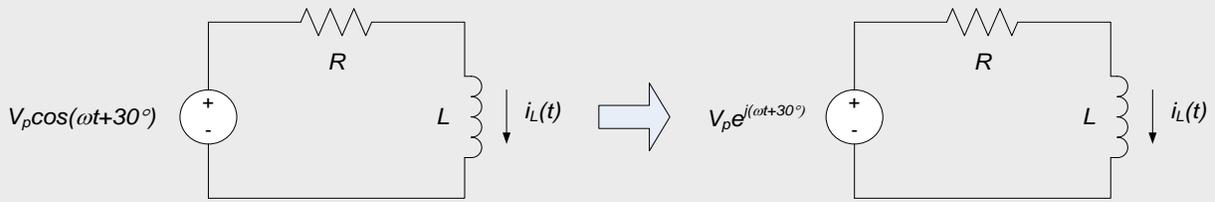
$$y(t) = \frac{5}{\sqrt{2}} \cos(1000t - 45^\circ)$$

A time-domain plot of the input and output are shown below. This plot emphasizes that the output lags the input, as indicated by our phasor diagram. The plot below replicates what would be seen from a measurement of the input and output voltages.



### Example 10.5: RL Circuit Sinusoidal Steady-state Response

In the circuit to the left below, the input voltage is  $V_P \cos(\omega t + 30^\circ)$  volts and the circuit response (or output) is the inductor current,  $i_L(t)$ . We want to find the steady-state response  $i_L(t \rightarrow \infty)$ .



The differential equation governing the circuit can be determined by applying KVL around the single loop:

$$L \frac{di_L(t)}{dt} + Ri_L(t) = u(t) \tag{Eq. 10.44}$$

We apply the conceptual input,  $u(t) = V_p e^{j(\omega t + 30^\circ)}$  as shown in the figure to the right above to this equation. We can represent this input in phasor form as:

$$u(t) = \underline{U} e^{j\omega t} \tag{Eq. 10.45}$$

Where the phasor  $\underline{U} = V_p \angle 30^\circ$ . Likewise, we represent the output in phasor form:

$$i_L(t) = \underline{I}_L e^{j\omega t} \tag{Eq. 10.46}$$

Where the phasor  $\underline{I}_L = |I_L| \angle \theta$ .

Substituting our assumed input and output in phasor form into equation (10.44) results in:

$$Lj\omega \underline{I}_L e^{j\omega t} + R \underline{I}_L e^{j\omega t} = \underline{U} e^{j\omega t} \tag{Eq. 10.47}$$

As in Example 10.4, we divide through by  $e^{j\omega t}$  to obtain the frequency domain governing equation:

$$Lj\omega \underline{I}_L + R \underline{I}_L = \underline{U} \tag{Eq. 10.48}$$

So that:

$$\underline{I}_L = \frac{\underline{U}}{R + j\omega L} = \frac{V_p \angle 30^\circ}{R + j\omega L} \tag{Eq. 10.49}$$

So that the phasor  $\underline{I}_L$  has magnitude and phase:

$$\begin{aligned} |I_L| &= \frac{V_p}{\sqrt{R^2 + (\omega L)^2}} \\ \theta &= 30^\circ - \tan^{-1}\left(\frac{\omega L}{R}\right) \end{aligned} \tag{Eq. 10.50}$$

The exponential form of the inductor current is therefore:

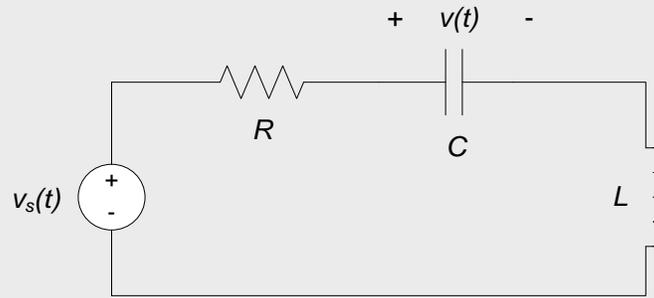
$$i_L(t) = \frac{V_p}{\sqrt{R^2 + (\omega L)^2}} e^{j[\omega t + 30^\circ - \tan^{-1}(\frac{\omega L}{R})]} \tag{Eq. 10.51}$$

And the actual physical inductor current is:

$$i_L(t) = \frac{V_p}{\sqrt{R^2 + (\omega L)^2}} \cos \left[ \omega t + 30^\circ - \tan^{-1} \left( \frac{\omega L}{R} \right) \right] \tag{Eq. 10.52}$$

### Example 10.6: Series RLC Circuit Sinusoidal Steady-state Response

Consider the circuit shown below. The input to the circuit is  $v_s(t) = 2 \cos(\omega t)$  volts. Find the output  $v(t)$ .



In section 8.1, it was determined that the differential equation governing the system is:

$$\frac{d^2v(t)}{dt^2} + \frac{R}{L} \frac{dv(t)}{dt} + \frac{1}{LC} v(t) = \frac{1}{LC} v_s(t) \quad \text{Eq. 10.53}$$

Assuming that the input is a complex exponential whose real part is the given  $v_s(t)$  provides:

$$v_s(t) = 2e^{j\omega t} \quad \text{Eq. 10.54}$$

The output is assumed to have the phasor form:

$$v(t) = \underline{V} e^{j\omega t} \quad \text{Eq. 10.55}$$

Where  $\underline{V}$  contains the (unknown) magnitude and phase of the output voltage. Substituting equations (10.54) and (10.55) into equation (10.53) results in:

$$-(j\omega)^2 \underline{V} e^{j\omega t} + \frac{R}{L} (j\omega) \underline{V} e^{j\omega t} + \frac{1}{LC} \underline{V} e^{j\omega t} = \frac{1}{LC} 2e^{j\omega t} \quad \text{Eq. 10.56}$$

Dividing through by  $e^{j\omega t}$  and noting that  $j^2 = -1$ , results in:

$$\left[ \frac{1}{LC} - \omega^2 + j \frac{R}{L} \omega \right] \underline{V} = \frac{2}{LC}$$

So that:

$$\underline{V} = \frac{\frac{2}{LC}}{\frac{1}{LC} - \omega^2 + j \frac{R}{L} \omega} \quad \text{Eq. 10.57}$$

The magnitude and phase of  $\underline{V}$  are:

$$|V| = \frac{\frac{2}{LC}}{\sqrt{\left(\frac{1}{LC} - \omega^2\right)^2 + \left(\frac{R}{L} \omega\right)^2}}$$

$$\angle \underline{V} = -\tan^{-1} \left( \frac{\frac{R\omega}{L}}{\frac{1}{LC} - \omega^2} \right)$$

And the capacitor voltage is:

$$v(t) = \frac{\frac{2}{LC}}{\sqrt{\left(\frac{1}{LC} - \omega^2\right)^2 + \left(\frac{R}{L} \omega\right)^2}} \cos \left\{ \omega t - \tan^{-1} \left( \frac{\frac{R\omega}{L}}{\frac{1}{LC} - \omega^2} \right) \right\} \quad \text{Eq. 10.58}$$

The complex arithmetic in this case becomes a bit tedious, but the complexity of the frequency-domain approach is nowhere near that of the time-domain solution of the second-order differential equation.

## Section Summary

- The steady-state response of a linear time invariant system to a sinusoidal input is a sinusoid with the same frequency as the input sinusoid. Only the amplitude and phase angle of the output sinusoid can be different from the input sinusoid, so the solution is entirely characterized by the magnitude and phase angle of the output sinusoid.
- The steady-state response of a system to a sinusoidal input can be determined by assuming a form of the solution, substituting the input signal and the output signal into the governing differential equation and solving for the amplitude and phase angle of the output sinusoid.
- The solution approach is simplified if the sinusoidal signals are represented as complex exponentials. The approach is further simplified if these complex exponentials are represented in phasor form – the phasor is a complex number which provides the amplitude and phase angle of the complex exponential.
- The above solution approaches convert the governing differential equation into an algebraic equation. If complex exponentials in phasor form are used to represent the signals of interest, the governing algebraic equation can have complex coefficients.
- The relationships between the steady state sinusoidal inputs and outputs are described by a relationship between the amplitudes (generally a ratio between the output amplitude and the input amplitude) and the phase angles (generally a difference between the output and input phase angles).
  - These relationships are often displayed graphically in a phasor diagram.

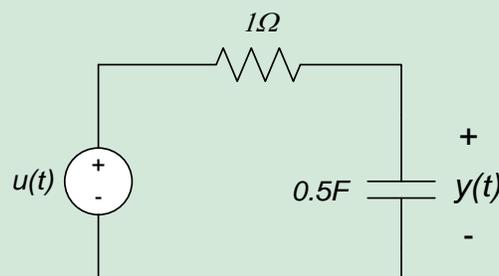
## 10.3 Exercises

1. The differential equation governing a circuit is:

$$2 \frac{dy(t)}{dt} + 6y(t) = u(t)$$

Where  $u(t)$  is the input and  $y(t)$  is the output. Determine the steady-state response of the circuit to an input  $u(t) = 2 \cos(3t)$ .

2. For the circuit shown below,  $u(t)$  is the input and  $y(t)$  is the output.
  - a. Write the differential equation relating  $u(t)$  and  $y(t)$ .
  - b. Determine  $y(t)$ ,  $t \rightarrow \infty$ , if  $u(t) = 3 \cos(2t)$



## 10.4 Phasor Representations of Circuit Elements

In section 10.3, we determined the sinusoidal steady-state response of an electrical circuit by transforming the circuit's governing differential equation into the *frequency domain* or *phasor domain*. This transformation converted the differential equation into an algebraic equation. This conversion significantly simplified the

subsequent circuit analysis, at the relatively minor expense of performing some complex arithmetic. In this module, we will further simplify this analysis by transforming the circuit itself directly into the frequency domain and writing the governing algebraic equations directly.

This approach eliminates the necessity of ever writing the differential equation governing the circuit (as long as we are only interested in the circuit's sinusoidal steady-state response). This approach also allows us to apply analysis techniques previously used only for purely resistive circuits to circuits containing energy storage elements.

### 10.4.1 Phasor Domain Voltage-current Relationships

In section 10.2, we introduced phasors as a method for representing sinusoidal signals. Phasors provide the magnitude and phase of the sinusoid. For example, the signal  $v(t) = V_p \cos(\omega t + \theta)$  has amplitude  $V_p$  and the phase angle  $\theta$ . This information can be represented in phasor form as:

$$\underline{V} = V_p e^{j\theta}$$

In which complex exponentials are used to represent the phase. Equivalently, the phase can be represented as an angle, and the phasor form of the signal can be written as:

$$\underline{V} = V_p \angle \theta$$

Note that the phasor does not provide the frequency of the signal,  $\omega$ . To include frequency information, the signal is typically written in complex exponential form as:

$$v(t) = \underline{V} e^{j\omega t}$$

In section 10.3, we used phasor representations to determine the steady-state sinusoidal response of electrical circuits by representing the signals of interest as complex exponentials in phasor form. When signals in the governing differential equation are represented in this form, the differential equation becomes an algebraic equation, resulting in a significant mathematical simplification. In section 10.3, it was also noted that the mathematics could be simplified further by representing the circuit itself directly in the phasor domain. In this section, we present the phasor form of voltage-current relations for our basic circuit elements: resistors, inductors, and capacitors. The voltage-current relations for these elements are presented individually in the following subsections.

### 10.4.2 Resistors

The voltage-current relationship for resistors is provided by Ohm's Law:

$$v(t) = R \cdot i(t) \tag{Eq. 10.59}$$

If the voltage and current are represented in phasor form as:

$$v(t) = \underline{V} e^{j\omega t} \tag{Eq. 10.60}$$

And:

$$i(t) = \underline{I} e^{j\omega t} \tag{Eq. 10.61}$$

Equation (10.59) can be written:

$$\underline{V} e^{j\omega t} = R \cdot \underline{I} e^{j\omega t} \tag{Eq. 10.62}$$

Cancelling the  $e^{j\omega t}$  term from both sides results in:

$$\underline{V} = R \cdot \underline{I}$$

Eq. 10.63

The voltage-current relationship for resistors (Ohm’s Law) is thus identical in the time and frequency domains. Schematically, the time- and frequency-domain representations of a resistor are as shown in Fig. 10.10.

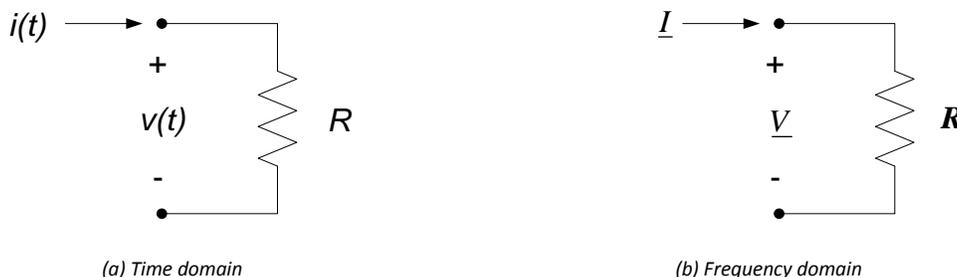


Figure 10.10. Voltage-current relations for a resistor.

Equation (10.63) shows that, in the frequency domain, the voltage and current in a resistor are related by a purely real, constant multiplicative factor. Thus, the sinusoidal voltage and current for a resistor are simply scaled versions of one another – there is no phase difference in the voltage and current for a resistor. This is shown graphically in Fig. 10.11.

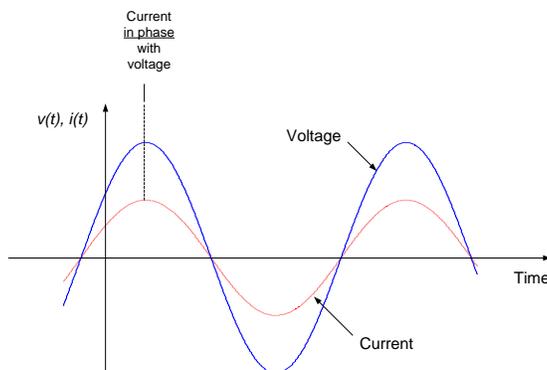


Figure 10.11. Voltage and current waveforms for a resistor.

A representative phasor diagram of the resistor’s voltage and current will appear as shown in Fig. 10.12 – the phasors representing voltage and current will always be in the same direction, though their lengths will typically be different.

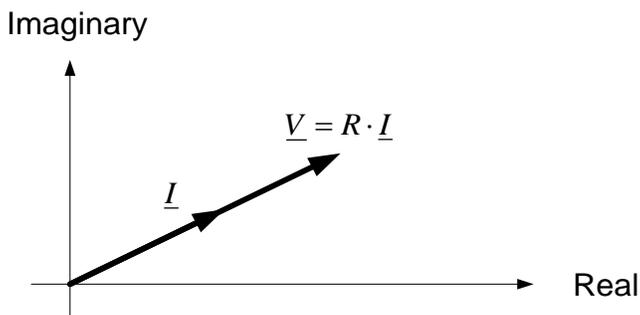


Figure 10.12. Voltage-current phasor diagram for resistor.

### 10.4.3 Inductors

The voltage-current relationship for inductors is:

$$v(t) = L \cdot \frac{di(t)}{dt} \quad \text{Eq. 10.64}$$

As with the resistive case presented above, we assume that the voltage and current are represented in phasor form as  $v(t) = \underline{V}e^{j\omega t}$  and  $i(t) = \underline{I}e^{j\omega t}$ , respectively. Substituting these expressions into equation (10.64) results in:

$$\underline{V}e^{j\omega t} = L \cdot \frac{d}{dt} [\underline{I}e^{j\omega t}] = L(j\omega)\underline{I}e^{j\omega t} \quad \text{Eq. 10.65}$$

Dividing equation (10.65) by  $e^{j\omega t}$  and re-arranging terms slightly results in the phasor domain or frequency domain representation of the inductor's voltage-current relationship:

$$\underline{V} = j\omega L \cdot \underline{I} \quad \text{Eq. 10.66}$$

In the frequency domain, therefore, the inductor's phasor voltage is proportional to its phasor current. The constant of proportionality is, unlike the case of the resistor, an imaginary number and is a function of the frequency,  $\omega$ . It is important to note that the differential relationship of equation (10.64) has been replaced with an algebraic voltage-current relationship. Schematically, the time- and frequency-domain representations of an inductor are as shown in Fig. 10.13.

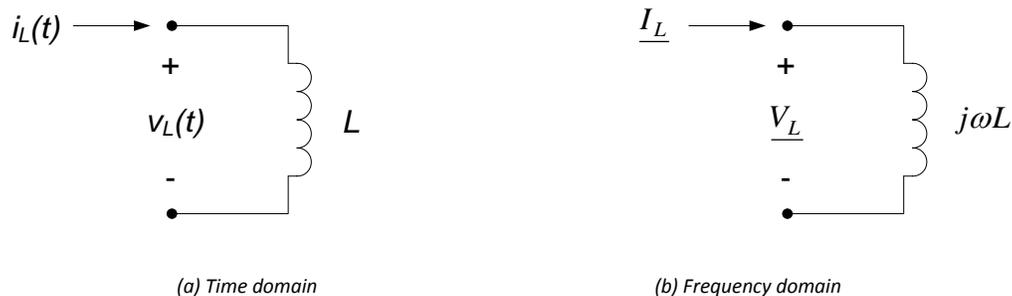


Figure 10.13. Inductor voltage-current relations.

The factor of  $j$  in the voltage-current relationship of equation (10.66) introduces a  $90^\circ$  phase shift between inductor voltage and current. Since  $j = e^{j90^\circ}$ , the voltage across an inductor leads the current by  $90^\circ$  (or, equivalently, the current lags the voltage by  $90^\circ$ ). The relative phase difference between inductor voltage and current are shown graphically in the time domain in Fig. 10.14. A representative phasor diagram of the inductor's voltage and current will appear as shown in Fig. 10.15 – the voltage phasor will always lead the current phasor by  $90^\circ$ , and the length of the voltage phasor will be a factor of  $\omega L$  times the length of the current phasor.

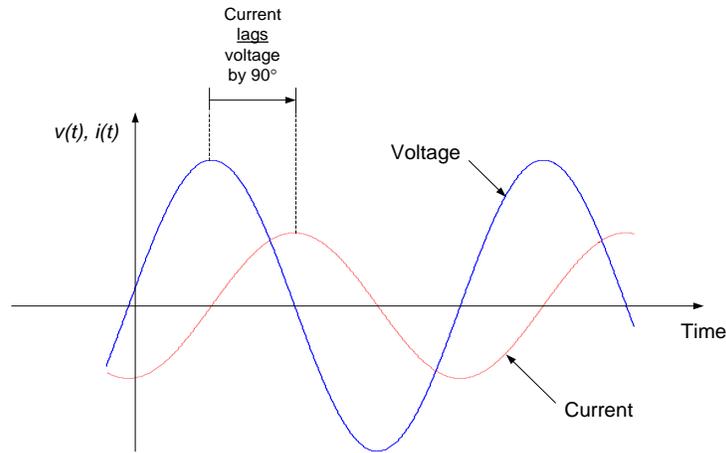


Figure 10.14. Voltage and current waveforms for an inductor.

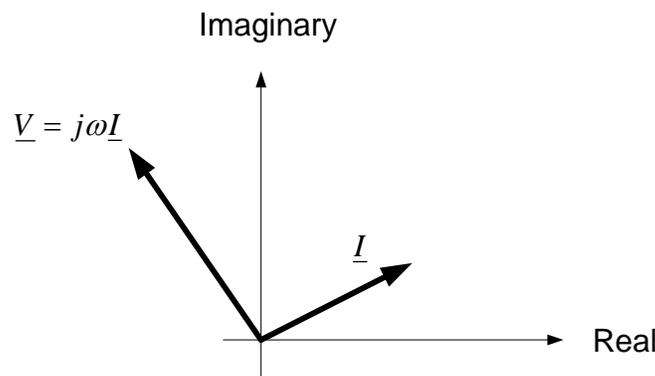


Figure 10.15. Voltage-current phasor diagram for inductor.

### 10.3.4 Capacitors

The voltage-current relationship for capacitors is:

$$i(t) = C \cdot \frac{dv(t)}{dt} \tag{Eq. 10.67}$$

As with the previous cases, we assume that the voltage and current are represented in phasor form as  $v(t) = \underline{V}e^{j\omega t}$  and  $i(t) = \underline{I}e^{j\omega t}$ , respectively. Substituting these expressions into equation (10.67) results in:

$$\underline{I}e^{j\omega t} = C \cdot \frac{d}{dt} [\underline{V}e^{j\omega t}] = C(j\omega)\underline{V}e^{j\omega t} \tag{Eq. 10.68}$$

Dividing the above by  $e^{j\omega t}$  results in the phasor domain or frequency domain representation of the capacitor's voltage-current relationship:

$$\underline{I} = j\omega C \cdot \underline{V} \tag{Eq. 10.69}$$

To be consistent with our voltage-current relationship for resistors and capacitors, we write the voltage in terms of the current. Thus,

$$\underline{V} = \frac{1}{j\omega C} \cdot \underline{I} \tag{Eq. 10.70}$$

In the frequency domain, therefore, the capacitor’s phasor voltage is proportional to its phasor current. The constant of proportionality is an imaginary number and is a function of the frequency,  $\omega$ . As with inductors, the differential voltage-current relationship has been replaced with an algebraic relationship. Schematically, the time- and frequency-domain representations of a capacitor are as shown in Fig. 10.16.

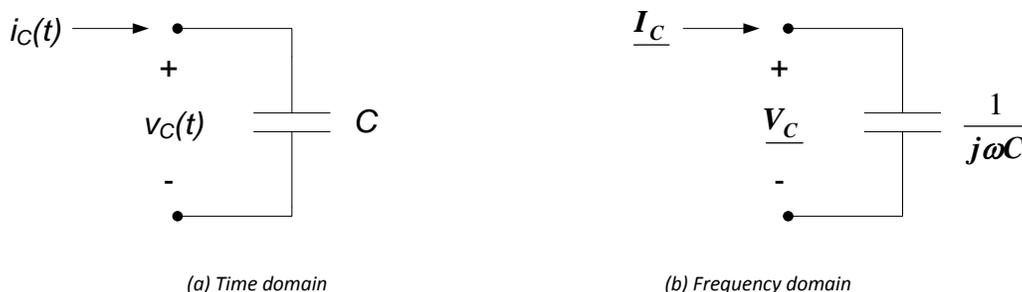


Figure 10.16. Capacitor voltage-current relations.

The factor of  $\frac{1}{j}$  in the voltage-current relationship of equation (10.70) introduces a  $90^\circ$  phase shift between inductor voltage and current. Since  $\frac{1}{j} = e^{-j90^\circ} = 1\angle -90^\circ$ , the voltage across a capacitor lags the current by  $90^\circ$  (or, equivalently, the current leads the voltage by  $90^\circ$ ). The relative phase difference between capacitor voltage and current are shown graphically in the time domain in Fig. 10.17. A representative phasor diagram of the capacitor’s voltage and current will appear as shown in Fig. 10.18 – the voltage phasor will always lag the current phasor by  $90^\circ$ , and the length of the voltage phasor will be a factor of  $\frac{1}{\omega C}$  times the length of the current phasor.

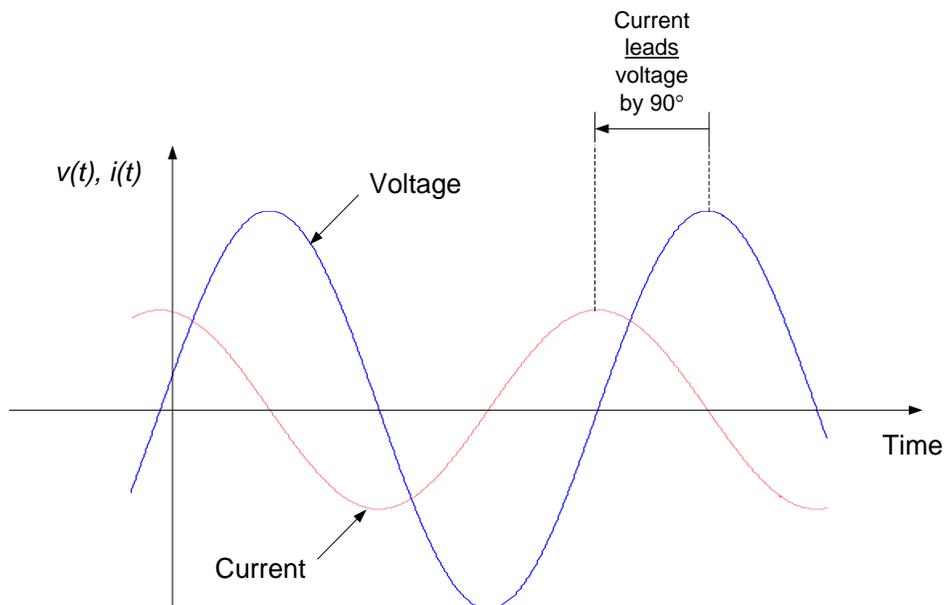


Figure 10.17. Voltage and current waveforms for a capacitor.

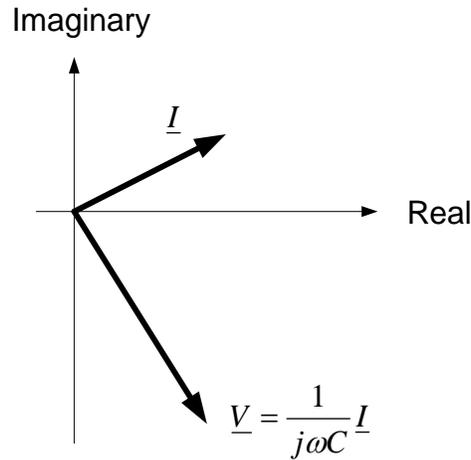


Figure 10.18. Voltage-current phasor diagram for capacitor.

### 10.3.5 Impedance and Admittance

The frequency domain voltage-current characteristics presented in the previous subsections indicate that the voltage difference across a circuit element can be written in terms of a multiplicative factor (which can be a complex number) times the current through the element. In order to generalize and formalize this concept, we define *impedance* as the ratio of phasor voltage to phasor current. Impedance is typically denoted as  $\underline{Z}$  and is defined mathematically as:

$$\underline{Z} = \frac{\underline{V}}{\underline{I}} \quad \text{Eq. 10.71}$$

Therefore, if the phasor voltage and current for a circuit element are given by:

$$\underline{V} = V_P e^{j\theta}$$

And:

$$\underline{I} = I_P e^{j\phi}$$

Then the impedance is:

$$\underline{Z} = \frac{\underline{V}}{\underline{I}} = \frac{V_P}{I_P} e^{j(\theta_Z)} \quad \text{Eq. 10.72}$$

Or alternatively,

$$\underline{Z} = \frac{V_P}{I_P} \angle \theta_Z \quad \text{Eq. 10.73}$$

Where  $\theta_Z$  is the angle of  $\underline{Z}$ . The magnitude of the impedance is the ratio of the magnitude of the voltage to the magnitude of the current:

$$|\underline{Z}| = \frac{V_P}{I_P} = \frac{|\underline{V}|}{|\underline{I}|} \quad \text{Eq. 10.74}$$

And the angle of the impedance is the difference between the voltage phase angle and the current phase angle:

$$\theta_Z = \angle \underline{Z} = \angle \underline{V} - \angle \underline{I} = \theta - \phi \quad \text{Eq. 10.75}$$

The impedance can also be represented in rectangular coordinates as:

$$\underline{Z} = R + jX$$

Eq. 10.76

Where  $R$  is the real part of the impedance (called the *resistance* or the *resistive component* of the impedance) and  $X$  is the imaginary part of the impedance (called the *reactance* or the *reactive part* of the impedance).  $R$  and  $X$  are related to  $|\underline{Z}|$  and  $\angle \underline{Z}$  by the usual rules relating rectangular and polar coordinates, so that:

$$|\underline{Z}| = \sqrt{R^2 + X^2}$$

$$\angle \underline{Z} = \tan^{-1} \left( \frac{X}{R} \right)$$

And:

$$R = \operatorname{Re}\{\underline{Z}\} = |\underline{Z}| \cos \theta_Z$$

$$X = \operatorname{Im}\{\underline{Z}\} = |\underline{Z}| \sin \theta_Z$$

Impedance is an extremely useful concept, in that it can be used to represent the voltage-current relations for any two-terminal electrical circuit element in the frequency domain, as indicated in Fig. 10.19.

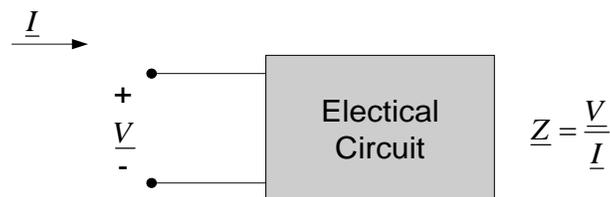


Figure 10.19. Impedance representation of two-terminal electric circuit.

The *admittance*,  $\underline{Y}$ , is defined as the reciprocal of impedance:

$$\underline{Y} = \frac{1}{\underline{Z}}$$
Eq. 10.77

Admittance is also a complex number, and is written in rectangular coordinates as:

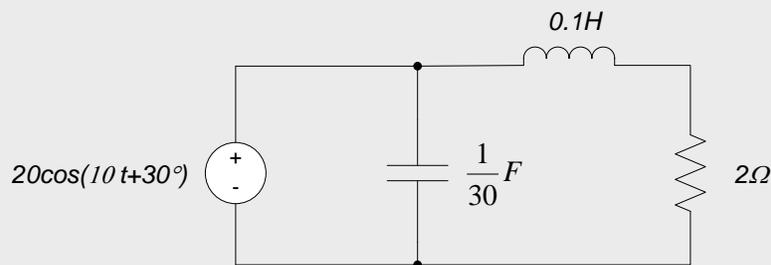
$$\underline{Y} = G + jB$$
Eq. 10.78

Where  $G$  is called the *conductance* and  $B$  is the *susceptance*.

Impedances and admittances for the three electrical circuit elements presented previously in this section are provided in Table 10.1 below. These results are readily obtained from the previously presented phasor domain voltage-current relationships and the definitions of impedance and admittance. The relations provided in Table 10.1 should be committed to memory.

Element	Impedance	Admittance
Resistor	$R$	$\frac{1}{R}$
Inductor	$j\omega L$	$\frac{1}{j\omega L}$
Capacitor	$\frac{1}{j\omega C}$	$j\omega C$

Table 10.1. Impedances and admittances for passive circuit elements.

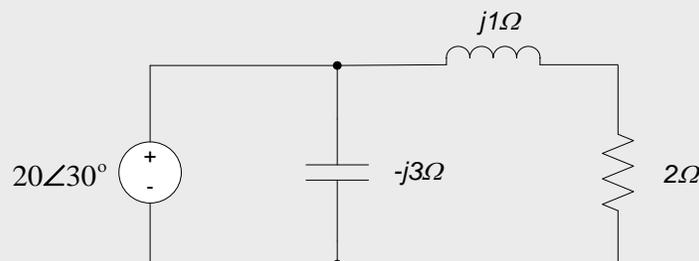
**Example 10.7: Provide the Phasor-domain Representation of the Circuit Below**

The input amplitude is 20 volts, and the input phase is  $30^\circ$ , so the phasor representation of the input voltage is  $20\angle 30^\circ$ .

The frequency of the input voltage is  $\omega = 10 \text{ rad/sec}$ . Thus, the impedances of the passive circuit elements are as follows:

- Resistor:  $\underline{Z} = R = 2\Omega$
- Inductor:  $\underline{Z} = j\omega L = j(10 \text{ rad/sec})(0.1H) = j1\Omega$
- Capacitor:  $\underline{Z} = \frac{1}{j\omega C} = \frac{1}{j(10 \text{ rad/sec})(\frac{1}{30}F)} = \frac{3}{j}\Omega = -j3\Omega$

The phasor-domain circuit is shown below.

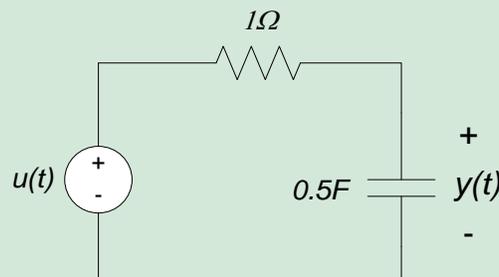
**Section Summary**

- Voltage-current relations for our passive circuit elements in the frequency domain are:
  - Resistor:  $\underline{V} = R\underline{I}$
  - Inductor:  $\underline{V} = j\omega L \cdot \underline{I}$
  - Capacitor:  $\underline{V} = \frac{1}{j\omega C} \cdot \underline{I}$

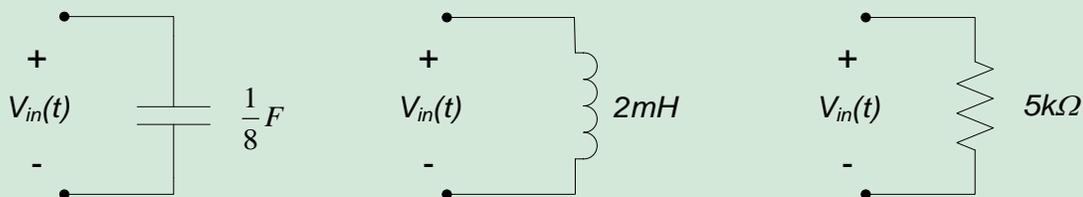
- The impedance of a circuit element is the ratio of the phasor voltage to the phasor current in that element:
  - Resistor:  $\underline{Z} = R$
  - Inductor:  $\underline{Z} = j\omega L$
  - Capacitor:  $\underline{Z} = \frac{1}{j\omega C}$
- Impedance is, in general, a complex number. Units of impedance are ohms ( $\Omega$ ). The real part of impedance is the *resistance*. The imaginary part of impedance is *reactance*. Impedance, for general circuit elements, plays the same role as resistance does for resistive circuit elements. In fact, for purely resistive circuit elements, impedance is simply the resistance of the element.
- Admittance is the inverse of impedance.
- Admittance is, in general, a complex number. The real part of admittance is *conductance*. The imaginary part of admittance is *susceptance*. For purely resistive circuits, admittance is the same as conductance.
- Impedance and admittance are, in general, functions of frequency.
- Impedance and admittance are not phasors. They are complex numbers – there is no sinusoidal time domain function corresponding to impedance or admittance. (Phasors, by definition, are a way to describe a time-domain sinusoidal function.)

## 10.4 Exercises

1. For the circuit shown below,  $u(t)$  is the input and  $y(t)$  is the output. Determine  $y(t)$ ,  $t \rightarrow \infty$ , if  $u(t) = 3\cos(2t)$ .



2. Sketch a diagram of the input and output phasors for exercise 1 above.
3. Determine the impedance of the circuit elements shown below if  $V_{in}(t) = 2\cos(4t)$ .



4. Determine the impedance of the circuit elements in exercise 3 if  $V_{in}(t) = 3\cos(8t)$

## 10.5 Direct Frequency Domain Circuit Analysis

In section 10.3, we determined the steady-state response of electrical circuits to sinusoidal signals using phasor representations of the signals involved, and time-domain representations of the circuit element voltage-current relations. Applying KVL and KCL in this manner resulted in governing equations in which the time dependence had been removed, which converted the governing equations from differential equations to algebraic equations. Unknowns in the resulting algebraic equations were the phasor representations of the signals. These equations

could then be solved to determine the desired signals in phasor form; these results could then be used to determine the time-domain representations of the signals.

In section 10.4, we replaced the time-domain voltage-current relations for passive electrical circuit elements with impedances, which provide voltage-current relations for the circuit elements directly in the frequency domain. At the end of section 10.4, we used these impedances to schematically represent a circuit directly in the frequency domain.

In this section, we will use this frequency-domain circuit representation to perform circuit analysis directly in the frequency domain using phasor representations of the signals and impedance representations of the circuit elements. This will allow us to write the algebraic equations governing the phasor representation of the circuit directly, without any reference to the time domain behavior of the circuit. As in section 10.3, these equations can be solved to determine the behavior of the circuit in terms of phasors, and the results transformed to the time domain.

Performing the circuit analysis directly in the frequency domain using impedances to represent the circuit elements can result in a significant simplification of the analysis. In addition, many circuit analysis techniques which were previously applied to resistive circuits (e.g. circuit reduction, nodal analysis, mesh analysis, superposition, Thevenin's and Norton's Theorems) are directly applicable in the frequency domain. Since these analysis techniques have been presented earlier for resistive circuits, in this section we will simply:

1. Provide examples of applying these analysis methods to frequency-domain circuits, and
2. Note any generalizations relative to using phasors in these analysis methods

Throughout this section, the reader should firmly keep in mind that we are dealing only with the steady-state responses of circuits to sinusoidal forcing functions. It is sometimes easy to lose track of this fact, since the sinusoidal nature of the signal is often not explicitly stated, but any time we deal with impedances and phasors, we are working with sinusoidal signals.

### 10.5.1 Kirchhoff's Voltage Law

Kirchhoff's Voltage Law states that the sum of the voltage differences around any closed loop is zero. Therefore, if  $v_1(t)$ ,  $v_2(t)$ , ...,  $v_N(t)$  are the voltages around some closed loop, KVL provides:

$$\sum_{k=1}^N v_k(t) = 0 \quad \text{Eq. 10.79}$$

Substituting the phasor representation of the voltages results in:

$$\sum_{k=1}^N \underline{V}_k e^{j\omega t} = 0 \quad \text{Eq. 10.80}$$

Dividing equation (10.80) by  $e^{j\omega t}$  results in:

$$\sum_{k=1}^N \underline{V}_k = 0 \quad \text{Eq. 10.81}$$

So that KVL states that the sum of the phasor voltages around any closed loop is zero.

### 10.5.2 Kirchhoff's Current Law

Kirchhoff's Current Law states that the sum of the currents entering any node is zero. Therefore, if  $i_1(t)$ ,  $i_2(t)$ , ...,  $i_N(t)$  are the currents entering a node, KCL provides:

$$\sum_{k=1}^N i_k(t) = 0 \quad \text{Eq. 10.82}$$

Substituting the phasor representation of the currents results in:

$$\sum_{k=1}^N \underline{I}_k e^{j\omega t} = 0 \quad \text{Eq. 10.83}$$

Dividing equation (10.83) by  $e^{j\omega t}$  results in:

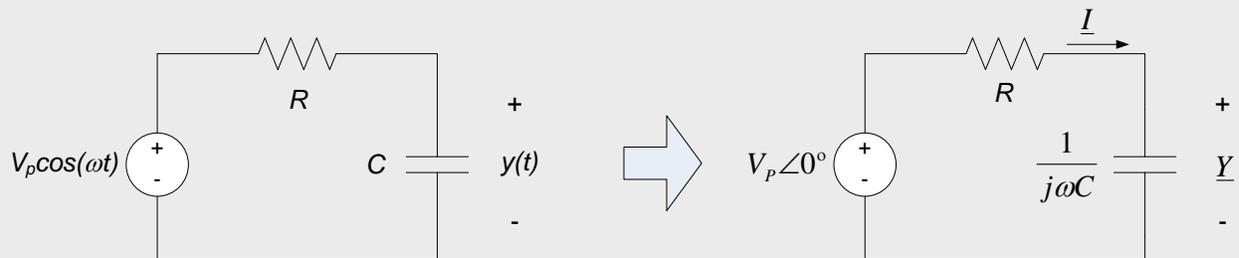
$$\sum_{k=1}^N \underline{I}_k = 0 \quad \text{Eq. 10.84}$$

So that KCL states that the sum of the phasor currents entering (or leaving) a node is zero.

**Important Result:** KVL and KCL apply directly in the frequency domain.

### Example 10.8: RC Circuit Steady-state Sinusoidal Response

In this example, we will revisit example 10.3. In that example, we determined the capacitor voltage in the circuit to the left below, using phasor analysis techniques applied to the circuit's time-domain governing equation. In this example, we will represent the circuit itself directly in the frequency domain, using impedance representations of the circuit element. The frequency-domain representation of the circuit is shown to the right below.



By the definition of impedance, we can determine the current through the capacitor to be:

$$\underline{I} = \frac{\underline{Y}}{Z_C} = \frac{\underline{Y}}{\frac{1}{j\omega C}} = j\omega C \underline{Y}$$

The voltage across the resistor can now, by the definition of impedance, be written as  $\underline{V}_R = R \cdot \underline{I} = R(j\omega C \underline{Y})$ . We now apply KVL for phasors to the circuit to the right above, which leads to:

$$V_p \angle 0^\circ = R(j\omega C \underline{Y}) + \underline{Y}$$

Solving for  $\underline{Y}$  in this equation provides  $\underline{Y} = \frac{V_p \angle 0^\circ}{1 + j\omega RC}$

By the rules of complex arithmetic, we can determine the magnitude and phase angle of  $\underline{Y}$  to be:

$$|\underline{Y}| = \frac{V_P}{\sqrt{1 + (\omega RC)^2}}$$

$$\angle \underline{Y} = -\tan^{-1}(\omega RC)$$

And the time-domain solution for  $y(t)$  is thus:

$$y(t) = \frac{V_P}{\sqrt{1 + (\omega RC)^2}} \cos[\omega t - \tan^{-1}(\omega RC)]$$

### 10.5.3 Parallel and Series Impedances & Circuit Reduction

Consider the case of N impedances connected in series, as shown in Fig. 10.20. Since the elements are in series, and since we have seen that KCL applies to phasors, the phasor current  $\underline{I}$  flows through each of the impedances. Applying KVL for phasors around the single loop, and incorporating the definition of impedance, we obtain:

$$\underline{V} = \underline{I}(\underline{Z}_1 + \underline{Z}_2 + \dots + \underline{Z}_N) = 0 \tag{Eq. 10.85}$$

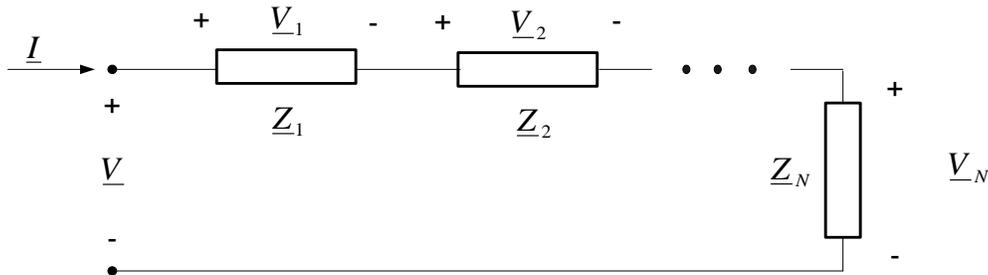


Figure 10.20. Series combination of impedances.

If we define  $\underline{Z}_{eq}$  as the equivalent impedance of the series combination, we have  $\underline{V} = \underline{I} \cdot \underline{Z}_{eq}$ , where:

$$\underline{Z}_{eq} = \underline{Z}_1 + \underline{Z}_2 + \dots + \underline{Z}_N \tag{Eq. 10.86}$$

So that impedances in series sum directly. Thus, impedances in series can be combined in the same way as resistances in series.

By extension of the above result, we can develop a voltage divider formula for phasors. Without derivation, we state that the phasor voltage across the k<sup>th</sup> impedance in a series combination of N impedances as shown in Fig. 10.20 can be determined as:

$$\underline{V}_k = \underline{V}(\underline{Y}_1 + \underline{Y}_2 + \dots + \underline{Y}_N) = 0 \tag{Eq. 10.88}$$

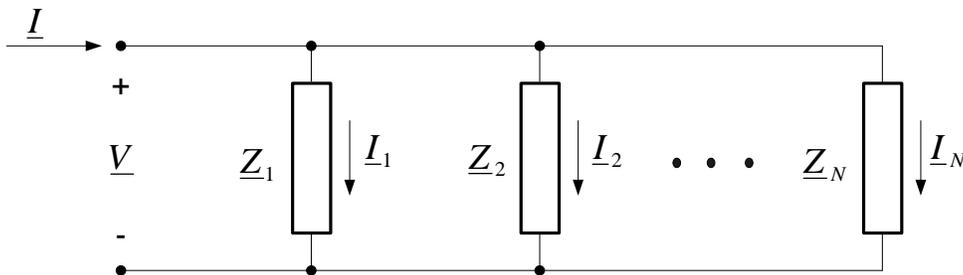


Figure 10.21. Parallel combination of impedances.

If we define  $\underline{Y}_{eq}$  as the equivalent admittance of the parallel combination, we have:

$$\underline{I} = \underline{V} \cdot \underline{Y}_{eq} \tag{Eq. 10.89}$$

Where:

$$\underline{Y}_{eq} = \underline{Y}_1 + \underline{Y}_2 + \dots + \underline{Y}_N \tag{Eq. 10.90}$$

So that admittances in parallel sum directly. Converting our admittances to impedances indicates that the equivalent impedance of a parallel combination of N impedances as shown in Fig. 10.21 is:

$$\underline{Z}_{eq} = \frac{1}{\frac{1}{\underline{Z}_1} + \frac{1}{\underline{Z}_2} + \dots + \frac{1}{\underline{Z}_N}} \tag{Eq. 10.91}$$

Thus, impedances in parallel can be combined in the same way as resistances in parallel.

By extension of the above result, we can develop a current divider formula for phasors. Without derivation, we state that the phasor current across the  $k^{\text{th}}$  impedance in a series combination of  $N$  impedances as shown in Fig. 10.21 can be determined as:

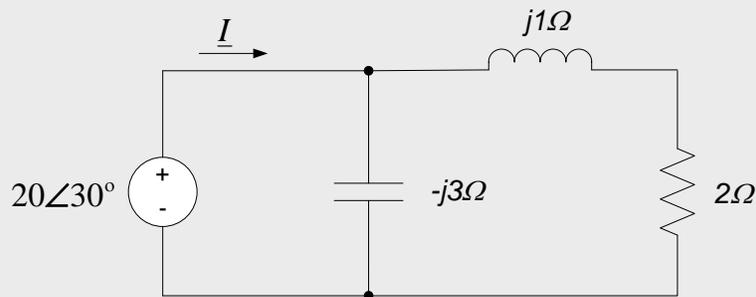
$$\underline{I}_k = \underline{I} \frac{\frac{1}{\underline{Z}_k}}{\frac{1}{\underline{Z}_1} + \frac{1}{\underline{Z}_2} + \dots + \frac{1}{\underline{Z}_N}} \tag{Eq. 10.92}$$

So that our current division relationships for resistors in parallel apply directly in the frequency domain for impedances in parallel.

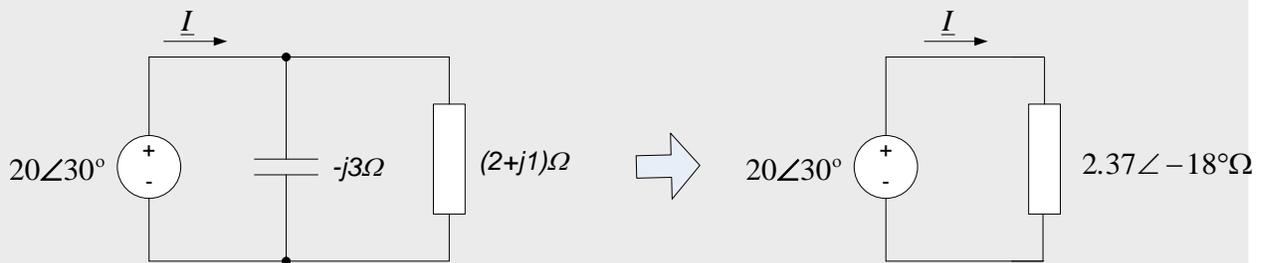
**Important Result:** All circuit reduction techniques for resistances apply directly to the frequency domain for impedances. Likewise, voltage and current divider relationships apply to phasor circuits in the frequency domain exactly as they apply to resistive circuits in the time domain.

### Example 10.9

Use circuit reduction techniques to determine the current phasor  $\underline{I}$  leaving the source in the circuit below. (Note: the circuit below is the frequency domain circuit we obtained in example 10.7.)



Since impedances in series add directly, the inductor and resistor can be combined into a single equivalent impedance of  $(2 + j1)\Omega$ , as shown in the figure to the left below. The capacitor is then in parallel with this equivalent impedance. Since impedances in parallel add in the same way as resistors in parallel, the equivalent impedance of this parallel combination can be calculated by dividing the product of the impedances by their sum, so  $\underline{Z}_{eq} = \frac{(-j3)(2+j1)}{(-j3)+(2+j1)} \Omega = \frac{3-j6}{2-j2} \Omega$ . Converting this impedance to polar form results in  $\underline{Z}_{eq} = 2.37\angle -18^\circ\Omega$ ; the final reduced circuit is shown in the figure to the right below.



Using the reduced circuit to the right above and the definition of impedance, we can see that:

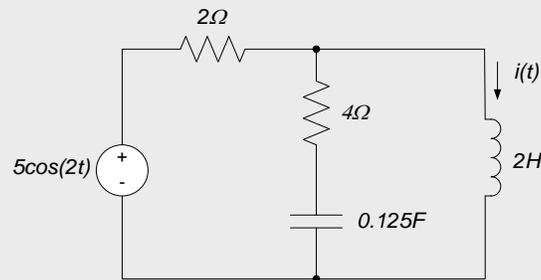
$$\underline{I} = \frac{20\angle 30^\circ}{2.37\angle -18^\circ} = \frac{2}{2.37}\angle [30^\circ - (-18^\circ)]A$$

So that:

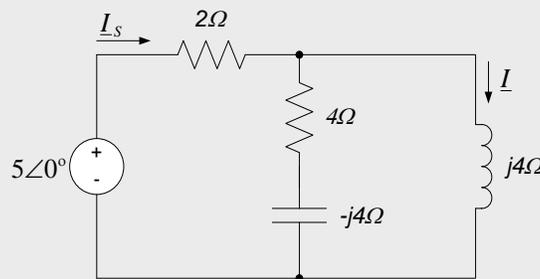
$$\underline{I} = 8.44\angle 48^\circ A$$

### Example 10.10

Use circuit reduction techniques to determine the current,  $i(t)$  through the inductor in the circuit below.

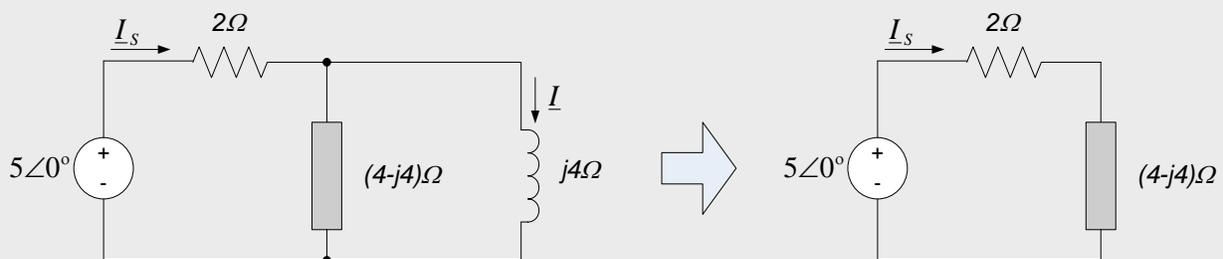


With  $\omega=2$  rad/sec, the frequency domain representation of the circuit is as shown in the figure to the left below; in that figure, we have also defined the current phasor  $\underline{I}_S$  leaving the source.



We now employ circuit reduction techniques to determine the phasor  $\underline{I}$ . To do this, we first determine the circuit impedance seen by the source; this impedance allows us to determine the source current  $\underline{I}_S$ . The current  $\underline{I}$  can be determined from a current divider relation and  $\underline{I}_S$ .

The impedances of the series combination of the capacitor and the  $4\Omega$  resistor is readily obtained by adding their individual impedances, as shown in the figure to the left below. This equivalent impedance is then in parallel with the inductor's impedance; the equivalent impedance of this parallel combination is as shown in the circuit to the right below.



The source current is then, by the definition of impedance,  $\underline{I}_S = \frac{5\angle 0^\circ}{(4-j4)\Omega + 2\Omega} = 0.69\angle -33.7^\circ$ .

The circuit to the left above, along with our voltage divider formula, provides:

$$\underline{I} = \frac{(4 - j4)\Omega}{(4 - j4)\Omega + j4\Omega} \cdot \underline{I}_S = (1 - j1)\Omega \cdot 0.69\angle -33.7^\circ = 0.98\angle -78.7^\circ$$

And the current  $i(t) = 0.98 \cos(2t - 78.7^\circ)$

### 10.5.4 Nodal and Mesh Analysis

Nodal analysis and mesh analysis techniques have been previously applied to resistive circuits in the time domain. In nodal analysis, we applied KCL at independent nodes and used Ohm's Law to write the resulting equations in terms of the node voltages. In mesh analysis, we applied KVL and used Ohm's Law to write the resulting equations in terms of the mesh currents.

In the frequency domain, as we have seen in previous sub-sections, KVL and KCL apply directly to the phasor representations of voltages and currents. Also, in the frequency domain, impedances can be used to represent voltage-current relations for circuit elements in the frequency domain in the same way that Ohm's Law applied to resistors in the time domain (the relation  $\underline{V} = \underline{I} \cdot \underline{Z}$  in the frequency domain corresponds exactly to the relation  $v(t) = R \cdot i(t)$  in the time domain). Thus, nodal analysis and mesh analysis apply to frequency domain circuits in exactly the same way as to time domain resistive circuits, with the following modifications:

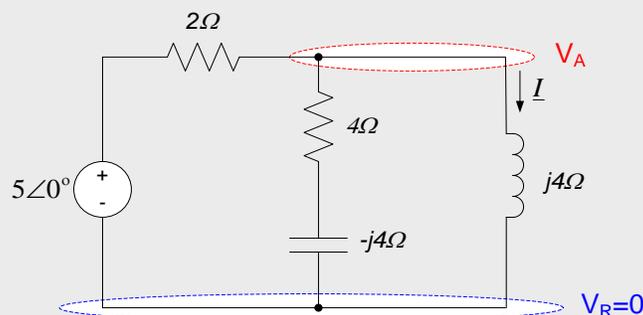
- The circuit excitations and responses are represented by phasors
- Phasor representations of node voltages and mesh currents are used
- Impedances are used in the place of resistances

Application of nodal and mesh analysis to frequency-domain circuit analysis is illustrated in the following examples.

#### Example 10.11

Use nodal analysis to determine the current  $i(t)$  in the circuit of example 10.10.

The desired frequency-domain circuit was previously determined in Example 10.10. Nodal analysis of the frequency-domain circuit proceeds exactly as was done in the case of resistive circuits. The reference voltage,  $V_R = 0$ , and our single node voltage,  $V_A$ , for this circuit are defined on the circuit below.



Applying KCL in phasor form at node A provides:

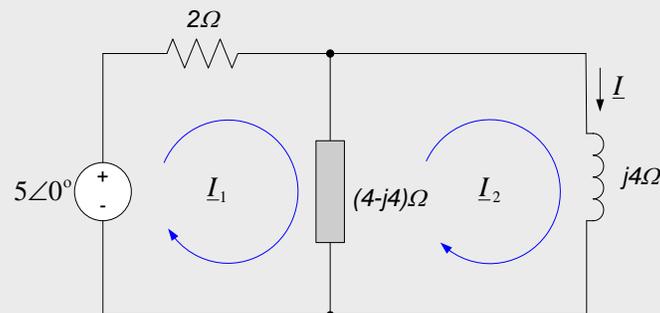
$$\frac{5\angle 0^\circ - V_A}{2\Omega} - \frac{V_A}{(4 - j4)\Omega} - \frac{V_A}{j4\Omega} = 0$$

Solving for  $\underline{V}_A$  gives  $\underline{V}_A = 3.92\angle 11.31^\circ\text{V}$ . By the definition of impedance, the desired current phasor  $\underline{I} = \frac{\underline{V}_A}{j4\Omega} = \frac{3.92\angle 11.31^\circ}{4\angle 90^\circ} = 0.98\angle -78.7^\circ$  so that  $i(t) = 0.98\cos(2t - 78.7^\circ)$ , which is consistent with our result obtained via circuit reduction in Example 3.

### Example 10.12

Use mesh analysis to determine the current  $i(t)$  in the circuit of examples 10.10 and 10.11.

The desired frequency-domain circuit was previously determined in Example 10.10. Mesh analysis of the frequency-domain circuit proceeds exactly as for resistive circuits. The figure below shows our choice of mesh loops; the series resistor-capacitor combination has been combined into a single equivalent resistance in the figure below, for clarity.



KVLO around the mesh loop  $\underline{I}_1$  provides:

$$5\angle 0^\circ - 2 \cdot \underline{I}_1 - (4 - j4)(\underline{I}_1 - \underline{I}_2) = 0$$

KVL around the mesh loop  $\underline{I}_2$  provides:

$$(4 - j4)(\underline{I}_2 - \underline{I}_1) + j4 \cdot \underline{I}_2 = 0$$

The second equation above can be simplified to provide:  $\underline{I}_2 = (1 - j)\underline{I}_1$ . Using this result to eliminate  $\underline{I}_1$  in the mesh equation for loop  $\underline{I}_1$  and simplifying provides:

$$5\angle 0^\circ = \left[ \frac{(6 - j4)}{1 - j} + (j4 - 4) \right] \underline{I}_2$$

So that  $\underline{I}_2 = 0.98\angle -78.7^\circ$ . The mesh current  $\underline{I}_2$  is simply the desired current  $\underline{I}$ , so in the time domain:

$$i(t) = 0.98 \cos(2t - 78.7^\circ)$$

Which is consistent without results from examples 10.10 and 10.11.

**Important Result:** Nodal and mesh analysis methods apply to phasor circuits exactly as they apply to resistive circuits in the time domain. Impedances simply replace resistances, and quantities of interest become complex valued.

## 10.5.5 Superposition

The extension of superposition to the frequency domain is an extremely important topic. Several common analysis techniques you will encounter later in this course and in future courses (frequency response, Fourier Series, and

Fourier Transforms, for example) will depend heavily upon the superposition of sinusoidal signals. In this subsection, we introduce the basic concepts involved.

In all of our steady-state sinusoidal analyses, we have required that the circuit is linear. (The statement that the steady state response to a sinusoidal input is a sinusoid at the same frequency requires the system to be linear. Nonlinear systems do not necessarily have this characteristic.) Thus, all phasor circuits are linear and superposition must apply. Thus, if a phasor circuit has multiple inputs, we can calculate the response of the circuit to each input individually and sum the results to obtain the overall response. It is important to realize, however, that the final step of summing the individual contributions to obtain the overall response can, in general, only be done in the time domain. Since the phasor representation of the circuit response implicitly assumes a particular frequency, the phasor representations cannot be summed directly. The time domain circuit response, however, explicitly provides frequency information, allowing those responses to be summed.

In fact, because the frequency-domain representation of the circuit depends upon the frequency of the input (in general, the impedances will be a function of frequency), the frequency domain representation of the circuit itself is, in general, different for different inputs. Thus, the only way in which circuits with multiple inputs at different frequencies can be analyzed in the frequency domain is with superposition.

In the special case in which all inputs share a common frequency, the circuit response can be determined by any of our previous analysis techniques (circuit reduction, nodal analysis, mesh analysis, superposition, etc.) In this case, if superposition is used, the circuit response to individual inputs can be summed directly in the frequency domain if desired.

Examples of the application of superposition to analysis of frequency-domain circuits are provided below.

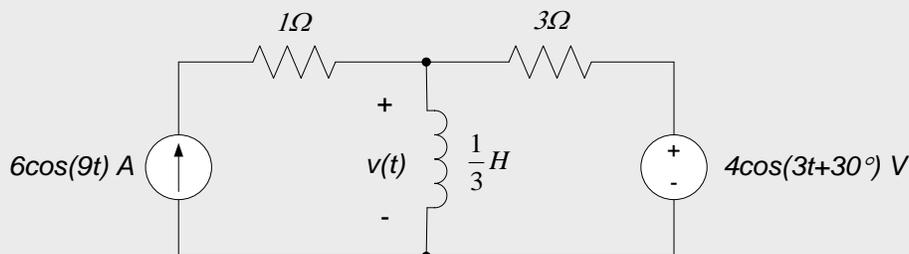
**Important Result:** In the case of multiple frequencies existing in the circuit, superposition is the only valid frequency-domain analysis approach.

Superposition applies directly in the frequency domain, insofar as contributions from individual sources can be determined by killing all other sources and analyzing the resulting circuit. In general, however, superimposing (summing) the contributions from the individual sources must be done in the time domain.

Superposition of responses to individual sources can be summed directly in the frequency domain (e.g. addition of the phasors representing the individual responses) is only appropriate if all sources have the same frequency. In this case (all source having the same frequency) any of our other modeling approaches are also valid.

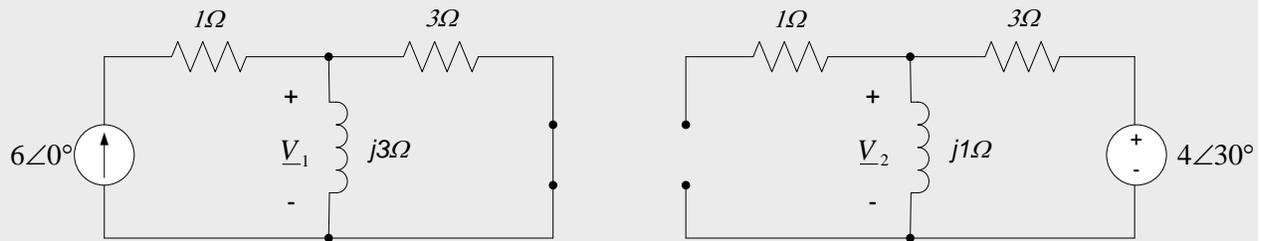
### Example 10.13

Determine the voltage  $v(t)$  across the inductor in the circuit below.



Since two different input frequencies are applied to the circuit, we must use superposition to determine the response. The circuit to the left below will provide the phasor response  $\underline{V}_1$  to the current source; the frequency is

$\omega = 9$  rad/sec and the voltage source is killed. The circuit to the right below will provide the phasor response  $\underline{V}_2$  to the voltage source; the frequency is  $\omega = 3$  rad/sec and the current source is killed.



To determine the voltage phasor resulting from the current source ( $\underline{V}_1$  in the circuit to the left above), we note that the inductor and the  $3\Omega$  resistor form a current divider. Thus, the current through the inductor resulting from the current source is  $\underline{I}_1 = \frac{3\Omega}{(3+j3)\Omega} \cdot 6\angle 0^\circ = \frac{3\angle 0^\circ \cdot \angle 0^\circ}{3\sqrt{2}\angle 45^\circ} = \frac{6}{\sqrt{2}}\angle -45^\circ$ . The voltage phasor  $\underline{V}_1$  can then be determined by multiplying this current times the inductor's impedance:

$$\underline{V}_1 = j3\Omega \cdot \frac{6}{\sqrt{2}}\angle -45^\circ = 3\angle 90^\circ \cdot \frac{6}{\sqrt{2}}\angle -45^\circ V$$

And the time-domain voltage across the inductor due to the current source is:

$$v_1(t) = 9\sqrt{2} \cos(9t + 45^\circ) V$$

To determine the voltage phasor resulting from the voltage source ( $\underline{V}_2$  in the circuit to the right above), we note that the inductor and the  $3\Omega$  resistor now form a voltage divider. Thus, the voltage  $\underline{V}_2$  can be readily determined by:

$$\underline{V}_2 = \frac{j1\Omega}{(3+j1)\Omega} \cdot 4\angle 30^\circ = \frac{1\angle 90^\circ \cdot 4\angle 30^\circ}{\sqrt{10}\angle 18.4^\circ} = \frac{4}{\sqrt{10}}\angle 101.6^\circ$$

So that the time-domain voltage across the inductor due to the voltage source is:

$$v_2(t) = \frac{4}{\sqrt{10}} \cos(3t + 101.6^\circ) V$$

The overall voltage is then the sum of the contributions from the two sources, in the time domain, so:

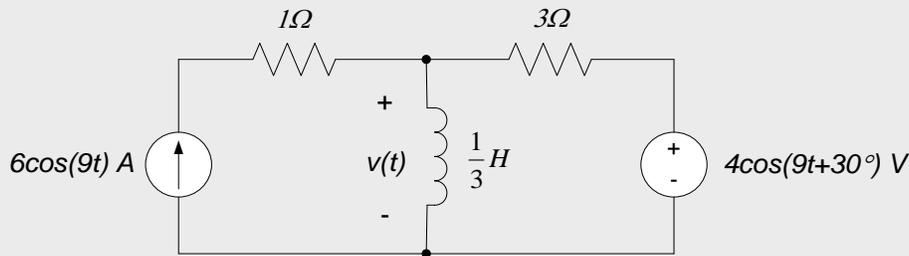
$$v(t) = v_1(t) + v_2(t)$$

And:

$$v(t) = 9\sqrt{2} \cos(9t + 45^\circ) + \frac{4}{\sqrt{10}} \cos(3t + 101.6^\circ) V$$

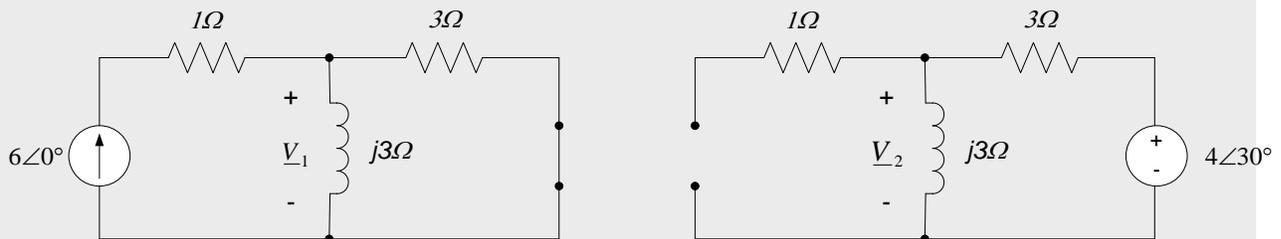
### Example 10.14

Determine the voltage  $v(t)$  across the inductor in the circuit below.



This circuit is essentially the same as the circuit of Example 10.13, with the important difference that the frequency of the voltage input has changed – the voltage source and current source both provide the same frequency input to the circuit, 9 rad/sec. We will first do this problem using superposition techniques. We will then use nodal analysis to solve the problem, to illustrate that multiple inputs at the same frequency do not require the use of superposition.

Individually killing each source in the circuit above results in the two circuits shown below. Note that the impedance of the inductor is now the same in both of these circuits.



The two circuits shown above will now be analyzed to determine the individual contributions to the inductor voltage; these results will then be summed to determine the overall inductor voltage.

The circuit to the left above has been analyzed in Example 6. Therefore, the voltage phasor  $\underline{V}_1$  is the same as determined in Example 10.13:

$$\underline{V}_1 = 9\sqrt{2}\angle 45^\circ V$$

The voltage  $\underline{V}_2$  in the circuit to the right above can be determined from application of the voltage divider formula for phasors:

$$\underline{V}_2 = \frac{j3\Omega}{(3 + j3)\Omega} \cdot 4\angle 30^\circ = \frac{3\angle 90^\circ \cdot 4\angle 30^\circ}{3\sqrt{2}\angle 45^\circ} = 2\sqrt{2}\angle 75^\circ$$

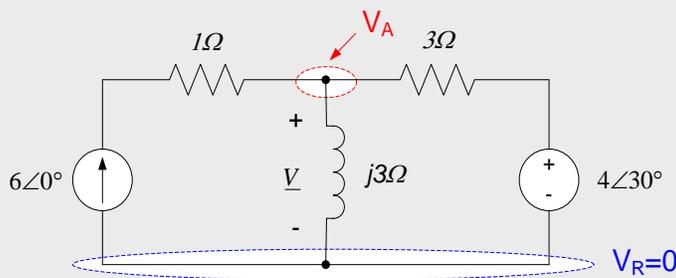
Since both inputs have the same frequency, we can superimpose the phasor results directly (we could, of course, also determine the individual time domain responses and superimpose those responses if we chose):

$$\underline{V} = \underline{V}_1 + \underline{V}_2 = 9\sqrt{2}\angle 45^\circ + 2\sqrt{2}\angle 75^\circ = 15.24\angle 50.3^\circ V$$

So that the time domain inductor voltage is  $v(t) = 15.24 \cos(9t + 50.3^\circ)V$ . Notice that the circuit response has only a single frequency component, since both inputs have the same frequency.

The superposition approach provided above is entirely valid. However, since both sources have the same input, we can choose any of our other analysis approaches to perform this problem. To emphasize this fact, we choose to do this problem using nodal analysis.

The frequency-domain circuit, with our definition of reference voltage and independent node, is shown in the figure below.



KCL at node A provides:

$$6\angle 0^\circ = \frac{V_A - 0}{j3\Omega} + \frac{V_A - 4\angle 30^\circ}{3\Omega}$$

Solving the above equation for  $\underline{V}_A$  provides  $\underline{V}_A = 15.24\angle 50.3^\circ V$  so that the inductor voltage as a function of time is:

$$v(t) = 15.24 \cos(9t + 50.3^\circ) V$$

Which is consistent with our result using superposition.

### 10.5.6 Thévenin’s & Norton’s Theorems, Source Transformations, and Maximum Power Transfer

Application of Thévenin’s and Norton’s Theorems to frequency domain circuits is identical to their application to time domain resistive circuits. The only differences are:

- The open circuit voltage ( $V_{oc}$ ) and short circuit current ( $i_{sc}$ ) determined for resistive circuits is replaced by their phasor representations,  $\underline{V}_{oc}$  and  $\underline{I}_{sc}$ .
- The Thévenin resistance,  $R_{TH}$ , is replaced by a Thévenin impedance,  $\underline{Z}_{TH}$ .

Thus, the Thévenin and Norton equivalent circuits in the frequency domain are as shown in Fig. 10.22.

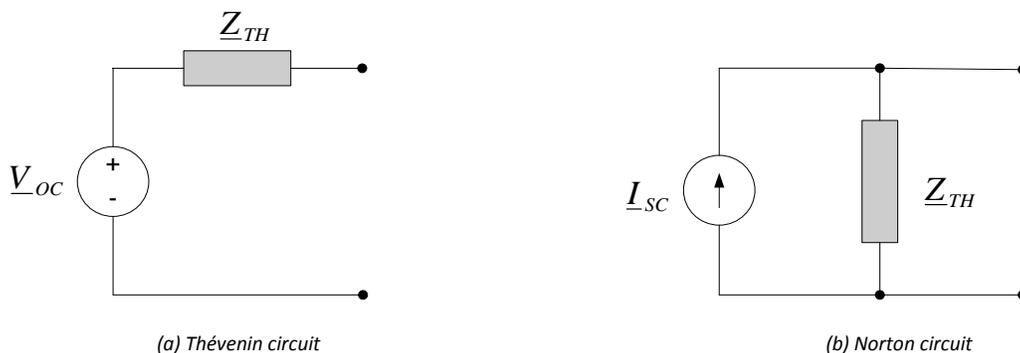


Figure 10.22. Thévenin and Norton equivalent circuits.

Since Thévenin’s and Norton’s Theorems both apply in the frequency domain, the approaches we used for source transformations in the time domain for resistive circuits translate directly to the frequency domain, with impedances substituted for resistances and phasors used for voltage and current terms.

In order to determine the load necessary to draw the maximum power from a Thévenin equivalent circuit, we must re-derive the maximum power result obtained previously for resistive circuits, substituting impedances for admittances and using phasors for source terms. We will not derive the governing relationship, but will simply state that, in order to transfer the maximum power to a load, the load impedance must be the complex conjugate of the Thévenin impedance of the circuit being loaded. Thus, if a Thévenin equivalent circuit has some impedance  $\underline{Z}_{TH}$  with a resistance  $R_{TH}$  and a reactance  $X_{TH}$ , the load which will draw the maximum power from this circuit must have resistance  $R_{TH}$  and a reactance  $-X_{TH}$ . The appropriate loaded circuit is shown in Fig. 10.23 below.

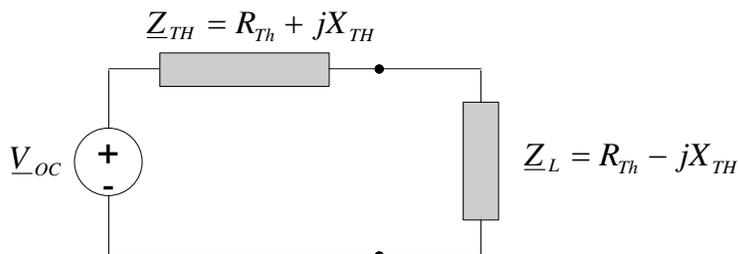
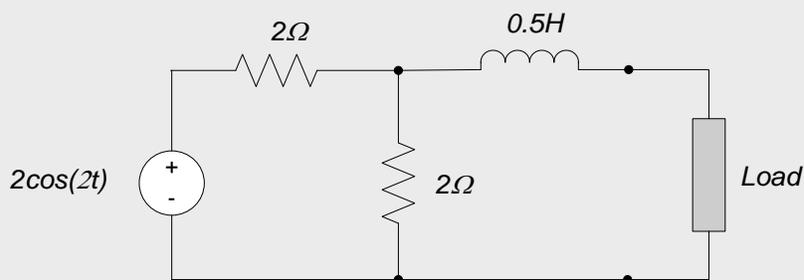


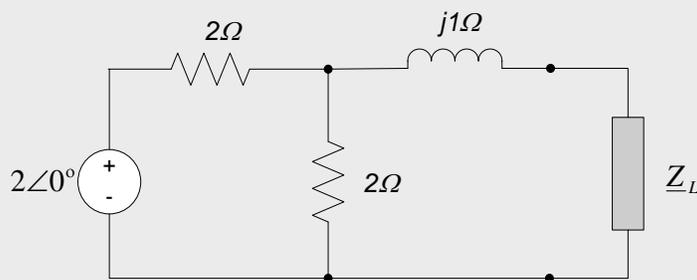
Figure 10.23. Load impedance to draw maximum power from a Thévenin circuit.

### Example 10.15

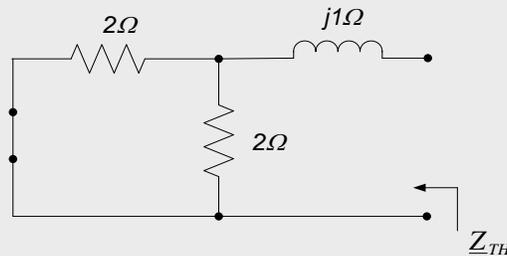
Determine the Thévenin equivalent circuit seen by the load in the circuit below



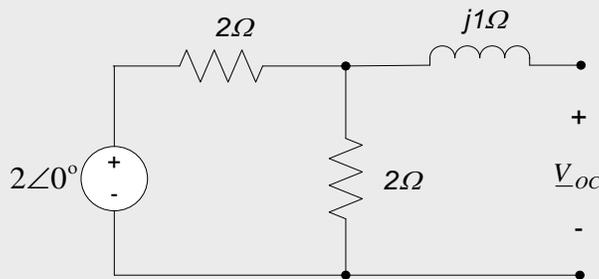
In the circuit below, we have used the input frequency,  $\omega=2$  rad/sec, to convert the circuit to the frequency domain.



Removing the load and killing the source allows us to determine the Thévenin resistance of the circuit. The appropriate circuit is:



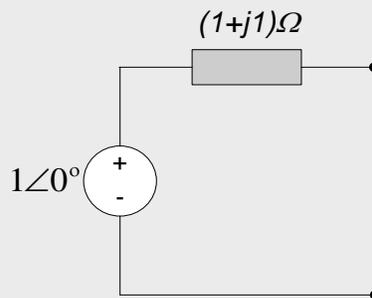
The parallel combination of two,  $2\Omega$  resistors have an equivalent resistance of  $1\Omega$ . This impedance, in series with the  $j1\Omega$  impedance, results in a Thévenin impedance  $Z_{TH} = (1 + j1)\Omega$ . Replacing the source, but leaving the load terminals open-circuited, as shown in the figure below, allows us to determine the open-circuit voltage  $V_{OC}$ .



Since there is no current through the inductor, due to the open-circuit condition,  $V_{OC}$  is determined from a simple resistive voltage divider formed by the two,  $2\Omega$  resistors. Thus, the open-circuit voltage is:

$$V_{OC} = \frac{2\Omega}{2\Omega + 2\Omega} \cdot 2\angle 0^\circ = 1\angle 0^\circ$$

The resulting Thévenin equivalent circuit is shown below:



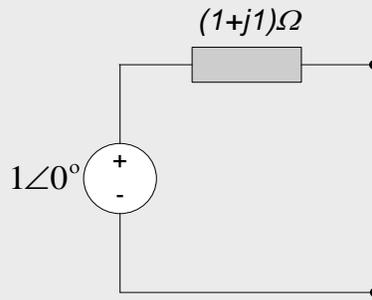
### Example 10.16

Determine the Norton equivalent circuit of the circuit of example 10.15.

Since we determined the Thévenin equivalent circuit in Example 10.15, a source transformation can be used to determine the Norton equivalent circuit. Consistent with our previous source transformation rules, the short-circuit current,  $I_{SC}$ , is equal to the open-circuit voltage divided by the Thévenin impedance:

$$I_{SC} = \frac{V_{OC}}{Z_{TH}} = \frac{1\angle 0^\circ}{(1 + j1)\Omega} = \frac{1\angle 0^\circ}{\sqrt{2}\angle 45^\circ} = \frac{1}{\sqrt{2}}\angle -45^\circ$$

Since the impedance doesn't change during a source transformation, the Norton equivalent circuit is therefore as shown below:



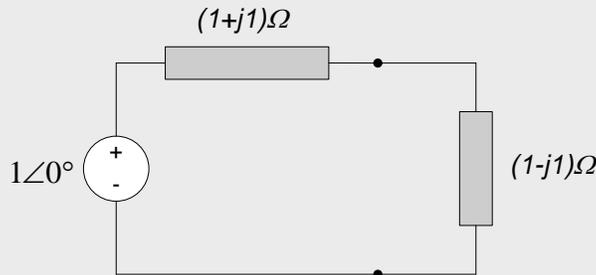
**Example 10.17**

Determine the load impedance for the circuit of Example 10.15 which will provide the maximum amount of power to be delivered to the load. Provide a physical realization (a circuit) which will provide this impedance.

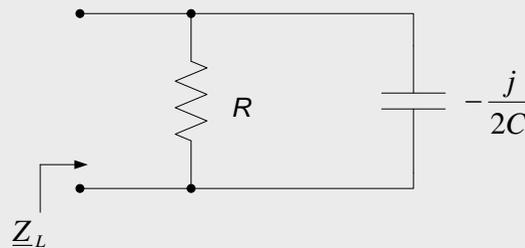
The maximum power is delivered to the load when the load impedance is the complex conjugate of the Thévenin impedance. Thus, the load impedance for maximum power transfer is:

$$\underline{Z}_L = (1 - j)\Omega$$

And the loaded Thévenin circuit is:



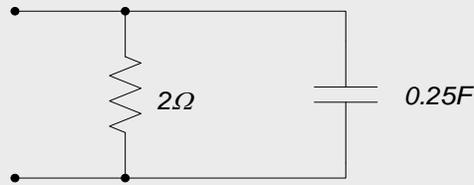
To implement this load, let us look at a parallel RC combination. With the frequency  $\omega=2$  rad/sec, the frequency domain load looks like:



Combining parallel impedances results in:

$$\underline{Z}_L = \frac{-j\left(\frac{R}{2C}\right)}{R - \frac{j}{2C}} \Omega = \frac{\frac{R}{4C^2} - j\frac{R^2}{2C}}{R^2 + \frac{1}{4C^2}} \Omega$$

Setting  $R = 2\Omega$  and  $C = 0.25F$  makes  $\underline{Z}_L = (1 - j)\Omega$ , as desired, so the physical implementation of our load is as shown below:



## Section Summary

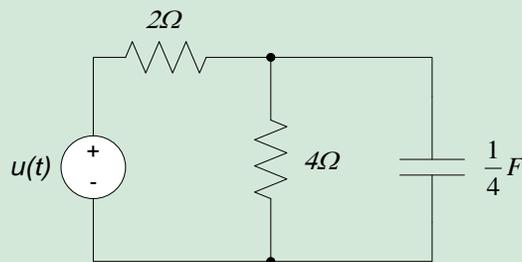
- The following analysis methods apply in the frequency domain exactly as they do in the time domain for purely resistive circuits
  - KVL and KCL
  - Voltage and current dividers
  - Circuit reduction techniques
  - Nodal and mesh analysis
  - Superposition, especially when multiple frequencies are present
  - Thévenin's and Norton's theorems

One simply uses phasor representations for the voltages and/or currents in the circuit and impedances to represent the circuit element voltage-current relationships. The analysis techniques presented Chapters 1 through 4 are then applied exactly as they were for resistive circuits.

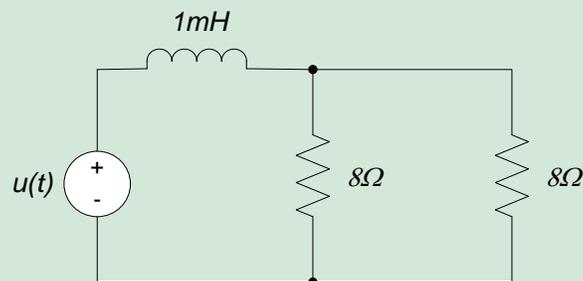
- One minor exception to the above statement is that, in order to draw maximum power from a circuit, the load impedance should be the complex conjugate of the impedance of the circuit's Thévenin equivalent.

## 10.5 Exercises:

1. Determine the impedance seen by the source for the circuit below if  $u(t) = 4\cos(t+30^\circ)$ .



2. Determine the impedance seen by the source for the circuit below if  $u(t) = 2\cos(4000t)$ .



3. For the circuit of exercise 1, determine the current delivered by the source.
4. For the circuit of exercise 2, determine the voltage across the  $8\Omega$  resistors.

## 10.6 Frequency Domain System Characterization

In Chapters 7 and 8, we wrote the differential equation governing the relationship between a circuit's input and output (the input-output equation) and used this differential equation to determine the response of a circuit to some input. We also characterized the time-domain behavior of the system by examining the circuit's natural and step responses. We saw that the behavior of a first order circuit can be characterized by its time constant and DC gain, while the response of a second order circuit is characterized by its natural frequency, damping ratio and DC gain. It is important to recognize that these characterizations were independent of specific input parameters; they depended upon the type of response (e.g. a step function or a natural response), but were independent of detailed information such as the amplitude of the step input or the actual values of the initial conditions.

We will now use the steady state sinusoidal response to characterize a circuit's behavior. As in the case of our time-domain characterization, this characterization will allow the system's behavior to be defined in terms of its response to sinusoidal inputs, but the characterization will be independent of details such as the input sinusoid's amplitude or phase angle. (The input sinusoid's frequency will, however, still be of prime importance.)

When a sinusoidal input is applied to a linear system, the system's forced response consists of a sinusoid with the same frequency as the input sinusoid, but in general having a different amplitude and phase from the input sinusoid. Figure 10.24 shows the general behavior, in block diagram form. Changes in the amplitude and phase angle between the input and output signals are often used to characterize the circuit's input-output relationship at the input frequency,  $\omega$ . In this chapter, we will demonstrate how this characterization is performed for inputs with discrete frequencies (as in the case of circuits with one or several inputs, each with a single frequency component). Later chapters will extend these concepts to the case in which frequency is considered to be a continuous variable.



Figure 10.24. Sinusoidal steady-state input-output relation for a linear time invariant system.

Previously in this chapter, we have (bit-by-bit) simplified the analysis of a system's steady state sinusoidal response significantly. We first represented the sinusoidal signals as complex exponentials in order to facilitate our analysis. We subsequently used phasors to represent our complex exponential signals, as shown in Fig. 10.25; this allowed us to represent and analyze the circuit's steady state sinusoidal response directly in the frequency domain.

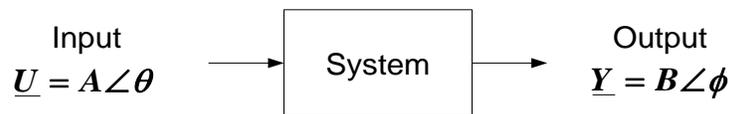


Figure 10.25. Phasor representation of sinusoidal inputs and outputs.

In the frequency domain analyses performed to date, we have generally determined the system's response to a specific input signal with a given frequency, amplitude, and phase angle. We now wish to characterize the system response to an input signal with a given frequency, but an arbitrary amplitude and phase angle. As indicated previously in section 10.1, we will see that the input-output relationship governing the system reduces to a relationship between the output and input signal amplitudes and the output and input signal phases. The circuit can thus be represented in phasor form as shown in Fig. 10.26. The system's effect on a sinusoidal input consists of an amplitude gain between the output and input signals ( $\frac{B}{A}$  in Fig. 10.26) and a phase difference between the output and input signals ( $\phi - \theta$  in Fig. 10.26).

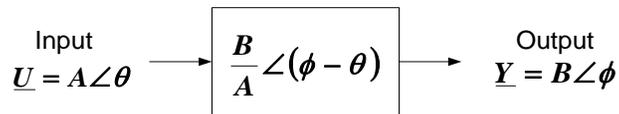
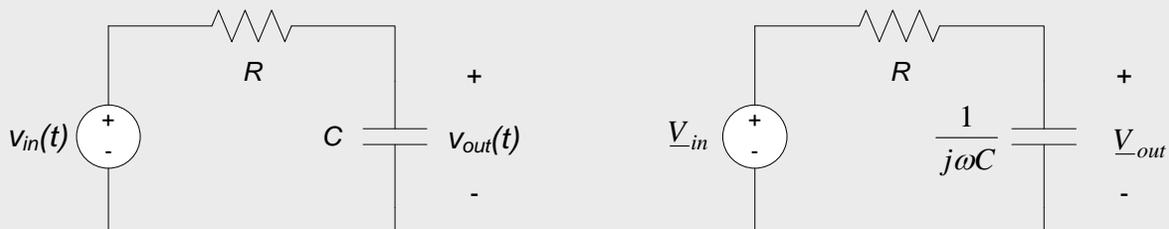


Figure 10.26. Frequency domain representation of circuit input-output relationship.

Rather than perform a rigorous demonstration of this property at this time, we will simply provide some simple examples to illustrate the basic concept.

### Example 10.18

A sinusoidal voltage,  $v_{in}(t)$ , is applied to the circuit to the left below. Determine the frequency-domain relationship between the phasor representing  $v_{in}(t)$  and the phasor representing the output voltage  $v_{out}(t)$ .



Since the frequency is unspecified, we leave frequency as an independent variable,  $\omega$ , in our analysis. In the frequency domain, therefore, the circuit can be represented as shown to the right above. The frequency domain circuit is a simple voltage divider, so the relation between input and output is:

$$V_{out} = \frac{1}{R + \frac{1}{j\omega C}} \cdot V_{in} = \frac{1}{1 + j\omega RC} \cdot V_{in}$$

The factor  $\frac{1}{1 + j\omega RC}$  is a complex number, for given values of  $\omega$ ,  $R$ , and  $C$ . It constitutes a multiplicative factor which, when applied to the input, results in the output. This multiplicative factor is often used to characterize the system's response at some frequency,  $\omega$ . We will call this multiplicative factor the frequency response function, and denote it as  $H(j\omega)$ . For a particular frequency,  $H(j\omega)$  is a complex number, with some amplitude,  $|H(j\omega)|$ , and phase angle,  $\angle H(j\omega)$ . For our example, the magnitude and phase of our frequency response function are:

$$|H(j\omega)| = \frac{1}{\sqrt{1 + (\omega RC)^2}}$$

$$\angle H(j\omega) = -\tan^{-1}(\omega RC)$$

According to the rules of multiplication of complex numbers, when two complex numbers are multiplied, the magnitude of the result is the product of the magnitudes of the individual numbers, and the phase angle of the result is the sum of the individual phase angles. Thus, if the input voltage is represented in phasor form as  $\underline{V}_{in} = |V_{in}| \cdot |H(j\omega)|$  and the output voltage is  $\underline{V}_{out} = |V_{out}| \angle \phi$ , it is easy to obtain the output voltage from the input voltage and the frequency response function:

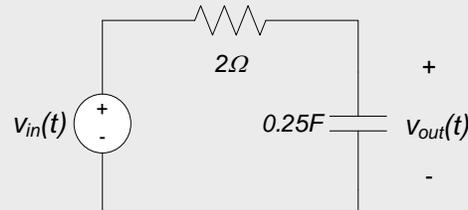
$$|V_{out}| = |V_{in}| \cdot |H(j\omega)|$$

$$\angle V_{out} = \angle V_{in} + \angle H(j\omega)$$

### Example 10.19

Use the frequency response function determined in Example 10.18 above to determine the response  $v_{out}(t)$  of the circuit shown below to the following input voltages:

- $v_{in}(t) = 3 \cos(2t + 20^\circ)$
- $v_{in}(t) = 7 \cos(4t - 60^\circ)$



When  $v_{in}(t) = 3 \cos(2t + 20^\circ)$ ,  $\omega = 2$  rad/sec,  $|V_{in}| = 3$  and  $\angle V_{in} = 20^\circ$ . For this value of  $\omega$ , and the given values of R and C, the magnitude and phase of the frequency response function are:

$$|H(j2)| = \frac{1}{\sqrt{1 + (\omega RC)^2}} = \frac{1}{\sqrt{1 + (2 \cdot 2\Omega \cdot 0.25F)^2}} = \frac{1}{\sqrt{1 + 1^2}} = \frac{1}{\sqrt{2}}$$

$$\angle H(j2) = -\tan^{-1}(\omega RC) = -\tan^{-1}(2 \cdot 2\Omega \cdot 0.25F) = -\tan^{-1}(1) = -45^\circ$$

The output amplitude is then the product of  $|V_{in}|$  and  $|H(j2)|$  and the output phase is the sum of  $\angle V_{in}$  and  $\angle H(j2)$ , so that:

$$|V_{out}| = |V_{in}| \cdot |H(j2)| = 3 \cdot \frac{1}{\sqrt{2}} = \frac{3}{\sqrt{2}}$$

$$\angle V_{out} = \angle V_{in} + \angle H(j2) = 20^\circ + (-45^\circ) = -25^\circ$$

And the time-domain output voltage is:

$$v_{out}(t) = \frac{3}{\sqrt{2}} \cos(2t - 25^\circ)$$

When  $v_{in} = 7 \cos(4t - 60^\circ)$ ,  $\omega = 4$  rad/sec,  $|V_{in}| = 7$  and  $\angle V_{in} = -60^\circ$ . For this value of  $\omega$ , and the given values of R and C, the magnitude and phase of the frequency response function are:

$$|H(j4)| = \frac{1}{\sqrt{1 + (\omega RC)^2}} = \frac{1}{\sqrt{5}}$$

And:

$$\angle H(j4) = -\tan^{-1}(\omega RC) = -63.4^\circ$$

The output amplitude is then the product of  $|V_{in}|$  and  $|H(j4)|$  and the output phase is the sum of  $\angle V_{in}$  and  $\angle H(j4)$  so that the time-domain output voltage in this case is:

$$V_{out}(t) = \frac{7}{\sqrt{5}} \cos(4t - 12.4^\circ)$$

From the above examples we can see that, once the frequency response function is calculated for a circuit as a function of frequency, we can determine the circuit's steady-state response to any input sinusoid directly from the frequency response function, without re-analyzing the circuit itself.

We conclude this section with one additional example, to illustrate the use of the frequency response function and superposition to determine a circuit's response to multiple inputs of different frequencies.

### Example 10.20

Use the results of examples 10.18 and 10.19 above to determine the response  $v_{out}(t)$  of the circuit shown below if the input voltage is  $v_{in}(t) = 3\cos(2t+20^\circ) + 7\cos(4t-60^\circ)$ . Plot the input and output waveforms.



Recall, from section 10.5, that superposition is the only valid approach for performing frequency domain analysis of circuits with inputs at multiple frequencies. Also recall that each frequency can be analyzed separately in the frequency domain, but that the superposition process (the summation of the individual contributions) must be done in the time domain. For this problem, we have contributions at two different frequencies: 2 rad/sec and 4 rad/sec. Luckily, we have determined the individual responses of the circuit to these two inputs in Example 10.19. Therefore, in the time domain, the two contributions to our output will be:

$$v_1(t) = \frac{3}{\sqrt{2}} \cos(2t - 25^\circ)$$

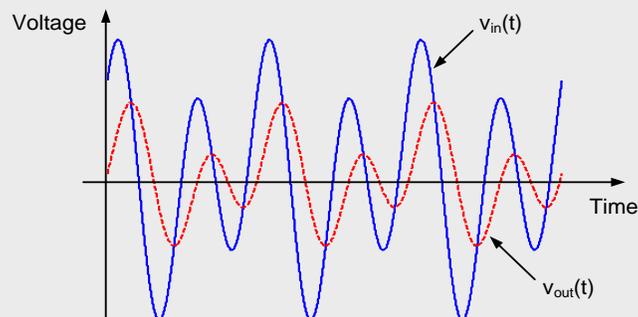
And:

$$v_2(t) = \frac{7}{\sqrt{5}} \cos(2t - 123.4^\circ)$$

The overall response is then:

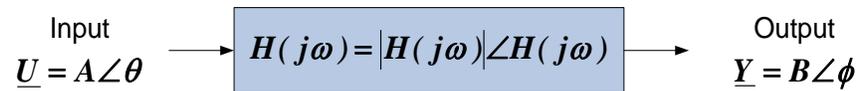
$$v_{out}(t) = v_1(t) + v_2(t) = \frac{3}{\sqrt{2}} \cos(2t - 25^\circ) + \frac{7}{\sqrt{5}} \cos(2t - 123.4^\circ)$$

A plot of the input and output waveforms is shown below:



## Section Summary

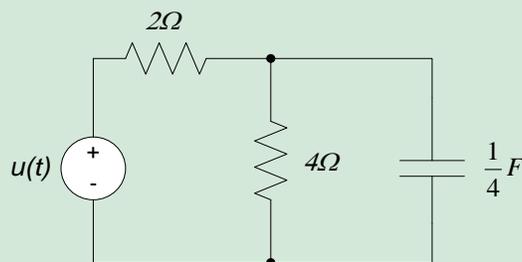
- The frequency response function or *frequency response* describes a circuit's input-output relationship directly in the frequency domain, as a function of frequency.
- The frequency response is a complex function of frequency  $H(j\omega)$  (that is, it is a complex number which depends upon the frequency). This complex function is generally expressed as a magnitude and phase,  $|H(j\omega)|$  and  $\angle H(j\omega)$ , respectively.  $|H(j\omega)|$  is called the *magnitude response* of the circuit, and  $\angle H(j\omega)$  is called the *phase response* of the circuit. The overall idea is illustrated in the block diagram below:



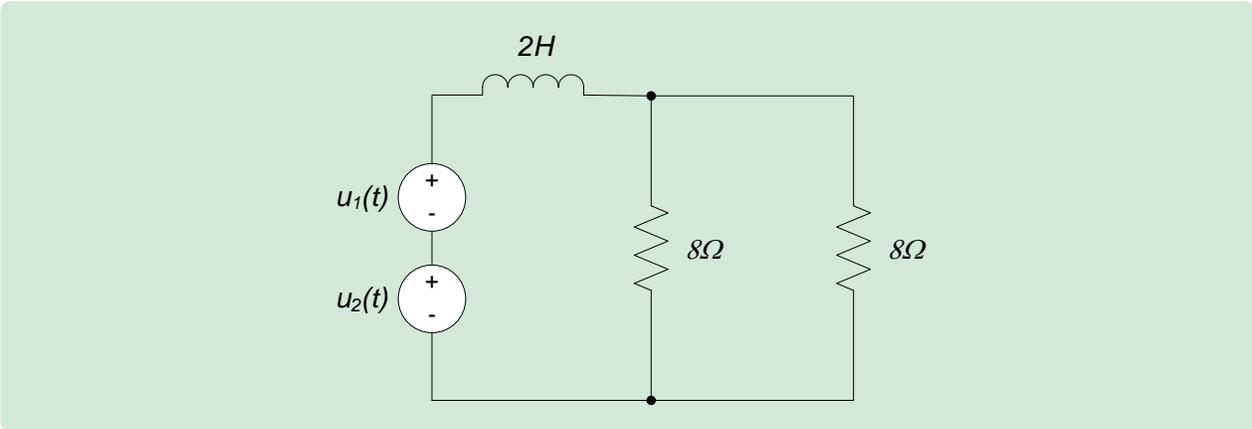
- The magnitude response of the circuit is the ratio of the output amplitude to the input amplitude. This is also called the gain of the system. Thus, in the figure above, the output amplitude  $B = |H(j\omega)| \cdot A$ . Note that the magnitude response or gain of the system is a function of frequency, so that inputs of different frequencies will have different gains.
- The phase response of the circuit is the difference between the output phase angle and the input phase angle. Thus, in the figure above, the output phase  $\phi = \angle H(j\omega) + \theta$ . Like the gain, the phase response is a function of frequency – inputs at different frequencies will, in general, have different phase shifts.
- Use of the frequency response to perform circuit analyses can be particularly helpful when the input signal contains a number of sinusoidal components at different frequencies. In this case, the response of the circuit to each individual component can be determined in the frequency domain using the frequency response and the resulting contributions summed in the time domain to obtain the overall response.

## 10.5 Exercises

1. Determine the voltage across the capacitor in the circuit below if  $u(t) = 4\cos(t+30^\circ) + 2\cos(2t-45^\circ)$ . (Hint: this may be easier if you find the response to the input as a function of frequency, evaluate the response for each of the above frequency components, and superimpose the results.)



2. Determine the voltage across the resistors in the circuit below if  $u_1(t) = 4\cos(2t)$  and if  $u_2(t) = \cos(4t)$ . (Hint: this may be easier if you find the response to the input as a function of frequency, evaluate the response for each of the above frequency components, and superimpose the results.)



# Real Analog Chapter 10: Lab Projects

## 10.4.1: Impedance

In this lab assignment, we measure impedances of resistors, capacitors, and inductors. The measured values will be compared with our expectations based on analyses.

### Before beginning this lab, you should be able to:

- Represent sinusoidal signals in phasor form

### After completing this lab, you should be able to:

- Measure impedances of passive circuit elements

### This lab exercise requires:

- Analog Discovery 2 module
- Digilent Analog Parts Kit
- Digital multimeter (optional)

### Symbol Key:

- DEMO** Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
- ANALYSIS** Analysis; include principle results of analysis in laboratory report.
- SIM** Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
- DATA** Record data in your lab notebook.

### General Discussion:

The concept of impedance is only appropriate in terms of the steady-state response of a circuit to a sinusoidal input. Impedance is a complex number which provides the relationship between voltage and current phasors in the circuit. Specifically, the impedance  $Z$  is the ratio of the voltage phasor to the current phasor:

$$Z = \frac{V}{I} = \frac{I e^{j\phi}}{V e^{j\theta}} = \left| \frac{V}{I} \right| e^{j(\phi-\theta)} \quad \text{Eq. 1}$$

where the voltage and current of interest,  $v(t)$  and  $i(t)$ , are assumed to be complex exponentials of the form:

$$v(t) = V e^{j(\omega t + \theta)} = \underline{V} e^{j\omega t} \quad \text{Eq. 2}$$

$$i(t) = I e^{j(\omega t + \phi)} = \underline{I} e^{j\omega t} \quad \text{Eq. 3}$$

$\underline{I}$  and  $\underline{V}$  are *phasors* representing the magnitude and phase of the current and voltage, respectively. Impedance is a very general concept which can be applied to any combination of voltage and current in a circuit. In this lab project, however, we will be interested only in the impedance of specific circuit elements: resistors, capacitors, and inductors.

In order to experimentally determine impedance, we must determine both voltage and current. Since oscilloscopes do not measure current, we will use the measured voltage across a known resistance in order to infer the current through the circuit element of interest. The appropriate circuit schematic is as shown in Fig. 1.

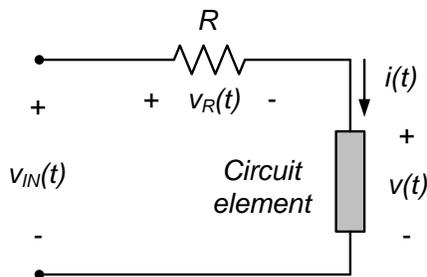


Figure 1. Circuit used for impedance measurements.

In the circuit of Fig. 1, we can measure the voltages  $v_R(t)$  and  $v(t)$ . The current through the circuit element of interest can be estimated from Ohm’s law as:

$$i(t) = \frac{v_R(t)}{R} \tag{Eq. 4}$$

By measuring the voltage  $v(t)$  and estimating the current  $i(t)$  for the circuit element in Fig. 1, we can determine the circuit element’s impedance from equation (1).

**Pre-lab:**

**ANALYSIS**

Assume that the voltages  $v_R(t)$  and  $v(t)$  in Figures 2 below are of the form:

$$v_R(t) = v_R \cos(\omega t + \theta)$$

$$v(t) = V \cos(\omega t + \varphi)$$

Determine the impedances of the impedances of the resistor R, the inductor L, and the capacitor C in Fig. 2 below in terms of the phasor representations of the voltages  $v_R(t)$  and  $v(t)$ . Express your results in terms of the magnitudes and phase angles of  $v_R(t)$  and  $v(t)$ .

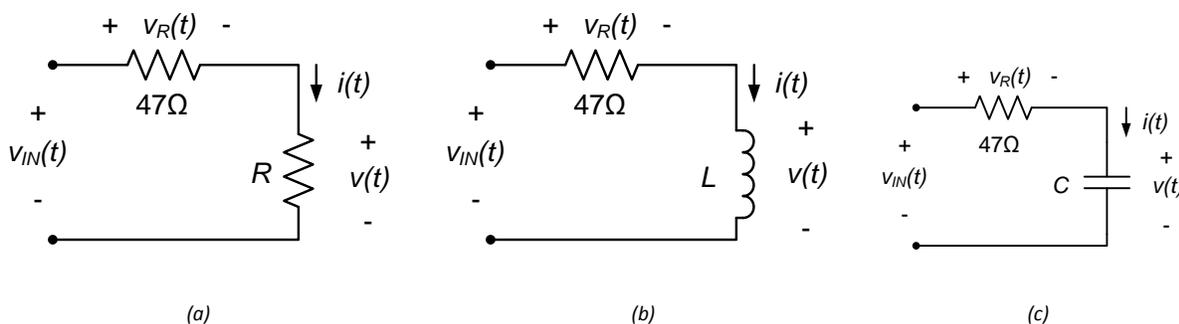


Figure 2. Circuits used in this lab project.

**Lab Procedures:**

- a. Construct the circuit of Fig. 2(a) with  $R = 100\Omega$ .
  - i. Use your function generator to apply a sinusoidal input voltage  $v_{IN}(t)$  with an amplitude of 2V and a 0V offset. Use your oscilloscope to measure the voltages  $v_R(t)$  and  $v(t)$ . Set up a math channel to display the current  $i(t)$ , according to equation (4). Record an image of the oscilloscope window, showing the signals  $v_R(t)$ ,  $v(t)$ , and  $i(t)$  for input signals with the following frequencies:
    1. 1kHz
    2. 5kHz

**DATA**

3. 10kHz

DATA

- ii. For each of the above three frequencies, tabulate:
- the amplitudes of  $v(t)$  and  $i(t)$ , and
  - the time difference between  $v(t)$  and  $i(t)$ .

ANALYSIS

- iii. Calculate the impedance of the resistor at the above three frequencies. Compare your results to your expectations from the pre-lab analyses. Include a percent difference between your expectations and your measured impedances. **Note: Appendix A of this lab assignment provides tips relative to gain and phase measurement of sinusoidal signals.**

DEMO

- iv. Demonstrate operation of your circuit to the TA and have them initial the appropriate pages of your lab notebook and the lab worksheet.

b. Construct the circuit of Fig. 2(b) with  $L = 1\text{mH}$ .

- i. Use your function generator to apply a sinusoidal input voltage  $v_{IN}(t)$  with an amplitude of 2V and a 0V offset. Use your oscilloscope to measure the voltages  $v_R(t)$  and  $v(t)$ . Set up a math channel to display the current  $i(t)$ , according to equation (4). Record an image of the oscilloscope window, showing the signals  $v_R(t)$ ,  $v(t)$ , and  $i(t)$  for input signals with the following frequencies:

4. 1kHz
5. 5kHz
6. 10kHz

DATA

- ii. For each of the above three frequencies, tabulate:
- the amplitudes of  $v(t)$  and  $i(t)$ , and
  - the time difference between  $v(t)$  and  $i(t)$ .

ANALYSIS

- iii. Calculate the impedance of the inductor at the above three frequencies. Compare your results to your expectations from the pre-lab analyses. Include a percent difference between your expectations and your measured impedances. **Note: Appendix A of this lab assignment provides tips relative to gain and phase measurement of sinusoidal signals.**

DEMO

- iv. Demonstrate operation of your circuit to the TA and have them initial the appropriate pages of your lab notebook and the lab worksheet.

c. Construct the circuit of Figure 2(c) with  $C = 100\text{nF}$ .

- i. Use your function generator to apply a sinusoidal input voltage  $v_{IN}(t)$  with an amplitude of 2V and a 0V offset. Use your oscilloscope to measure the voltages  $v_R(t)$  and  $v(t)$ . Set up a math channel to display the current  $i(t)$ , according to equation (4). Record an image of the oscilloscope window, showing the signals  $v_R(t)$ ,  $v(t)$ , and  $i(t)$  for input signals with the following frequencies:

7. 1kHz
8. 5kHz
9. 10kHz

DATA

- ii. For each of the above three frequencies, tabulate:
- the amplitudes of  $v(t)$  and  $i(t)$ , and
  - the time difference between  $v(t)$  and  $i(t)$ .

ANALYSIS

- iii. Calculate the impedance of the capacitor at the above three frequencies. Compare your results to your expectations from the pre-lab analyses. Include a percent difference between your expectations and your measured impedances. **Note: Appendix A of this lab assignment provides tips relative to gain and phase measurement of sinusoidal signals.**

DEMO

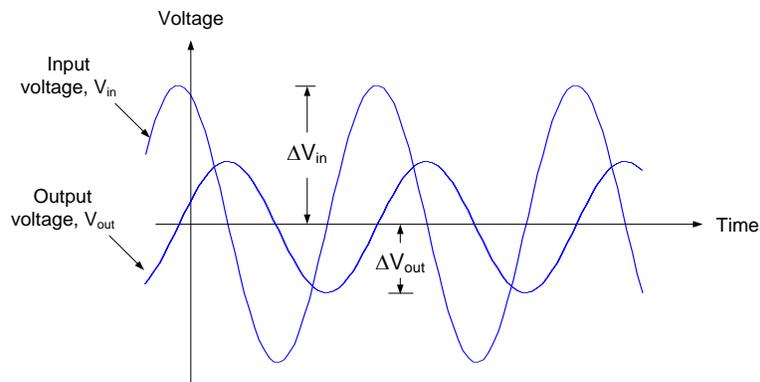
- iv. Demonstrate operation of your circuit to the TA and have them initial the appropriate pages of your lab notebook and the lab worksheet.

## Appendix A: Measuring Gain and Phase:

The gain of a system at a particular frequency is the ratio of the magnitude of the output voltage to the magnitude of the input voltage at that frequency, so that:

$$Gain = \frac{\Delta V_{out}}{\Delta V_{in}}$$

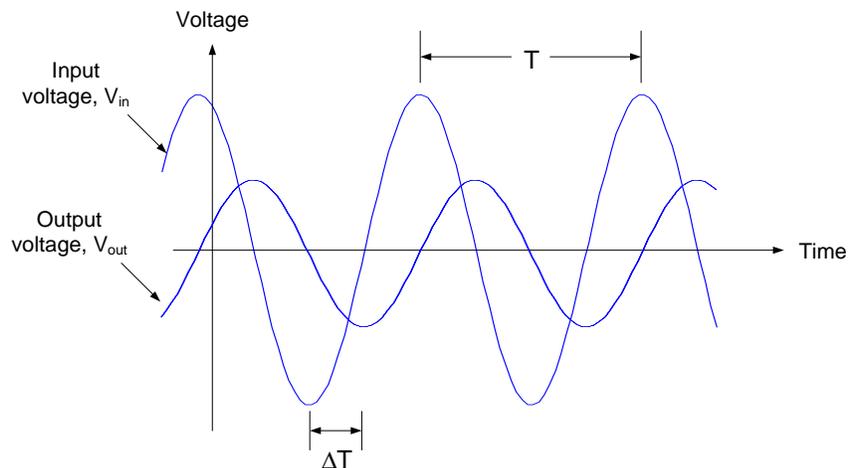
Where  $\Delta V_{out}$  and  $\Delta V_{in}$  can be measured from the sinusoidal input and output voltages as shown in the figure below.



The phase of a system at a particular frequency is a measure of the time shift between the output and input voltage at that frequency, so that:

$$Phase = \frac{\Delta T}{T} \times 360^\circ$$

where  $\Delta T$  and  $T$  can be measured from the sinusoidal input and output voltages as shown in the figure below.









## Real Analog Chapter 10: Lab Projects

### 10.6.1: Passive RL Circuit Response

In this lab assignment, we will be concerned with the steady-state response of electrical circuits to sinusoidal inputs. Figure 1(a) shows a block-diagram representation of the system. The input and output signals both have the same frequency, but the two signals can have different amplitudes and phase angles.

The analysis of the circuit of Fig. 1(a) can be simplified by representing the sinusoidal signals as *phasors*. The phasors provide the amplitude and phase information of sinusoidal signals. By comparing the phasors representing the input and output signals, the effect of the circuit can be represented as an amplitude gain between the output and input signals and a phase difference between the output and input signals, as shown in Fig. 1(b).

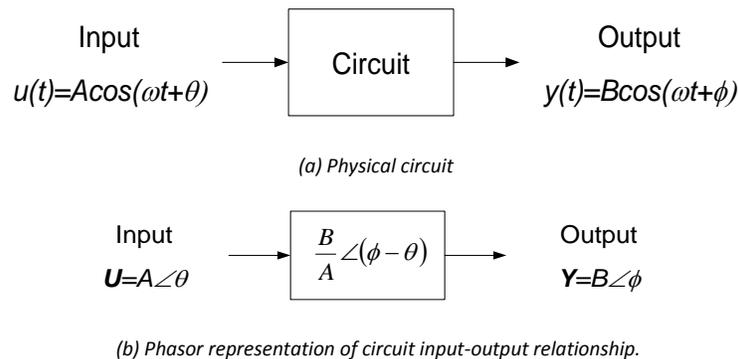


Figure 1. Steady-state sinusoidal circuit analysis

In this lab assignment, we will measure the gain and phase responses of a passive RL circuit and compare these measurements with expectations based on analysis.

#### Before beginning this lab, you should be able to:

- Represent sinusoidal signals in phasor form
- Determine electrical circuit steady-state sinusoidal responses in phasor form

#### After completing this lab, you should be able to:

- Measure phasor form of circuit steady-state sinusoidal response
- Measure input impedance of electrical circuit

#### This lab exercise requires:

- Analog Discovery 2 module
- Digilent Analog Parts Kit
- Digital multimeter (optional)

#### Symbol Key:

- DEMO** Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
- ANALYSIS** Analysis; include principle results of analysis in laboratory report.
- SIM** Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
- DATA** Record data in your lab notebook.

## General Discussion:

Consider the RL circuit shown in Fig. 2 below. The input to the circuit is an applied voltage and we choose the current supplied by the source to be the system output. The differential equation relating the applied voltage  $v_{IN}(t)$  to the input current  $i_{IN}(t)$  can be obtained by applying KVL around the single loop:

If we assume that the input voltage and current are complex exponentials of the form:

$$v_{IN}(t) = V e^{j(\omega t + \theta)} \quad \text{Eq. 1}$$

$$i_{IN}(t) = I e^{j(\omega t + \varphi)} \quad \text{Eq. 2}$$

We can write the circuit's input-output relation as a ratio between the current and the voltage:

$$\frac{I}{V} = \frac{I e^{j\varphi}}{V e^{j\theta}} = \frac{1}{R + j\omega L} \quad \text{Eq. 3}$$

Where  $I$  and  $V$  are *phasors* representing the magnitude and phase of the input current and input voltage to the circuit, respectively. This input-output relation can be written in terms of an amplitude gain and a phase shift:

$$\left| \frac{I}{V} \right| = \left| \frac{1}{R + j\omega L} \right| = \frac{1}{\sqrt{R^2 + \omega^2 L^2}} \quad \text{Eq. 4}$$

$$\varphi - \theta = -\tan^{-1} \left( \frac{\omega L}{R} \right) \quad \text{Eq. 5}$$

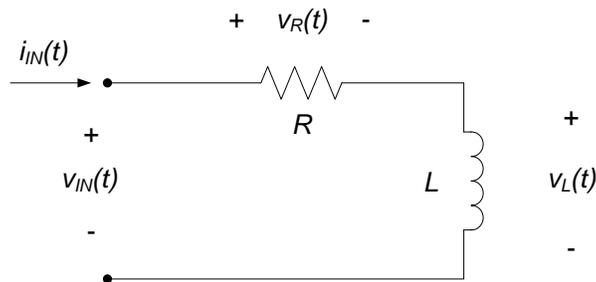


Figure 2. RL circuit.

## Pre-lab:

### ANALYSIS

a. Show that the amplitude gain and phase difference between the input voltage and the input current are as shown in equations (4) and (5).

### ANALYSIS

b. The cutoff frequency for the circuit of Fig. 2 is given to be  $\omega_c = \frac{R}{L}$ . Calculate the cutoff frequency for the circuit of Fig. 2 if  $L = 1\text{mH}$  and  $R = 47\Omega$ .

### ANALYSIS

c. Determine the gain and phase difference for the RL circuit for frequencies  $\omega \approx 0$ ,  $\omega \rightarrow \infty$ , and  $\omega = \omega_c$  if  $L = 1\text{mH}$  and  $R = 47\Omega$ .

### ANALYSIS

d. Do your low and high frequency gain results in part (c) agree with your expectations based on the inductor's low and high frequency behavior? (e.g. calculate the inductor impedance at low and high frequencies, substitute these impedances into the circuit of Fig. 2, calculate the response of the resulting resistive circuit, and compare to the results of part (c).)

## Notes:

In this lab assignment, we will measure  $v_{IN}(t)$  and  $v_L(t)$ . These measurements will be used to estimate the gain and phase difference between  $v_{IN}(t)$  and  $i_{IN}(t)$  and the gain and phase difference between  $v_L(t)$  and  $i_{IN}(t)$ . These results will be compared with our expectations based on the pre-lab analyses. We do not have the ability to directly measure a time-varying current, so we will infer  $i_{IN}(t)$  by measuring  $v_{IN}(t) - v_L(t)$  and determining  $i_{IN}(t)$  by:

$$i_{IN}(t) = \frac{v_{IN}(t) - v_L(t)}{R}$$

Eq. 6

All signals we will be dealing with are sinusoidal. Appendix A of this lab assignment provides tips relative to gain and phase measurement of sinusoidal signals.

### Lab Procedures:

Construct the circuit of Fig. 2 with  $L = 1\text{mH}$  and  $R = 47\Omega$ .

#### DATA

- a. Use your function generator to apply a sinusoidal input at  $v_{IN}(t)$ . Use your oscilloscope to display both  $v_{IN}(t)$  and  $v_L(t)$ . Use the oscilloscope's math operation to display the input current,  $i_{IN}(t)$ , as provided by equation (6). Record the amplitude of  $v_{IN}(t)$  and  $i_{IN}(t)$  and the time delay between  $v_{IN}(t)$  and  $i_{IN}(t)$  for the following input voltage frequencies:

- $\omega \approx \frac{\omega_c}{10}$  (low frequency input)
- $\omega \approx 10\omega_c$  (high frequency input)
- $\omega \approx \omega_c$  (corner frequency input)

#### DEMO

- b. Demonstrate operation of your circuit to the TA and have them initial the appropriate page(s) of your lab notebook and the lab worksheet.

#### ANALYSIS

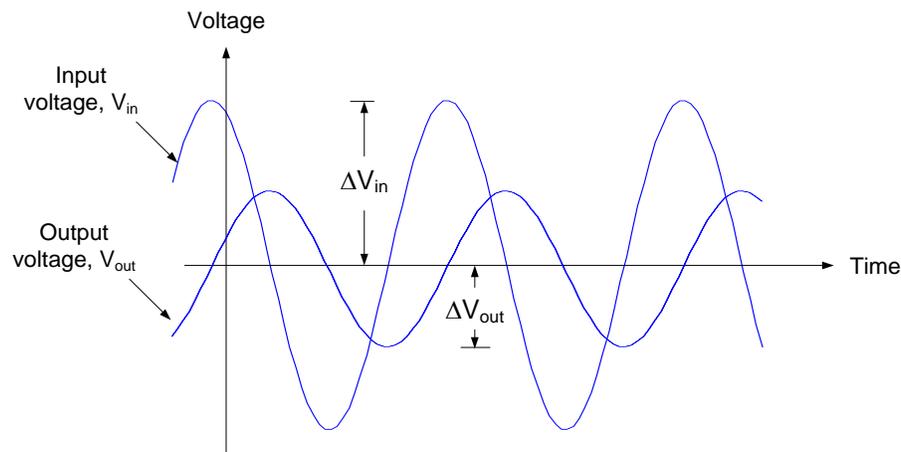
- c. Calculate the measured gains and phase differences between  $i_{IN}(t)$  and  $v_{IN}(t)$  for the three frequencies listed in part (b) above. Compare your measured results with your expectations from the pre-lab. Comment on your results.

## Appendix A: Measuring Gain and Phase

The gain of a system at a particular frequency is the ratio of the magnitude of the output voltage to the magnitude of the input voltage at that frequency, so that:

$$\text{Gain} = \frac{\Delta V_{out}}{\Delta V_{in}}$$

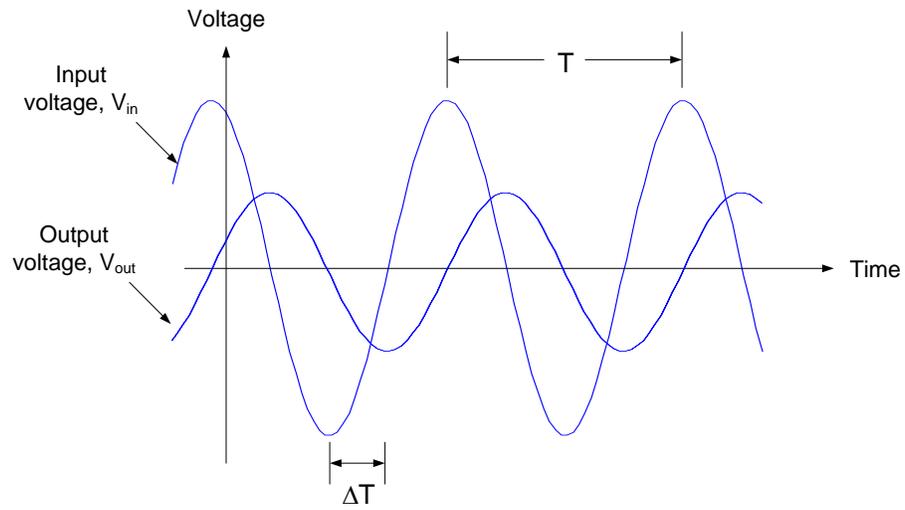
where  $\Delta V_{out}$  and  $\Delta V_{in}$  can be measured from the sinusoidal input and output voltages as shown in the figure below.



The phase of a system at a particular frequency is a measure of the time shift between the output and input voltage at that frequency, so that:

$$\text{Phase} = \frac{\Delta T}{T} \times 360^\circ$$

where  $\Delta V_{in}$  and  $T$  can be measured from the sinusoidal input and output voltages as shown in the figure below.



## Real Analog Chapter 10: Lab Worksheets

### 10.6.1: Passive RL Circuit Response (40 points total)

1. Attach, to this worksheet, your derivation of gain and phase expressions (equations (5) and (6)) (3 pts)
2. In the space below, provide the cutoff frequency calculated in part (b) of the pre-lab. (3 pts)
3. In the space below, provide the gain ( $\frac{I}{V}$ ) and phase ( $\angle I - \angle V$ ) for the RL circuit at low, high, and corner frequencies as determined from part (c) of your pre-lab analysis. (9 pts)
4. Comment below on the inductor physical behavior at low and high frequencies vs. expressions provided in (2) above. (2 pts)
5. In the space below, provide a table listing  $v_{IN}(t)$  and  $i_{IN}(t)$  and time delays between  $v_{IN}(t)$  and  $i_{IN}(t)$  for the three frequencies of interest in part (a) of the lab procedures. (10 pts)

6. In the space below, provide a table listing the measured gains and phase differences between  $i_{IN}(t)$  and  $v_{IN}(t)$  and  $i_{IN}(t)$  and  $v_L(t)$  for the three frequencies of interest. (8 pts)

7. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your system's operation. (5 pts total)

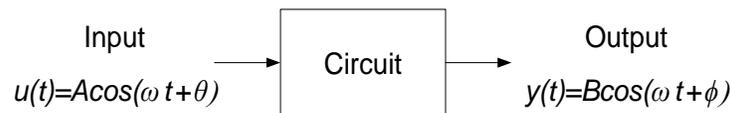
TA Initials: \_\_\_\_\_

## Real Analog Chapter 10: Lab Projects

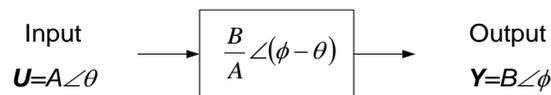
### 10.6.2: Passive RC Circuit Response

In this lab assignment, we will be concerned with the steady-state response of electrical circuits to sinusoidal inputs. Figure 1(a) shows a block-diagram representation of the system. The input and output signals both have the same frequency, but the two signals can have different amplitudes and phase angles.

The analysis of the circuit of Fig. 1(a) can be simplified by representing the sinusoidal signals as *phasors*. The phasors provide the amplitude and phase information of sinusoidal signals. By comparing the phasors representing the input and output signals, the effect of the circuit can be represented as an amplitude gain between the output and input signals and a phase difference between the output and input signals, as shown in Fig. 1(b).



(a) Physical circuit



(b) Phasor representation of circuit input-output relationship.

Figure 1. Steady-state sinusoidal circuit analysis

In this lab assignment, we will measure the gain and phase responses of a passive RC circuit and compare these measurements with expectations based on analysis. These measurements will be used to estimate the impedance of the overall RC circuit.

**Before beginning this lab, you should be able to:**

- Represent sinusoidal signals in phasor form
- Determine electrical circuit steady-state sinusoidal responses in phasor form

**After completing this lab, you should be able to:**

- Measure phasor form of circuit steady-state sinusoidal response
- Measure input impedance of electrical circuit

**This lab exercise requires:**

- Analog Discovery 2 module
- Digilent Analog Parts Kit
- Digital multimeter (optional)

## Symbol Key:

- DEMO** Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
- ANALYSIS** Analysis; include principle results of analysis in laboratory report.
- SIM** Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
- DATA** Record data in your lab notebook.

## General Discussion:

In this lab assignment, we will determine the input impedance of the passive RC circuit shown in Fig. 1. The input impedance of a circuit is defined as the ratio of input voltage to input current. Thus, for the circuit of Fig. 1, the input impedance is represented in phasor form as:

$$\underline{Z}_{IN} = \frac{\underline{V}_{IN}}{\underline{I}_{IN}} \quad \text{Eq. 1}$$

Where  $\underline{V}_{IN}$  is the phasor representation of the circuit input voltage and  $\underline{I}_{IN}$  is the phasor representation of the input current to the circuit.

The cutoff frequency for the circuit of Fig. 1 is:

$$\omega_c = \frac{1}{RC} \quad \text{Eq. 2}$$

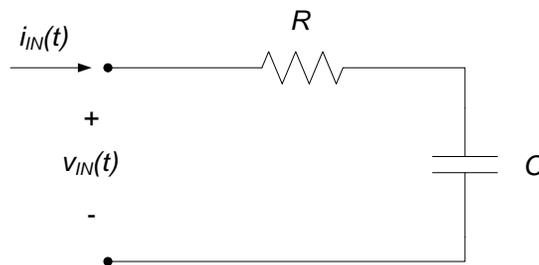


Figure 1. Passive RC circuit.

## Pre-lab:

- ANALYSIS** a. Determine an expression for the input impedance of the circuit of Fig. 1 in terms of  $R$ ,  $C$ , and  $\omega$ .
- ANALYSIS** b. If  $R = 100\Omega$  and  $C = 1\mu\text{F}$ , determine the cutoff frequency for the circuit. Also determine the input impedance for frequencies of:
- $\omega = \frac{\omega_c}{10}$  (low frequency input)
  - $\omega = 10\omega_c$  (high frequency input)
  - $\omega = \omega_c$  (corner frequency input)
- ANALYSIS** c. Check your low and high frequency results in part (b) relative to your expectations based on the capacitor's low and high frequency behavior.

## Lab Procedures:

Construct the circuit of Fig. 3, using  $R = 100\Omega$  and  $C = 1\mu\text{F}$ .

**DATA**

- a. Measure the input voltage amplitude, the input current amplitude, and the time delay between the input voltage and the input current for the following frequencies:
- $\omega \approx \frac{\omega_c}{10}$  (low frequency input)
  - $\omega \approx 10\omega_c$  (high frequency input)
  - $\omega \approx \omega_c$  (corner frequency input)

Use your data to calculate the input impedance (magnitude and phase) of the circuit for the above frequencies. Create a table providing the measured data and the calculated input impedances at the above frequencies.

**ANALYSIS**

- b. Compare your measured results with your expectations based on the analysis you did in the pre-lab.

**DEMO**

- c. Demonstrate operation of your circuit to the TA and have them initial the appropriate page(s) of your lab notebook and the lab worksheet.

**Hint:**

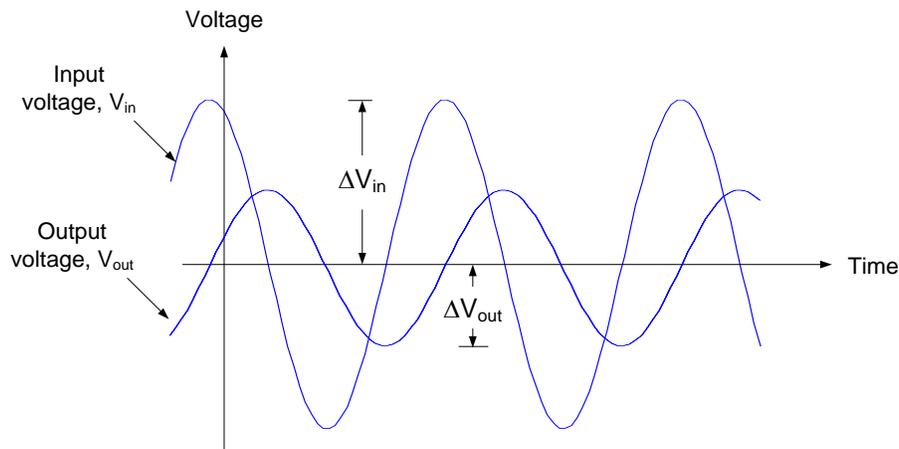
The process to perform the above lab procedures is comparable to the process performed in lab assignment 10.6.1. Be sure to record all necessary data and any calculations you perform to obtain your results in your lab notebook. Appendix A of this lab assignment provides tips relative to gain and phase measurement of sinusoidal signals.

## Appendix A: Measuring Gain and Phase:

The gain of a system at a particular frequency is the ratio of the magnitude of the output voltage to the magnitude of the input voltage at that frequency, so that:

$$\text{Gain} = \frac{\Delta V_{out}}{\Delta V_{in}}$$

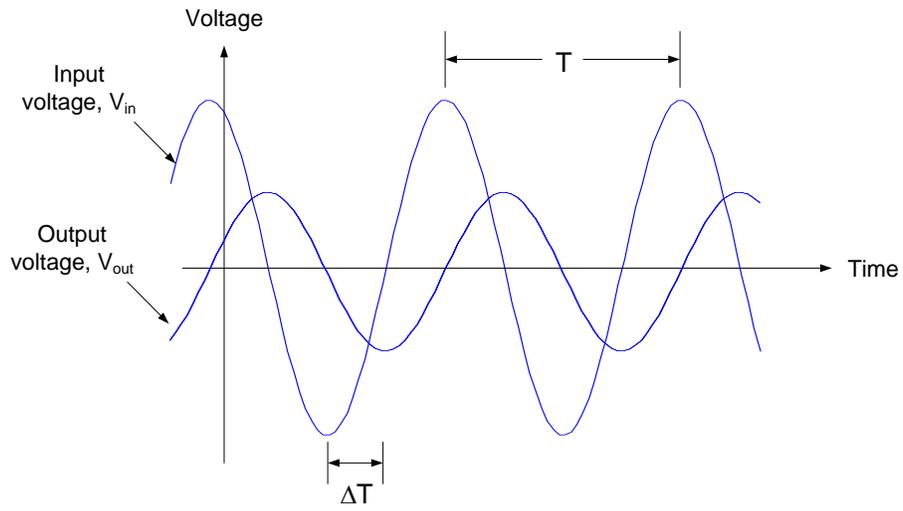
where  $\Delta V_{out}$  and  $\Delta V_{in}$  can be measured from the sinusoidal input and output voltages as shown in the figure below.



The phase of a system at a particular frequency is a measure of the time shift between the output and input voltage at that frequency, so that:

$$\text{Phase} = \frac{\Delta T}{T} \times 360^\circ$$

where  $\Delta T$  and  $T$  can be measured from the sinusoidal input and output voltages as shown in the figure below.







## Real Analog Chapter 10: Lab Projects

### 10.6.3: Inverting Voltage Amplifier

In this lab assignment, we will be concerned with the steady-state response of electrical circuits to sinusoidal inputs. Figure 1(a) shows a block-diagram representation of the system. The input and output signals both have the same frequency, but the two signals can have different amplitudes and phase angles.

The analysis of the circuit of Fig. 1(a) can be simplified by representing the sinusoidal signals as *phasors*. The phasors provide the amplitude and phase information of sinusoidal signals. By comparing the phasors representing the input and output signals, the effect of the circuit can be represented as an amplitude gain between the output and input signals and a phase difference between the output and input signals, as shown in Fig. 1(b).

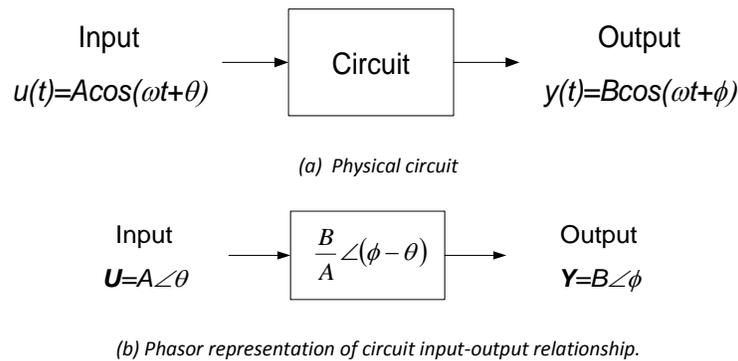


Figure 1. Steady-state sinusoidal circuit analysis

In this lab assignment, we will measure the gain and phase responses of an inverting voltage amplifier circuit and compare these measurements with expectations based on analysis.

#### Before beginning this lab, you should be able to:

- Represent sinusoidal signals in phasor form
- Represent electrical circuit steady-state sinusoidal responses in phasor form
- Analyze operational amplifier-based circuits

#### After completing this lab, you should be able to:

- Measure phasor form of circuit steady-state sinusoidal response
- Measure input impedance of electrical circuit

#### This lab exercise requires:

- Analog Discovery 2 module
- Digilent Analog Parts Kit
- Digital multimeter (optional)

#### Symbol Key:

- DEMO** Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
- ANALYSIS** Analysis; include principle results of analysis in laboratory report.
- SIM** Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
- DATA** Record data in your lab notebook.

## General Discussion:

In this lab assignment, we will measure the frequency domain input-output relation governing the inverting voltage amplifier shown in Fig. 1. The frequency domain input-output relation for the circuit of Fig. 1 is:

$$\frac{V_{OUT}}{V_{IN}} = -\frac{1}{j\omega RC + 1} \quad \text{Eq. 1}$$

So that the amplitude gain between the output and input is:

$$\left| \frac{V_{OUT}}{V_{IN}} \right| = -\frac{1}{\sqrt{(\omega RC)^2 + 1}} \quad \text{Eq. 2}$$

And the phase difference between the output and input is:

$$\angle V_{OUT} - \angle V_{IN} = 180^\circ - \tan^{-1}\left(\frac{1}{\omega RC}\right) \quad \text{Eq. 3}$$

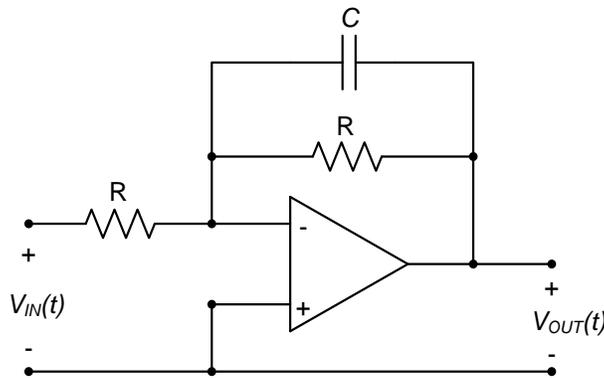


Figure 1. Inverting voltage amplifier.

## Pre-lab:

### ANALYSIS

- a. Show that equation (1) is the input-output relation for the circuit of Fig. 1. Also verify equations (2) and (3) above.

### ANALYSIS

- b. Determine the cutoff frequency of the circuit if  $R = 10\text{k}\Omega$  and  $C = 10\text{nF}$ . Also determine the amplitude gain and the phase difference between the circuit's input and output voltages for the circuit<sup>42</sup>. Also determine the input impedance for frequencies of:

- $\omega = \frac{\omega_c}{10}$  (low frequency input)
- $\omega = 10\omega_c$  (high frequency input)
- $\omega = \omega_c$  (corner frequency input)

### ANALYSIS

- c. Check your low and high frequency results in part (b) relative to your expectations based on the capacitor's low and high frequency behavior.

## Lab Procedures:

Construct the circuit of Fig. 2, using  $R = 10\text{k}\Omega$  and  $C = 10\text{nF}$ .

### DATA

- a. Use the waveform generator to apply a sinusoidal signal with 2V amplitude and 0V offset to the circuit. Set up the oscilloscope to measure both the input and output voltages. Measure the

<sup>42</sup> Be sure to use units of radians/second for  $\omega$  in equations (2) and (3)!

amplitudes of the input and output voltage signal, and the time delay between the input and output signal for inputs with the following frequencies:

- 100 Hz
- 1 KHz
- 5 KHz

**DATA**

- b. Record an image of the oscilloscope window, showing the signals  $V_{IN}(t)$  and  $V_{OUT}(t)$ , for each of the above frequencies.

**ANALYSIS**

- c. Use your measurements to calculate the amplitude gain and phase difference of the circuit for the above three frequencies. Compare your measured results with your expectations based on the analysis you did in the pre-lab.

**DEMO**

- d. Demonstrate operation of your circuit to the TA and have them initial the appropriate page(s) of your lab notebook and the lab worksheet.

**Hint:**

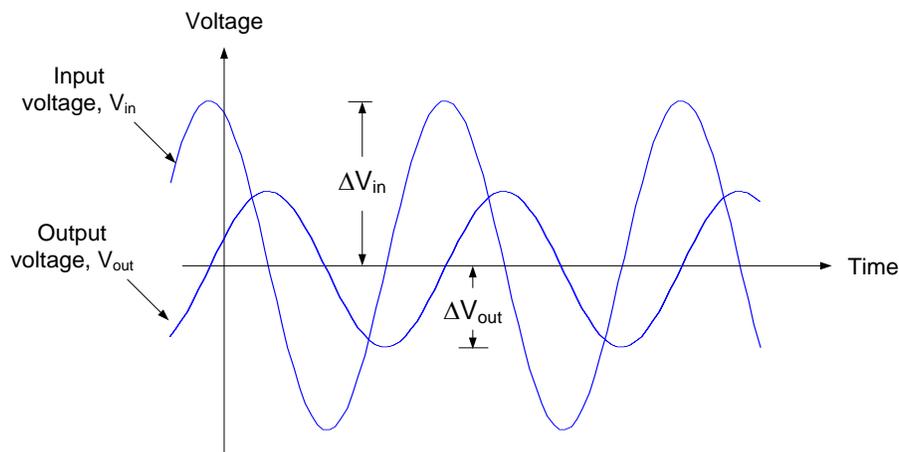
Be sure to record all necessary data and any calculations you perform to obtain your results in your lab notebook. Appendix A of this lab assignment provides tips relative to gain and phase measurement of sinusoidal signals.

## Appendix A: Measuring Gain and Phase:

The gain of a system at a particular frequency is the ratio of the magnitude of the output voltage to the magnitude of the input voltage at that frequency, so that:

$$\text{Gain} = \frac{\Delta V_{out}}{\Delta V_{in}}$$

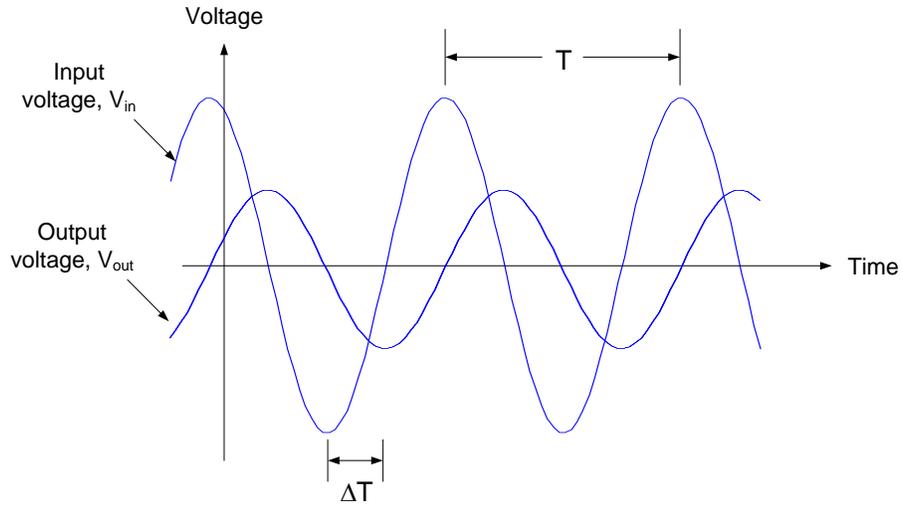
Where  $\Delta V_{out}$  and  $\Delta V_{in}$  can be measured from the sinusoidal input and output voltages as shown in the figure below.



The phase of a system at a particular frequency is a measure of the time shift between the output and input voltage at that frequency, so that:

$$\text{Phase} = \frac{\Delta T}{T} \times 360^\circ$$

Where  $\Delta T$  and  $T$  can be measured from the sinusoidal input and output voltages as shown in the figure below.



## Real Analog Chapter 10: Lab Worksheets

### 10.6.3: Inverting Voltage Amplifier (45 points total)

1. Attach, to this worksheet, your derivation of equations (1), (2), and (3). (3 pts)
2. In the space below, provide the cutoff frequency calculated in part (b) of the pre-lab. (3 pts)
3. In the space below, provide the gain, phase, and impedance for the circuit at low, high, and corner frequencies as determined from part (c) of your pre-lab analysis. (6 pts)
4. Comment below on the capacitor physical behavior at low and high frequencies vs. expressions provided in (3) above. (2 pts)
5. Attach to this worksheet images of the oscilloscope window, showing the input and output voltages as functions of time for each of the three specified frequencies. (6 pts, 2 pts each image)



## Real Analog Chapter 10: Lab Projects

### 10.6.4: Non-inverting Voltage Amplifier

In this lab assignment, we will be concerned with the steady-state response of electrical circuits to sinusoidal inputs. Figure 1(a) shows a block-diagram representation of the system. The input and output signals both have the same frequency, but the two signals can have different amplitudes and phase angles.

The analysis of the circuit of Fig. 1(a) can be simplified by representing the sinusoidal signals as *phasors*. The phasors provide the amplitude and phase information of sinusoidal signals. By comparing the phasors representing the input and output signals, the effect of the circuit can be represented as an amplitude gain between the output and input signals and a phase difference between the output and input signals, as shown in Fig. 1(b).

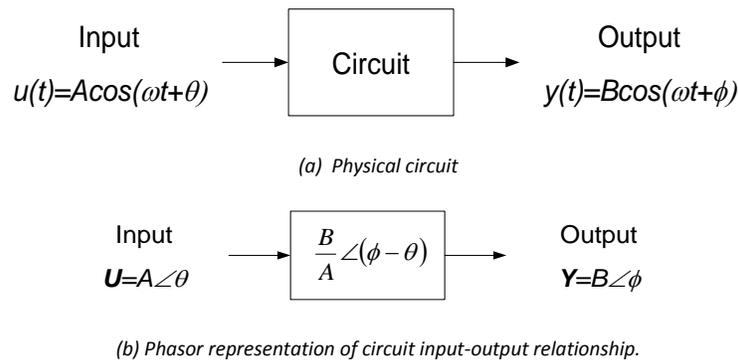


Figure 1. Steady-state sinusoidal circuit analysis

In this lab assignment, we will measure the gain and phase responses of a non-inverting voltage amplifier circuit and compare these measurements with expectations based on analysis.

#### Before beginning this lab, you should be able to:

- Represent sinusoidal signals in phasor form
- Represent electrical circuit steady-state sinusoidal responses in phasor form
- Analyze operational amplifier-based circuits

#### After completing this lab, you should be able to:

- Measure phasor form of circuit steady-state sinusoidal response
- Measure input impedance of electrical circuit

#### This lab exercise requires:

- Analog Discovery 2 module
- Digilent Analog Parts Kit
- Digital multimeter (optional)

#### Symbol Key:

- DEMO** Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
- ANALYSIS** Analysis; include principle results of analysis in laboratory report.
- SIM** Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
- DATA** Record data in your lab notebook.

## General Discussion:

In this lab assignment, we will measure the frequency domain input-output relation governing the voltage amplifier shown in Fig. 1. The frequency domain input-output relation for the circuit of Fig. 1 is:

$$\frac{V_{OUT}}{V_{IN}} = \frac{R_1 + R_2}{R_1} \frac{1}{j\omega + \frac{1}{R_3 C}} \quad \text{Eq. 1}$$

So that the amplitude gain and phase difference between the output and input are:

$$\left| \frac{V_{OUT}}{V_{IN}} \right| = 2 \frac{\frac{1}{RC}}{\sqrt{\omega^2 + \left(\frac{1}{RC}\right)^2}} \quad \text{Eq. 2}$$

$$\angle V_{OUT} - \angle V_{IN} = -\tan^{-1}(\omega RC) \quad \text{Eq. 3}$$

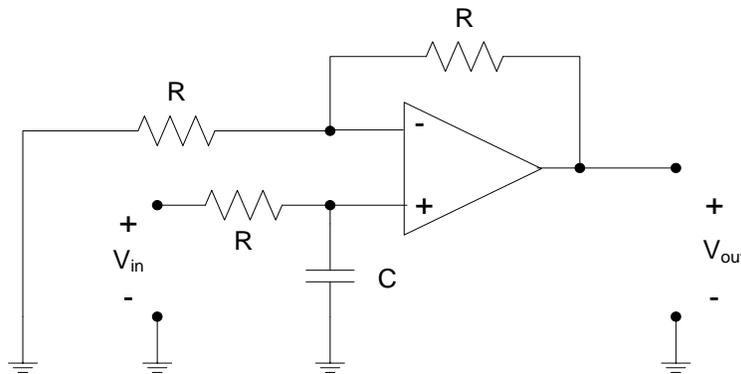


Figure 1. Non-inverting voltage amplifier.

## Pre-lab:

- ANALYSIS**
- Show that equation (1) is the input-output relation for the circuit of Fig. 1. Also verify equations (2) and (3) above.
- ANALYSIS**
- If  $R = 10\text{k}\Omega$  and  $C = 10\text{nF}$ , determine the amplitude gain and the phase difference between the circuit's input and output voltages for the circuit for input frequencies of 100Hz, 5kHz, and 10kHz<sup>43</sup>.
- ANALYSIS**
- Check your low and high frequency results in part (b) relative to your expectations based on the capacitor's low and high frequency behavior.

## Lab Procedures:

Construct the circuit of Fig. 2, using  $R = 10\text{k}\Omega$  and  $C = 10\text{nF}$ .

- DATA**
- Use the waveform generator to apply a sinusoidal signal with 1V amplitude and 0V offset to the circuit. Set up the oscilloscope to measure both the input and output voltages. Measure the amplitudes of the input and output voltage signal, and the time delay between the input and output signal for inputs with the following frequencies:
    - 100 Hz
    - 5 KHz
    - 10 KHz

<sup>43</sup> Be sure to use units of radians/second for  $\omega$  when evaluating equations (2) and (3)!

## DATA

- b. Record an image of the oscilloscope window, showing the signals  $V_{IN}(t)$  and  $V_{OUT}(t)$ , for each of the above frequencies.

## ANALYSIS

- c. Use your measurements to calculate the amplitude gain and phase difference of the circuit for the above three frequencies. Compare your measured results with your expectations based on the analysis you did in the pre-lab.

## DEMO

- d. Demonstrate operation of your circuit to the TA and have them initial the appropriate page(s) of your lab notebook and the lab worksheet.

## Hint:

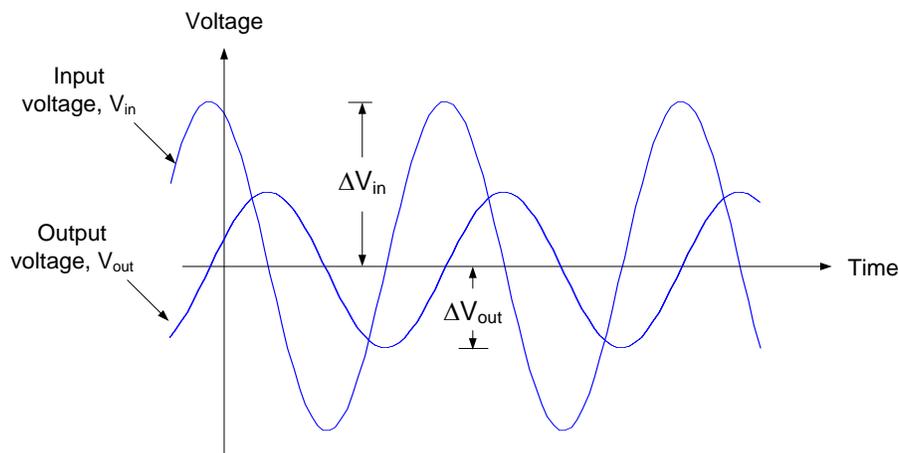
Be sure to record all necessary data and any calculations you perform to obtain your results in your lab notebook. Appendix A of this lab assignment provides tips relative to gain and phase measurement of sinusoidal signals.

## Appendix A: Measuring Gain and Phase

The gain of a system at a particular frequency is the ratio of the magnitude of the output voltage to the magnitude of the input voltage at that frequency, so that:

$$\text{Gain} = \frac{\Delta V_{out}}{\Delta V_{in}}$$

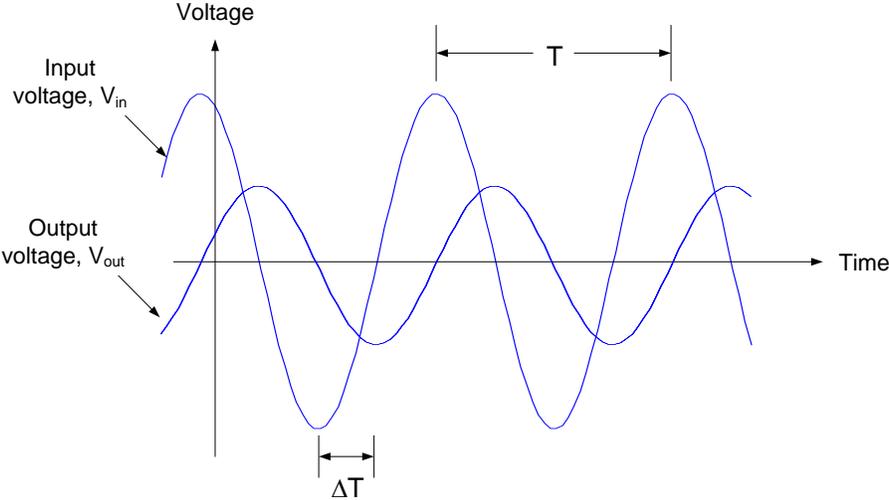
Where  $\Delta V_{out}$  and  $\Delta V_{in}$  can be measured from the sinusoidal input and output voltages as shown in the figure below.



The phase of a system at a particular frequency is a measure of the time shift between the output and input voltage at that frequency, so that:

$$\text{Phase} = \frac{\Delta T}{T} \times 360^\circ$$

Where  $\Delta T$  and  $T$  can be measured from the sinusoidal input and output voltages as shown in the figure below.



## Real Analog Chapter 10: Lab Worksheets

### 10.6.4: Non-inverting Voltage Amplifier (40 points total)

1. Attach, to this worksheet, your derivation of equations (1), (2), and (3). (3 pts)
2. In the space below, provide the calculated gain and phase for the circuit at frequencies of 100Hz, 5kHz, and 10kHz. (Part (b) of the pre-lab.) (4 pts)
3. Comment below on the capacitor physical behavior at low and high frequencies vs. expressions provided in (2) above. (2 pts)
4. Attach to this worksheet images of the oscilloscope window, showing the input and output voltages as functions of time for each of the three specified frequencies. (6 pts, 2 pts each image)



## Real Analog Chapter 10: Homework

10.1 A circuit is described by the differential equation:

$$2 \frac{di(t)}{dt} + 10i(t) = 10v(t)$$

If  $v(t) = 3\cos(5t)$ , determine the steady-state response of  $i(t)$ .

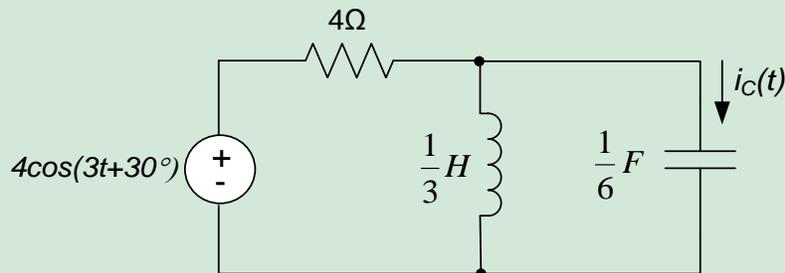
10.2 The differential equation governing a circuit is:

$$3 \frac{di(t)}{dt} + 6i(t) = v_s(t)$$

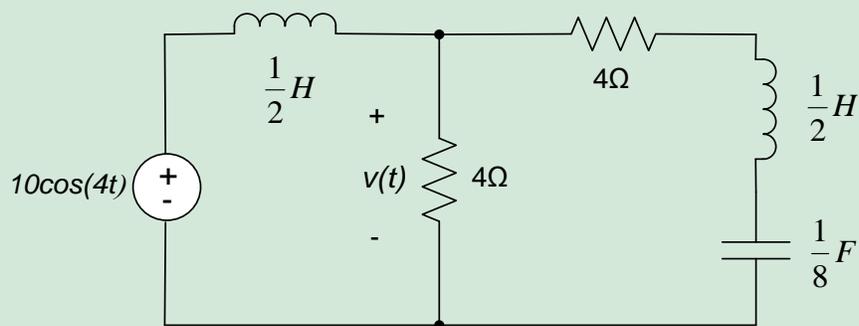
Where  $v_s(t)$  is the input and  $i(t)$  is the output. Determine the steady-state response of the circuit to an input  $v_s(t) = 5\cos(4t + 30^\circ)$ .

10.3 For the circuit below,

- The equivalent impedance seen by the source.
- $i_C(t)$ ,  $t \rightarrow \infty$ .

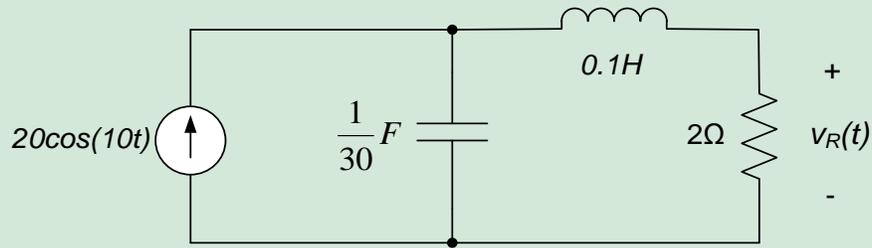


10.4 For the circuit below, find  $v(t)$ ,  $t \rightarrow \infty$ .



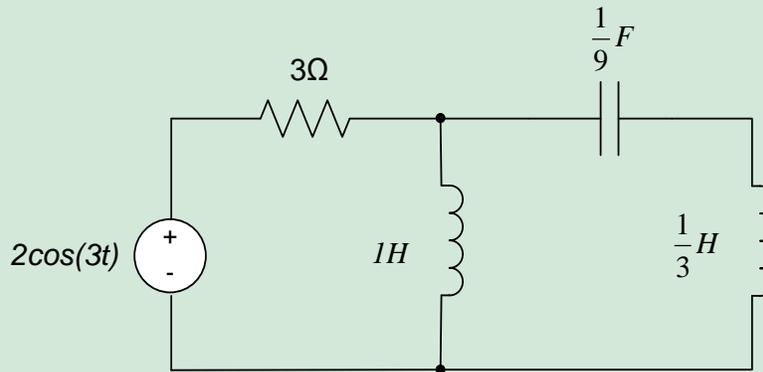
10.5 For the circuit shown, find

- The equivalent impedance seen by the source.
- The steady-state response of the voltage across the resistor,  $v_R(t)$ .



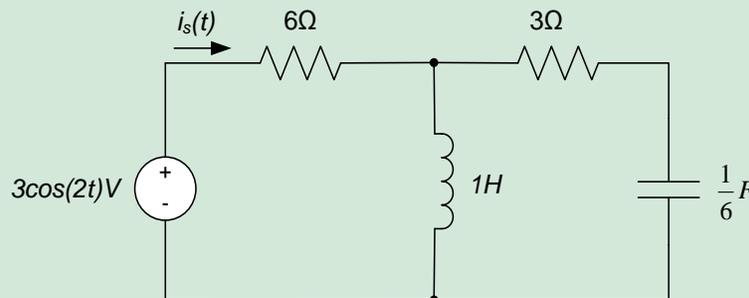
10.6 For the circuit shown, find

- The equivalent impedance seen by the source.
- The steady-state current delivered by the source.



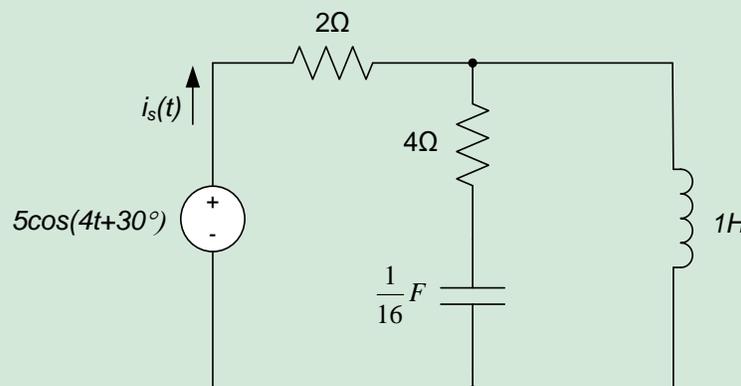
10.7 For the circuit shown, determine

- The equivalent impedance seen by the source.
- The steady-state current out of the source,  $i_s(t \rightarrow \infty)$ .

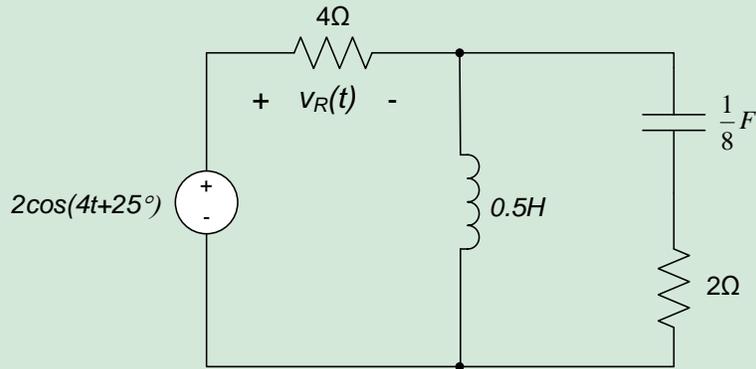


10.8 For the circuit shown, find

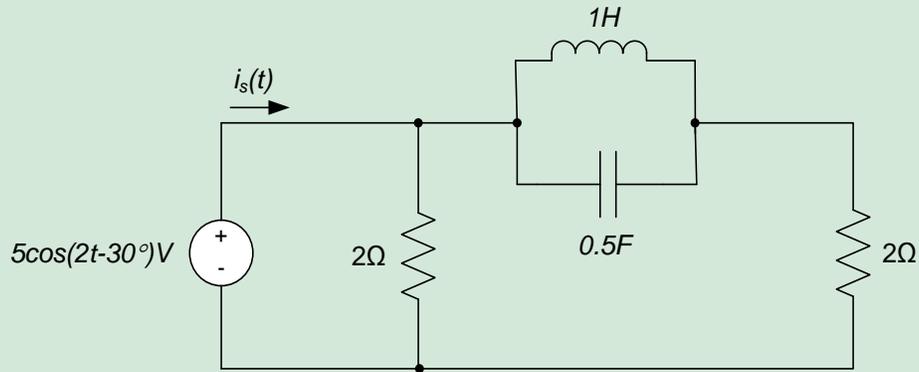
- The equivalent impedance seen by the source.
- $i_s(t), t \rightarrow \infty$ .



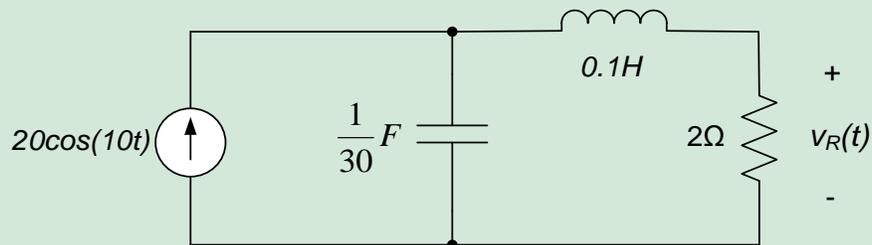
- 10.9 For the circuit shown, find
- The equivalent impedance seen by the source.
  - $v_R(t \rightarrow \infty)$ .



- 10.10 For the circuit shown, find  $i_s(t)$ ,  $t > 0$ .



- 10.11 For the circuit shown, find
- The equivalent impedance seen by the source.
  - The steady-state response of the voltage across the resistor,  $v_R(t)$ .



# Chapter 11: Frequency Response and Filtering

## 11 Introduction and Chapter Objectives

In section 10.6, we saw that a system's frequency response provided a steady-state input-output relationship for a system, as a function of frequency. We could apply this frequency response to the phasor representation of the input signal in order to determine the system's steady-state sinusoidal response – we simply evaluated the frequency response at the appropriate frequencies to determine the effect of the system on the input sinusoids. This approach had the potential for simplifying our analysis considerably, particularly for the case in which the input signal contained multiple sinusoids with different frequencies.

In Chapter 10, the signals we considered consisted only of individual sinusoids. It is more useful, however, in some ways to think in terms of the inputs and outputs of the system as functions of frequency, in the same way in which we considered the frequency response of the system to be a function of frequency in section 10.6. We can then perform our analysis of the system entirely in terms of the frequencies involved. This leads to the use of the system's frequency response directly as a design and analysis tool. In many cases, this means that the actual time-domain behavior of the system or signal is of limited interest (or in some cases, not considered at all). Some examples of frequency domain analyses are:

1. *Determining dominant sinusoidal frequency components in a measured signal.* Complex signals can often be represented as a superposition of several sinusoidal components with different frequencies. Identifying sinusoidal components with large amplitudes (the so-called dominant frequencies) can help with many design problems. One application of this is in the area of combustion instability – combustion processes in rocket engines can become unstable due to a variety of reasons, any of which can result in catastrophic failure of the engine. The type of instability which occurs is generally linked to a particular frequency; identification of the frequency of the pressure oscillations associated with the combustion instability is generally the first step in determining the cause of the instability.
2. *Designing systems to provide a desired frequency response.* Audio components in stereo systems are generally designed to produce a desired frequency response. A graphic equalizer, for example, can be used to boost (or amplify) some frequency ranges and attenuate other frequency ranges. When adjusting the settings on an equalizer, you are essentially directly adjusting the system's frequency response to provide a desired system response.

This chapter begins in section 11.1 with a brief review of frequency responses and an overview of the use of the frequency response in system analysis and design. In section 11.2, we discuss representation of signals in terms of their frequency content. At this time, we will also represent the frequency content of the input and output signals and the frequency response of the system in graphical format – this helps us visualize the frequency content of the signals and system. This leads us to think in terms of using a system to create a signal with a desired frequency content – this process is called filtering and is discussed in section 11.3. Using logarithmic scales to represent the signal and system frequency responses can – in many cases – simplify the analysis or design process; this format of presentation is called a Bode plot, and they are very briefly introduced in section 11.4. We will discuss Bode plots in more depth in later chapters.

It is important to keep in mind that, when we are performing frequency domain analyses, we are restricting our attention to the steady-state sinusoidal response of the system. Frequency domain design and analysis methods are so pervasive that they are often used to infer the system's transient response and/or its response to non-sinusoidal signals, so it is sometimes possible to forget the origins and limitations of the original concepts!

After completing this chapter, you should be able to:

- Use the frequency response of a system to determine the frequency domain response of a system to a given input
- State from memory the definition of *signal spectrum*
- Create plots of given signal spectra
- Plot a circuit's magnitude and phase responses
- Check a circuit's amplitude response at low and high frequencies against the expected physical behavior of the circuit
- Graphically represent a system's frequency domain response from provided signal spectra plots and plots of the system's frequency response
- Identify low pass and high pass filters
- Calculate a system's cutoff frequency
- Determine the DC gain of an electrical circuit
- Write, from memory, the equation used to convert gains to decibel form
- Sketch straight-line amplitude approximations to Bode plots
- Sketch straight-line phase approximations to Bode plots

## 11.1 Introduction to Steady-state Sinusoidal Analysis

In section 10.6, we defined the frequency response  $H(j\omega)$  of a system as a complex function of frequency which describes the relationship between the steady state sinusoidal response of a system and the corresponding sinusoidal input. Thus, if a sinusoidal input with some frequency  $\omega_0$  is applied to a system with frequency response  $H(j\omega)$ , the amplitude of the output sinusoid is the input sinusoid's amplitude multiplied by the magnitude response of the system, evaluated at the frequency  $\omega_0$ . The phase angle of the output sinusoid is the sum of the input sinusoid's phase and the phase response of the system, evaluated at the frequency  $\omega_0$ . The overall idea is presented in block diagram form in Fig. 11.1 below.

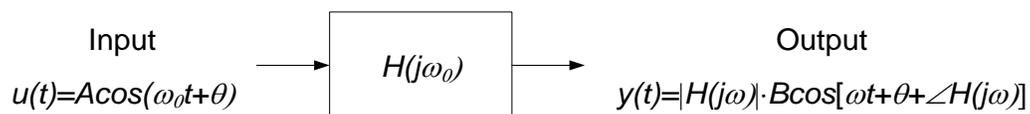


Figure 11.1. Frequency response used to determine steady state sinusoidal system response.

The true power of the frequency response is, however, if we consider both the system's input and output phasors to be complex functions of frequency, in the same way that the frequency response is a complex function of frequency. In this case, the block diagram of Fig. 11.1 can be represented as shown in Fig. 11.2.

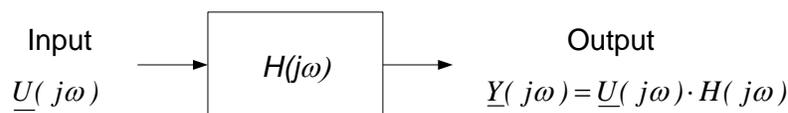


Figure 11.2. Frequency response used to determine system response as a function of frequency.

In Fig. 11.2, the output is determined by multiplying the phasor representation of the input by the system's frequency response. It is important to keep in mind that the arguments of this multiplication are complex functions of frequency – both the input and the frequency response at any frequency are complex numbers, so the output at any frequency is also a complex number. We typically use polar form to represent these complex numbers, so the amplitude of the output signal is the product of the amplitude of the input signal and the magnitude response

of the system and the phase of the output signal is the sum of the phase of the input and the phase response of the system. Mathematically, these are expressed as:

$$|\underline{Y}(j\omega)| = |\underline{U}(j\omega)| \cdot |H(j\omega)| \quad \text{Eq. 11.1}$$

And:

$$\angle \underline{Y}(j\omega) = \angle \underline{U}(j\omega) + \angle H(j\omega) \quad \text{Eq. 11.2}$$

We now present two examples of the process defined by equations (11.1) and (11.2) above.

### Example 11.1

Determine the phasor representation for  $v_{out}(t)$  in the circuit shown below as a function of frequency, if the input voltage is  $v_{in}(t) = 3\cos(2t+20^\circ) + 7\cos(4t-60^\circ)$ . (Note: this problem is the same as that of Example 10.19 of chapter 10.6; the difference is primarily philosophical.)



The frequency response of this circuit, for arbitrary resistance and capacitance values, was determined in example 10.18 of chapter 10.6. For our specific resistor and capacitor values, this becomes:

$$H(j\omega) = \frac{1}{1 + j\omega RC} = \frac{1}{1 + j\omega(2\Omega)(0.25F)} = \frac{1}{1 + j\omega(0.5)} = \frac{2}{2 + j\omega}$$

We can represent the input as a piecewise function of frequency:

$$\underline{V}_{in} = \begin{cases} 3\angle 20^\circ, & \omega = 2\text{rad/sec} \\ 7\angle -60^\circ, & \omega = 4\text{rad/sec} \\ 0, & \text{otherwise} \end{cases}$$

The input phasor is now considered to be a function of frequency, whose only nonzero components are at frequencies of 2 rad/sec and 4 rad/sec.

The phasor output is simply the product of the input phasor as a function of frequency and the frequency response. For frequencies, other than 2 rad/sec and 4 rad/sec, the input is zero and the frequency response is finite, so the output is zero. We determined the output phasor at frequencies of 2 and 4 rad/sec in example 10.19 in section 10.6; using those results allows us to write the output phasor directly as:

$$\underline{V}_{out} = \begin{cases} \frac{3}{\sqrt{2}}\angle -25^\circ, & \omega = 2\text{rad/sec} \\ \frac{7}{\sqrt{5}}\angle -123.4^\circ, & \omega = 4\text{rad/sec} \\ 0, & \text{otherwise} \end{cases}$$

### Example 11.2

The frequency response of a system,  $H(j\omega)$ , and the frequency domain input to the system,  $\underline{U}(j\omega)$ , are given below. The frequency response is dimensionless, the input has units of volts, and the units of frequency are rad/sec. Determine the system output  $\underline{Y}(j\omega)$ .

$$\underline{U}_{j\omega} = \begin{cases} j\omega, & 0 < \omega \leq 1 \\ j(2 - \omega), & 1 < \omega < 2 \\ 0, & \text{otherwise} \end{cases}$$

$$\underline{H}_{j\omega} = \begin{cases} 0, & 0 < \omega \leq 0.5 \\ 1, & 0.5 < \omega \leq 1.5 \\ 0, & \text{otherwise} \end{cases}$$

The system output is determined from a point-by-point multiplication of the input and the frequency response. In this case, both the input to the system and the system frequency response are defined as piecewise functions of frequency, so we must perform a piecewise multiplication to obtain the output. The output, for various ranges of frequency, is obtained below:

$0 < \omega \leq 0.5$ :

- In this range,  $H(j\omega) = 0$  and the input is finite, so  $\underline{Y}(j\omega) = 0$ .

$0.5 < \omega \leq 1.5$ :

- In this range,  $H(j\omega) = 1$  and  $\underline{U}(j\omega) = j\omega$ , so the output is:  $\underline{Y} = (1)(j\omega) = j\omega$ .

$1 < \omega \leq 1.5$ :

- In this range,  $H(j\omega) = 0$  and the input is finite, so  $\underline{Y}(j\omega) = 0$ .

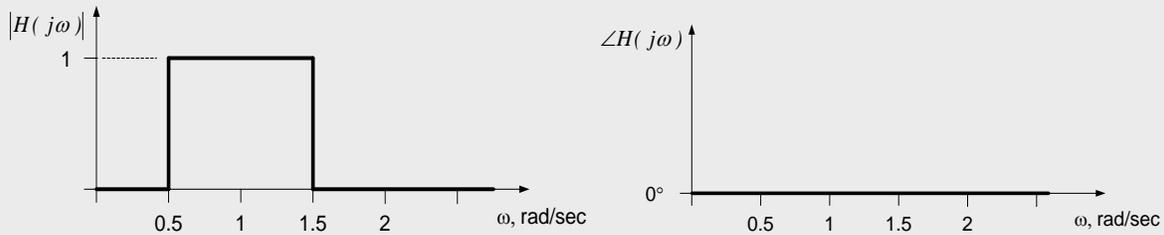
For any other value of frequency, both the input and the frequency response are zero, so the output is obviously zero.

The above results allow us to define the system output in a piecewise fashion as:

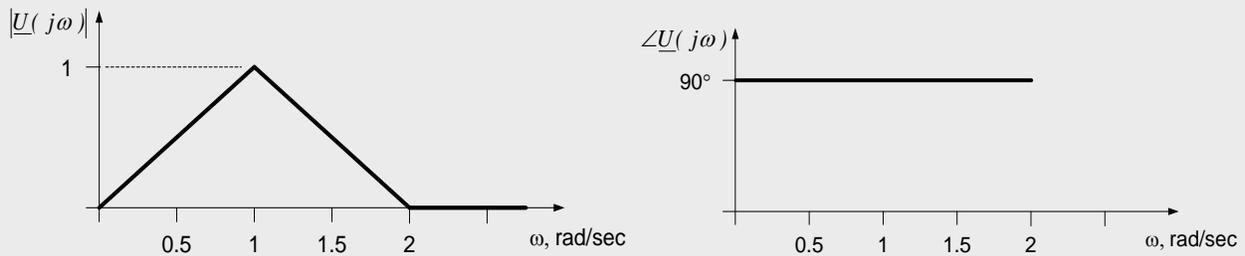
$$\underline{Y}_{j\omega} = \begin{cases} j\omega, & 0.5 < \omega \leq 1 \\ j(2 - \omega), & 1 < \omega \leq 1.5 \\ 0, & \text{otherwise} \end{cases}$$

### Example 11.3

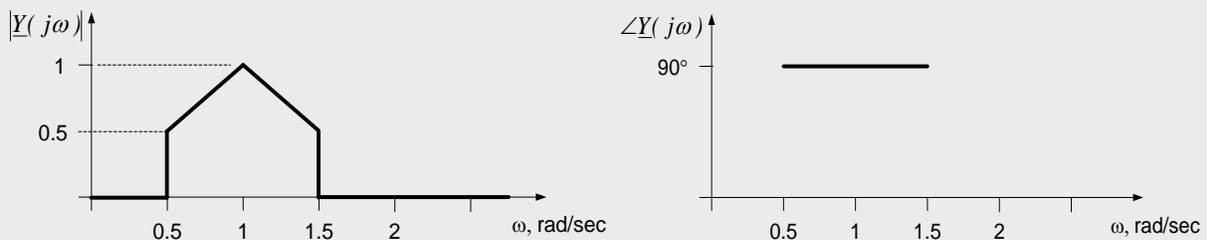
For the signals of example 11.2, plot: the magnitude and phase responses of the system and the magnitude and phase of both the input and output signals. The magnitude and phase response of the system are shown to the left and right below. The phase of the input is indicated as zero for all frequencies, since the input is real-valued for all frequencies. Strictly speaking, however, the phase is not well defined when the magnitude response is zero. (The phase angle provides the direction of a number from the origin of the complex plane, it is difficult to tell what direction “zero” is from itself!)



The magnitude and phase of the frequency domain input are shown below. The 90° phase shift over the range of 0 to 2 rad/sec is due to the factor of “j” in the frequency response. The phase is not indicated where the input amplitude is zero; this is again because the phase angle of “zero” is not well defined.



The magnitude and phase of the output are shown below. It is easy to see from the above figures that the magnitude of the output is the product of the input magnitude and the magnitude response and the phase of the output is the sum of the input phase and the phase response.



The above system is performing a potentially useful operation. Any sinusoidal signals with frequencies between 0.5 and 1.5 rad/sec are passed through the system (they appear at the output, unchanged). Sinusoids with frequencies outside this range are eliminated by the system – they are not present in the system’s output. This system is performing what is called a *band-pass operation*; frequencies within a certain frequency band are passed through the system, while all other frequencies are stopped. This type of operation can be useful, for example, in communication systems – signals from different radio stations should not overlap or they will interfere with one another.

### Section Summary

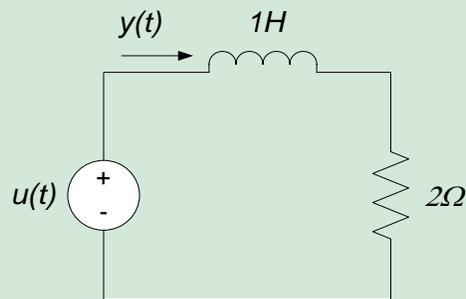
A very powerful analysis and design techniques for linear systems consists of considering the input and output signals to be phasors which are functions of frequency, so that the amplitude and phase of the input and output signals are both defined at each value of frequency. If this viewpoint is taken,

- The amplitude of the output signal is the frequency-by-frequency product of the amplitude of the input signal and the magnitude response of the system, and

- The phase of the output signal is the frequency-by-frequency sum of the phase of the input and the phase response of the system.
- Calculate the frequency response for the circuit below, if  $u(t)$  is the input and  $y(t)$  is the output.

### 11.1 Exercises

1. Calculate the frequency response for the circuit below, if  $u(t)$  is the input and  $y(t)$  is the output.



2. The input to a system is  $u(t) = 3\cos(2t-60^\circ) + 4\cos(4t+30^\circ) + 7\cos(6t+45^\circ)$ . The system frequency response is given by the piecewise function:

$$H(j\omega) = \begin{cases} 0, & 0 < \omega < 3 \\ 1, & 3 \leq \omega \leq 5 \\ 0, & \omega > 5 \end{cases}$$

What is the system output,  $y(t)$ ?

## 11.2 Signal Spectra and Frequency Response Plots

In previous sections, we used the frequency response of a system to determine the system output, when the input to the system consists of signals comprised of one or more sinusoidal components. This analysis approach is extremely powerful, since it turns out that nearly any signal can be represented as a superposition of sinusoids<sup>44</sup>. We will ultimately, therefore, use our frequency domain analysis approaches on a very broad range of input signals, many of which may have very little resemblance to sinusoidal signals.

Often, a graphical representation of the system's frequency response and the frequency content of the signals of interest can facilitate analysis and provide insight into the overall system behavior. We have seen an example of this in example 10.3. In this chapter, we introduce the concept of a signal's *spectrum* – the frequency content of a signal – and we will look more closely at the representation of frequency responses in graphical form. We will conclude this section with an example of the use of signal spectra and frequency response plots to obtain a qualitative representation of a system's frequency domain response to some input.

<sup>44</sup> Later in this textbook, we will use *Fourier Series* to represent arbitrary periodic signals and *Fourier Transforms* to represent non-periodic signals. Computer methods, based on these analysis approaches, are commonly used to determine the frequency components present in measured signals. A detailed discussion of these approaches is currently beyond our capabilities; at the moment, we simply ask that you believe that nearly any time domain signal can be represented to some extent in the frequency domain. For the immediate future, we will assume that the frequency domain representation of any signals of interest is directly available.

## 11.2.1 Signal Spectra

The signals currently of interest to us are sinusoidal. Any sinusoidal signal can be written in the form:

$$f(t) = A\cos(\omega_0 t + \theta) \quad \text{Eq. 11.3}$$

The signal is completely defined by its amplitude,  $A$ , its frequency,  $\omega_0$ , and its phase angle,  $\theta$ . Our primary interest in these signals is as inputs and outputs to systems. As indicated in section 10.1, we can characterize systems by their magnitude and phase responses as functions of frequency. To be consistent with this frame of mind, we will consider sinusoidal signals of the type shown in equation (11.3) to be functions of frequency as well. We will thus begin to consider the frequency,  $\omega$ , to be an independent variable, much in the same way that time was treated as an independent variable when we determined time domain responses of first and second order systems.

With frequency treated as an independent variable, the sinusoidal signal of equation (11.3) can be re-written in terms of  $\omega$ :

$$Fj\omega = \begin{cases} A\angle\theta, & \omega = \omega_0 \\ 0, & \text{otherwise} \end{cases} \quad \text{Eq. 11.4}$$

The frequency domain representation of a sinusoidal signal has two dependent variables: the amplitude and the phase. Our immediate goal is to represent the frequency content of signals graphically – we will, therefore, need to use two plots: amplitude as a function of frequency and phase as a function of frequency.

A signal's amplitude and phase as functions of frequency is called the *spectrum* of the signal. If we are provided with the spectrum of the signal, we have all the information necessary to completely define the signal. Signal spectra are often presented graphically in terms of two plots; we will refer to the plot of amplitude as a function of frequency as the *magnitude spectrum*, while the plot of phase as a function of frequency will be called the *phase spectrum*. As an example, the magnitude and phase spectra of the signal provided in equation (11.4) are shown in Fig. 11.3 below.

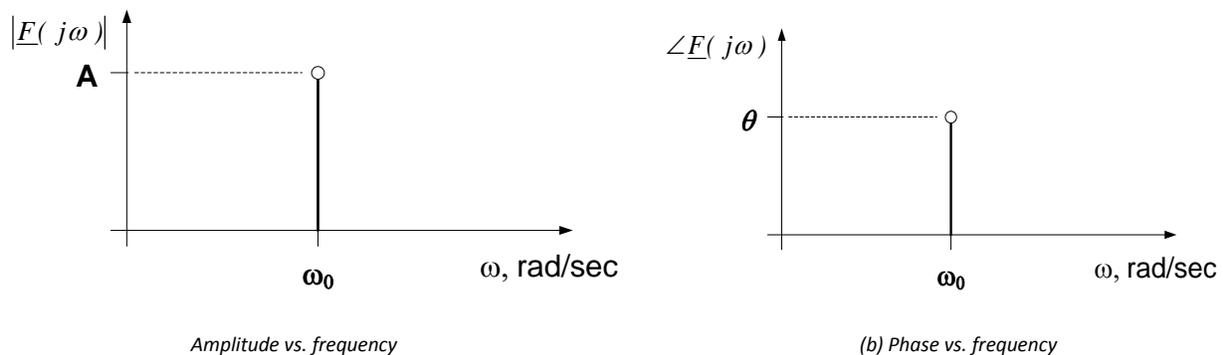
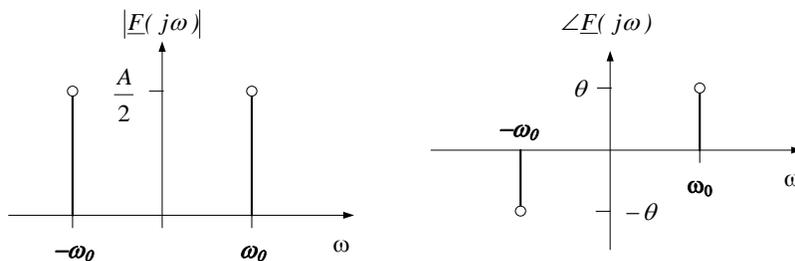


Figure 11.3. Spectrum for the sinusoidal signal of equation (11.4).

### Notes:

- The term spectrum is most commonly applied to the plot of amplitude and phase vs. frequency of signals in complex exponential form. For example, the signal of equation (11.1) can be written in terms of complex exponentials as:  $f(t) = \frac{Ae^{j(\omega_0 t + \theta)} + Ae^{-j(\omega_0 t + \theta)}}{2} = \frac{A}{2}e^{j\theta}e^{j\omega_0 t} + \frac{A}{2}e^{-j\theta}e^{-j\omega_0 t}$ . The spectral plot of the signal in this form is shown below. Note that spectra in this form have both positive and negative frequencies. In our discussions in this chapter, we will present spectra only in terms of having positive frequencies.



- The time domain representation of a signal has only one dependent variable – the value of the signal at any time. Time domain representations are therefore represented graphically as a single plot with time on the horizontal axis. This difference is, fundamentally, due to the fact that we do not work with complex functions of time; if we could measure a signal with both real and imaginary parts, two plots would be required.)

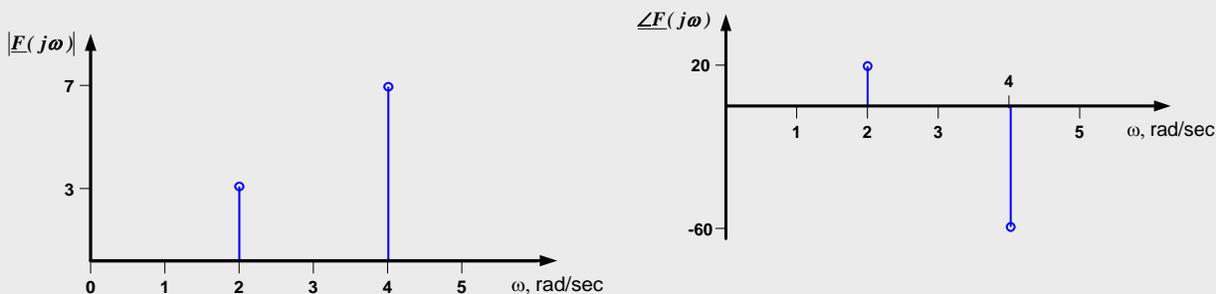
### Example 11.4

Plot the spectrum of the voltage signal  $v_{in}(t) = 3 \cos(2t + 20^\circ) + 7 \cos(4t - 60^\circ)$ . (Note: this is the same signal as that used in Example 10.20; for a time-domain plot of the signal, see that example.)

The phasor form of this signal can be expressed as a function of frequency as:

$$\underline{V}_{in}(j\omega) = \begin{cases} 3\angle 20^\circ, & \omega = 2 \text{ rad/sec} \\ 7\angle -60^\circ, & \omega = 4 \text{ rad/sec} \\ 0, & \text{otherwise} \end{cases}$$

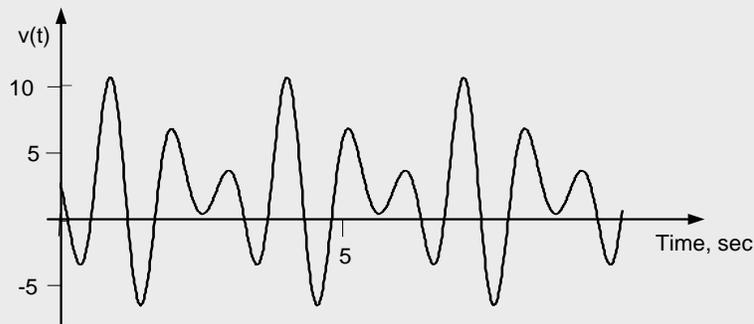
The spectrum is shown below:



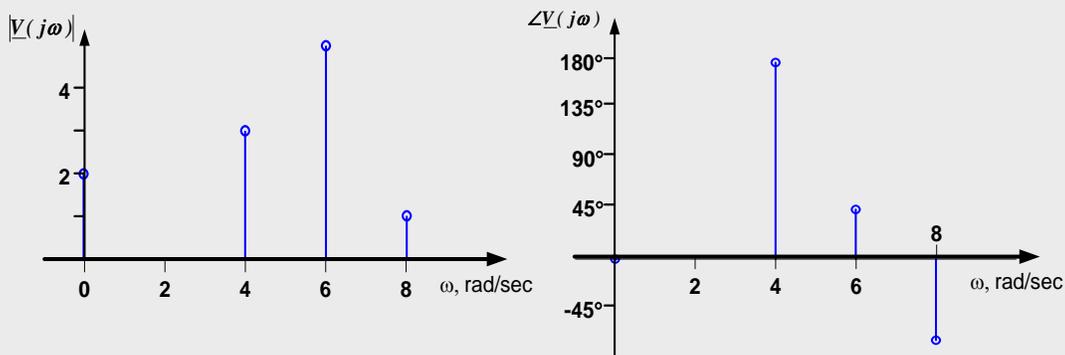
### Example 11.5

Plot the voltage signal  $v(t) = 2 - 3 \cos(4t) + 5 \cos(6t + 45^\circ) + \cos(8t - 75^\circ)$  and its spectrum.

A plot of the time domain signal is shown below.



A couple of notes should be made about the spectrum of this signal. (1) The constant (DC) value of two corresponds to a frequency of zero rad/sec, since we can write  $2 = 2\cos(0t)$ . (2) Sinusoidal amplitudes are by definition positive, so the negative sign in the  $-3\cos(4t)$  term must be accounted for in the phase. Therefore, we re-write this term as  $+3\cos(4t + 180^\circ)$ . The spectrum of the signal can then be plotted as shown below:

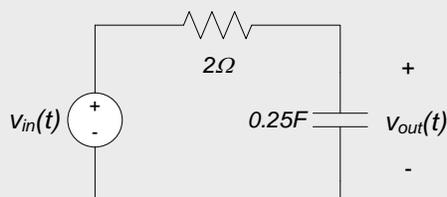


### 11.2.2 Frequency Response Plots

We have previously seen that the frequency response of a system consists of the system’s magnitude response and phase response. The magnitude response provides the gain of the system (the ratio between the amplitudes of the output and input sinusoids) as a function of frequency, while the phase response provides the change in phase between the input and output sinusoids, as a function of frequency. In this section, we will emphasize the presentation of this information graphically. We do this by plotting the frequency response of two simple first order circuits in the examples below.

#### Example 11.6

Plot the frequency response for the circuit shown below. The voltage  $v_{in}(t)$  is the input and the *capacitor voltage*  $v_{out}(t)$  is the *desired response*.



A mathematical expression for the frequency response of this circuit was determined in example 11.1 to be:

$$H(j\omega) = \frac{2}{2 + j\omega}$$

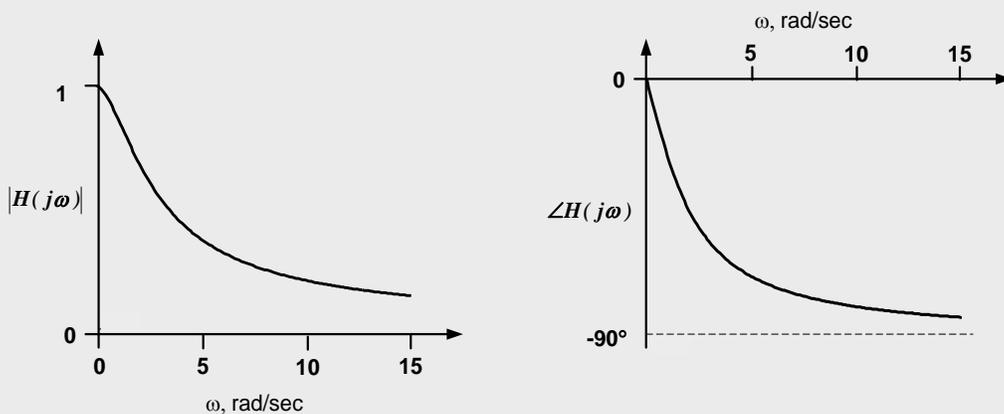
The magnitude and phase responses, as functions of frequency are, therefore:

$$|H(j\omega)| = \frac{2}{\sqrt{4 + \omega^2}}$$

And:

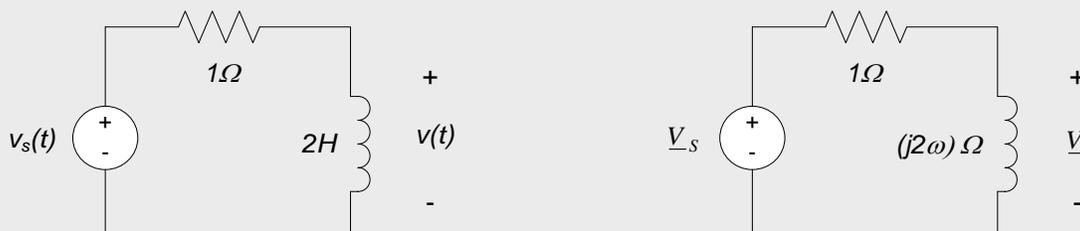
$$\angle H(j\omega) = -\tan^{-1}\left(\frac{\omega}{2}\right)$$

Plotting these functions results in the graphical frequency response shown below:



### Example 11.7

Plot the frequency response for the circuit shown to the left below. The source voltage  $v_s(t)$  is the input and the inductor voltage  $v(t)$  is the output.



We begin by converting the circuit to the frequency domain. Representing the input and output signals as phasors and the circuit elements as impedances results in the circuit to the right above. This circuit suggests that the output voltage can be determined from the input voltage via a voltage divider formula:

$$\underline{V} = \frac{j2\omega}{1 + j2\omega} \cdot \underline{V}_S$$

The circuit frequency response,  $H(j\omega)$ , is the ratio of the output phasor to the input phasor:

$$H(j\omega) = \frac{\underline{V}}{\underline{V}_S} = \frac{j2\omega}{1 + j2\omega}$$

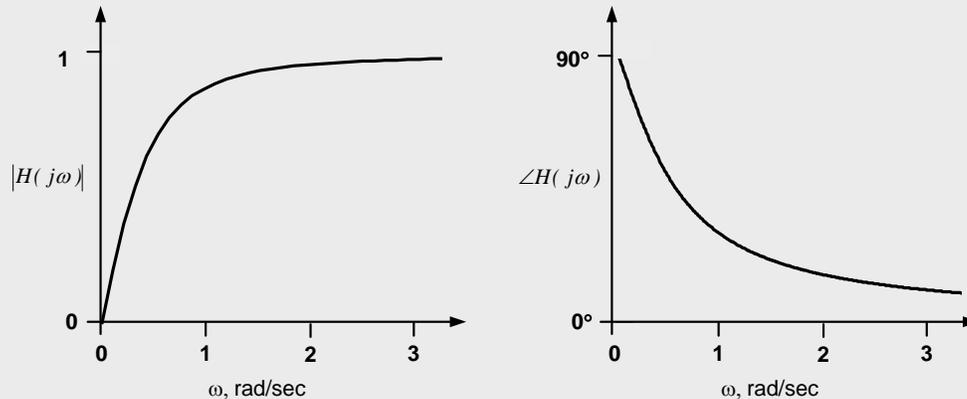
So that the magnitude and phase response of the circuit are:

$$|H(j\omega)| = \frac{2\omega}{\sqrt{1^2 + (2\omega)^2}}$$

And:

$$\angle H(j\omega) = 90^\circ - \tan^{-1}(2\omega)$$

Plots of these functions are shown below.



### 11.2.3 Checking the Frequency Response

A circuit's amplitude response is relatively easy to determine at low and high frequencies. For very low ( $\omega \rightarrow 0$ ) and very high ( $\omega \rightarrow \infty$ ) frequencies, the circuit can be modeled as a purely resistive network. Since resistive networks are relatively easy to analyze (no complex arithmetic is required), this can provide a valuable tool for checking results or predicting expected behavior.

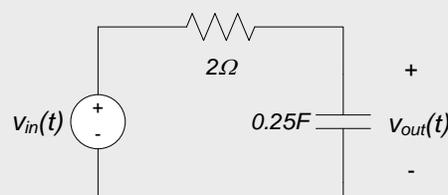
- Capacitors at low and high frequencies: A capacitor's impedance is  $Z_C = \frac{1}{j\omega C}$ . At low frequencies ( $\omega \rightarrow 0$ ), the impedance  $Z_C \rightarrow \infty$ , and the capacitor behaves as an open circuit. At high frequencies ( $\omega \rightarrow \infty$ ) the impedance  $Z_C \rightarrow 0$  and the capacitor behaves like a short circuit.
- Inductors at low and high frequencies: an inductor's impedance is  $Z_L = j\omega L$ . At low frequencies ( $\omega \rightarrow 0$ ), the impedance  $Z_L \rightarrow 0$ , and the inductor behaves as a short circuit. At high frequencies ( $\omega \rightarrow \infty$ ) the impedance  $Z_L \rightarrow \infty$  and the inductor behaves like an open circuit.

Please note that the above statements are relative only to the amplitude response.

#### Example 11.8

Use the circuits' low and high frequency behavior to check the amplitude response plots for example 11.6.

The circuit of interest is shown below.

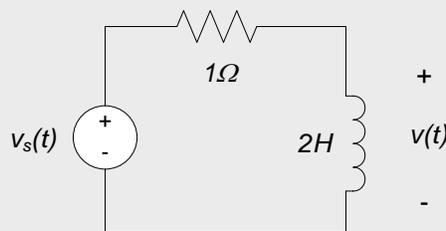


- At low frequencies, the capacitor becomes an open circuit, no current flows through the resistor, and  $v_{out}=v_{in}$ . Thus, the amplitude of the output is the same as the amplitude of the input and the gain as  $\omega \rightarrow 0$  is one.
- At high frequencies, the capacitor becomes a short circuit, so that  $v_{out}=0$ . Since the output amplitude is zero, for a non-zero input, the gain of the circuit as  $\omega \rightarrow \infty$  is zero.

The expected behavior at low and high frequencies agrees with the amplitude response of example 11.6; the amplitude response is one at  $\omega=0$ , and approaches zero as frequency increases.

### Example 11.9

Use the circuits' low and high frequency behavior to check the amplitude response plots for example 11.7. The circuit of interest is shown below:

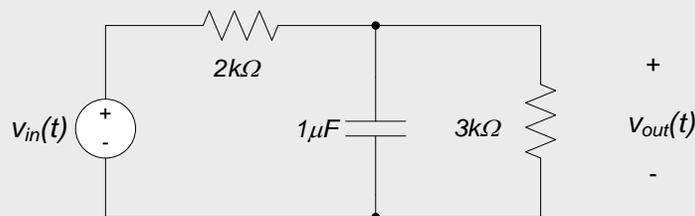


- At low frequencies, the inductor becomes a short circuit, so that  $v=0$ . Since the output amplitude is zero, for a non-zero input, the gain of the circuit as  $\omega \rightarrow 0$  is zero.
- At high frequencies, the inductor becomes an open circuit, no current flows through the resistor, and  $v=v_s$ . Thus, the amplitude of the output is the same as the amplitude of the input and the gain as  $\omega \rightarrow \infty$  is one.

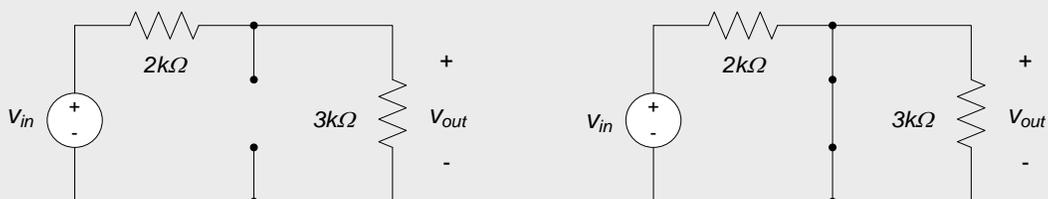
The expected behavior at low and high frequencies agrees with the amplitude response of example 11.7; the amplitude response is zero at  $\omega=0$ , and approaches one as frequency increases.

### Example 11.10

What is the gain of the circuit shown below as  $\omega \rightarrow 0$  and  $\omega \rightarrow \infty$ ? The input is  $v_{in}(t)$  and the output is  $v_{out}(t)$ .



As  $\omega \rightarrow 0$ , the capacitor becomes an open circuit, and the overall circuit becomes as shown to the left below. As  $\omega \rightarrow \infty$ , the capacitor is replaced by a short circuit and the overall circuit becomes as shown to the right below.



As  $\omega \rightarrow 0$ , the circuit becomes a voltage divider, and  $v_{out} = \frac{3k\Omega}{2k\Omega+3k\Omega} \cdot v_{in}$ , so that the circuit's gain is  $\frac{3}{5}$ . As  $\omega \rightarrow \infty$ ,  $v_{out}$  is measured across a short circuit, and the gain is zero.

### 11.2.4 Graphical Representation of System Response

We can use graphical depictions of the input signal spectrum and the frequency response of the system to obtain a graphical representation of a system's response. This representation can be especially useful in interpreting and understanding the effect of a system on an input signal. This qualitative interpretation of a system's response can be an invaluable aid in system design.

The frequency-domain representation of a system's response is shown in Fig. 11.4. The frequency-domain output is simply a point-by-point multiplication between the input signal and the system's frequency response. Thus, the amplitude spectrum of the output is simply the point-by-point product of the input signal's amplitude spectrum with the system's magnitude response. The phase spectrum of the output is the sum of the input signal's phase spectrum and the system's phase response. An example of this process and its use in interpreting a system's response is provided in the example below.

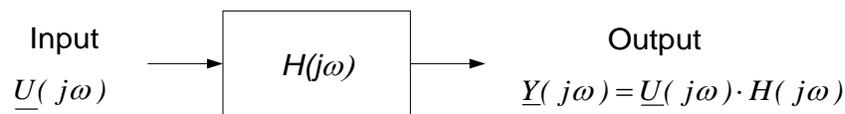
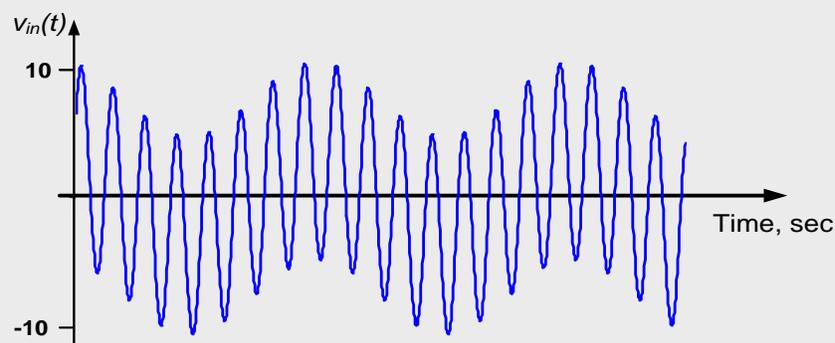


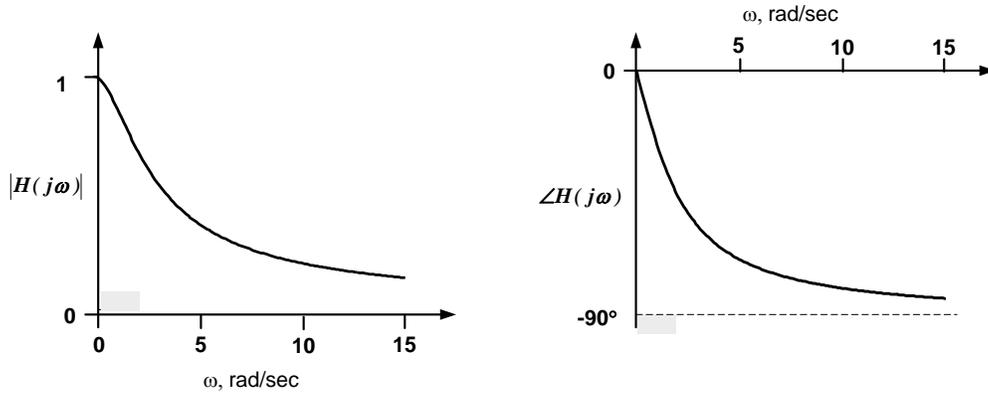
Figure 11.4. Frequency-domain system block diagram.

#### Example 11.11

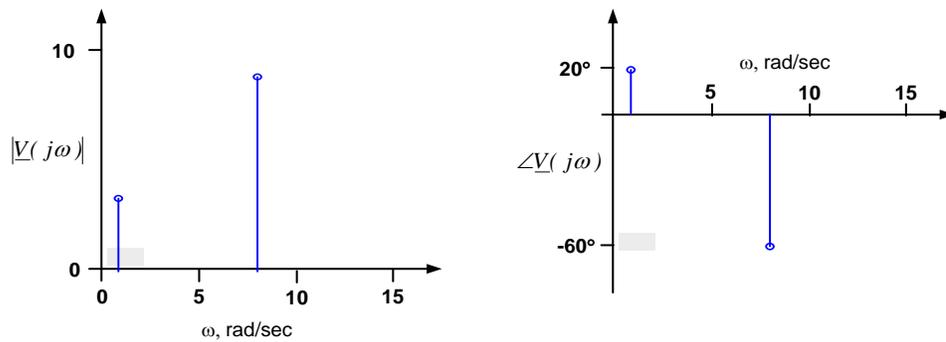
Use graphical methods to interpret the response of the circuit of example 11.6 to an input voltage  $v_{in}(t) = 3 \cos(t + 20^\circ) + 8 \cos(8t - 60^\circ)$ . The input voltage time-domain signal is shown below. The signal consists of a relatively small amplitude, low frequency sinusoid superimposed with a large amplitude, higher frequency signal.



The frequency response of the circuit was determined in example 11.6; the frequency response plots are repeated below:

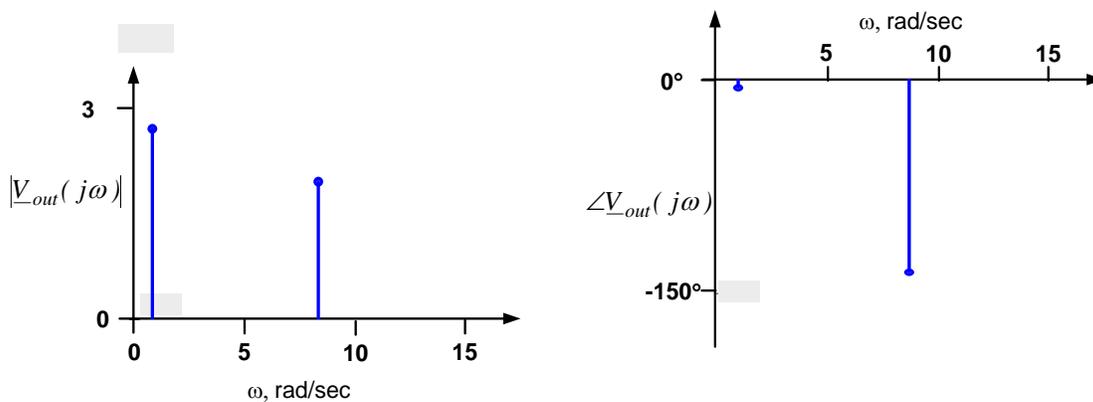


The input signal spectrum,  $\underline{V}(j\omega)$  is shown below.

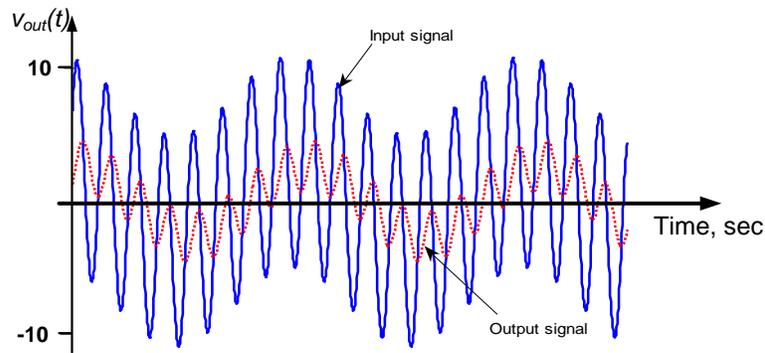


We can obtain the spectrum of the output signal from a frequency-by-frequency product of the input amplitude and the circuit's amplitude response and a frequency-by-frequency sum of the input phase and the circuit's phase response. Keep in mind that when the input amplitude is zero, the output amplitude is also zero and the phase is undetermined.

This process results in the output spectrum shown below. Notice that the circuit attenuates the magnitude of the high frequency component of the signal relative to the low frequency component. The low frequency component is said to be "passed through" the circuit, while the higher frequency component is "stopped" by the circuit.



The time-domain signal of the output signal is shown below, superimposed over the input signal. It is clear that the amplitudes of the higher frequency component of the output signal has been reduced relative to the amplitude of the lower frequency component. This circuit is said to be *frequency selective* – it selects low frequencies to pass and high frequencies to stop.



## Section Summary

- *Signal spectra* are a primarily a graphical representation of the frequency content of a signal. The spectrum consists, in general, of two plots: amplitude as a function of frequency and phase as a function of frequency.
- The signal spectrum is useful, since many signals of interest can be expressed as a superposition of sinusoidal components. This allows signals which do not appear “sinusoidal” to be analyzed using steady-state sinusoidal analysis techniques.
- *Frequency response plots* are a graphical representation of the frequency response of a system. Frequency response plots also consist of two parts: the system’s magnitude response as a function of frequency and the system’s phase response as a function of frequency.
- The combination of the signal spectrum and the frequency response plots can provide valuable insight into a system’s operation. The amplitude of the output signal is the point-by-point product of the input signal’s amplitude and the system’s magnitude response. The phase of the output signal is the point-by-point sum of the input signal’s phase and the system’s phase response.
  - Perhaps even more importantly, the signal spectra and frequency response plots can be used as an effective design tool: if the frequency content of the input signal and the desired frequency content of the output signal are both known, we can determine the system frequency response necessary to provide the desired output signal. This allows us to design our system to perform the desired task
- When calculating a system’s frequency response, it is always important to check your result relative to the expected behavior of the system. This can be done by determining the magnitude response at very low ( $\omega \rightarrow 0$ ) and very high ( $\omega \rightarrow \infty$ ) frequencies. In these extreme conditions the circuit can be approximated as a purely resistive circuit, since energy storage elements can be replaced by either short-circuits or open-circuits as follows:
  - Capacitors at low and high frequencies: A capacitor’s impedance is  $Z_C = \frac{1}{j\omega C}$ . At low frequencies ( $\omega \rightarrow 0$ ), the impedance  $Z_C \rightarrow \infty$ , and the capacitor behaves as an open circuit. At high frequencies ( $\omega \rightarrow \infty$ ) the impedance  $Z_C \rightarrow 0$  and the capacitor behaves like a short circuit.
  - Inductors at low and high frequencies: A inductor’s impedance is  $Z_L = j\omega L$ . At low frequencies ( $\omega \rightarrow 0$ ), the impedance  $Z_L \rightarrow 0$ , and the inductor behaves as a short circuit. At high frequencies ( $\omega \rightarrow \infty$ ) the impedance  $Z_L \rightarrow \infty$  and the inductor behaves like an open circuit.

## 11.2 Exercises

- The input to a system is  $u(t) = 3 \cos(2t - 60^\circ) + 4 \cos(4t + 30^\circ) + 7 \cos(6t + 45^\circ)$ . The system frequency response is given by the piecewise function:

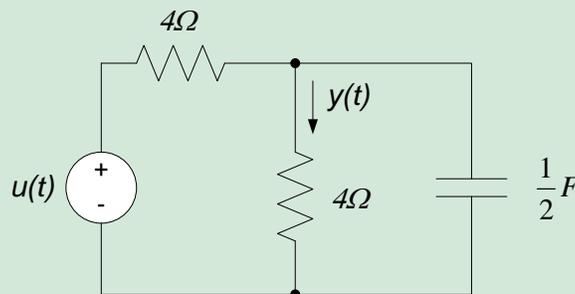
$$H(j\omega) = \begin{cases} 0, & 0 < \omega < 3 \\ 1, & 3 \leq \omega \leq 5 \\ 0, & \omega > 5 \end{cases}$$

Sketch:

- The spectrum of the input signal (magnitude and phase)
- The frequency response of the system (magnitude response and phase response)
- And the output signal spectrum.

Compare your sketches to your results of exercise 2 in section 1.11.

- Sketch the spectrum for the signal  $v(t) = 4 + 3 \cos(2t - 45^\circ) + 2 \cos(4t + 45^\circ)$ .
- Use the circuit behavior at high and low frequencies to check the frequency response you calculated for the circuit of exercise 1 in section 11.1.
- Determine the steady-state response of the output  $y(t)$  for the circuit below at very low and very high frequencies.



## 11.3 Frequency Selective Circuits and Filters

It is common to categorize circuits by the overall “shape” of their magnitude response. As we saw in example 11.11, in some frequency ranges the output amplitude may be high relative to the input amplitude, while in other frequency ranges the output amplitude will be low relative to the input amplitude. If the output amplitude at some frequency is high relative to the input amplitude, the magnitude response at that frequency is “large” and that frequency is said to be *passed* by the circuit. Conversely, if the output amplitude at some frequency is low relative to the input amplitude, the magnitude response at that frequency is “small” and that frequency is said to be *stopped* by the circuit.

Circuits which select certain frequencies to pass and other frequencies to stop are called *frequency selective* circuits or *filters* (since they tend to “filter out” certain frequency ranges of the input signal). The range of frequencies which are passed are called the *passband* of the filter, and the range of frequencies which are stopped are called the *stopband* of the filter. There are four primary categories of filters:

- Low-pass filters* pass low frequencies and stop high frequencies
- High-pass filters* pass high frequencies and stop low frequencies
- Band-pass filters* pass a range of frequencies between two ranges of stopped frequencies
- Band-reject filters* stop a range of frequencies between two ranges of passed frequencies

Filters are also categorized by their *order*. The order of the filter is simply the order of the differential equation governing the filter. Thus, a first-order filter is governed by a first-order differential equation, a second-order filter is governed by a second-order differential equation, and so on. Low-pass and high-pass filters can be any order, while band-pass and band-stop filters must be at least second order.

In this chapter, we restrict our attention to first order filters, so we will consider only low pass and high pass filters.

### 11.3.2 Ideal Low-pass and High-pass Filters

We will first introduce the basic concepts relative to first order filters in the context of ideal filters. It must be clearly understood that ideal filters are not *physically realizable* – that is, we cannot construct a physical system which can perform this way<sup>45</sup>. Ideal filters entirely stop all input signals in the stopband and completely pass all signals in the passband. Thus, the magnitude response of an ideal filter is exactly one in the passband and exactly zero in the stop band. First order filters can be either high-pass or low-pass filters.

An ideal low pass filter has a magnitude response as shown in Fig. 11.5. The passband is shown as the shaded area under the magnitude response. The magnitude response is discontinuous – it goes from one to zero instantaneously. The *cutoff frequency*,  $\omega_c$ , defines the boundary between the passband and the stopband. Any signal with a frequency below  $\omega_c$  is passed through the filter without any attenuation; any signal with a frequency above  $\omega_c$  is entirely stopped by the filter – it is not present in the output signal.

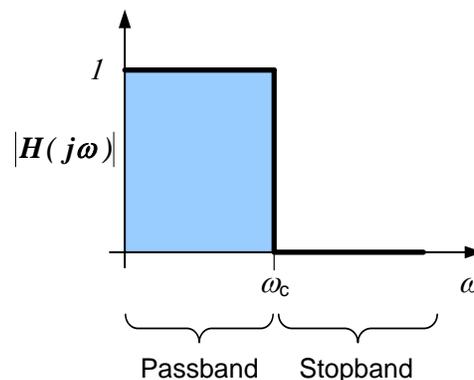


Figure 11.5. Magnitude response of an ideal low-pass filter.

An ideal high-pass filter has a magnitude response as shown in Fig. 11.6. The passband is again shown as the shaded area under the magnitude response. The magnitude response is discontinuous – it goes from zero to one instantaneously. The *cutoff frequency*,  $\omega_c$ , again defines the boundary between the passband and the stopband. Any signal with a frequency below  $\omega_c$  is entirely stopped by the filter while any signal with a frequency above  $\omega_c$  is passed through the filter with no amplitude change.

As previously noted, it is impossible to physically implement an ideal filter. Thus, all electrical circuits implement non-ideal filters. Non-ideal filters do not provide an instantaneous transition between the pass band and the stop band. Non-ideal first order filters are discussed in the following subsections.

<sup>45</sup> In order to implement an ideal filter, the circuit would need to know values of the input signal before the values are applied – that is, the circuit would need to be able to see into the future. The technical term is that the filter would need to be non-causal. It is, of course, impossible to build a device that predicts the future.

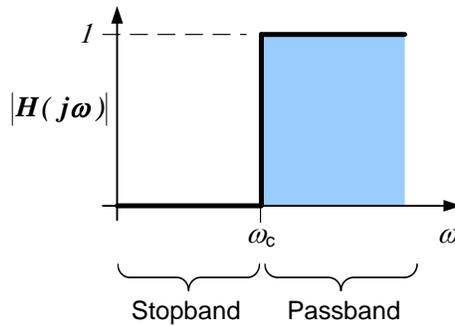


Figure 11.6. Magnitude response of an ideal low-pass filter.

### 11.3.3 First Order Low-pass Filters

The form of the governing differential equation for a first order low pass filter can be written as:

$$\frac{dy(t)}{dt} + \omega_c y(t) = K \cdot u(t) \quad \text{Eq. 11.5}$$

The frequency response of the filter can be determined to be:

$$H(j\omega) = \frac{K}{j\omega + \omega_c} \quad \text{Eq. 11.6}$$

The magnitude response of the filter is thus:

$$|H(j\omega)| = \frac{K}{\sqrt{\omega^2 + \omega_c^2}} \quad \text{Eq. 11.7}$$

The maximum magnitude response is  $\frac{K}{\omega_c}$  when  $\omega=0$  and the magnitude response is zero as  $\omega \rightarrow \infty$ . Thus, the filter is passing low frequencies and stopping high frequencies. The differential equation describes a low-pass filter.

The magnitude response of the filter is shown in Fig. 11.7. The frequency response is a smooth curve, rather than the discontinuous function shown in Fig. 11.5. There is no single frequency that obviously separates the passband from the stopband, so we must choose a relatively arbitrary point to define the boundary between the passband and the stopband. By consensus, the cutoff frequency for a low-pass filter is defined as the frequency at which the magnitude response is  $\frac{1}{\sqrt{2}}$  times the magnitude response at  $\omega=0$ . For the magnitude response given by equation (11.7), the cutoff frequency is  $\omega=\omega_c$ . This point is indicated on Fig. 11.7.

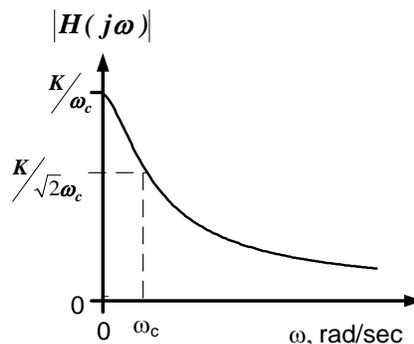


Figure 11.7. Magnitude response of first order low-pass filter.

### 11.3.4 First Order High-pass Filters

The form of the governing differential equation for a first order low pass filter can be written as:

$$\frac{dy(t)}{dt} + \omega_c y(t) = K \cdot \frac{du(t)}{dt} \quad \text{Eq. 11.8}$$

The frequency response of the filter can be determined to be:

$$H(j\omega) = \frac{jK\omega}{j\omega + \omega_c} \quad \text{Eq. 11.9}$$

The magnitude response of the filter is thus:

$$|H(j\omega)| = \frac{K\omega}{\sqrt{\omega^2 + \omega_c^2}} \quad \text{Eq. 11.10}$$

The maximum magnitude response is approximately  $K$  when  $\omega \rightarrow \infty$  and the magnitude response is zero at  $\omega=0$ . Thus, the filter is passing high frequencies and stopping low frequencies. The differential equation describes a high-pass filter.

The magnitude response of the filter is shown in Fig. 11.8. As with the non-ideal low-pass filter, the frequency response is a smooth curve, rather than the discontinuous function shown in Fig. 11.6. Again, there is no single frequency that obviously separates the passband from the stopband, so we must choose a relatively arbitrary point to define the boundary between the passband and the stopband. Consistent with our choice of cutoff frequency for the low-pass filter, the cutoff frequency for a high-pass filter is defined as the frequency at which the magnitude response is  $\frac{1}{\sqrt{2}}$  times the magnitude response at  $\omega \rightarrow \infty$ . For the magnitude response given by equation (1.10), the cutoff frequency is  $\omega=\omega_c$ . This point is indicated on Fig. 11.8.

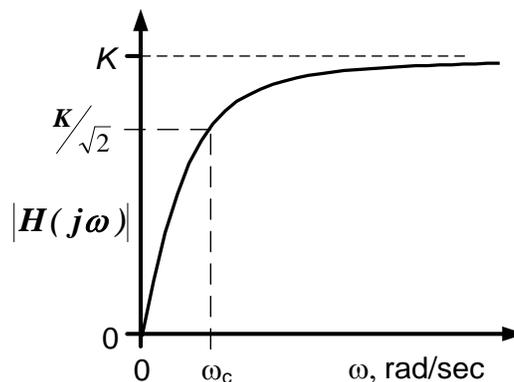


Figure 11.8. Magnitude response of first order high-pass filter.

#### Notes:

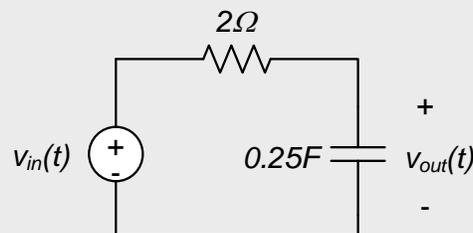
- The cutoff frequency is also called the corner frequency, the 3dB frequency, or the half-power point.
- The cutoff frequency for both low-pass and high-pass filters is defined as the frequency at which the magnitude is  $\frac{1}{\sqrt{2}}$  times the maximum value of the magnitude response.
- It can be seen from examples in section 11.2 that the phase response of a first order low-pass filter is  $0^\circ$  at  $\omega=0$  and decreases to  $-90^\circ$  as  $\omega \rightarrow \infty$ . The phase response is  $-45^\circ$  at the cutoff frequency.
- It can be seen from examples in section 11.2 that the phase response of a first order high-pass filter is  $90^\circ$  at  $\omega=0$  and decreases to  $0^\circ$  as  $\omega \rightarrow \infty$ . The phase response is  $45^\circ$  at the cutoff frequency.

- For both low-pass and high-pass filters, the cutoff frequency is the inverse of the time constant for the circuit, so that  $\omega_c = \frac{1}{\tau}$ .
- The circuit's response at zero frequency is generally an important parameter to consider. This is called the DC gain, and is the ratio of the output amplitude to the input amplitude for a constant input. A constant input corresponds to a cosine with zero frequency. Low pass filters have a relatively high DC gain and a correspondingly large response to a constant input. High pass filters have a low DC gain; they have little or no response to constant inputs.

We conclude this section with examples of circuits from section 10.2 which implement low-pass and high-pass filter operations.

### Example 11.12: First Order Low-pass Filter

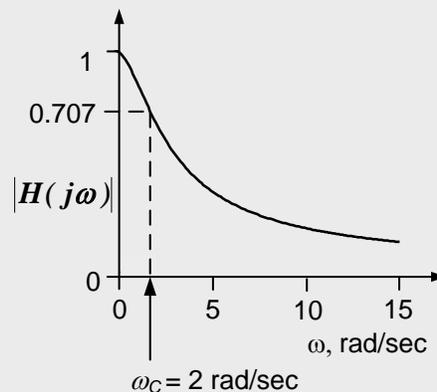
The circuit below is the circuit from example 11.6. The input is  $v_{in}(t)$  and the output is  $v_{out}(t)$ .



In example 11.6, the frequency response was determined to be:

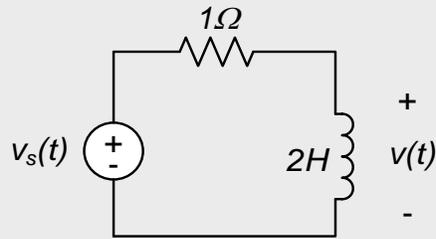
$$H(j\omega) = \frac{2}{2 + j\omega}$$

The maximum value of the magnitude response is one at a frequency of zero radians/second and the magnitude response goes to infinity as  $\omega \rightarrow \infty$ , so the circuit acts as a low-pass filter. Comparing the amplitude response above with equation (2) above, it can be seen that the cutoff frequency is  $\omega_c = 2$  rad/sec. The amplitude response, with the cutoff frequency labeled, is shown below.



### Example 11.13: First Order High-pass Filter

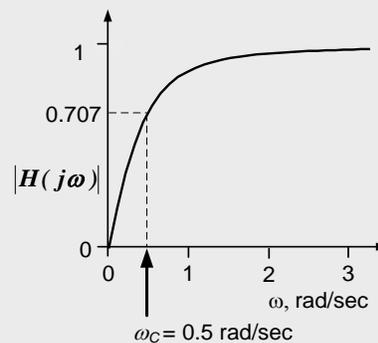
The circuit below is the circuit from example 11.7. The input is  $v_s(t)$  and the output is  $v(t)$ .



The frequency response of the circuit was previously determined to be:

$$H(j\omega) = \frac{V}{V_S} = \frac{j2\omega}{1 + j2\omega}$$

The maximum value of the magnitude response is one as  $\omega \rightarrow \infty$  and goes to zero at a frequency of zero radians/second, so the circuit acts as a low-pass filter. Comparing the amplitude response above with equation (4) above, it can be seen that the cutoff frequency is  $\omega_c = 0.5$  rad/sec. The amplitude response, with the cutoff frequency labeled, is shown below.

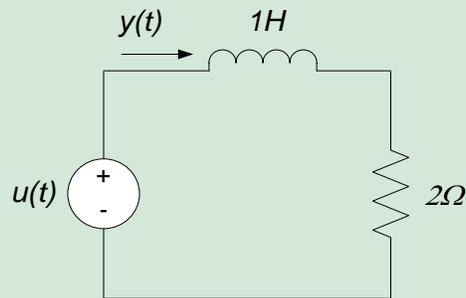


## Section Summary

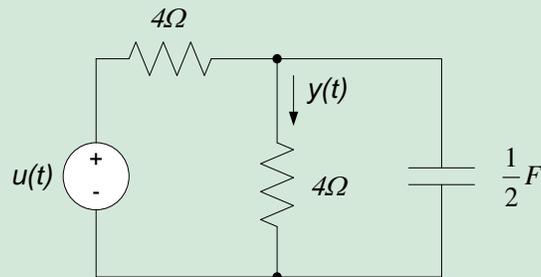
- Filters are frequency-selective systems. These systems provide a desired relationship between the input signal spectrum and the output signal spectrum. The filter does this by *passing* certain frequencies from the input to the output and *stopping* some frequencies from propagating from the input to the output. Nomenclature is as follows:
  - The range of frequencies which are passed is called the *passband* of the filter
  - The range of frequencies which are stopped is called the *stopband* of the filter
- Filters are broadly categorized as follows:
  - *Low-pass filters* pass low frequencies and stop high frequencies
  - *High-pass filters* pass high frequencies and stop low frequencies
  - *Band-pass filters* pass a range of frequencies between two ranges of stopped frequencies
  - *Band-reject filters* stop a range of frequencies between two ranges of passed frequencies
- In this chapter, we were concerned only with first-order low-pass and high-pass filters. These filters are primarily characterized by the following parameters:
  - Cutoff frequency: the cutoff is defined as the frequency at which the magnitude is  $\frac{1}{\sqrt{2}}$  times the maximum value of the magnitude response.
  - *DC gain*: the DC gain is the ratio of the output amplitude to the input amplitude for a constant input (a cosine function with zero frequency). Low pass filters have a relatively high DC gain and a correspondingly large response to a constant input. High pass filters have a low DC gain; they have little or no response to constant inputs.

### 11.3 Exercises

1. What is the cutoff frequency of the circuit below? (You may want to use your results from exercise 1 of section 11.1.)



2. Use the circuit behavior at high and low frequencies and your time constant calculated in exercise 1 above to sketch the frequency response of the circuit of exercise 1. Label your frequency response to include DC gain and cutoff frequency. Is the circuit a high-pass or low-pass filter?
3. Calculate the time constant and the cutoff frequency for the circuit below, if  $u(t)$  is the input and  $y(t)$  is the output. Verify that the cutoff frequency is the inverse of the time constant. Use the circuit behavior at high and low frequencies to determine whether the circuit is a high-pass or low-pass filter.



4. Calculate the frequency response of the circuit of exercise 3. Compare your frequency response to your results of exercise 3. Resolve any differences between the two.

### 11.4 Introduction to Bode Plots

Plotting a systems' frequency response on a linear scale, as done in sections 11.3 and 11.4, has a number of drawbacks, especially for higher-order systems<sup>46</sup>. An alternate format for plotting frequency responses, called a Bode plot, is therefore commonly used<sup>47</sup>. On Bode plots, the amplitude response is essentially presented as a log-log plot, while the phase response is a semi-log plot. Some reasons for this are:

- Use of logarithms converts the operation of multiplication and division to addition and subtraction. This can simplify the creation of frequency response plots for higher order systems.
- Frequencies and amplitudes of interest commonly span many orders of magnitude. Logarithmic scales improve the presentation of this type of data.

<sup>46</sup> Higher order systems are often modeled as a series of lower-order systems (or in the technical parlance, *cascaded* lower-order systems). A combination of cascaded frequency responses is multiplicative. Multiplying several frequency responses together can be tedious

<sup>47</sup> Bode plots use logarithmic scales. Logarithms convert multiplication to addition. Thus, a cascaded set of frequency responses in a logarithmic scale add – adding frequency responses is significantly simpler than multiplying them.

- Human senses are fundamentally logarithmic. The use of logarithmic scales is therefore more “natural”. (This is the reason for use of the Richter scale in measuring earthquake intensity, and the decibel scale in measuring sound levels. It is also the reason that increasing a musical tone by one octave corresponds to doubling its frequency.)

### 11.4.1 Properties of Logarithms

Since Bode plots employ logarithms extensively, we will briefly review some of the basic properties of logarithms before proceeding further. Bode plots rely upon base-10 logarithms ( $\log_{10}$ ), so we will restrict our attention to base-10 logarithms.

A plot of  $\log_{10}(x)$  vs.  $x$  is shown in Fig. 11.9 below. A few important features to note are:

- $\log_{10}(x)$  is a real number only for positive values of  $x$ .
- $\log_{10}(x)$  asymptotically approaches  $-\infty$  as  $x \rightarrow 0$ . The slope of  $\log_{10}(x)$  becomes very large as  $x \rightarrow 0$ .
- The slope of  $\log_{10}(x)$  becomes small as  $x \rightarrow \infty$ .
- From the comments above relative to the slope of  $\log_{10}(x)$ , it can be seen that the sensitivity of  $\log_{10}(x)$  to variations in  $x$  decreases as  $x$  increases (this is the reason why logarithmic scales are used when large variations in  $x$  are encountered – as in Richter scales and musical scales).
- $\log_{10}(1) = 0$

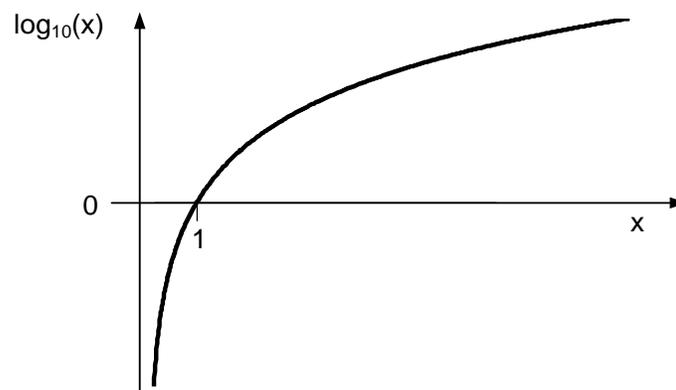


Figure 11.9. Plot of  $\log_{10}(x)$  vs.  $x$

The basic defining property of a base-10 logarithm is that  $x=10^y$ , then  $y=\log_{10}x$ . This property leads to the following rules governing logarithmic operations:

1. Logarithms convert multiplication and division to addition and subtraction, respectively. Thus,

$$\log_{10}(xy) = \log_{10} x + \log_{10} y$$

And:

$$\log_{10}\left(\frac{x}{y}\right) = \log_{10} x - \log_{10} y$$

This property is especially useful for us, since determining the spectrum of an output signal results from the product of an input signal’s spectrum with the frequency response. Thus, the output spectrum on a logarithmic scale can be obtained from a simple addition.

2. Logarithms convert exponentiation to multiplication by the exponent, so that:

$$\log_{10}(x^n) = n \log_{10}x$$

### 11.4.2 Decibel Scales

Magnitude responses are often presented in terms of decibels (abbreviated dB). Decibels are a logarithmic scale. A magnitude response is presented in units of decibels according to the following conversion:

$$|H(j\omega)|_{dB} = 20 \log_{10}(|H(j\omega)|) \quad \text{Eq. 11.11}$$

Strictly speaking, magnitudes in decibels are only appropriate if the amplitude response is unitless (e.g. the units of the input and output must be the same in order for the logarithm to be a mathematically appropriate operation). However, in practice, magnitude responses are often presented in decibels regardless of the relative units of the input and output – thus, magnitude responses are provided in decibels even if the input is voltage and the output is current or vice-versa.

#### Brief Historical Note

Decibel units are related to the unit “bel”, which are named after Alexander Graham Bell. Units of bels are, strictly speaking, applicable only to power. Power in bels is expressed as  $\log_{10}\left(\frac{P}{P_{ref}}\right)$ , where  $P_{ref}$  is a “reference” power. Bels are an inconveniently large unit, so these were converted to decibels, or tenths of a bel. Thus, power in decibels is  $10 \log_{10}\left(\frac{P}{P_{ref}}\right)$ . Since the units of interest to electrical engineers are generally voltages or currents, which must be squared to obtain power, we obtain  $20 \log_{10}(|H(j\omega)|)$ . The significant aspect of the decibel unit for us is not, however, the multiplicative factor of “20”, but the fact that the unit is logarithmic.

We conclude this subsection with a table of common values for  $|H(j\omega)|$  and their associated decibel values.

$ H(j\omega) $	$ H(j\omega) $ in decibels
10	20
1	0
0.1	-20
$\frac{1}{\sqrt{2}}$	-3
$\frac{1}{2}$	-6

### 11.4.3 Bode Plots

Bode plots are simply plots of the magnitude and phase response of a system using a particular set of axes. For Bode plots,

- Units of frequency are on a base-10 logarithm scale.
- Amplitudes (or magnitudes) are in decibels (dB)
- Phases are presented on a linear scale

#### Notes

- Since frequencies are on a logarithmic scale, frequencies separated by the same multiplicative factor are evenly separated on a logarithmic scale. Some of these multiplicative factors have special names. For

example, frequencies separated by a factor of two are said to be separated by *octaves* on a logarithmic scale and frequencies separated by a factor of 10 are said to be separated by *decades*.

- Since decibels are intrinsically a logarithmic scale, magnitudes which are separated by the same multiplicative factor are evenly separated on a decibel scale. For example, magnitudes which are separated by a factor of 10 are separated by 20dB on a decibel scale.

One convenient aspect of the presentation of frequency responses in terms of Bode plots is the ability to generate a reasonable sketch of a frequency response very easily. In general, this approach consists of approximating the Bode plot of a system by its asymptotic behavior as a set of straight lines. This is called a “straight line approximation” of the Bode plot; the approach is illustrated for a typical low-pass filter in the following subsection.

### 11.4.4 Bode Plots for First Order Low-pass Filters

The frequency response of a general first order low-pass filter is provided in section 11.3 as:

$$H(j\omega) = \frac{K}{j\omega + \omega_c} \quad \text{Eq. 11.12}$$

Thus, magnitude response of the circuit is:

$$|H(j\omega)| = \frac{K}{\sqrt{\omega^2 + \omega_c^2}} \quad \text{Eq. 11.13}$$

And the phase response of the circuit is:

$$\angle H(j\omega) = -\tan^{-1}\left(\frac{\omega}{\omega_c}\right) \quad \text{Eq. 11.14}$$

To estimate the asymptotic behavior of the frequency response, we consider the behavior of equations (11.13) and (11.14) for the low frequency and high frequency cases. In general, we consider “low” frequencies to be frequencies which are less than a factor of 10 below the cutoff frequency (i.e.  $\omega < \frac{\omega_c}{10}$ , or frequencies more than a decade below the cutoff frequency). High frequencies are typically assumed to be frequencies which are more than a factor of 10 above the cutoff frequency (i.e.  $\omega > 10\omega_c$ , or more than a decade above the cutoff frequency). We consider the high and low frequency cases separately below.

- Low frequencies:
  - The magnitude response given by equation (2) is  $|H(j\omega)| = \frac{K}{\sqrt{\omega^2 + \omega_c^2}}$ . If  $\omega \ll \omega_c$ , the denominator is approximately  $\sqrt{\omega_c^2} = \omega_c$  and the magnitude response  $|H(j\omega)| \approx \frac{K}{\omega_c}$ . If  $\omega \ll \omega_c$ ,  $\frac{\omega}{\omega_c} \approx 0$  and the phase is approximately  $\angle H(j\omega) \approx -\tan^{-1}(0) = 0^\circ$ .
- High Frequencies:
  - If  $\omega \gg \omega_c$ , the denominator of the amplitude response is  $\sqrt{\omega^2 + \omega_c^2} \approx \sqrt{\omega^2} = \omega$ . Therefore, for high frequencies, the magnitude response  $|H(j\omega)| \approx \frac{K}{\omega}$ . If, for high frequencies, we increase the frequency by a factor of 10, we reduce the magnitude response by 20dB (since  $|H(j \cdot 10\omega)| \approx \frac{K}{10\omega} = 0.1 \frac{K}{\omega}$  and the multiplicative factor of 0.1 corresponds to -20dB). Thus, for frequencies well above the cutoff frequency, the magnitude response, presented in Bode plot form, decreases by 20dB/decade. When  $\omega \gg \omega_c$ , the phase response is given by  $\angle H(j\omega) = -\tan^{-1}\left(\frac{\omega}{\omega_c}\right) \approx -\tan^{-1}(\infty) = -90^\circ$ .

### Summary: Low-pass Filter Straight-line Bode Plot Approximations

The straight line approximation to the magnitude response is constant below the cutoff frequency, with a value (in decibels) of  $20 \log_{10} \left( \frac{K}{\omega_c} \right)$ . Above the cutoff frequency, the Bode plot straight-line approximation has a constant slope of  $-20 \text{ dB/decade}$ .

The straight-line approximation to the phase response is zero degrees up to a frequency of  $\frac{\omega_c}{10}$  and is  $-90^\circ$  above a frequency of  $10\omega_c$ . A straight line is used to connect the  $\frac{\omega_c}{10}$  and  $10\omega_c$  frequencies.

A straight-line approximation to the Bode plot for a typical low-pass circuit, with  $K = \omega_c$  (so that the frequency response is  $H(j\omega) = \frac{\omega_c}{j\omega + \omega_c}$  and the DC gain is 1, or 0dB) along with an exact curve is provided below in Fig. 11.10.

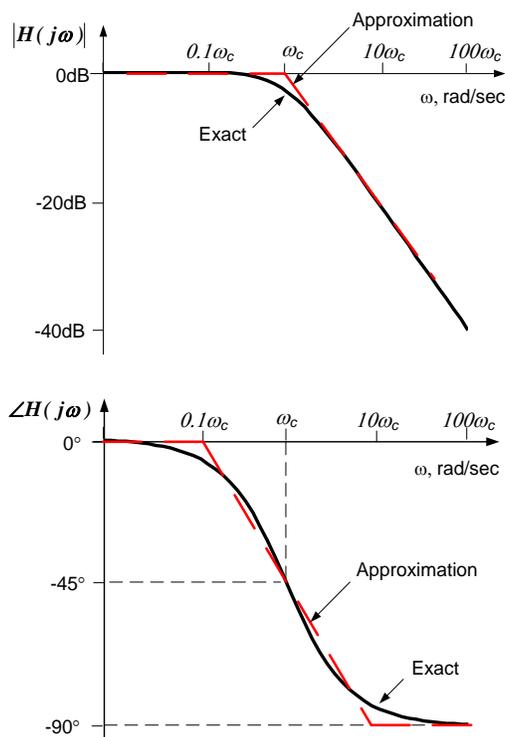
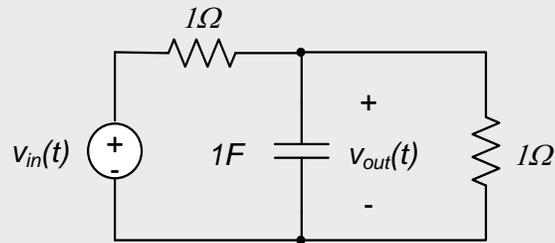


Figure 11.10. Low pass filter straight-line approximation and exact Bode plot.

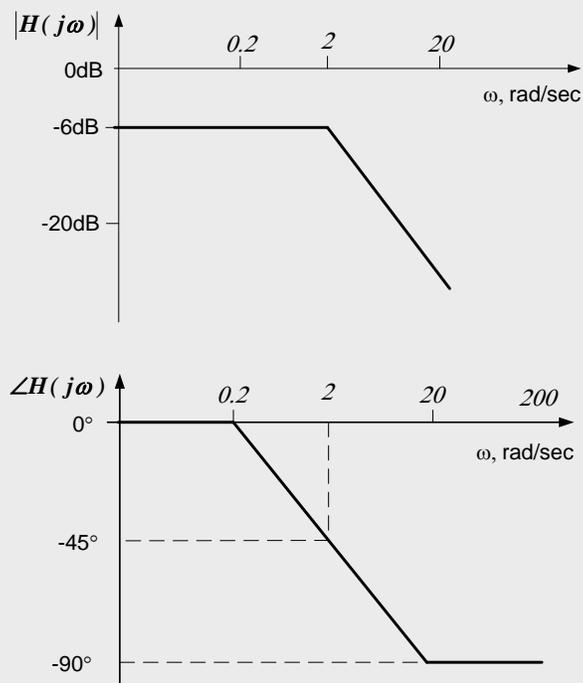
We conclude this section with a numerical example of the straight-line approximation to a Bode plot for a specific circuit.

#### Example 11.14

Sketch a straight-line approximation to the Bode plot for the circuit below. The input is  $v_{in}(t)$  and the output is  $v_{out}(t)$ .



The frequency response for this circuit is  $H(j\omega) = \frac{1}{j\omega+2}$ . Therefore, the cutoff frequency is  $\omega_c=2$  rad/sec and the gain in decibels at low frequencies is  $|H(j0)|_{dB} = 20 \log_{10}\left(\frac{1}{2}\right) \approx -6dB$ . Thus, the straight-line magnitude response is -6dB below the cutoff frequency and decreases by 20dB/decade above the cutoff frequency. The straight-line phase response is  $0^\circ$  below 0.2rad/sec,  $-90^\circ$  above 20 rad/sec and a straight line between these frequencies. The associated plots are shown below.



## Section Summary

- Bode plots are a very useful format for plotting frequency responses. Bode plots provide magnitude responses and phase responses in the following format:
  - Units of frequency are on a base-10 logarithm scale.
  - Amplitudes (or magnitudes) are in decibels (dB)
  - Phases are presented on a linear scale.
  - Magnitude responses in decibels are calculated according to:

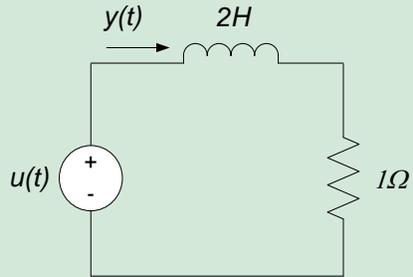
$$|H(j\omega)|_{dB} = 20 \log_{10}(|H(j\omega)|)$$

- The use of logarithmic scales in Bode plots has a number of advantages. Logarithms convert multiplication and division into addition and subtraction, respectively. This provides a significant mathematical simplification in the analysis of higher-order systems. Logarithms are also, in some sense, more “natural” to interpret, since human senses are fundamentally logarithmic.

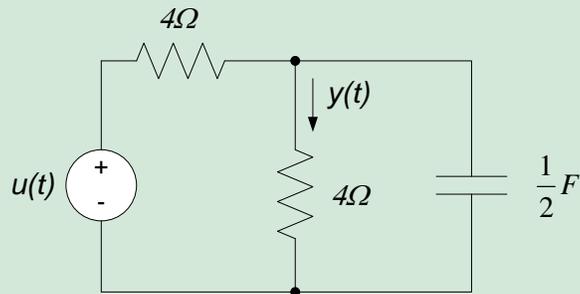
- Bode plots also have the advantage of being approximated fairly well by straight-line approximations. This allows the engineer to sketch a fairly accurate frequency response plot with only a minimal number of calculations.

## 11.4 Exercises

1. Sketch a Bode plot (straight-line approximation) for the circuit below.



2. Sketch a Bode plot (straight-line approximation) for the circuit below.



## Real Analog Chapter 11: Lab Projects

### 11.2.1: Signals with Multiple Frequency Components

In this lab project, we will calculate the magnitude response of an electrical circuit and use this information to infer the effect of the circuit on some relatively complex input signals. In particular, we will apply the following input signal types to the circuit:

- A signal composed of multiple sinusoidal waves of different frequencies
- A sinusoidal signal with a time-varying frequency (a sinusoidal *sweep*)

We will subsequently measure the response of the circuit to these input signals and compare them to our expectations. The goal of this lab project is to see how a circuit's magnitude response affects the "shape" of a signal applied to it. This is a preliminary step to designing a circuit to provide a magnitude response which has a desired effect on an input signal.

#### Before beginning this lab, you should be able to:

- Calculate the frequency response of a passive electrical circuit
- Calculate the magnitude and phase responses of a passive electrical circuit
- Use a circuit's magnitude and phase responses to determine the response to a sinusoidal input

#### After completing this lab, you should be able to:

- Measure the response of a circuit to an input signal with multiple sinusoidal components
- Apply a sinusoidal sweep to an electrical circuit

#### This lab exercise requires:

- Analog Discovery 2 module
- Digilent Analog Parts Kit
- Digital multimeter (optional)

#### Symbol Key:

<b>DEMO</b>	Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
<b>ANALYSIS</b>	Analysis; include principle results of analysis in laboratory report.
<b>SIM</b>	Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
<b>DATA</b>	Record data in your lab notebook.

#### General Discussion:

This lab assignment concerns the circuit shown in Fig. 1. The overall behavior in response to sinusoidal inputs is relatively simple to understand. At very low frequencies, the capacitor has infinite impedance, the circuit becomes a simple voltage divider, and the output voltage phasor amplitude is simply half of the input voltage amplitude:

$$\omega = 0 \Rightarrow \left| \frac{V_{OUT}}{V_{IN}} \right| = \frac{1}{2} \quad \text{Eq. 1}$$

At very high frequencies, the capacitor's impedance is zero, the capacitor behaves like a short circuit, and the output voltage is zero regardless of the input voltage amplitude:

$$\omega = \infty \Rightarrow \left| \frac{V_{OUT}}{V_{IN}} \right| = 0 \quad \text{Eq. 2}$$

Since the circuit's output lets low frequency signals appear in the output (to some extent) and does not let high frequency inputs appear in the output, the circuit is said to "pass" low frequencies and "stop" high frequencies<sup>48</sup>.

In this assignment, we will see how we can infer a circuit's response to relatively complex input voltage waveforms by using simple observations like those of equations (1) and (2).

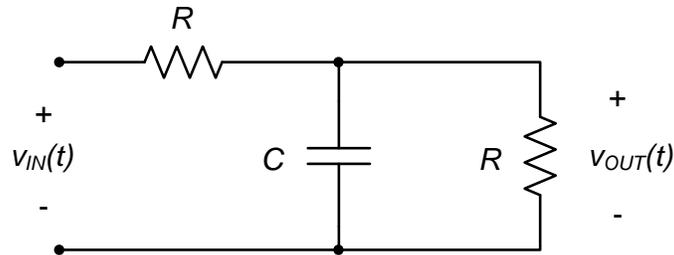


Figure 1. RC circuit.

### Pre-lab:

#### ANALYSIS

Determine the magnitude response of the circuit shown in Fig. 1. If  $R = 1\text{k}\Omega$  and  $C = 100\text{nF}$ , calculate the magnitude response (the ratio of the amplitude of the output sinusoid to the input sinusoid) of the circuit at frequencies of 500Hz, 1000Hz, and 10kHz. Do your magnitude responses at these frequencies agree with the expectations of the circuit response as expressed by equations (1) and (2)?

### Lab Procedures:

#### DATA

- a. Construct the circuit of Fig. 1, using  $R = 1\text{k}\Omega$  and  $C = 100\text{nF}$ .
  - i. Use your oscilloscope to measure both  $v_{IN}(t)$  and  $v_{OUT}(t)$ . Use the waveform generator to apply a custom waveform to the circuit. The waveform will be defined by the expression:

$$v_{IN}(t) = 20[\sin(100\pi t) + \sin(2000\pi t) + \sin(20,000\pi t)] \quad \text{Eq. 3}$$

Notice that the first term in this series has a frequency of 500Hz, the second term a frequency of 1000Hz, and the third term a frequency of 10,000Hz. Instructions for creating and applying a custom signal based on the mathematical expression above are provided in Appendix A of this project.

#### DATA

- ii. Record an image of the oscilloscope window, showing the voltages  $v_{IN}(t)$  and  $v_{OUT}(t)$ .
- iii. Demonstrate operation of your circuit to the TA and have them initial the appropriate page(s) of your lab notebook and the lab worksheet.

#### DEMO

#### ANALYSIS

- iv. Comment on the overall shape of the input and output signals, relative to your expectations based on the pre-lab analyses and equations (1) and (2).

b. With the same circuit as in part (a):

- i. Use the waveform generator to apply a sinusoidal sweep to the circuit. A sinusoidal sweep typically has a frequency which changes linearly with time. The signal we will use starts at a frequency of 100Hz; the frequency increases to 10kHz in 20msec. Detailed instructions for creating and applying this signal are provided in Appendix B of this project.

<sup>48</sup> The circuit is said to be a *low pass filter*.

DATA

ii. Record an image of the oscilloscope window, showing the voltages  $v_{IN}(t)$  and  $v_{OUT}(t)$ .

DEMO

iii. Demonstrate operation of your circuit to the TA and have them initial the appropriate page(s) of your lab notebook and the lab worksheet.

ANALYSIS

iv. Comment on the overall shape of the input and output signals, relative to your expectations based on the pre-lab analyses and equations (1) and (2).

## Appendix A: Creating a Custom Mathematical Waveform

Click on the mode drop-down of the waveform generator, select **Custom**, and click **new**. A WaveGen-custom window should open, providing a number of ways to create custom waveforms.

1. Select the **Math** tab on the **WaveGen-custom** window and create the mathematical expression corresponding to the given function. This will require several steps:
  - The horizontal range of values ( $X$ , in the waveform generator) are set in the second set of text boxes in the math tab. Your range of  $X$  values should be from 0.000 to 1.000, as shown in Fig. A1.
  - Type the mathematical expression for the signal being created. Multiplication is denoted by  $*$ ,  $\text{PI} = \pi$ , and  $\text{sin} = \text{sine}$ . Operator precedence is as in most calculators. Thus, the mathematical expression in equation (3) can be implemented as:

$$20 * (\sin(2 * \text{PI} * X) + \sin(4 * \text{PI} * X) + \sin(40 * \text{PI} * X))$$

Note that the arguments of the sinusoid do not agree with the original mathematical expression.

Actually, the only thing that really matters about these arguments is that the second is twice the first, and the third is twenty times the first. The arguments will all be scaled once we have the waveform generator “play” the signal.

- Click **Generate**. The final result should look something like that shown in Fig. A1. Don’t worry about the axes in the plot window; the horizontal axis is in “samples” and the vertical axis is “percent of full scale”. Both of these will be scaled when we play back the signal.
- Click **OK**. The **WaveGen-custom** window should close.

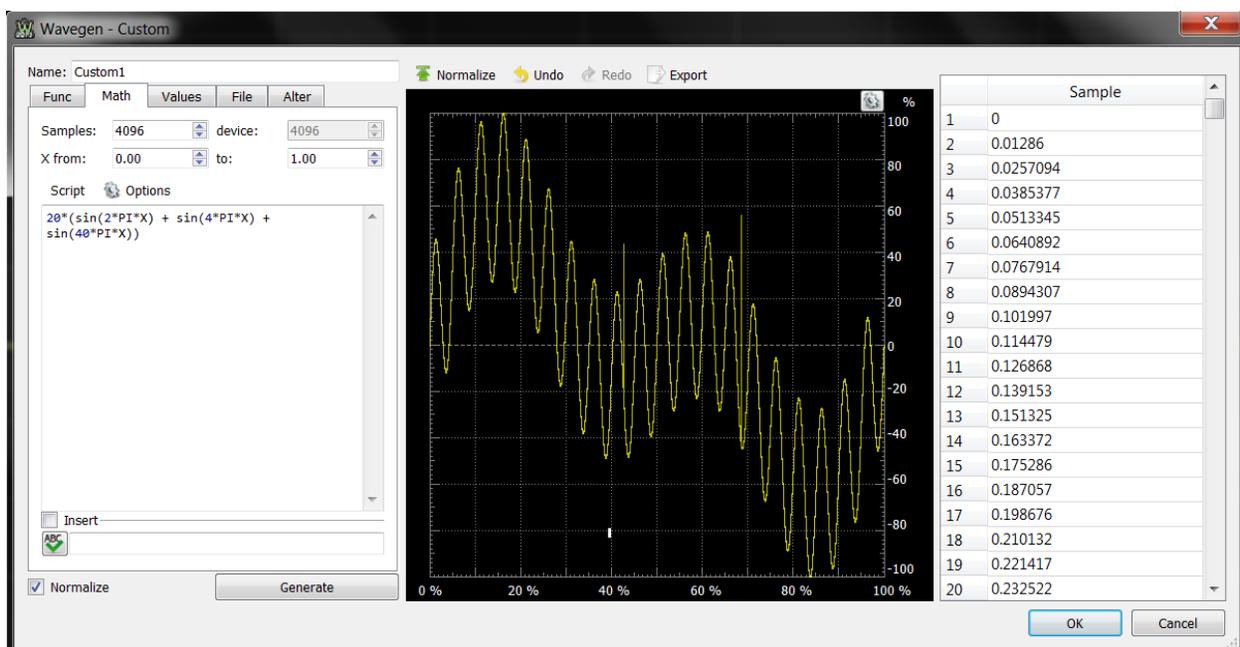
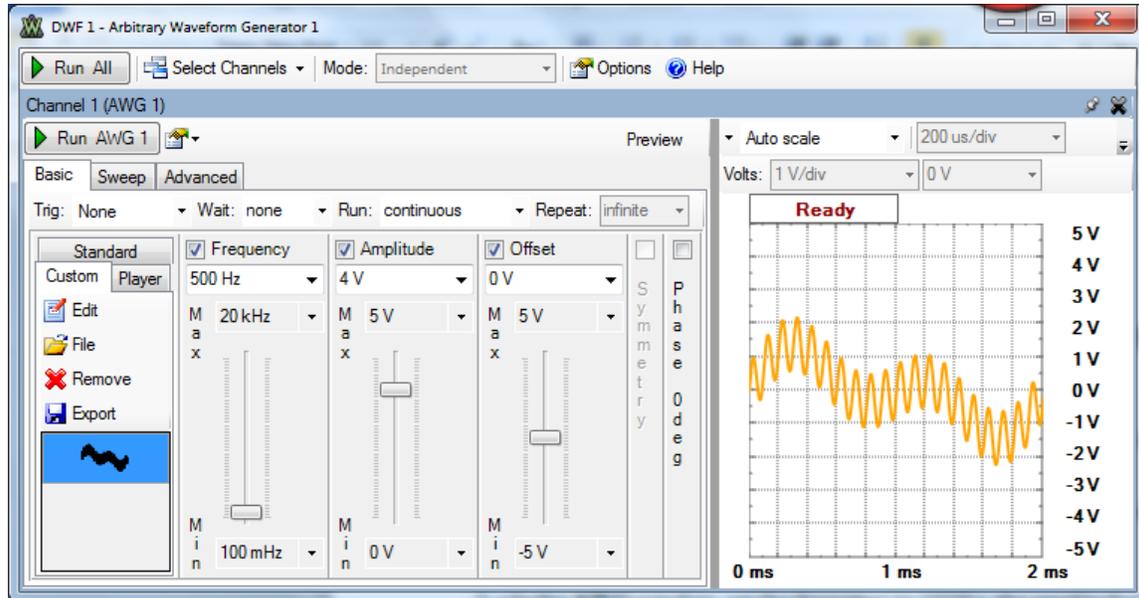


Figure A1. Sample AWG Editor window.

- In the **AWG** window, set the frequency to 500Hz, the amplitude to 4V, and the offset to 0V. Setting the frequency to 500Hz causes the waveform we created to be played back 500 times per second. This causes the frequencies of the sinusoids to be 500Hz, 1000Hz, and 10,000Hz, as desired. The AWG window should look approximately as shown in Figure A2; notice that the plot window indicates that the period of the signal is 2msec, consistent with the 500Hz frequency of the lowest frequency component in the signal. Click on “Run AWG” to play back the signal.



## Appendix B: Creating a Sinusoidal Sweep

- To create a sinusoidal sweep, select “Sweep” from the mode drop-down in the Arbitrary Waveform Generator Window. The following steps will set up the waveform as desired for this part of the lab project.
  - In the **Type** text drop-down, select “Sine”. Make sure “Sweep to” is checked and “Damp to” is unchecked.
  - Next to **Frequency**, set the range of frequencies used in your sweep, and the time duration over which the sweep occurs. In the leftmost text box, choose 100Hz (this is the initial frequency in the sweep). In the next textbox to the right, choose 10kHz (this is the final frequency in the sweep). In the last text box, choose 20ms (the duration of the sweep is ten seconds). Thus, the range of frequencies is from 100Hz to 10kHz, over a time of 20 milli-seconds. This pattern repeats itself until the user interrupts it.
  - Next to **Amplitude**, select 3.3V. (This will result in a maximum value of 3.3V and a minimum value of -3.3V.)
  - All other text boxes can be left at their default values.
  - The sinusoidal sweep we will apply is displayed in the plot window. Since our sweep changes frequency rapidly, the plot will not provide much detail as to the actual wave shape. If desired, you can adjust the plot parameters to get a better feeling as to the actual shape of the waveform. The **Arbitrary Waveform Generator** window, with the plot parameters chosen above, is shown in Fig. B1; your window should look.

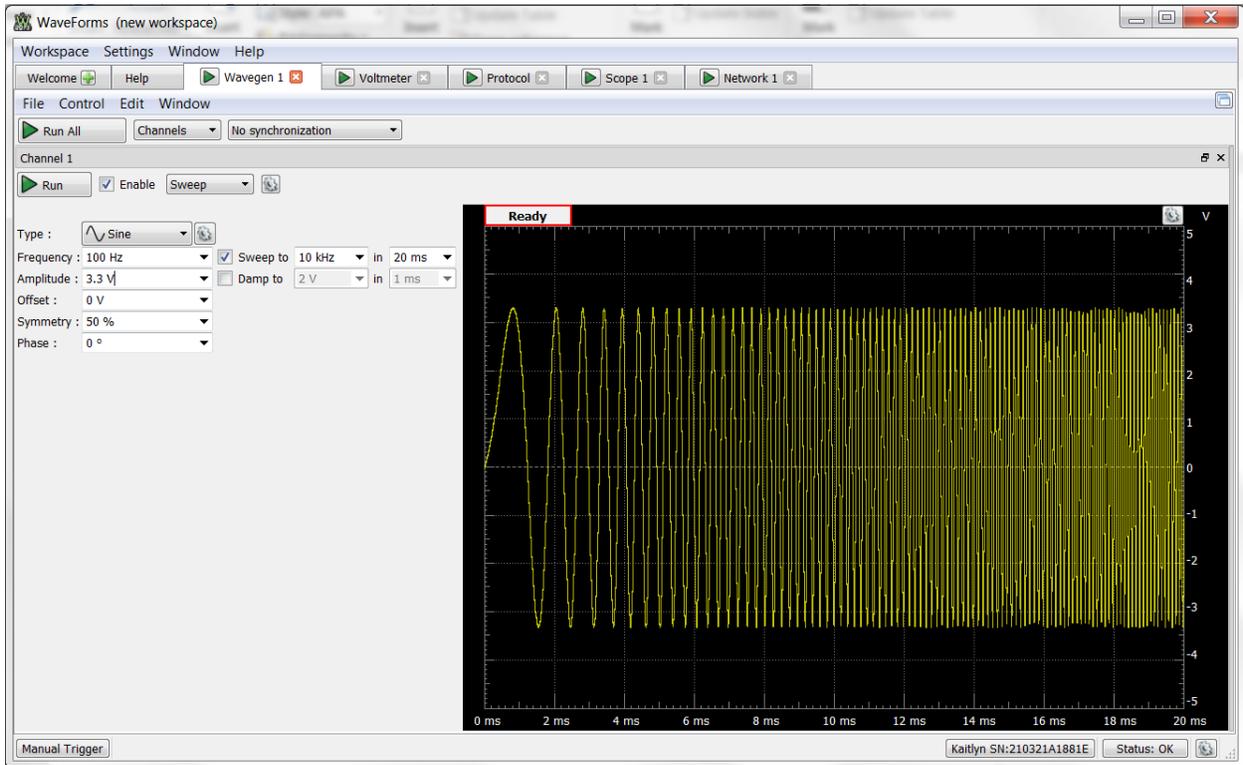


Figure B1. Example arbitrary waveform generator window.

2. Click on “Run” to play back the signal.



6. In the space below, briefly discuss the circuit's response to the sinusoidal sweep input voltage, relative to your expectations based on the circuit's magnitude response and the input voltage. (7 pts)

7. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuit's response to the sinusoidal sweep input. (4 pts total)

**TA Initials:** \_\_\_\_\_

## Real Analog Chapter 11: Lab Projects

### 11.3.1: Passive RL Filter

This assignment concerns a passive filtering circuit consisting of a series combination of a resistor and an inductor. If the voltage across the inductor is taken as the filter output, the circuit will act as a high-pass filter. However, if we take the voltage across the resistor as the filter output, the circuit acts as a low-pass filter<sup>49</sup>. We will measure both the high-pass and low-pass characteristics of this circuit and compare them to our expectations based on the analytically determined frequency response.

#### Before beginning this lab, you should be able to:

- Calculate the frequency response of a passive electrical circuit
- Calculate the magnitude and phase responses of a passive electrical circuit
- Determine the DC gain, high frequency gain, and cutoff frequency of a passive first order filter

#### After completing this lab, you should be able to:

- Measure the magnitude and phase responses of first order filter circuits

#### This lab exercise requires:

- Analog Discovery 2 module
- Digilent Analog Parts Kit
- Digital multimeter (optional)

#### Symbol Key:

- |   |   |
|---|---|
|  | Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable. |
|  | Analysis; include principle results of analysis in laboratory report.   |
|  | Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.                      |
|  | Record data in your lab notebook.   |

#### General Discussion:

This lab assignment concerns the circuit shown in Fig. 1. We will measure the response of both the voltages  $v_R(t)$  and  $v_L(t)$  to the input voltage  $v_{IN}(t)$  and plot the frequency response – both magnitude and phase – of both voltages. Obviously, since we are interested in the frequency response of the circuit, the input voltage will consist of sinusoids.

<sup>49</sup> If the energy in the input does not appear in the inductor voltage, it must be somewhere else (it can't simply disappear). Thus, if the inductor voltage stops low frequencies, the energy in these frequencies must show up somewhere else – in this case, it will be apparent in the resistor voltage (or, equivalently, the inductor current).

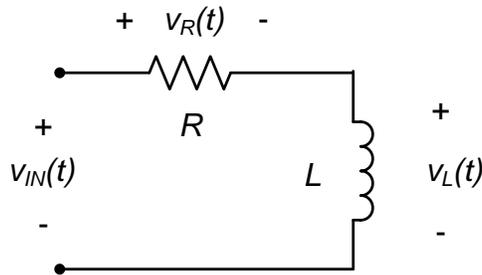


Figure 1. RL circuit.

**Pre-lab:****ANALYSIS**

Determine the frequency responses of both  $v_L(t)$  and  $v_R(t)$  in the circuit shown in Fig. 1, as a function of  $R$  and  $C$ . Use the voltage  $v_{IN}(t)$  as the input in both cases<sup>50</sup>. Calculate the cutoff frequency of the circuit (both outputs share the same cutoff frequency). Also calculate the DC gain and high frequency gain of both outputs.

**ANALYSIS**

Sketch the magnitude and phase responses of both outputs in the circuit of Fig. 1. Label your sketches with the DC gain, high frequency gain, and cutoff frequency.

**Lab Procedures:**

- a. Construct the circuit of Fig. 1, using  $R = 100\Omega$  and  $L = 1\text{mH}$ .
- Use your oscilloscope to measure both  $v_{IN}(t)$  and  $v_R(t)$ .

**DATA**

In order to measure the frequency response (amplitude and phase) of your circuit, use the function generator to apply sinusoidal inputs to the circuit. Apply inputs for at least the following frequencies:  $\frac{\omega_c}{10}$ ,  $\frac{\omega_c}{8}$ ,  $\frac{\omega_c}{4}$ ,  $\frac{\omega_c}{2}$ ,  $\omega_c$ ,  $2\omega_c$ ,  $4\omega_c$ ,  $8\omega_c$ , and  $10\omega_c$ , where  $\omega_c$  is the cutoff frequency of the circuit, as determined in the pre-lab. For each frequency, record the frequency, the input voltage amplitude, the output voltage amplitude, and the time difference between the input and output sinusoids.

**DEMO**

- Demonstrate operation of your circuit to the TA and have them initial the appropriate page(s) of your lab notebook and the lab worksheet.

**DATA**

- b. Repeat the procedures of part (a):
- Except use your oscilloscope to measure  $v_{IN}(t)$  and  $v_L(t)$ . (In this case, the inductor voltage,  $v_L(t)$ , is our output.) Again, for each of the frequencies listed in part (a), record the frequency, the input voltage amplitude, the output voltage amplitude, and the time difference between the input and output sinusoids.

**DEMO**

- Demonstrate operation of your circuit to the TA and have them initial the appropriate page(s) of your lab notebook and the lab worksheet.

**Post-lab Exercises:****SIM**

- a. Use your favorite mathematical analysis software package (MATLAB, Octave, Excel...) to plot the theoretical frequency response determined in the pre-lab, with  $R = 100\Omega$  and  $L = 1\text{mH}$ . Use  $v_R(t)$  as your output, and plot both magnitude and phase vs. frequency. Use a range of frequencies

<sup>50</sup> If the phasor representation of  $v_{IN}(t)$ ,  $v_L(t)$ , and  $v_R(t)$  are  $\underline{V}_{IN}$ ,  $\underline{V}_L$ , and  $\underline{V}_R$ , respectively, we are calculating  $\frac{V_L}{V_{IN}}$  and  $\frac{V_R}{V_{IN}}$ .

from  $\frac{\omega_c}{10}$  to  $10\omega_c$ . Overlay the data you acquired in part (a) of the lab procedures on your plot. Comment on the agreement between the data and the theoretical response. Does the circuit behave as a high-pass or low-pass filter when the output is the resistor voltage?

**SIM**

- b. Use your favorite mathematical analysis software package (MATLAB, Octave, Excel...) to plot the theoretical frequency response determined in the pre-lab, with  $R = 100\Omega$  and  $L = 1\text{mH}$ . Use  $v_L(t)$  as your output, and plot both magnitude and phase vs. frequency. Use a range of frequencies from  $\frac{\omega_c}{10}$  to  $10\omega_c$ . Overlay the data you acquired in part (b) of the lab procedures on your plot. Comment on the agreement between the data and the theoretical response. Does the circuit behave as a high-pass or low-pass filter when the output is the inductor voltage?



6. In the space below, provide a table showing the input frequencies, the amplitudes of the voltages  $v_{IN}(t)$  and  $v_L(t)$ , and the time difference between  $v_{IN}(t)$  and  $v_R(t)$  for the frequencies specified in part (a) of the lab procedures. (10 pts)
  
  
  
  
  
  
  
  
  
  
7. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your signal acquisition from the sensor. (4 pts total)  
  
**TA Initials:** \_\_\_\_\_
  
  
  
  
  
  
  
  
  
  
8. Attach to this worksheet a plot of the theoretical magnitude and phase response with  $v_R(t)$  as the circuit output, overlaid with the data from part 4 above. In the space below, comment on the agreement between the theoretical frequency response and the measured data. Also characterize the circuit as either a high-pass or a low-pass filter. (6 pts)
  
  
  
  
  
  
  
  
  
  
9. Attach to this worksheet a plot of the theoretical magnitude and phase response with  $v_L(t)$  as the circuit output, overlaid with the data from part 6 above. In the space below, comment on the agreement between the theoretical frequency response and the measured data. Also characterize the circuit as either a high-pass or a low-pass filter. (6 pts)

## Real Analog Chapter 11: Lab Projects

### 11.3.2: Passive RC Filter

This assignment concerns a passive filtering circuit consisting of a series combination of a resistor and a capacitor. If the voltage across the capacitor is taken as the filter output, the circuit will act as a low-pass filter. However, if we take the voltage across the resistor as the filter output, the circuit acts as a high-pass filter<sup>51</sup>. We will measure both the high-pass and low-pass characteristics of this circuit and compare them to our expectations based on the analytically determined frequency response.

#### Before beginning this lab, you should be able to:

- Calculate the frequency response of a passive electrical circuit
- Calculate the magnitude and phase responses of a passive electrical circuit
- Determine the DC gain, high frequency gain, and cutoff frequency of a passive first order filter

#### After completing this lab, you should be able to:

- Measure the magnitude and phase responses of first order filter circuits

#### This lab exercise requires:

- Analog Discovery 2 module
- Digiilent Analog Parts Kit
- Digital multimeter (optional)

#### Symbol Key:

- DEMO** Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
- ANALYSIS** Analysis; include principle results of analysis in laboratory report.
- SIM** Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
- DATA** Record data in your lab notebook.

#### General Discussion:

This lab assignment concerns the circuit shown in Fig. 1. We will measure the response of both the voltages  $v_R(t)$  and  $v_C(t)$  to the input voltage  $v_M(t)$  and plot the frequency response – both magnitude and phase – of both voltages. Obviously, since we are interested in the frequency response of the circuit, the input voltage will consist of sinusoids.

<sup>51</sup> If the energy in the input does not appear in the capacitor voltage, it must be somewhere else (it can't simply disappear). Thus, if the capacitor voltage stops low frequencies, the energy in these frequencies must show up somewhere else – in this case, it will be apparent in the resistor voltage (or, equivalently, the capacitor current).

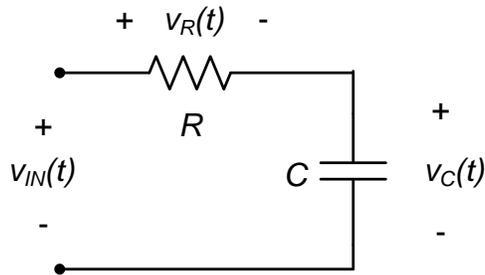


Figure 1. RC circuit.

**Pre-lab:****ANALYSIS**

Determine the frequency responses of both  $v_C(t)$  and  $v_R(t)$  in the circuit shown in Fig. 1, as a function of  $R$  and  $C$ . Use the voltage  $v_{IN}(t)$  as the input in both cases<sup>52</sup>. Calculate the cutoff frequency of the circuit (both outputs share the same cutoff frequency). Also calculate the DC gain and high frequency gain of both outputs.

**ANALYSIS**

Sketch the magnitude and phase responses of both outputs in the circuit of Fig. 1. Label your sketches with the DC gain, high frequency gain, and cutoff frequency.

**Lab Procedures:****DATA**

- a. Construct the circuit of Fig. 1, using  $R=2.2\text{k}\Omega$  and  $C = 4.7\text{nF}$
- Use your oscilloscope to measure both  $v_{IN}(t)$  and  $v_R(t)$ .

In order to measure the frequency response (amplitude and phase) of your circuit, use the function generator to apply sinusoidal inputs to the circuit. Apply inputs for at least the following frequencies:  $\frac{\omega_c}{10}$ ,  $\frac{\omega_c}{8}$ ,  $\frac{\omega_c}{4}$ ,  $\frac{\omega_c}{2}$ ,  $\omega_c$ ,  $2\omega_c$ ,  $4\omega_c$ ,  $8\omega_c$ , and  $10\omega_c$ , where  $\omega_c$ , is the cutoff frequency of the circuit, as determined in the pre-lab. For each frequency, record the frequency, the input voltage amplitude, the output voltage amplitude, and the time difference between the input and output sinusoids.

**DEMO**

- Demonstrate operation of your circuit to the TA and have them initial the appropriate page(s) of your lab notebook and the lab worksheet.

**DATA**

- b. Repeat the procedures of part (a):

- Except use your oscilloscope to measure  $v_{IN}(t)$  and  $v_C(t)$ . (In this case, the capacitor voltage,  $v_C(t)$ , is our output.) Again, for each of the frequencies listed in part (a), record the frequency, the input voltage amplitude, the output voltage amplitude, and the time difference between the input and output sinusoids.

**DEMO**

- Demonstrate operation of your circuit to the TA and have them initial the appropriate page(s) of your lab notebook and the lab worksheet.

**Post-lab Exercises:****SIM**

- a. Use your favorite mathematical analysis software package (MATLAB, Octave, Excel...) to plot the theoretical frequency response determined in the pre-lab, with  $R = 2.2\text{k}\Omega$  and  $C = 4.7\text{nF}$ . Use  $v_R(t)$  as your output, and plot both magnitude and phase vs. frequency. Use a range of frequencies

<sup>52</sup> If the phasor representation of  $v_{IN}(t)$ ,  $v_C(t)$ , and  $v_R(t)$  are  $\underline{V}_{IN}$ ,  $\underline{V}_C$ , and  $\underline{V}_R$ , respectively, we are calculating  $\frac{V_C}{V_{IN}}$  and  $\frac{V_R}{V_{IN}}$ .

from  $\frac{\omega_c}{10}$  to  $10\omega_c$ . Overlay the data you acquired in part (a) of the lab procedures on your plot. Comment on the agreement between the data and the theoretical response. Does the circuit behave as a high-pass or low-pass filter when the output is the resistor voltage?

**SIM**

- b. Use your favorite mathematical analysis software package (MATLAB, Octave, Excel...) to plot the theoretical frequency response determined in the pre-lab, with  $R = 100\Omega$  and  $L = 1\text{mH}$ . Use  $v_c(t)$  as your output, and plot both magnitude and phase vs. frequency. Use a range of frequencies from  $\frac{\omega_c}{10}$  to  $10\omega_c$ . Overlay the data you acquired in part (b) of the lab procedures on your plot. Comment on the agreement between the data and the theoretical response. Does the circuit behave as a high-pass or low-pass filter when the output is the capacitor voltage?



6. In the space below, provide a table showing the input frequencies, the amplitudes of the voltages  $v_{IN}(t)$  and  $v_C(t)$ , and the time difference between  $v_{IN}(t)$  and  $v_C(t)$  for the frequencies specified in part (a) of the lab procedures. (10 pts)
7. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuit's operation. (4 pts total)
- TA Initials:** \_\_\_\_\_
8. Attach to this worksheet a plot of the theoretical magnitude and phase response with  $v_R(t)$  as the circuit output, overlaid with the data from part 4 above. In the space below, comment on the agreement between the theoretical frequency response and the measured data. Also characterize the circuit as either a high-pass or a low-pass filter. (4 pts)
9. Attach to this worksheet a plot of the theoretical magnitude and phase response with  $v_C(t)$  as the circuit output, overlaid with the data from part 6 above. In the space below, comment on the agreement between the theoretical frequency response and the measured data. Also characterize the circuit as either a high-pass or a low-pass filter. (4 pts)

## Real Analog Chapter 11: Lab Projects

### 11.3.3: Active Low-pass Filter

Active filter circuits provide a number of the same advantages as our previous active circuits:

- As we saw in Lab projects 7.5.1 and 7.5.3, applying a load to an active circuit can have less of effect on the circuit's behavior than if the circuit were passive
- For an active circuit, the output voltage can be larger than the input voltage – the active circuit can amplify a signal.
- The high input impedance of operational amplifiers can help reduce the effect that the passive filter has on the signal source. This can be particularly important in instrumentation applications; since many sensors provide extremely limited power, it is important to process the sensor's output with a circuit which draws little or no power from the sensor.

In this lab project, we will design an active low-pass filter to provide a desired DC gain, cutoff frequency, and input resistance. The circuit used in this project also *inverts* the input – the DC output voltage has the opposite sign as the input voltage.

#### Before beginning this lab, you should be able to:

- Calculate the frequency response of an active electrical circuit
- Determine the DC gain, high frequency gain, and cutoff frequency of a first order filter
- Measure the magnitude and phase responses of first order filter circuits (Labs 11.3.1, 11.3.2)

#### After completing this lab, you should be able to:

- Calculate the magnitude and phase responses of an active electrical circuit
- Design an active filter to provide a desired DC gain, cutoff frequency, and input impedance

#### This lab exercise requires:

- Analog Discovery 2 module
- Digilent Analog Parts Kit
- Digital multimeter (optional)

#### Symbol Key:

- DEMO** Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
- ANALYSIS** Analysis; include principle results of analysis in laboratory report.
- SIM** Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
- DATA** Record data in your lab notebook.

#### General Discussion:

The circuit of Fig. 1 is an inverting low pass filter. The magnitude response of the circuit is

$$\left| \frac{V_{OUT}}{V_{IN}} \right| = - \frac{\frac{R_2}{R_1}}{\sqrt{1+(\omega R_2 C)^2}} \quad \text{Eq. 1}$$

In this assignment, we will choose values of  $R_1$ ,  $R_2$ , and  $C$  in the circuit of Fig. 1 to meet design requirements set on:

- **Input resistance:** the input resistance is the ratio of input voltage to input current for a circuit. In the circuit of Fig. 1, the input resistance essentially specifies the resistance  $R_1$ .

- DC gain: the DC gain is, from equation (1), the ratio of  $R_2$  to  $R_1$ . Once  $R_1$  is determined from the input impedance requirement, the DC gain specifies the required value for  $R_2$ .
- Cutoff frequency: once  $R_2$  is known, the cutoff frequency requirement specifies the value of the capacitor  $C$ .

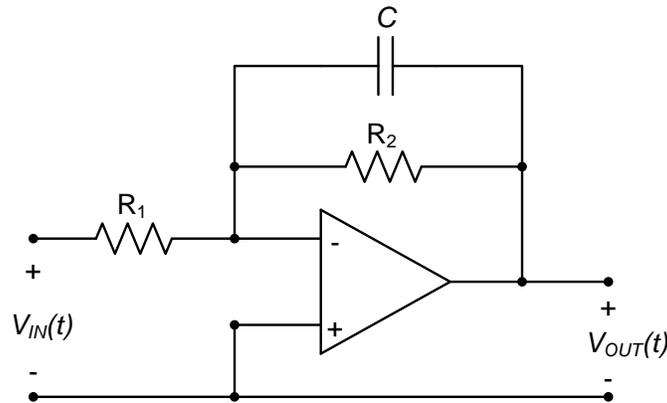


Figure 1. Inverting low-pass filter circuit.

### Pre-lab:

#### ANALYSIS

Design the circuit of Fig. 1 (e.g. choose values for  $R_1$ ,  $R_2$ , and  $C$ ) to meet the following design requirements:

- Input impedance  $\geq 10\text{k}\Omega$
- DC gain = 2
- Cutoff frequency = 10kHz

### Lab Procedures:

#### DATA

Construct the circuit you designed in the pre-lab. Be sure to measure the actual values.

#### DATA

- Measure the magnitude response of the circuit over a range of frequencies  $\frac{\omega_c}{10} < \omega < 10\omega_c$ , where  $\omega_c$  is the cutoff frequency of the circuit<sup>53</sup>. Make sure you measure the response for at least six different frequencies.

#### DEMO

- Demonstrate operation of your circuit to the TA and have them initial the appropriate page(s) of your lab notebook and the lab worksheet.

#### ANALYSIS

- From your measured data, determine the actual cutoff frequency and DC gain of your circuit. Compare your measured values to the design requirements. Comment on the differences between the design requirements and your measured values.

<sup>53</sup> Keep in mind that the units of  $\omega$  are radians/second, while the design requirement on the cutoff frequency is given in Hz.



## Real Analog Chapter 11: Lab Projects

### 11.3.4: Signal Conditioning – Audio Application

When making time-varying measurements, the sensor being used often has at least a few undesirable characteristics. Common shortcomings of sensors are lower than desired sensitivity, excessive noise levels, and DC offsets. *Frequency selective circuits* are often used to *condition* the sensor's output signal to reduce the effects of these shortcomings. *Low-pass filters*, for example, can be used to increase the sensor's low-frequency sensitivity while reducing the high-frequency noise components in the sensor output signal.

In this lab assignment, we will condition the output from a microphone. The microphone is a sensor which converts acoustic waves to electrical signals. The microphone we will be using has fairly low output levels (on the order of 20mV) and a relatively large DC offset. Our goal is to design a filter to amplify the frequencies of interest – those frequencies associated with audio signals – and suppress the DC offset.

#### Before beginning this lab, you should be able to:

- Calculate the frequency response of a active electrical circuit
- Determine the DC gain, high frequency gain, and cutoff frequency of a first order filter
- Categorize frequency-selective circuits as high pass or low pass filters
- Measure the magnitude and phase responses of first order filter circuits (Labs 11.3.1, 11.3.2)

#### After completing this lab, you should be able to:

- Design an active filter high pass filter to provide a desired high frequency gain, cutoff frequency, and input impedance
- Use a frequency selective filter to process a microphone's output signal

#### This lab exercise requires:

- Analog Discovery 2 module
- Digilent Analog Parts Kit
- Digital multimeter (optional)

#### Symbol Key:

- DEMO** Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
- ANALYSIS** Analysis; include principle results of analysis in laboratory report.
- SIM** Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
- DATA** Record data in your lab notebook.

## 1 Microphone

A microphone converts sound to an electrical signal. We will be using an ADMP504 microphone in this lab project. The ADMP504 microphone is in a surface-mount package; this package does not allow direct implementation in our solderless breadboard. The ADMP504 part in your analog parts kit has been mounted onto a circuit board with pins which can be inserted directly in your breadboard. Figure 1 provides top and bottom views of this circuit board. The ADPM504 microphone is visible on the bottom view of the board, as shown in Fig. 1(a). The pin indicator showing the location of pin 1 can also be seen in the bottom view. Pins are consecutively numbered clockwise from pin 1, also as shown in Fig. 1(a). In the top view of the chip, a small hole is visible – this hole allows

sound to contact the ADMP504 sensor. The hole, and the pin locations seen in the top view are shown in Figure 1(b).

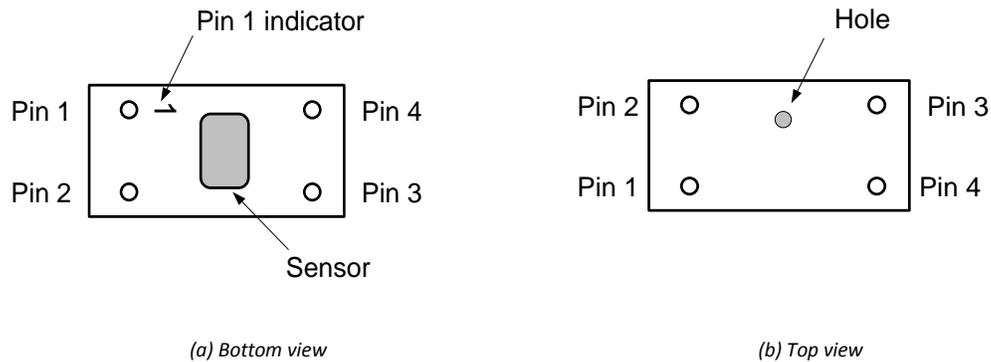


Figure 1. Top and bottom views of the analog parts kit chip on which the microphone is mounted.

The descriptions of the pins shown in Fig. 1 are as follows:

- **Pin 1:** Output voltage. This pin provides a voltage which indicates the audio signal applied to the microphone.
- **Pin 2:** Ground.
- **Pin 3:** Voltage source,  $V_{DD}$ . This power source is necessary for the microphone to work. We will use  $V_{DD} = 3.3V$ .
- **Pin 4:** Not connected.

Additional information about the ADMP504 microphone can be found on the Analog Devices website, <http://www.analog.com>.

### Pre-lab:

None

### Lab Procedures:

#### DATA

- In this portion of the lab project, we will measure the response of the sensor and use this data to determine the amplification necessary to provide us with an output signal of the desired sensitivity.

Use the arbitrary waveform generator to apply 3.3V to pin 3 of the microphone board<sup>54</sup>. Connect pin 2 of the chip to ground. Use one channel of your oscilloscope to measure the voltage difference between pin 1 and ground. Make a sound<sup>55</sup> and verify that you are receiving a signal on the oscilloscope from the sensor. At this stage, you will probably want to use a fairly large time base on your oscilloscope (greater than 100msec/division).

#### DATA

- With a relatively large vertical scale (500 mV/division or so) measure and record the DC offset of the sensor.

#### DATA

- With a relatively large vertical offset (approximately the negative of the DC voltage measured in part i above), set the vertical scale on your oscilloscope channel to be on the order of 10 – 20 msec/division. Measure and record the amplitude of the time-varying signal

<sup>54</sup> The waveform generator can be used to apply a constant voltage if the amplitude is set to 0V and the offset is set to 3.3V.

<sup>55</sup> Whistling or snapping your fingers tend to be good sound sources; they are fairly repeatable.

resulting from your sound source. It may be useful to decrease the time base on your oscilloscope significantly in order to accurately measure this value. Setting a trigger to acquire the waveform may assist you once you have reduced the time base.

- b. In Part II of this project, we want design a filter which uses the sensor output voltage to produce a minimum 1V peak-to-peak signal with no DC offset. Use the time varying data acquired in part ii above to estimate the amplification necessary to produce the desired 1V amplitude.

**ANALYSIS**

## 2 Signal Conditioning Circuit

The circuit of Fig. 2 is an inverting high pass filter. The frequency response of the circuit is:

$$\frac{V_{OUT}}{V_{IN}} = -\frac{R_2}{R_1} \frac{j\omega}{j\omega + \frac{1}{R_1 C}} \quad \text{Eq. 1}$$

So that the magnitude response is:

$$\left| \frac{V_{OUT}}{V_{IN}} \right| = -\frac{R_2}{R_1} \frac{\omega}{\sqrt{\omega^2 + \left(\frac{1}{R_1 C}\right)^2}} \quad \text{Eq. 2}$$

In this assignment, we will choose values of  $R_1$ ,  $R_2$ , and  $C$  in the circuit of Fig. 2 to meet design requirements set on:

- **Input resistance:** the input resistance is the ratio of input voltage to input current for a circuit. In the circuit of Fig. 1, the input resistance is dependent upon both the capacitance and the frequency. However, at high frequencies, the capacitor behaves like a short circuit, and the input resistance is essentially the resistance  $R_1$ .
- **High frequency gain:** the high frequency gain (the gain as  $\omega \rightarrow \infty$ ) is, from equation (2), the ratio of  $R_2$  to  $R_1$ . Once  $R_1$  is determined from the input resistance requirement, the high frequency gain specifies the required value for  $R_2$ .
- **Cutoff frequency:** once  $R_2$  is known, the cutoff frequency requirement specifies the value of the capacitor  $C$ .

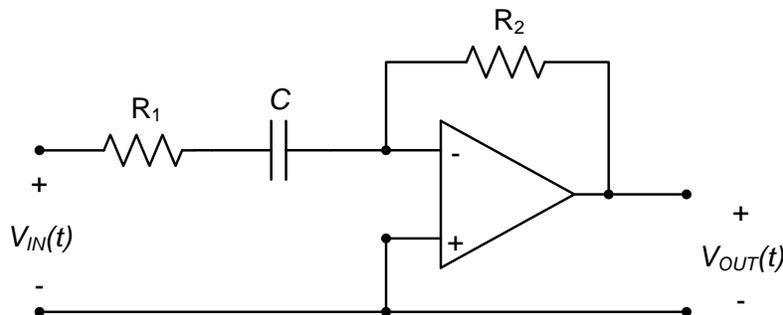


Figure 1. Inverting high-pass filter circuit.

### Pre-lab:

**ANALYSIS**

Design the circuit of Fig. 1 (e.g. choose values for  $R_1$ ,  $R_2$ , and  $C$ ) to meet the following design requirements:

- $R_1 \geq 2k\Omega$ . (This essentially sets the input resistance for the filter.)
- High frequency gain (gain as  $\omega \rightarrow \infty$ ) = Amplification value necessary to produce 1V peak-to-peak output voltage, as determined in the Lab Procedures for Part I.
- Cutoff frequency  $\leq 500\text{Hz}$

**ANALYSIS**

Determine the response of the circuit of Fig. 1 to a DC input voltage. What will the circuit's response be to the DC voltage provided by the sensor?

**Lab Procedures:****DATA**

Construct the circuit you designed in the pre-lab. Be sure to measure the actual values.

**DATA**

- i. Measure the magnitude response of the circuit over a range of frequencies  $\frac{\omega_c}{10} < \omega < 10\omega_c$ , where  $\omega_c$  is the cutoff frequency of the circuit<sup>56</sup>. Make sure you measure the response for at least six different frequencies.

**DEMO**

- ii. Demonstrate operation of your circuit to the TA and have them initial the appropriate page(s) of your lab notebook and the lab worksheet.

**ANALYSIS**

- iii. From your measured data, determine the actual cutoff frequency and high frequency gain of your circuit. Compare your measured values to the design requirements. Comment on the differences between the design requirements and your measured values.

### 3 Overall System Integration

We will now integrate the signal conditioning circuit designed and built in Part II with the sensor of Part I. The goal is to amplify the “important” part of the response of the sensor – the time-varying signal corresponding to the sound – and remove the undesirable DC level in the sensor output. One possibly important drawback to this approach, of course, is that desirable lower-frequency information will also be removed from the data.

**Pre-lab:**

None

**Lab Procedures:****DATA**

- i. Apply the sensor output voltage to the input terminals of the signal conditioning circuit,  $V_{IN}(t)$ . Using the oscilloscope, measure both  $V_{in}(t)$  from the sensor and the signal conditioning unit's output voltage,  $V_{OUT}(t)$  in Fig. 2. Make a sound (clap your hands, whistle,...) and display the resulting waveform on the oscilloscope main window. Record an image of the oscilloscope window, showing the voltages  $V_{IN}(t)$  and  $V_{OUT}(t)$ . Comment on your results relative to your expectations.

**DEMO**

- ii. Demonstrate operation of your circuit to the TA and have them initial the appropriate page(s) of your lab notebook and the lab worksheet.

<sup>56</sup> Keep in mind that the units of  $\omega$  are radians/second, while the design requirement on the cutoff frequency is given in Hz.

## Real Analog Chapter 11: Lab Worksheets

### 11.3.4: Signal Conditioning – Audio Application (100 points total)

#### 1. Microphone (20 points total)

1. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your signal acquisition from the microphone. (8 pts)

TA Initials: \_\_\_\_\_

2. In the space below, provide the DC offset and the amplitude of the time-varying portion of the microphone output. (7 pts)

3. In the space below, provide your estimate of the amplification necessary to provide the desired 1V output amplitude. (5 pts)

#### 2. Signal Conditioning Circuit (50 points total)

1. Provide below a schematic showing your signal conditioning circuit. Include desired resistance and capacitance values (3 pts)





## Real Analog Chapter 11: Lab Projects

### 11.3.5: Signal Conditioning – Vibration Measurement

When making time-varying measurements, the sensor being used often has lower than desired sensitivity and a higher than desired noise level. *Frequency selective circuits* are therefore often used to *condition* the sensor's output signal. *Low-pass filters*, for example, can be used to increase the sensor's low-frequency sensitivity while reducing the high-frequency noise components in the sensor output signal.

In this lab assignment, we will condition the output from a piezoelectric sensor with a low-pass filter. Our goal will be to design a filter to amplify the frequencies of interest – those frequencies below the fundamental vibration frequency of the beam – and suppress higher-frequency components of the signal, which we will interpret as noise.

#### Before beginning this lab, you should be able to:

- Perform frequency-domain analysis of electrical circuits
- Represent sinusoidal signals in phasor form
- Analyze operational amplifier based electrical circuits

#### After completing this lab, you should be able to:

- Use a piezoelectric sensor to measure the vibration of a cantilever beam.
- Estimate the fundamental frequency in a measured signal
- Design and implement a first order low-pass signal conditioning circuit
- Measure the frequency response (gain and phase) of a system

#### This lab exercise requires:

- Analog Discovery 2 module
- Digilent Analog Parts Kit
- Digital multimeter (optional)

#### Symbol Key:

- DEMO** Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
- ANALYSIS** Analysis; include principle results of analysis in laboratory report.
- SIM** Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
- DATA** Record data in your lab notebook.

## 1 Piezoelectric Sensor Vibration

The *dynamic response* (or time-varying response) of a mechanical system can be extremely important in the determination of the structural loads in the system. The dynamic response can often be the dominant contributing factor in the stresses in the structure and, as such, can be the primary factor in a structure's failure. The famous Tacoma Narrows bridge failure was caused by the bridge's dynamic response to wind loads. A loading condition is considered to be dynamic if the loading condition changes relatively rapidly compared to how quickly the structure can respond to the load. "Plucking" a guitar string or striking a tuning for, for example, are dynamic loading conditions, since they set up responses which persist much longer than the actual application of the input.

A system's dynamic response is often interpreted in terms of *vibration*. Vibration is essentially an oscillatory mechanical displacement. In our guitar string and tuning fork examples above, it is apparent that the systems oscillate as a result of the applied input. In general, structural vibrations do not consist of a single, sinusoidal, frequency component. (The tuning fork example is an exception to this rule; tuning forks are designed to vibrate at a single frequency, thus producing a pure auditory tone.) If multiple frequency components are present in a system's dynamic response, these frequency components are generally described in terms of the system's *modes* of vibration. The modes of vibration of the system are the *independent* ways in which the natural response of the system can vibrate. Some modes will be more important than others, from the standpoint of their contribution to the overall dynamic displacement of the system; the most important modes are often called the *dominant* or *fundamental* modes. In the guitar string example above, there are a number of frequencies which contribute to the sound we hear from the guitar (the sound is not a pure tone); these contributions are due to the different vibrational modes of the guitar string.

Vibrations are often measured using piezoelectric sensors. Piezoelectric devices produce a voltage when they are deflected. By bonding the device to a vibrating system, the output voltage from the sensor will indicate the vibration levels the system is experiencing. In this part of the lab assignment, we will induce a vibration in the piezoelectric sensor itself and measure its response. This data will be used to estimate the fundamental mode of vibration of the sensor. The design of the low-pass filter we will use later as our signal conditioning system will be based on this fundamental mode of vibration.

### Pre-lab:

Read the material in Appendix A relative to piezoelectric sensors.

### Lab Procedures:

- a. Connect the leads of one channel of your oscilloscope to the sensor contacts (the tabs protruding from the piezoelectric sensor). "Flick" the sensor and verify that you are receiving a signal on the oscilloscope from the sensor. Adjust the time and amplitude scales on your oscilloscope until the oscilloscope displays what you feel is a reasonable representation of the sensor's deflection as a function of time on the oscilloscope.
- b. Demonstrate operation of your circuit to the TA and have them initial the appropriate page(s) of your lab notebook and the lab worksheet.
- c. Verify that the piezoelectric sensor provides no steady-state response to a constant input. To do this, use the oscilloscope to monitor the output voltage from the sensor as above. Bend the sensor to a constant deflection (bend it and leave it bent) and monitor the output of the sensor on your oscilloscope. The sensor output voltage should return to zero volts, even though the sensor is still deflected. Comment on your results in your lab notebook. Include a qualitative discussion as to what frequencies the sensor responds to. What type of frequency-selective circuit does the sensor resemble (low-pass, high-pass, band-pass, etc.)?
- d. Measure the natural response of the sensor. To do this, hold the end of the sensor to which the leads are attached and deflect the other end of the sensor slightly. This corresponds to an initial condition on the sensor deflection. Release the sensor tip suddenly, this allows the sensor to respond to this initial condition. Since no external forces act on the sensor after it is released, this corresponds to the natural response of the sensor.

#### DEMO

#### ANALYSIS

#### DATA

#### ANALYSIS

Acquire the natural response waveform on your oscilloscope. (You may wish to use the **run/stop** button or the **single sequence** capability on your oscilloscope to do this. **Run/stop** will require you to **manually stop the oscilloscopes data acquisition when the signal is acquired**; single sequence will use the oscilloscope trigger to acquire the waveform) Save the waveform to a file. Estimate the dominant frequency in the signal and note it in your lab notebook. (Note: this will correspond to the fundamental mode of the sensor; this *mode shape* consists of the sensor

vibrating in a “shape” that we typically associate with the motion of a swimming pool diving board, as shown in Fig. 1.) Also note that the recorded waveform contains frequency components other than the dominant frequency; the signal is not a pure sinusoid with a single frequency.

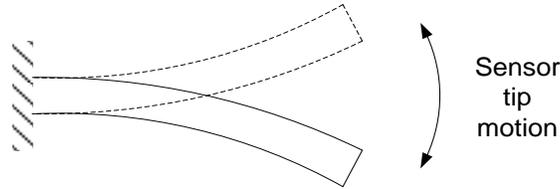


Figure 1. Dominant mode shape for sensor vibration (side view).

## 2 Signal Conditioning Circuit

The voltage output from many measurement systems suffers from two primary shortcomings:

1. The output voltage can be noisy.
2. The sensitivity of the output voltage can be lower than desired.

To overcome the above problems, we will design and implement an electrical circuit to *condition* the output voltage from the sensor before displaying the signal on the oscilloscope. The circuit we will use performs two primary functions, each of which is intended to compensate for one of the above shortcomings. The circuit will:

1. *Low-pass filter* the output signal from the sensor. This will reduce the high-frequency noise in the sensor’s output voltage.
2. *Amplify* the output signal from the sensor. This will increase the sensitivity of the overall measurement.

We will implement the above operations using the circuit shown in Fig. 2. The frequency response of the circuit shown in Fig. 2 is:

$$\frac{V_{OUT}}{V_{IN}} = \frac{R_1 + R_2}{R_1} \frac{\frac{1}{R_3 C}}{j\omega + \frac{1}{R_3 C}} \quad \text{Eq. 1}$$

So the amplitude response of the circuit is:

$$\left| \frac{V_{OUT}}{V_{IN}} \right| = \frac{R_1 + R_2}{R_1} \frac{\frac{1}{R_3 C}}{\sqrt{\omega^2 + \frac{1}{R_3 C}}} \quad \text{Eq. 2}$$

The amplitude response of the overall signal conditioning circuit of Fig. 2 is shown in Fig. 3. The low frequency gain (as  $\omega \rightarrow 0$ ) of the circuit is  $\frac{R_1 + R_2}{R_1}$ , and the filter’s output goes to zero at high frequencies (as  $\omega \rightarrow \infty$ ). The *cutoff frequency* of the circuit indicates at what frequency the filter’s output begins to decrease rapidly; for our circuit, the cutoff frequency is  $\omega_c = \frac{1}{R_3 C}$ . The DC gain can be used to amplify the output of the piezoelectric sensor in the pass band, while the stop band can be used to eliminate the noise in the signal at high frequencies. (Signals entering the circuit with frequencies below the cutoff frequency – the pass band – are amplified; signals entering the circuit with frequencies above the cutoff frequency – the stop band – are attenuated.)

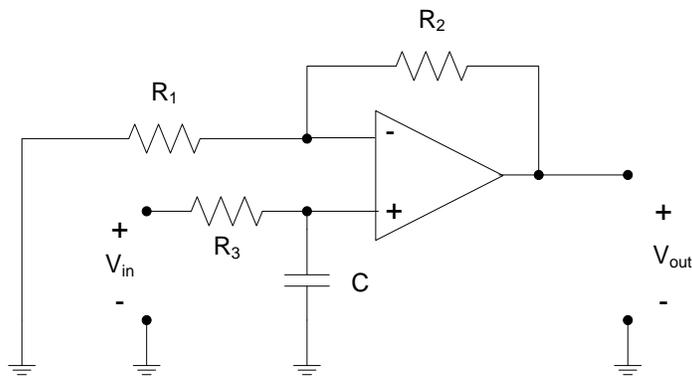


Figure 2. Signal conditioning circuit.

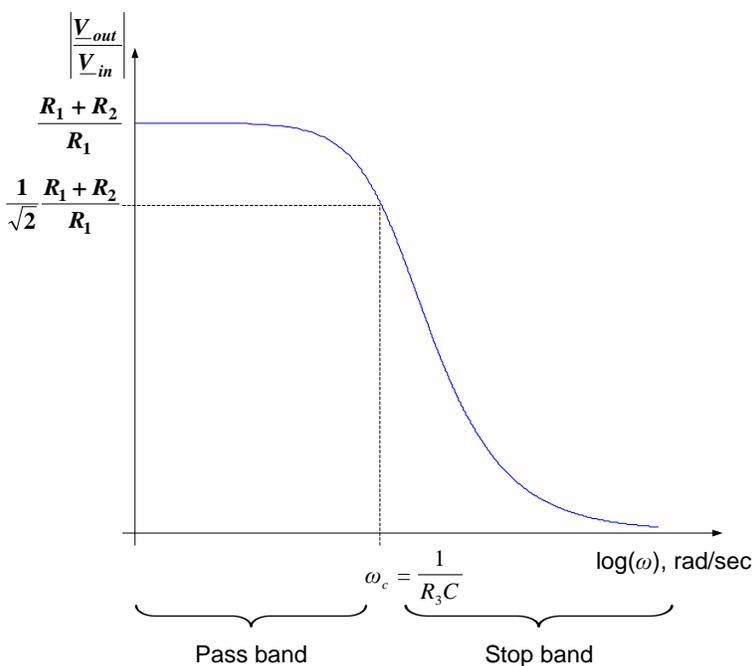


Figure 3. Signal conditioning circuit amplitude response.

**Pre-lab:**

**ANALYSIS**  
**ANALYSIS**  
**ANALYSIS**

- Using equation (1) as your starting point, show that the amplitude response of the circuit of Fig. 2 is as provided in equation (2)
- Using equation (1) as your starting point, determine the phase response of the circuit of Fig. 3
- Determine the cutoff frequency of the circuit of Fig. 2. What is the gain and phase of the circuit at the cutoff frequency?

**Lab Procedures:**

**ANALYSIS**  
**DATA**

- Design a circuit like that shown in Fig. 2 (e.g. choose  $R_1$ ,  $R_2$ ,  $R_3$  and  $C$ ) to provide a DC gain of approximately two and a cutoff frequency of roughly twice the sensor’s dominant vibration frequency as determined in Part I of this lab assignment.
- Construct the circuit you designed in part (a). Record actual resistance and capacitance values.

## DATA

- c. Measure the frequency response (amplitude and phase) of your circuit. To do this, use the function generator to apply sinusoidal inputs to the circuit. Record input voltage amplitude, output voltage amplitude, the time difference between the two, and frequency for at least 5 or 6 values of frequency; make sure you use a range of frequencies which includes your cutoff frequency. Note: Appendix B of this lab assignment provides tips relative to gain and phase measurement.

## DEMO

- d. Demonstrate operation of your circuit to the TA and have them initial the appropriate page(s) of your lab notebook and the lab worksheet.

## ANALYSIS

- e. Calculate the gain and phase of your circuit for the frequencies measured in part (c). Plot the amplitude response in your lab notebook; use a logarithmic scale on the frequency axes of your plots. Discuss your measured response vs. the expected response from your pre-lab calculations. In particular, compare the actual and expected gain and phase at low frequencies, high frequencies, and the cutoff frequency.

### 3 Overall System Integration

We will now integrate the signal conditioning circuit designed and built in Part II with the sensor of Part I. Since our signal conditioning circuit's cutoff frequency is approximately twice the natural frequency of the sensor, most of the low frequency components in the sensor's output should lie within the pass band of the signal conditioning circuit. The goal is to amplify the "important" part of the response of the sensor and remove the (hopefully) less significant higher frequency content in the sensor output – high frequency noise, for example, will be removed by the low-pass filter. One possibly important drawback to this approach, of course, is that desirable higher-frequency information will also be removed from the data.

#### Pre-lab:

None

#### Lab Procedures:

## DATA

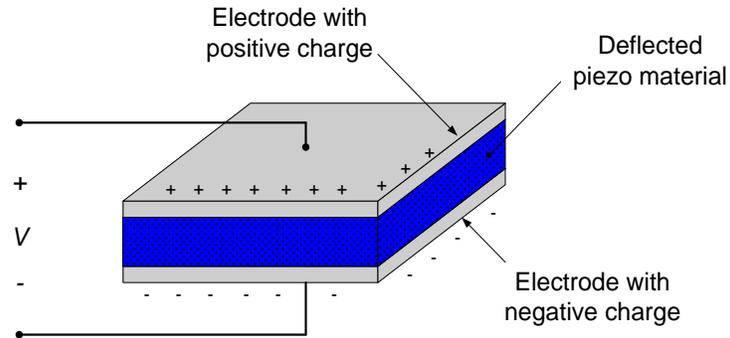
- a. Apply the sensor output voltage to the input terminals of the signal conditioning circuit,  $V_{IN}(t)$ . Using the oscilloscope, measure both  $V_{in}(t)$  from the sensor and the signal conditioning unit's output voltage,  $V_{OUT}(t)$  in Fig. 2. "Flick" the sensor and record an image of the oscilloscope window, showing the voltages  $V_{IN}(t)$  and  $V_{OUT}(t)$ . Comment on your results relative to your expectations.

## DEMO

- b. Demonstrate operation of your circuit to the TA and have them initial the appropriate page(s) of your lab notebook and the lab worksheet.

### Appendix A: Piezoelectric sensors

Some materials (certain crystals, for example) produce a charge when they are deflected; this is called a *piezoelectric* effect; materials which exhibit this property are called *piezoelectric materials* or *piezo materials*. If a piezoelectric material is sandwiched between two conductors, or *electrodes*, a voltage difference is produced between the electrodes when the material is deflected. A typical arrangement is shown in the figure below.



A constant (or *static*) deflection of a piezoelectric material will result in a fixed charge at the sensor's electrodes. *Leakage effects*, either within the piezoelectric material or the electronics associated with the sensor, cause this charge to dissipate with time. Thus, piezoelectric sensors cannot generally be used for *static* measurements (measurement of constant values) since the sensor's output voltage will decay to zero if the piezoelectric material's deflection is constant. Piezoelectric devices do, however, make excellent *dynamic* sensors (sensors which record time-varying phenomena) in which the piezoelectric material deflects rapidly relative to the leakage rate.

Piezoelectric sensors are often used in the measurement of time-varying pressures, accelerations, and forces. In these applications, the sensor is set up so that the process to be measured results in deflection of the piezoelectric material; the resulting voltage is used to indicate the desired physical parameter. A force applied to the material, for example, induces a stress in the material with a corresponding deformation of the material.

The piezoelectric sensor provided in the analog parts kit consists of a very thin piezoelectric film sandwiched between two printed electrodes and laminated to a polyester substrate. Contacts are provided to make connections to measure the response voltage. The device is shown below.

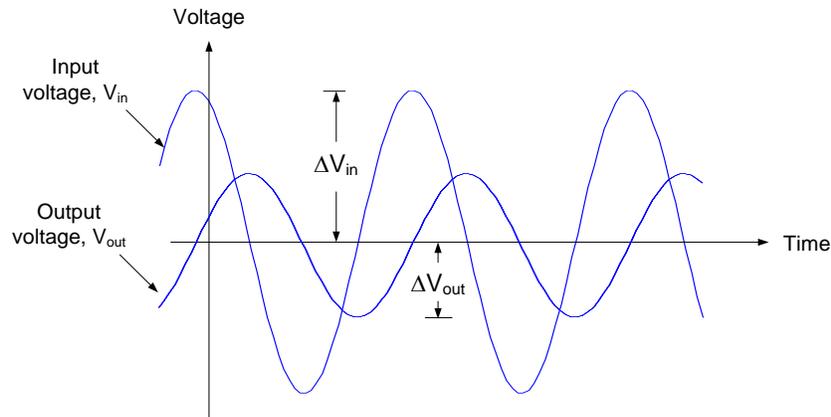


## Appendix B: Measuring Gain and Phase

The gain of a system at a particular frequency is the ratio of the magnitude of the output voltage to the magnitude of the input voltage at that frequency, so that:

$$\text{Gain} = \frac{\Delta V_{out}}{\Delta V_{in}}$$

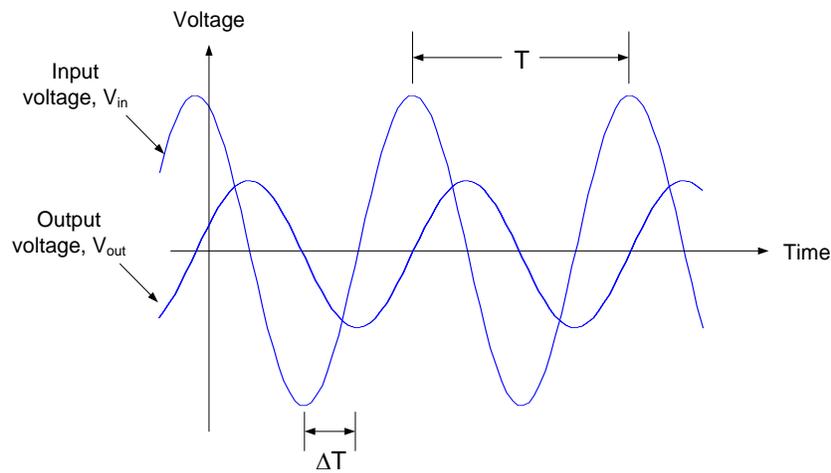
Where  $\Delta V_{out}$  and  $\Delta V_{in}$  can be measured from the sinusoidal input and output voltages as shown in the figure below.



The phase of a system at a particular frequency is a measure of the time shift between the output and input voltage at that frequency, so that:

$$\text{Phase} = \frac{\Delta T}{T} \times 360^\circ$$

Where  $\Delta T$  and  $T$  can be measured from the sinusoidal input and output voltages as shown in the figure below.



## Real Analog Chapter 11: Lab Worksheets

### 11.3.5: Signal Conditioning – Vibration Measurement (100 points total)

#### 1. Piezoelectric Sensor Vibration (20 points total)

1. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your signal acquisition from the sensor. (8 pts)

**TA Initials:** \_\_\_\_\_

2. Attach, to this worksheet, a plot of the sensor's natural response. (7 pts)
3. In the space below, provide your estimate of the dominant frequency of the sensor. Annotate your plot of part 3 to indicate how this dominant frequency was determined. (5 pts)

#### 2. Signal Conditioning Circuit (50 points total)

1. Provide below a schematic showing your signal conditioning circuit. Include desired resistance and capacitance values (3 pts)
2. Attach, to this worksheet, analyses which support your circuit design. These analyses should include (at a minimum) your desired low frequency gain and cutoff frequency; chosen resistance and capacitance values, and how these values were chosen. (15 pts)

3. In the space below, provide a table containing the following. (15 pts)
- The frequencies at which the response data was measured
  - The measured data characterizing your signal processing circuit (the input and output voltage amplitudes; time difference)
  - The calculated amplitude and phase response.
4. Attach, to this worksheet, a plot of the measured gain and phase vs. frequency for your signal conditioning circuit. In the space below, provide a discussion of measured vs. expected performance. (7 pts)
5. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuit's operation. (10 pts total)

**TA Initials:** \_\_\_\_\_



## Real Analog Chapter 11: Lab Projects

### 11.4.1: Passive Low-pass Filter

Bode plots are a convenient way to represent frequency responses. In particular, straight-line approximations to Bode plots are extremely easy to generate. In this lab assignment, we will generate a straight line approximation to the Bode plot for a passive low-pass filter. We will then measure the frequency response of the circuit at several discrete frequencies and compare these to our straight-line approximation. Finally, we will use the Analog Discovery 2's Bode Transfer Function instrument to automatically generate a Bode plot for the circuit.

#### Before beginning this lab, you should be able to:

- Generate straight line approximations to the Bode plot for a first order circuit
- Determine the DC gain, high frequency gain, and cutoff frequency of a first order filter
- Measure the magnitude and phase responses of first order filter circuits (Labs 11.3.1, 11.3.2)

#### After completing this lab, you should be able to:

- Measure the magnitude and phase responses of an electrical circuit at discrete frequencies
- Use the Bode Transfer Function instrument to automatically measure the Bode plot for a circuit

#### This lab exercise requires:

- Analog Discovery 2 module
- Digilent Analog Parts Kit
- Digital multimeter (optional)

#### Symbol Key:

**DEMO**

Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.

**ANALYSIS**

Analysis; include principle results of analysis in laboratory report.

**SIM**

Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.

**DATA**

Record data in your lab notebook.

#### General Discussion:

The circuit of interest in this lab project is shown in Fig. 1.

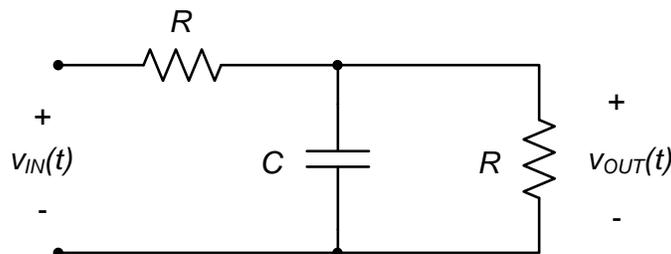


Figure 1. Passive low-pass filter circuit.

## Pre-lab:

## ANALYSIS

Sketch a straight-line approximation to the Bode plot for the circuit of Fig. 1 if  $R = 2.2\text{k}\Omega$  and  $C = 100\text{nF}$ .

## Lab Procedures:

## DATA

- a. Construct the circuit of Figure 1 if  $R = 2.2\text{k}\Omega$  and  $C = 100\text{nF}$ .
- Be sure to measure and record the actual values for the two resistors. If your DMM has a capacitance measurement, record the actual value for  $C$ ; if you do not have the ability to measure capacitance, assume that the nominal value of your capacitor is correct.

## DATA

- Measure the magnitude response of the circuit at frequencies approximately equal to  $\frac{\omega_c}{8}$ ,  $\frac{\omega_c}{4}$ ,  $\frac{\omega_c}{2}$ ,  $\omega_c$ ,  $2\omega_c$ ,  $4\omega_c$ , and  $8\omega_c$ , where  $\omega_c$  is the cutoff frequency of the circuit<sup>57</sup>.

## DEMO

- Demonstrate operation of your circuit to the TA and have them initial the appropriate page(s) of your lab notebook and the lab worksheet.
- b. Use the Bode Transfer Function instrument to automatically generate a Bode plot for the circuit. To do this, follow the steps below:
- Click on the **Network Analyzer** in the WaveForms main window. A window will open, which will allow you to automatically create a Bode plot for the system.
  - Bode plot parameters are set using the options shown in Fig. 2 below. The available options are:
    - Start and Stop:** provide the range of frequencies which will be plotted. Generally, a range of frequencies from about an order of magnitude below the cutoff frequency to about an order of magnitude above the cutoff frequency is a good starting point.
    - WaveGen Offset and Amplitude:** The offset and amplitude of the input voltage. Use the offset and amplitude that you used to measure the response in part (a).
    - Samples:** The number of data points that will be used to generate the plot and the maximum gain that can be measured. More points will result in a smoother curve, but will take longer to generate.
    - Magnitude and Phase Settings:** The drop-down menus here allow you to choose the scale on both the magnitude and phase responses.
    - Channel 1 and Channel 2 Settings:** Choose the channels to be displayed.

## DATA

Example parameters for this lab project are shown in Fig. 2. Record an image of the Network analyzer window. Also save the data to a .csv file for later processing by selecting the **File** option on the menu bar and clicking on **Export**.



Figure 2. Example parameters for this lab project.

## Post-lab Exercises:

<sup>57</sup> Keep in mind that the units of  $\omega$  are radians/second, while the design requirement on the cutoff frequency is given in Hz.

**SIM**

Use your favorite mathematical software package (MATLAB, Octave, Excel,...) to plot the frequency response data (gain and phase vs. frequency) acquired by the Bode Transfer Function instrument in part (b) of the lab procedures. Overlay on this plot the data you acquired in part (a) of the lab procedures. Compare the responses from parts (a) and (b) and comment on any differences.

**ANALYSIS****ANALYSIS**

Compare the data you acquired in parts (a) and (b) of the lab procedures with your expectations from the pre-lab. Especially provide an assessment as to whether the original design requirements were met by your design.

## Real Analog Chapter 11: Lab Worksheets

### 11.4.1: Passive Low-pass Filter (45 points total)

1. Attach to this worksheet a sketch of the straight-line approximation to the Bode plot for the circuit of Fig. 1 if  $R = 2.2\text{k}\Omega$  and  $C = 100\text{nF}$ . (5 pts)
2. In the space below, sketch the circuit of Fig. 1 with measured values for the resistances and capacitance (if available). (3 pts)
3. In the space below, tabulate the input frequencies and the magnitude response of your circuit (in decibels) at each of these frequencies. (Note: feel free to include additional data in your table. It may result in partial credit.) (10 pts)
4. **DEMO:** Have a teaching assistant initial this sheet, indicating that they have observed your circuit's operation. (4 pts total)

TA Initials: \_\_\_\_\_

5. Attach to this worksheet an image showing the output of the Network Analyzer window. (8 pts)

6. Attach to this worksheet a plot showing the Network Analyzer data overlaid with the tabulated data from part 3 above. (10 pts)
  
7. In the space below, comment on the agreement between the agreement between the data acquired in part 3 and the data acquired with the Network Analyzer in part 6. Also comment on the agreement between your expectations based on your pre-lab analysis and both sets of data. (5 pts)

## Real Analog Chapter 11: Lab Projects

### 11.4.2: Non-inverting Low-pass Filter

Bode plots are a convenient way to represent frequency responses. In particular, straight-line approximations to Bode plots are extremely easy to generate. In this lab assignment, we will generate a straight line approximation to the Bode plot for an active low-pass filter. We will then measure the frequency response of the circuit at several discrete frequencies and compare these to our straight-line approximation. Finally, we will use the Analog Discovery 2's Bode Transfer Function instrument to automatically generate a Bode plot for the circuit.

#### Before beginning this lab, you should be able to:

- Generate straight line approximations to the Bode plot for a first order circuit
- Determine the DC gain, high frequency gain, and cutoff frequency of a first order filter
- Measure the magnitude and phase responses of first order filter circuits (Labs 11.3.1, 11.3.2)

#### After completing this lab, you should be able to:

- Measure the magnitude and phase responses of an active electrical circuit at discrete frequencies
- Use the Bode Transfer Function instrument to automatically measure the Bode plot for a circuit
- Design an active filter to provide a desired DC gain and cutoff frequency

#### This lab exercise requires:

- Analog Discovery 2 module
- Digilent Analog Parts Kit
- Digital multimeter (optional)

#### Symbol Key:

<b>DEMO</b>	Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
<b>ANALYSIS</b>	Analysis; include principle results of analysis in laboratory report.
<b>SIM</b>	Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
<b>DATA</b>	Record data in your lab notebook.

#### General Discussion:

The circuit of Fig. 1 is a non-inverting low pass filter. The DC gain of the circuit of Fig. 1 is:

$$\omega_c = \frac{1}{R_3 C} \quad \text{Eq. 1}$$

And the DC gain of the circuit is:

$$DC \text{ Gain} = \frac{R_2}{R_1} \quad \text{Eq. 2}$$

In this assignment, we will choose values of  $R_1$ ,  $R_2$ , and  $C$  in the circuit of Fig. 1 to meet design requirements set on the cutoff frequency and DC gain.

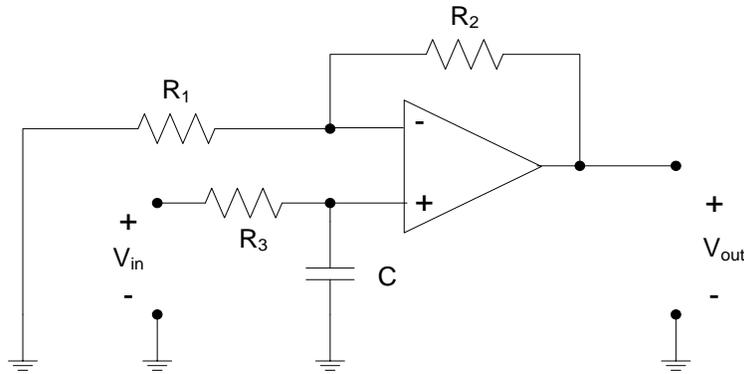


Figure 1. Non-inverting low-pass filter circuit.

### Pre-lab:

#### ANALYSIS

Design the circuit of Fig. 1 (e.g. choose values for  $R_1$ ,  $R_2$ ,  $R_3$ , and  $C$ ) to meet the following design requirements:

- DC gain =  $3 \pm 10\%$
- Cutoff frequency =  $1.5\text{kHz} \pm 10\%$

#### ANALYSIS

Sketch a straight-line approximation to the Bode Plot for your design.

### Lab Procedures:

#### DATA

- Construct the circuit you designed in the pre-lab.
  - Be sure to measure and record the actual values for  $R_1$ ,  $R_2$ , and  $R_3$ . If your DMM has a capacitance measurement, record the actual value for  $C$ ; if you do not have the ability to measure capacitance, assume that the nominal value of your capacitor is correct.
  - Measure the magnitude response of the circuit at frequencies approximately equal to  $\frac{\omega_c}{8}$ ,  $\frac{\omega_c}{4}$ ,  $\frac{\omega_c}{2}$ ,  $\omega_c$ ,  $2\omega_c$ ,  $4\omega_c$ , and  $8\omega_c$ , where  $\omega_c$  is the cutoff frequency of the circuit<sup>58</sup>.
  - Demonstrate operation of your circuit to the TA and have them initial the appropriate page(s) of your lab notebook and the lab worksheet.
- Use the Bode Transfer Function instrument to automatically generate a Bode plot for the circuit. To do this, follow the steps below:
  - Click on the **Network Analyzer** in the WaveForms main window. A window will open, which will allow you to automatically create a Bode plot for the system.
  - Bode plot parameters are set using the options shown in Fig. 2 below. The available options are:
    - **Start and Stop:** provide the range of frequencies which will be plotted. Generally, a range of frequencies from about an order of magnitude below the cutoff frequency to about an order of magnitude above the cutoff frequency is a good starting point.
    - **WaveGen Offset and Amplitude:** The offset and amplitude of the input voltage. Use the offset and amplitude that you used to measure the response in part (a).
    - **Samples:** The number of data points that will be used to generate the plot and the maximum gain that can be measured. More points will result in a smoother curve, but will take longer to generate.

#### DATA

#### DEMO

<sup>58</sup> Keep in mind that the units of  $\omega$  are radians/second, while the design requirement on the cutoff frequency is given in Hz.

- **Magnitude and Phase Settings:** The drop-down menus here allow you to choose the scale on both the magnitude and phase responses.
- **Channel 1 and Channel 2 Settings:** Choose the channels to be displayed.

**DATA**

Example parameters for this lab project are shown in Fig. 2. Record an image of the Network Analyzer window. Also save the data to a .csv file for later processing by selecting the **File** option on the menu bar and clicking on **Export**.

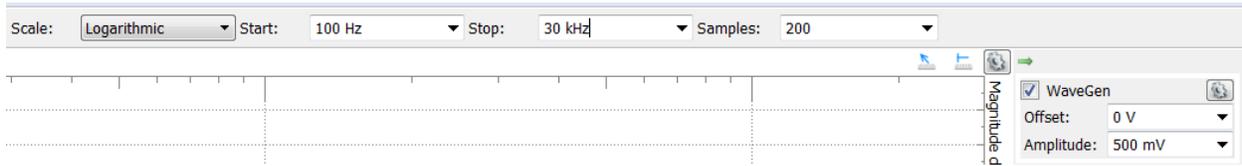


Figure 2. Example parameters for this lab project.

**Post-lab Exercises:**
**SIM**

Use your favorite mathematical software package (MATLAB, Octave, Excel,...) to plot the frequency response data (gain and phase vs. frequency) acquired by the Bode Transfer Function instrument in part (b) of the lab procedures. Overlay on this plot the data you acquired in part (a) of the lab procedures. Compare the responses from parts (a) and (b) and comment on any differences.

**ANALYSIS**

Compare the data you acquired in parts (a) and (b) of the lab procedures with your expectations from the pre-lab. Especially provide an assessment as to whether the original design requirements were met by your design.



6. Attach to this worksheet a plot showing the Network Analyzer data overlaid with the tabulated data from part 3 above. (10 pts)
  
7. In the space below, comment on the agreement between the agreement between the data acquired in part 3 and the data acquired with the Network Analyzer in part 6. Also comment on the agreement between your expectations based on your pre-lab analysis and both sets of data. (5 pts)

## Real Analog Chapter 11: Lab Projects

### 11.4.3: Non-inverting High-pass Filter

Bode plots are a convenient way to represent frequency responses. In particular, straight-line approximations to Bode plots are extremely easy to generate. In this lab assignment, we will generate a straight line approximation to the Bode plot for an active high-pass filter. We will then measure the frequency response of the circuit at several discrete frequencies and compare these to our straight-line approximation. Finally, we will use the Analog Discovery 2's Bode Transfer Function instrument to automatically generate a Bode plot for the circuit.

#### Before beginning this lab, you should be able to:

- Generate straight line approximations to the Bode plot for a first order circuit
- Determine the DC gain, high frequency gain, and cutoff frequency of a first order filter
- Measure the magnitude and phase responses of first order filter circuits (Labs 11.3.1, 11.3.2)

#### After completing this lab, you should be able to:

- Measure the magnitude and phase responses of an active electrical circuit at discrete frequencies
- Use the Bode Transfer Function instrument to automatically measure the Bode plot for a circuit
- Design an active filter to provide a desired DC gain and cutoff frequency

#### This lab exercise requires:

- Analog Discovery 2 module
- Digilent Analog Parts Kit
- Digital multimeter (optional)

#### Symbol Key:

- DEMO** Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable.
- ANALYSIS** Analysis; include principle results of analysis in laboratory report.
- SIM** Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.
- DATA** Record data in your lab notebook.

#### General Discussion:

In this project, we will sketch a straight line approximation to the Bode plot for the circuit. The frequency response of the circuit will then be measured and compared with our expectations.

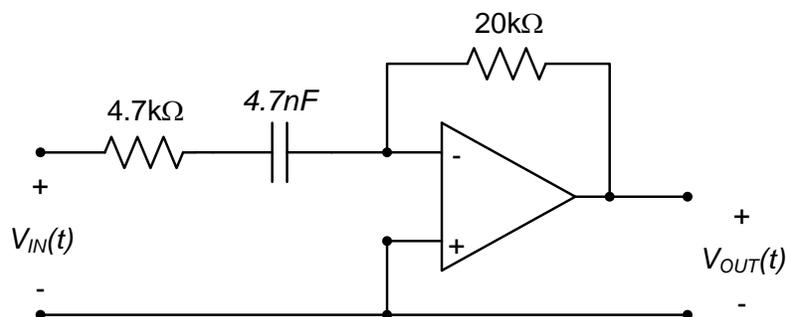


Figure 1. High-pass filter circuit.

## Pre-lab:

## ANALYSIS

Calculate the frequency response for the circuit shown in Fig. 1 and sketch the straight-line approximation to the circuit's Bode plot. Label the plot to show the high frequency gain and the cutoff frequency.

## Lab Procedures:

## DATA

a. Construct the circuit shown in Fig. 1.

i. Measure and record the actual values for  $R_1$ ,  $R_2$ , and  $R_3$ . If your DMM has a capacitance measurement, record the actual value for  $C$ ; if you do not have the ability to measure capacitance, assume that the nominal value of your capacitor is correct.

## DATA

ii. Measure the magnitude response of the circuit at frequencies approximately equal to  $\frac{\omega_c}{8}$ ,  $\frac{\omega_c}{4}$ ,  $\frac{\omega_c}{2}$ ,  $\omega_c$ ,  $2\omega_c$ ,  $4\omega_c$ , and  $8\omega_c$ , where  $\omega_c$  is the cutoff frequency of the circuit<sup>59</sup>.

## DEMO

iii. Demonstrate operation of your circuit to the TA and have them initial the appropriate page(s) of your lab notebook and the lab worksheet.

b. Use the Bode Transfer Function instrument to automatically generate a Bode plot for the circuit. To do this, follow the steps below:

i. Click on **Network Analyzer** in the WaveForms main window. A window will open, which will allow you to automatically create a Bode plot for the system.

ii. Bode plot parameters are set using the options shown in Fig. 2 below. The available options are:

- **Start and Stop:** provide the range of frequencies which will be plotted. Generally, a range of frequencies from about an order of magnitude below the cutoff frequency to about an order of magnitude above the cutoff frequency is a good starting point.
- **WaveGen Offset and Amplitude:** The offset and amplitude of the input voltage. Use the offset and amplitude that you used to measure the response in part (a).
- **Samples:** The number of data points that will be used to generate the plot and the maximum gain that can be measured. More points will result in a smoother curve, but will take longer to generate.
- **Magnitude and Phase Settings:** The drop-down menus here allow you to choose the scale on both the magnitude and phase responses.
- **Channel 1 and Channel 2 Settings:** Choose the channels to be displayed.

## DATA

Example parameters for this lab project are shown in Fig. 2. Record an image of the Network Analyzer window. Also save the data to a .csv file for later processing by selecting the **File** option on the menu bar and clicking on **Export**.

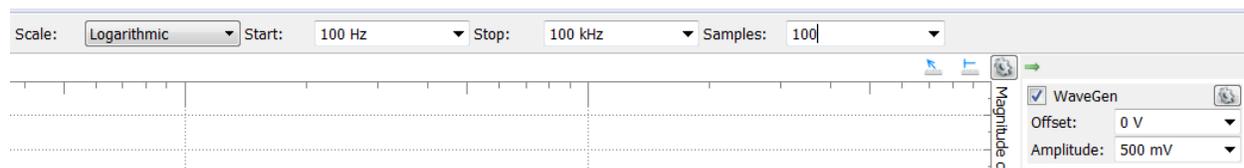


Figure 2. Example parameters for this lab project.

<sup>59</sup> Keep in mind that the units of  $\omega$  are radians/second, while the design requirement on the cutoff frequency is given in Hz.

**Post-lab Exercises:**

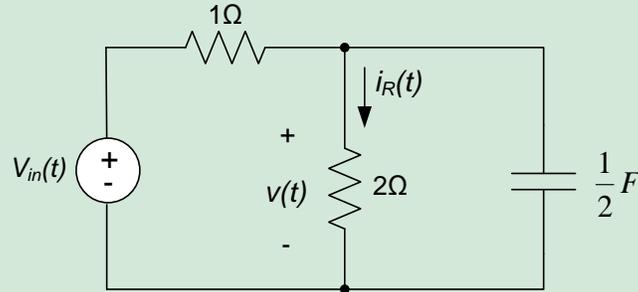
- SIM** Use your favorite mathematical software package (MATLAB, Octave, Excel,...) to plot the frequency response data (gain and phase vs. frequency) acquired by the Bode Transfer Function instrument in part (b) of the lab procedures. Overlay on this plot the data you acquired in part (a) of the lab procedures. Compare the responses from parts (a) and (b) and comment on any differences.
- ANALYSIS** Compare the data you acquired in parts (a) and (b) of the lab procedures with your expectations from the pre-lab. Especially provide an assessment as to whether the original design requirements were met by your design.



7. Attach to this worksheet a plot showing the Network Analyzer data overlaid with the tabulated data from part 4 above. (10 pts)
  
8. In the space below, comment on the agreement between the agreement between the data acquired in part 4 and the data acquired with the Network Analyzer in part 7. Also comment on the agreement between your expectations based on your pre-lab analysis and both sets of data. (5 pts)

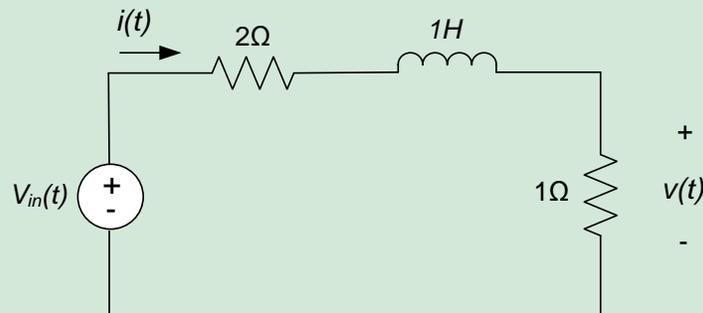
## Real Analog Chapter 11: Homework

- 11.1 For the circuit below, the input voltage  $V_{in}(t) = 3\cos(2t-20^\circ) - 2\cos(3t+30^\circ) + \cos(4t)$ . Determine:
- The steady-state response of the voltage  $v(t)$ .
  - The steady-state response of the current  $i_R(t)$ . (Hint: take advantage of your results from part a)

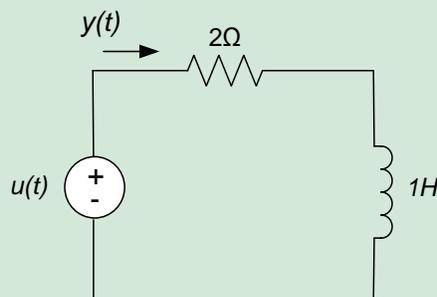


- 11.2 For the circuit of problem 1, sketch the magnitude and phase response of the voltage  $v(t)$ . Label your sketch at  $\omega = 0$ ,  $\omega \rightarrow \infty$  and the circuit's cutoff frequency. Does the circuit behave as a low-pass or high-pass filter? Why?

- 11.3 For the circuit below, the input is the voltage  $V_{in}(t)$ . If the output is the current  $i(t)$ , sketch a straight-line approximation to the Bode plot for the circuit. Label your sketch to include the DC gain and the cutoff frequency of the circuit.

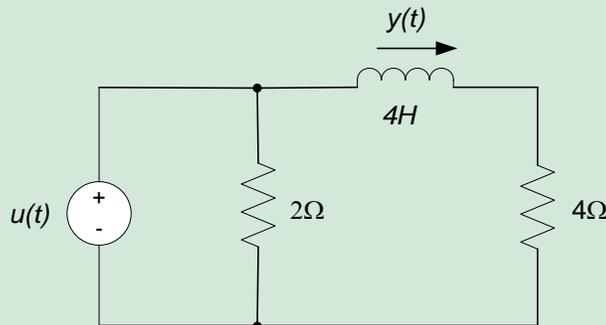


- 11.4 For the circuit below, the input is the voltage  $u(t)$  and the output is the current  $y(t)$ . Sketch a straight-line approximation to the Bode plot. Label your sketch to include the DC gain and the cutoff frequency of the circuit.



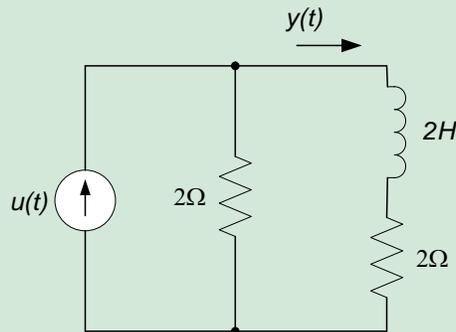
11.5 For the circuit of problem 11.4, sketch a straight-line approximation to the Bode plot for the circuit if the output is the voltage  $v(t)$ . Label your sketch to include the DC gain and the cutoff frequency of the circuit.

- 11.6 For the circuit shown, the input is the voltage source  $u(t)$  and the output is the current through the inductor,  $y(t)$ .
- Determine the frequency response of the circuit.
  - Sketch the magnitude response of the circuit. Label your sketch to show the DC gain and the cutoff frequency of the circuit.
  - Does the circuit behave as a high pass or low pass filter? Explain.



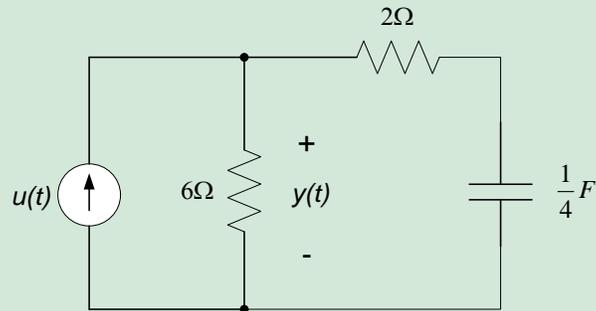
- 11.7 For the circuit of problem 1, the input is the voltage source  $u(t)$ . If the output is the voltage across the inductor,
- Determine the frequency response of the circuit.
  - Sketch the magnitude response of the circuit. Label your sketch to show the DC gain and the cutoff frequency of the circuit.
  - Does the circuit behave as a high pass or low pass filter? Explain.

- 11.8 For the circuit shown, the input is the current source  $u(t)$  and the output is the current through the inductor,  $y(t)$ .
- Determine the frequency response of the circuit.
  - Sketch the magnitude response and phase response of the circuit. Label your sketch to show the DC gain and the cutoff frequency of the circuit.
  - Does the circuit behave as a high pass or low pass filter? Explain.



11.9 For the circuit shown, the input is the current source  $u(t)$  and the output is the voltage across the  $6\Omega$  resistor,  $y(t)$ .

- Determine the frequency response of the circuit.
- Does the frequency response you calculated above agree with your expectations as to the circuit's response at high ( $\omega \rightarrow \infty$ ) and low ( $\omega \rightarrow 0$ ) frequencies? Why or why not?



## Chapter 12: Steady-state Sinusoidal Power

### 12 Introduction and Chapter Objectives

In this chapter we will address the issue of power transmission via sinusoidal (or AC) signals. This topic is extremely important, since the vast majority of power transmission in the world is performed using AC voltages and currents.

For the most part, the topic of AC power transmission focuses on the average power delivered to a load over time. In general, it is not productive to focus on the power transmission at a particular time since, if the load contains energy storage elements (such as capacitors and inductors), it is possible that at times the load will absorb power and at other times the load will release power. This characteristic leads to the concepts of *average power* and *reactive power* – average power is typically the power that is converted by the load to useful work, while reactive power is the power that is simply exchanged by energy storage elements. Power companies cannot really charge customers for power which is not absorbed by the load, so one primary goal in AC power transmission is to reduce the reactive power that is sent to the load.

In this chapter we introduce the basic topics relative to calculation of AC power. In section 12.1, we introduce the basic concepts associated with AC power, including the notion that a load containing energy storage elements may alternately absorb and release power. This discussion will lead to the concepts of average power and reactive power, which are discussed in section 12.2. Power calculations are often presented in terms of RMS values; these are introduced in section 12.3. The relative roles of average and reactive power are often characterized by the apparent power and the power factor, which are presented in section 12.4. In section 12.5, we will use complex numbers to simultaneously quantify the average power, the reactive power, the apparent power, and the power factor. Finally, in section 12.6, we examine approaches to reduce the reactive power which is exchanged between the power company and the user. This technique is called power factor correction.

#### After completing this chapter, you should be able to:

- Define instantaneous power, average power, and reactive power
- Define real power, reactive power, and complex power
- Define RMS signal values and calculate the RMS value of a given sinusoidal signal
- State, from memory, the definition of power factor and calculate the power factor from a given combination of voltage and current sinusoids
- Draw a power triangle
- Correct the power factor of an inductive load to a desired value

## 12.1 Instantaneous Power

We will begin our study of steady-state sinusoidal power by examining the power delivered by a sinusoidal signal as a function of time. We will see that, since all the signals involved are sinusoidal, the delivered power varies sinusoidally with time. This time-varying power is called instantaneous power, since it describes the power delivered to the load at every instant in time. The instantaneous power will not, in general, be directly useful to us in later sections but it does provide the basis for understanding the concepts presented throughout this chapter.

In chapter 1, we saw that power is the product of voltage and current, so that power as a function of time is:

$$p(t) = v(t) \cdot i(t) \quad \text{Eq. 12.1}$$

Power as a function of time is often called *instantaneous power*, since it provides the power at any instant in time. So far, this is the only type of power with which we have been concerned. If our voltages and currents are sinusoidal, as is the case for AC power, we can write  $v(t)$  and  $i(t)$  as:

$$v(t) = V_m \cos(\omega t + \theta_v) \quad \text{Eq. 12.2}$$

And:

$$i(t) = I_m \cos(\omega t + \theta_i) \quad \text{Eq. 12.3}$$

Where, of course,  $V_m$  and  $\theta_v$  are the amplitude and phase angle of the voltage signal while  $I_m$  and  $\theta_i$  are the amplitude and phase angle of the current signal. It should be noted at this point that the voltage and current signals of equations (12.2) and (12.3) are not independent of one another. Figure 12.1 shows the overall system being analyzed – the voltage  $v(t)$  and the current  $i(t)$  are the voltage and current applied to some load. If the load has some impedance,  $Z_L$ , the voltage and current are related through this impedance. Thus, if we represent  $v(t)$  and  $i(t)$  in phasor form as  $\underline{V} = \underline{V} e^{j\omega t}$  and  $\underline{I} = \underline{I} e^{j\omega t}$ , then  $\underline{V} = Z_L \cdot \underline{I}$ .

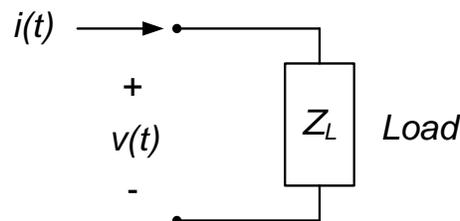


Figure 12.1. Voltage and current applied to a load.

Substituting equations (12.2) and (12.3) into equation (12.1) results in:

$$p(t) = V_m I_m \cos(\omega t + \theta_v) \cos(\omega t + \theta_i) \quad \text{Eq. 12.4}$$

Equation (12.4) can be re-written, using some algebra and trigonometric identities, as:

$$p(t) = \frac{V_m I_m}{2} \{ \cos(\theta_v - \theta_i) + \cos(2\omega t + \theta_v + \theta_i) \} \quad \text{Eq. 12.5}$$

Since  $V_m$ ,  $\theta_v$ ,  $I_m$ , and  $\theta_i$  are all constants, we can see that equation (12.5) is the sum of two terms: a constant value,  $\frac{V_m I_m}{2} \cos(\theta_v - \theta_i)$ , and a sinusoidal component,  $\frac{V_m I_m}{2} \cos(2\omega t + \theta_v + \theta_i)$ . The time-domain relationship of equation (12.5) is plotted in Fig. 12.2. The signal's average value is  $\frac{V_m I_m}{2} \cos(\theta_v - \theta_i)$  and has a sinusoidal component with an amplitude of  $\frac{V_m I_m}{2}$ .

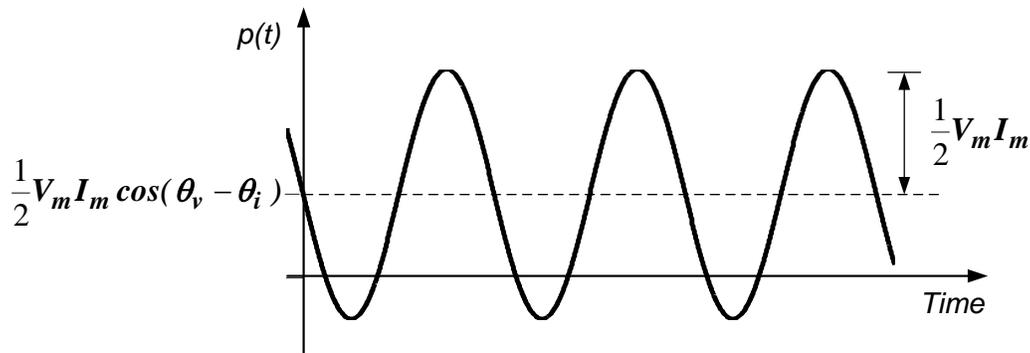


Figure 12.2. Plot of instantaneous power vs. time.

### Section Summary:

- The power delivered to a load by a sinusoidal (or AC) signal has two components: an average value and a sinusoidal component.
- Both the average value and the sinusoidal component are dependent upon the amplitude and phase angles of the voltage and current delivered to the load. These values are, in turn, set by the impedance of the load.
- The average power is dissipated or absorbed by the load. This power is electrical power which is converted to heat or useful work.
- The sinusoidal component of the power is due to energy storage elements in the load; this power is exchanged (in some sense) between the load and the system supplying power. It is purely electrical energy which the load is not using to perform useful work.

### 12.1 Exercises

1. The current and voltage delivered to a load are  $i(t) = 2\cos(100t)$  and  $v(t) = 120\cos(100t+65^\circ)$ , respectively. Calculate the average power delivered to the load and the amplitude of the sinusoidal component of the power.

## 12.2 Average and Reactive Power

Examination of equation (12.5) and Fig. 12.2 indicates that the instantaneous power can be either positive or negative, so the load is alternately absorbing or releasing power. The overall amount of power absorbed vs. power released is dependent primarily upon the  $\cos(\theta_v - \theta_i)$  term. If the voltage and current are in phase,  $\theta_v - \theta_i$ ,  $\cos(\theta_v - \theta_i) = 1$  and the instantaneous power is never negative. The voltage and current have the same phase if the load is purely resistive – a resistor always absorbs power. If the voltage and current are  $90^\circ$  out of phase, as is the case for a purely capacitive or purely inductive load,  $\cos(\theta_v - \theta_i) = 0$ . In this case, the instantaneous power curve is a pure sinusoid with no DC offset, so on average no power is delivered to the load. This is consistent with our models of capacitors and inductors as energy storage elements, which do not dissipate any energy.

The concepts presented in the paragraph above can be mathematically presented by rearranging equation (12.5) yet again. Application of additional trigonometric identities and performing more algebra results in:

$$p(t) = \frac{V_m I_m}{2} \cos(\theta_v - \theta_i) [1 + \cos(2\omega t)] + \frac{V_m I_m}{2} \sin(\theta_v - \theta_i) [\sin(2\omega t)] \quad \text{Eq. 12.6}$$

The two terms in equation (12.6) are plotted separately in Fig. 12.3. The first term,  $\frac{V_m I_m}{2} \cos(\theta_v - \theta_i) [1 + \cos(2\omega t)]$ , has an average value of  $\frac{V_m I_m}{2} \cos(\theta_v - \theta_i)$ , while the second term,  $\frac{V_m I_m}{2} \sin(\theta_v - \theta_i) [\sin(2\omega t)]$ , has an average value of zero. Thus, the average power delivered to a load is:

$$P = \frac{V_m I_m}{2} \cos(\theta_v - \theta_i) \quad \text{Eq. 12.7}$$

The amplitude of the term  $\frac{V_m I_m}{2} \sin(\theta_v - \theta_i) [\sin(2\omega t)]$ , which provides no average power to the load, is termed the *reactive power*,  $Q$ :

$$Q = \frac{V_m I_m}{2} \sin(\theta_v - \theta_i) \quad \text{Eq. 12.8}$$

The reactive power is a measure of the amount of power which is delivered to the load, but is not absorbed by the load – the load returns this power to the source!

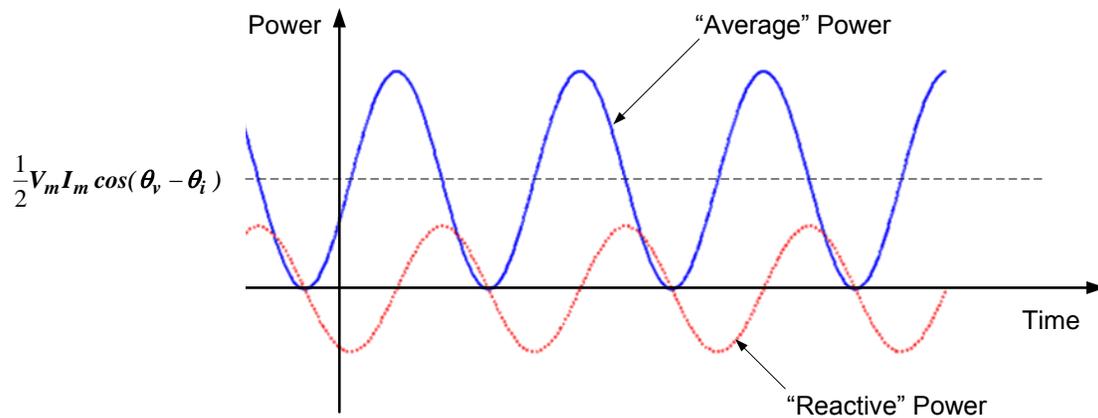


Figure 12.3. Components of instantaneous power vs. time.

### Section Summary:

- The *average* power delivered to a load is:

$$P = \frac{V_m I_m}{2} \cos(\theta_v - \theta_i)$$

where  $V_m$  and  $\theta_v$  are the amplitude and phase angle of the voltage while  $I_m$  and  $\theta_i$  are the amplitude and phase angle of the current. The average power is often also called the *real* power. The average power is also the power dissipated by any resistive elements in the load. Units of average power are Watts (abbreviated W).

- The *reactive* power delivered to a load is:

$$Q = \frac{V_m I_m}{2} \sin(\theta_v - \theta_i)$$

The reactive power is not actually absorbed by the load; it is stored by the energy storage elements in the load and then returned to the source. The units of reactive power are taken to be Volt-Amperes Reactive (abbreviated VAR). Note that technically watts are the same as volt-amps, but we have changed terminology to avoid any confusion between real and reactive power.

## 12.2 Exercises:

1. The current and voltage delivered to a load are  $i(t) = 2\cos(100t)$  and  $v(t) = 120\cos(100t+65^\circ)$ , respectively. Calculate the average power delivered to the load and the reactive power delivered to the load.

## 12.3 RMS Values

In sections 12.1 and 12.2, we introduced some basic quantities relative to delivery of power using sinusoidal signals. We saw that power dissipated by a load (essentially, any energy which is converted to non-electrical energy such as heat or work) is the *average power*. *Reactive power* results from energy which is stored by capacitors and inductors in the load and is then returned to the source without dissipation.

In this chapter, we continue our study of AC power analysis. We will introduce the concept of the root-mean-square (RMS) value of a signal as a way to represent the power of a time-varying signal. We will also introduce *complex power* as a way to conveniently represent both average and reactive power as a single complex number. We also introduce power factor as a way to represent the efficiency of the transfer of power to a load.

It is often desirable to compare different types of time-varying signals (for example, square waves vs. triangular waves vs. sinusoidal waves) using a very simple metric. Different types of signals are often compared by their RMS (root-mean-squared) values. The general idea behind the RMS value of a time-varying signal is that we wish to determine a constant value, which delivers the same average power to a resistive load.

The average value,  $P$ , of an instantaneous power  $p(t)$  is defined to be:

$$P = \frac{1}{T} \int_{t_0}^{t_0+T} p(t) dt \quad \text{Eq. 12.9}$$

The power delivered to a resistive load by a constant voltage or current source is, from chapter 1.1,

$$P = R \cdot I_{eff}^2 = \frac{v_{eff}^2}{R} \quad \text{Eq. 12.10}$$

$I_{eff}$  and  $V_{eff}$  are the effective (or constant) current and voltage, respectively, applied to the resistive load,  $R$ . It is our goal to equate equations (12.9) and (12.10) to determine the effective voltage or current values, which deliver the same average power to a resistive load as some time-varying waveform.

Assuming that a current is applied to a resistive load, the instantaneous power is  $p(t) = R \cdot i^2(t)$ . Substituting this into equation (12.9) and equating to equation (12.10) results in:

$$R \cdot I_{eff}^2 = \frac{1}{T} \int_{t_0}^{t_0+T} R \cdot i^2(t) dt \quad \text{Eq. 12.11}$$

Solving this for  $I_{eff}$  results in:

$$I_{eff} = I_{RMS} = \sqrt{\frac{1}{T} \int_{t_0}^{t_0+T} i^2(t) dt} \quad \text{Eq. 12.12}$$

And the effective current is the square root of the mean of the square of the time-varying current. This is also called the RMS (or root-mean-square) value, for rather obvious reasons.

A similar process can be applied to the voltage across a resistive load, so that  $p(t) = \frac{v^2(t)}{R}$ . Equating this expression to equation (12.9) results in:

$$V_{eff} = V_{RMS} = \sqrt{\frac{1}{T} \int_{t_0}^{t_0+T} v^2(t) dt} \quad \text{Eq. 12.13}$$

So that the definition of an RMS voltage is equivalent to the definition of an RMS current.

Equations (12.12) and (12.13) are applicable to any time-varying waveform; the waveforms of interest to us are sinusoids, with zero average values (per equations (12.10) and (12.11)). In this particular case, the RMS values can be calculated to be:

$$V_{eff} = V_{RMS} = \frac{V_m}{\sqrt{2}} \quad \text{Eq. 12.14}$$

And

$$I_{eff} = I_{RMS} = \frac{I_m}{\sqrt{2}} \quad \text{Eq. 12.15}$$

Where  $V_m$  and  $I_m$  are the peak (or maximum) values of the voltage and current waveforms, per equations (12.10) and (12.11). Please note that equations (12.14) and (12.15) are applicable only to sinusoidal signals with zero average values.

The average and reactive powers given by equations (12.16) and (12.17) can be written in terms of the RMS values of voltage and current as follows:

$$P = V_{RMS} I_{RMS} \cos(\theta_v - \theta_i) \quad \text{Eq. 12.16}$$

And

$$Q = V_{RMS} I_{RMS} \sin(\theta_v - \theta_i) \quad \text{Eq. 12.17}$$

### Section Summary:

- The RMS value of a sinusoidal signal  $f(t) = F_m \cos(\omega t + \theta)$  is given by:

$$f_{RMS} = \frac{F_m}{\sqrt{2}}$$

- The above formula cannot be used for any signal other than a pure sinusoid with no offset.

### 12.3 Exercises

- The current and voltage delivered to a load are  $i(t) = 2\cos(100t)$  And  $v(t) = 120\cos(100t + 65^\circ)$ , e=respectively. What are the RMS values of voltage and current?

### 12.4 Apparent Power and Power Factor

In the previous subsections, we have seen that average power can be represented in terms of either the magnitudes of the voltage and current or the RMS values of the voltage and current and a multiplicative factor consisting of the cosine of the difference between the voltage phase and the current phase:

$$P = \frac{V_m I_m}{2} \cos(\theta_v - \theta_i) = V_{RMS} I_{RMS} \cos(\theta_v - \theta_i)$$

It is sometimes convenient to think of the average power as being the product of *apparent power* and a *power factor* (abbreviated *pf*). These are defined below:

- The apparent power is defined as either  $\frac{V_m I_m}{2}$  or  $V_{RMS} I_{RMS}$ . (the two terms are, of course, equivalent.) Units of apparent power are designated as volt-amperes (abbreviated VA) to differentiate apparent power from either average power or reactive power.
- The power factor is defined as  $\cos(\theta_v - \theta_i)$ . Since cosine is an even function (the sign of the function is independent of the sign of the argument), the power factor does not indicate whether the voltage is leading or lagging the current. Thus, power factor is said to be either *leading* (if current leads voltage) or *lagging* (if current lags voltage).

It should be emphasized again at this point that the voltage and current are not independent quantities; they are related by the load impedance. For the system of Figure 12.1, for example, the voltage and current phasors are:

$$\underline{V} = Z_L \cdot \underline{I} \quad \text{Eq. 12.18}$$

where  $\underline{V}$  is the voltage phasor across the load,  $\underline{I}$  is the current phasor through the load, and  $Z_L$  is the load impedance. Thus, the difference between the voltage and current phase angles is simply the phase angle of the load impedance:  $\theta_v - \theta_i = \angle \theta_{Z_L}$ . Therefore, the load impedance sets the power factor. If the load is purely resistive,  $\theta_v = \theta_i$ , the power factor is one, and the average power is the same as the apparent power. If the load is purely imaginary (as with purely inductive or purely capacitive loads) the power factor is zero and there is no average or real power absorbed by the load.

## Section Summary

- *Apparent power* and *power factor* provide an alternate method for characterizing the average power delivered to the load. If the average power is:

$$P = \frac{V_m I_m}{2} \cos(\theta_v - \theta_i),$$

The apparent power is  $\frac{V_m I_m}{2}$  and the power factor is  $\cos(\theta_v - \theta_i)$ .

## 12.4 Exercises

1. The current and voltage delivered to a load are  $i(t) = 2\cos(100t)$  and  $v(t) = 120\cos(100t + 65^\circ)$ , respectively. What are the apparent power and power factor of the power delivered to the load?
2. A load consumes 100kW with a power factor pf = 0.85 (lagging). If the load current is 256A (RMS), find the load voltage.
3. An industrial plant has a load which consumes 20kW of power from a 220V<sub>RMS</sub> line. If the power factor is 0.9 (lagging), what is the difference in angle between the load voltage and the load current?

## 12.5 Complex Power

Apparent power, average power, reactive power, and power factor can all be represented simultaneously in a single parameter called complex power. If we define complex power as:

$$S = \frac{\underline{V} \underline{I}^*}{2} \quad \text{Eq. 12.19}$$

Where  $\underline{V}$  is the phasor representing the voltage,  $\underline{I}$  is the phasor representing the current, and the superscript \* denotes complex conjugation (simply changing the sign on the imaginary part of the phasor).

If we substitute the magnitude and phase angle representations for the phasors in equation (12.19), we obtain (since complex conjugation simply changes the sign of the phase angle of a complex number):

$$S = \frac{1}{2}(V_m \angle \theta_v)(I_m \angle -\theta_i) = \frac{V_m I_m}{2} \angle (\theta_v - \theta_i) \quad \text{Eq. 12.20}$$

So that the complex power  $S$  is a complex number with magnitude  $\frac{V_m I_m}{2}$  (or, equivalently,  $V_{RMS} I_{RMS}$ ) and phase angle  $\theta_v - \theta_i$ . It is easy to see that the magnitude of the complex power is simply the apparent power. If we represent  $S$  in rectangular coordinates, we obtain:

$$S = P + jQ \quad \text{Eq. 12.21}$$

Where  $P$  is the average power,

$$P = \frac{V_m I_m}{2} \cos(\theta_v - \theta_i) = V_{RMS} I_{RMS} \cos(\theta_v - \theta_i)$$

as before, and  $Q$  is the reactive power,

$$Q = \frac{V_m I_m}{2} \sin(\theta_v - \theta_i) = V_{RMS} I_{RMS} \sin(\theta_v - \theta_i)$$

also as before.

The complex power, real power, reactive power, and apparent power can be represented graphically in the complex plane as a *power triangle*, as shown in Fig. 12.4 below.

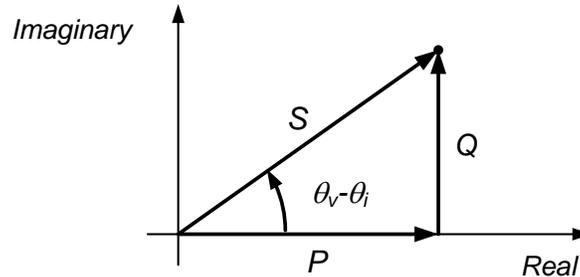


Figure 12.4. Power triangle.

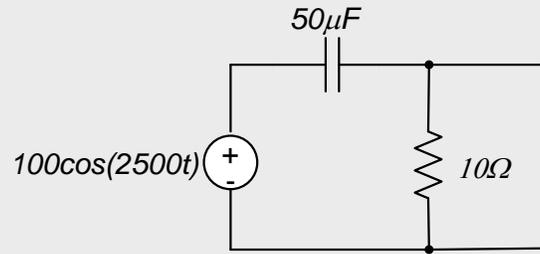
One important thing to note about Fig. 12.4 is that this figure differs from a phasor diagram – the components shown on the power triangle are not phasors, since they do not provide magnitude and phase information about sinusoidal signals. The vectors shown in Fig. 12.4 are simply complex numbers.

We conclude this section with an example.

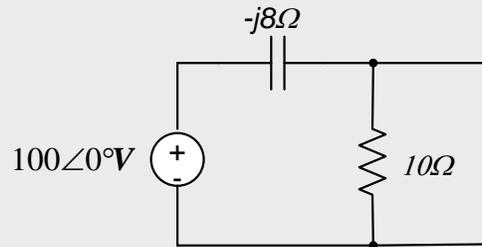
### Example 12.1

For the circuit below,

- find the average power delivered by the source
- find the power absorbed by the resistor
- find the apparent, real, and reactive powers delivered by the source
- sketch a power triangle for the source



The frequency domain circuit is shown below.



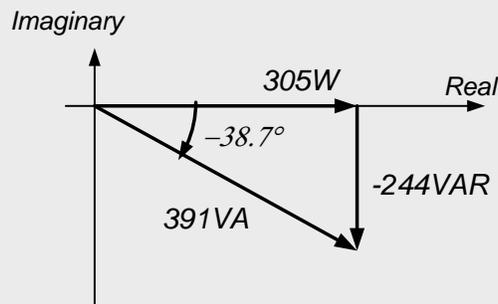
The current delivered by the source is therefore:

$$\underline{I} = \frac{100\angle 0^\circ}{(10 - j8)\Omega} = 7.8 \angle 38.66^\circ A$$

- a. The average power delivered by the source is therefore:

$$P = \frac{(100V)(7.8A)}{2} \cos(0^\circ - 38.66^\circ) = 305W$$

- b. The power absorbed by the resistor is the same as the average power delivered by the source, so  $P_{10\Omega} = 305W$ , which is the same as  $R \cdot I_{RMS}^2$ .
- c. The apparent power is  $\frac{(100V)(7.8A)}{2} = 391 VA$ , the reactive power is  $Q = \frac{(100V)(7.8A)}{2} \sin(0^\circ - 38.66^\circ) = -244 VAR$ , and the real power is simply the average power,  $P = 305W$ .
- d. The power triangle is shown below:



### Section Summary:

- It is often convenient to express power as a complex number,  $S$ . This quantity expresses, simultaneously, the average power, the reactive power, the apparent power, and the power factor. Complex power can be determined from the voltage and current phasors as:

$$S = \frac{VI^*}{2},$$

Where \* denotes complex conjugation.

- As with any other complex number, complex power can be expressed in either rectangular or polar coordinates.
  - In rectangular coordinates, the complex power readily provides the real power ( $P$ ) and reactive power ( $Q$ ):

$$S = P + jQ$$

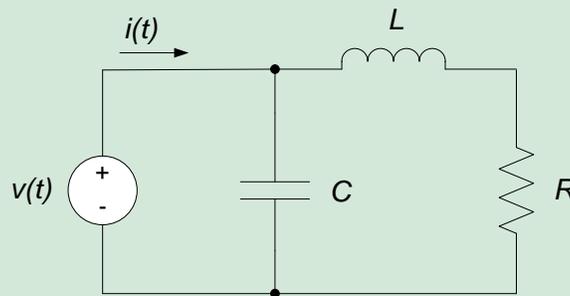
- In polar coordinates, the complex power readily provides the apparent power and power factor:

$$S = \frac{V_m I_m}{2} \angle(\theta_v - \theta_i)$$

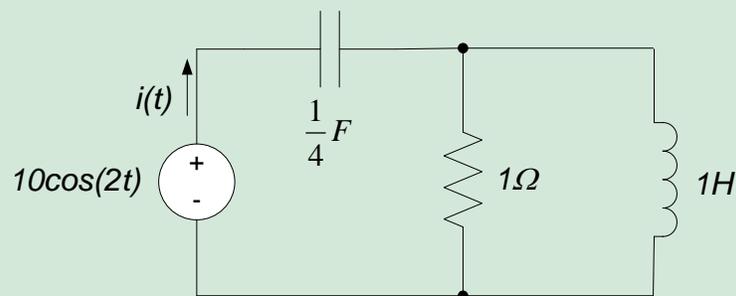
Where the apparent power is  $\frac{V_m I_m}{2}$  and the power factor is given by  $\cos(\theta_v - \theta_i)$ .

## 12.5 Exercises

1. The current and voltage delivered to a load are  $i(t) = 2\cos(100t - 30^\circ)$  and  $v(t) = 120\cos(100t + 65^\circ)$ , respectively. What is the complex power delivered to the load?
2. For the circuit shown,  $i(t) = 2\cos(100t)$  and  $v(t) = 120\cos(100t + 65^\circ)$ . Find:
  - a. The complex power delivered by the source
  - b. The average power delivered by the source
  - c. The power dissipated by the resistor



3. For the circuit shown, find
  - a.  $i(t)$
  - b. The complex power delivered by the source
  - c. The average power delivered by the source



## 12.6 Power Factor Correction

In previous sections, we saw that power can be considered to be real (average power) or imaginary (reactive power). Reactive power is power which is provided to energy storage elements such as inductors or capacitors; this power is returned to the power source, since inductors and capacitors have no energy dissipation mechanism. The power factor provides a metric for assessing the amount of real power relative to the reactive power delivered to a load. The higher the power factor, the larger the amount of real power relative to reactive power – ideally, a power factor of one means that no reactive power is provided to the load.

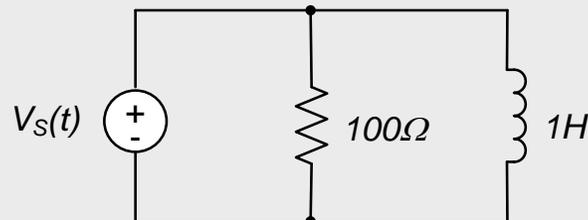
Power companies in general cannot charge customers for power which is returned to the power company, so delivering reactive power to a customer is not productive from a power company's standpoint. In fact, since transmission losses will typically result in real power losses due to transmission of reactive power, the power company actually loses power when transmitting reactive power, for which no one pays! Unsurprisingly, this is not popular with power companies.

Large power users, such as factories, may have requirements placed upon them by the power company to provide a minimum power factor for their loads. If the factory cannot meet the required power factor, the power company can refuse to supply power or charge an increased rate for the power they do provide. This chapter illustrates how to re-design an inductive load to increase its power factor.

Most large power users' loads are inductive in nature. Therefore, in this section, we will only consider power factor correction for inductive loads. We illustrate the overall process in the context of an example.

### Example 12.2

Determine the power factor for the circuit below if  $v_s(t) = 100\cos(377t)$ . Re-design the load so that the power factor is one.



The load in this case consists of a  $100\Omega$  resistor in parallel with a  $1H$  inductor. To determine the power factor, we need to determine the current delivered by the source; we do this by determining the equivalent impedance of the load,  $Z_L$ :

$$Z_L = \frac{(100\Omega)(j377\Omega)}{100\Omega + j377\Omega} = 96.67\angle 14.86^\circ$$

Therefore, the current phasor delivered by the source is:

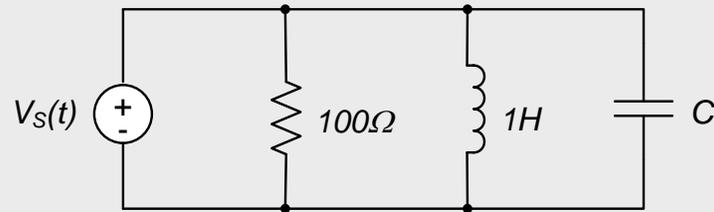
$$\underline{I_s} = \frac{V_s}{Z_L} = \frac{100\angle 0^\circ}{96.67\angle 14.86^\circ} = 1.03\angle -14.86^\circ$$

The power factor is:

$$pf = \cos(\theta_v - \theta_i) = \cos(0^\circ - (-14.86^\circ)) = 0.967 \text{ (lagging)}$$

Note that the power factor is lagging, since current lags voltage, this is because the load is inductive in nature.

We can change the power factor of the load by adding a capacitor in parallel with the inductive load. The impedance of the capacitor can be used, in essence, to cancel out the inductive impedance of the load. The re-designed circuit is as shown below. The capacitance,  $C$ , must be chosen to provide the desired power factor.



In order to determine the necessary size of the capacitor, we first determine the reactive power of the uncompensated system. Without the capacitor, the reactive power is:

$$Q = \frac{V_m I_m}{2} \sin(\theta_v - \theta_i) = \frac{(100V)(1.03A)}{2} \sin(14.86^\circ) = 13.26 \text{ VAR}$$

Our goal is to use the capacitor to change this reactive power so that the resulting power factor is as desired, without changing the real power delivered to the load. The capacitor will induce a negative reactive power; in order to achieve a power factor of exactly one, the reactive power introduced by the capacitor must exactly cancel the reactive power of the original load. Thus, the capacitor's reactive power must be:

$$Q_C = -13.26 \text{ VAR}$$

For the parallel combination of the resistor, capacitor, and inductor, we can claim:

$$|Q_C| = \frac{V_{RMS}^2}{|Z_C|} = \frac{V_{RMS}^2}{\frac{1}{\omega C}} = \omega C V_{RMS}^2$$

Where  $|Q_C|$  is the magnitude of the desired reactive power, and  $|Z_C|$  is the magnitude of the capacitor's impedance. Solving the above expression for the desired capacitance provides:

$$C = \frac{|Q_C|}{\omega V_{RMS}^2} = \frac{13.26 \text{ VAR}}{(377 \text{ rad/sec}) \left(\frac{100}{\sqrt{2}}\right)^2} = 7 \mu F$$

And a  $7\mu F$  capacitor placed in parallel with the load will give the desired  $pf=1$ .

### Section Summary:

- A small power factor means that a large portion of the power delivered by the power company to a user is in the form of reactive power. The user does not pay for reactive power, since it is essentially returned to the power company. The power company, however, incurs costs in exchanging reactive power with the user, since transmission of the reactive power results in power dissipation. Power companies, therefore, may require that a user meet a minimum power factor requirement.
- The power factor of an inductive load can be increased. The goal is to increase the power factor without changing the average power delivered to the load.

## Real Analog Chapter 12: Lab Projects

### 12.4.1: Apparent Power and Power Factor

This lab assignment emphasizes the use of apparent power and power factor to quantify the AC power delivered to a load and the power dissipated by the process of transmitting this power.

Initially, our load will be inductive in nature. The power delivered to the load will be transmitted via a resistive network. The apparent power, average power, and power factor associated with the circuit will be measured and related to expectations based on analysis of the circuit. It will be seen that the difference between the apparent power and the average power, as quantified by the power factor, influences the amount of power which is dissipated during power delivery relative to the power provided to the load. We will see that a low power factor corresponds to a situation in which a relatively large amount of power is dissipated in the process of delivering power to a load.

After examining the inductive load, we will modify the load impedance by adding a capacitive component to the load. This will increase the power factor of the load, which will result in more efficient power delivery to the load.

#### Before beginning this lab, you should be able to:

- Perform frequency-domain analysis of electrical circuits
- Use peak and RMS values to calculate average power
- Calculate power factor from given voltage and current waveforms
- Use peak and RMS values to calculate apparent power

#### After completing this lab, you should be able to:

- Measure the power factor and apparent power delivered to a load by an AC source
- State the effect of power factor on the power dissipation due to transmitting power to a load

#### This lab exercise requires:

- Analog Discovery 2 module
- Digilent Analog Parts Kit
- Digital multimeter (optional)

#### Symbol Key:

- |   |   |
|---|---|
|  | Demonstrate circuit operation to teaching assistant; teaching assistant should initial lab notebook and grade sheet, indicating that circuit operation is acceptable. |
|  | Analysis; include principle results of analysis in laboratory report.   |
|  | Numerical simulation (using PSPICE or MATLAB as indicated); include results of MATLAB numerical analysis and/or simulation in laboratory report.                      |
|  | Record data in your lab notebook.   |

#### General Discussion:

A typical power delivery system is shown in Fig. 1. The source provides power to some load via a transmission line. The transmission line typically has some resistance,  $R_T$ . The process of delivering power to the load results in undesirable power losses due to the transmission line resistance. One primary goal of power delivery is to provide the necessary average power to a load with a minimum of power lost during the process of the power delivery. Unfortunately, the reactive power (which is not useful to the load) causes power losses during transmission. Thus,

it is desirable to minimize the reactive power delivered to the load in order to reduce the average power dissipated during transmission, relative to the average power delivered to the load. The apparent power and the power factor make it relatively easy to quantify these effects.

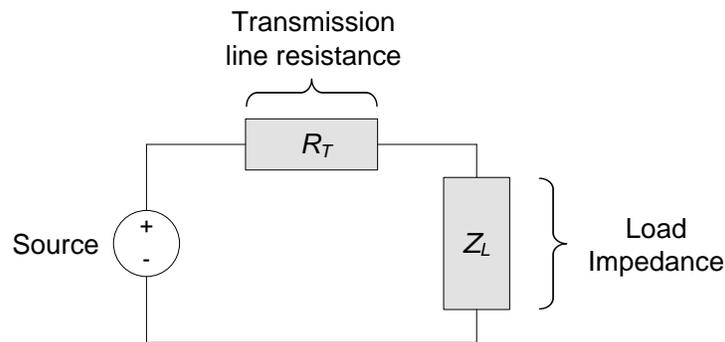


Figure 1. Power delivery system block diagram.

In this lab assignment, we will emulate the system of Fig. 1 with the circuit shown in Fig. 2. The load is composed of a series combination of an inductor,  $L$ , and a load resistance,  $R_L$ . The “transmission line” resistance is represented by the resistor  $R_T$ . In this lab assignment, we will examine the effects of the load’s power factor on the power dissipated by the resistor  $R_T$ . For the purposes of this lab assignment, we will emphasize the transmission line effects by making the resistance  $R_T$  large relative to the load impedance.

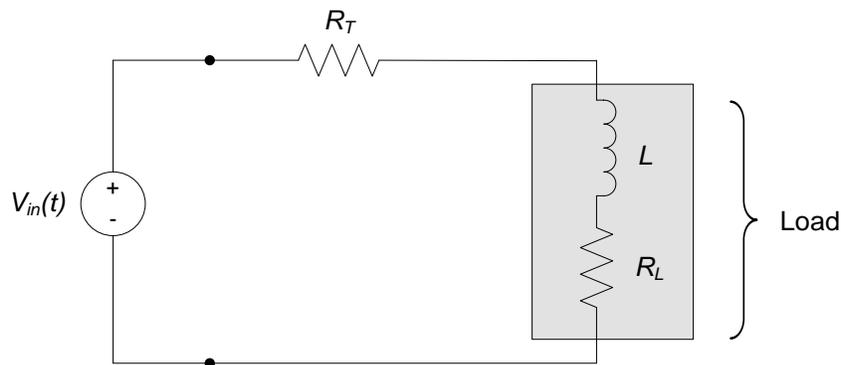


Figure 2. Circuit used to implement the system of Figure 1.

## Pre-lab:

Nominal values for the parameters in the circuit of Fig. 2 are:

- $V_{in}(t) = 1\cos(2\pi \cdot 5000t)$  (e.g. the input sinusoid has a  $1V_{0-P}$  amplitude and a 5kHz frequency.)
- $L = 1\text{mH}$
- $R_T = 10\Omega$
- $R_L = 10\Omega, 47\Omega, \text{ and } 100\Omega$ .

### ANALYSIS

Analyze the circuit of Fig. 2 to determine, for each value of  $R_L$ <sup>60</sup>,

- a. the RMS current delivered by the source,
- b. the RMS load voltage,

<sup>60</sup> The calculations are simple, but rather tedious. The MATLAB m-file associated with this lab can be used to streamline these calculations. Run the m-file and input the appropriate resistance values when prompted. Assume that the inductor resistance is zero for the pre-lab calculations.

- c. the average power delivered to the load,
- d. the apparent power delivered to the load,
- e. the load's power factor,
- f. the average power dissipated by the resistor  $R_T$ <sup>61</sup>, and
- g. the ratio between the average power dissipated by the resistor  $R_T$  and the average power delivered to the load.

Create a table providing the above parameters for each load resistance value.

### Lab Procedures:

- a. Construct the circuit of Fig. 2 using the  $10\Omega$  load resistance and the other component values as given in the pre-lab.

#### DATA

- i. Be sure to measure and record actual resistance values of the components, including the inductor resistance.
- ii. Use one channel of your arbitrary waveform generator to apply the sinusoidal input voltage provided in the pre-lab. Do not turn on the waveform generator yet.
- iii. Use channel 1 (CH1) of your oscilloscope to measure the input voltage and channel 2 (CH2) of your oscilloscope to measure the voltage across the load. Set up a math channel as follows:
  - Channel M1: the current through the load. This can be calculated from the voltage difference across the resistor  $R_T$ , divided by the resistance  $R_T$ .
- iv. Create measurements using the "Measure" button to display:
  - The RMS value of the load current (channel M1)
  - The RMS value of the load voltage (CH2)

- b. Turn on the waveform generator to apply power to your circuit.

#### DATA

- i. Record an image of the oscilloscope window, showing the three waveforms and two measurements listed in part a above. Record the following measurements in your lab notebook:
  - The RMS value of the load voltage
  - The RMS value of the load current
  - The phase difference between the load voltage and the load current
- ii. Repeat the measurements of part b, for the other two load resistors provided in the pre-lab. ( $R_L = 47\Omega$  and  $100\Omega$ .)
- iii. Replace the  $10\Omega$  load resistor in the circuit. Modify the load to include a  $1\mu\text{F}$  capacitor in parallel with the previous load, as shown in Fig. 3 below. Repeat the measurements of part b above.

#### DEMO

- iv. Demonstrate operation of your circuit to the TA and have them initial the appropriate page(s) of your lab notebook and the lab worksheet.

<sup>61</sup> The power dissipated by the transmission line resistance can be determined from  $P = R \cdot I_{RMS}^2$ .

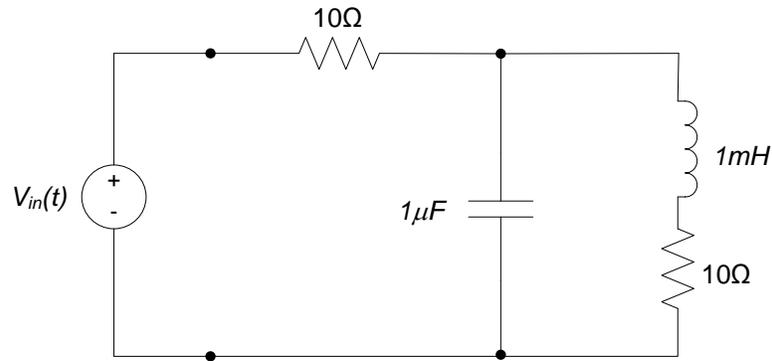


Figure 3. Circuit with capacitor in parallel with load.

### Post-lab Exercises:

1. Re-run the pre-lab calculations using your measured values of  $R_T$ ,  $R_L$ , and inductor resistance, for each value of load resistance.
2. Use the measurements you made in the lab procedures to calculate the following parameters, for each value of load resistance in parts a through c of the lab procedures:
  - a. The power factor of the load
  - b. The apparent power delivered to the load, as determined from the RMS load voltage and the RMS load current:  $|S| = V_{RMS} I_{RMS}$
  - c. The average power delivered to the load, as determined from the RMS load voltage, the RMS load current, and the power factor:  $P_L = V_{RMS} I_{RMS} \cos(\theta_v - \theta_i)$ .
  - d. The power dissipated by the transmission line resistor  $R_T$ :  $P_T = R_T \cdot I_{RMS}^2$ .
  - e. The ratio between the average power dissipated by the transmission resistor and the average power delivered to the load,  $\frac{P_T}{P_L}$ .

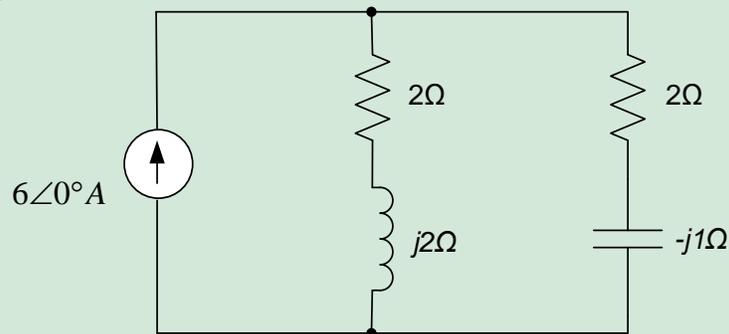
#### ANALYSIS

#### ANALYSIS

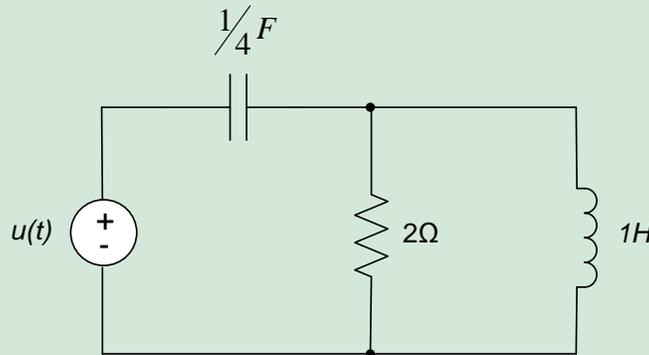
Also perform the above calculations for the circuit of part d of the lab procedures. (e.g., the circuit with the capacitor in parallel with the load.)

## Real Analog Chapter 12: Homework

- 12.1 The circuit shown is described in the frequency domain. Determine the average power absorbed or generated by each circuit element.



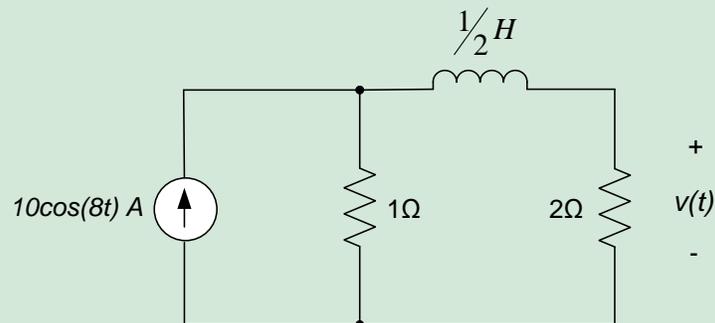
- 12.2 Find the average power generated by the source in the circuit below, if  $u(t) = 10\cos(2t)$ .



- 12.3 For the circuit of problem 2, find the power absorbed by the resistor.

- 12.4 For the circuit below, find:

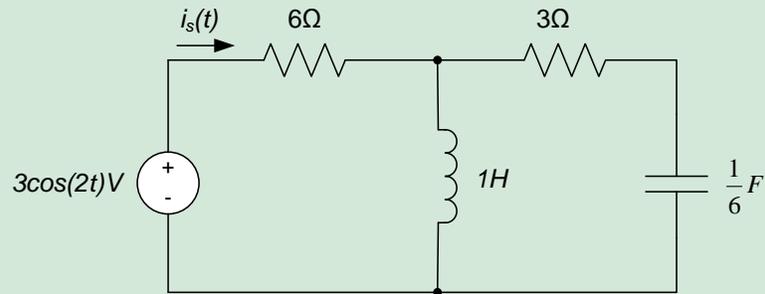
- The voltage  $v(t)$
- The average power delivered by the source.
- The complex power delivered by the source
- The inductor power



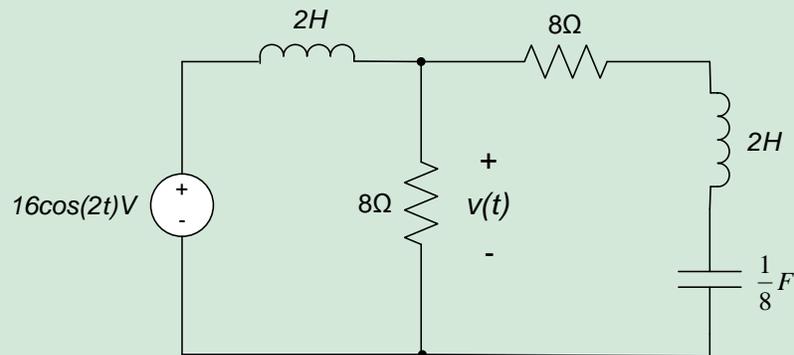
- 12.5 A load consumes 150kW with a power factor  $\text{pf} = 0.7$  (lagging). If the load current is 240A (RMS), find the load voltage.

- 12.6 An industrial plant has an inductive load which consumes 10kW of power from a 220VRMS line. If the power factor is 0.8, what is the difference in angle between the load voltage and the load current?

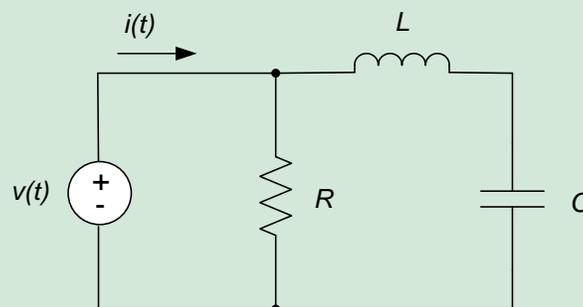
- 12.7 For the circuit below,
- Find the complex power delivered by the source.
  - Sketch a power triangle for the power delivered by source.
  - Find the average power delivered by the source.
  - Find the total power absorbed by both resistors.
  - Find the power absorbed by the  $3\Omega$  resistor.



- 12.8 For the circuit shown, find:
- $v(t)$ ,  $t \rightarrow \infty$
  - The complex power delivered by the source.
  - The average power delivered by the source.
  - The reactive power delivered by the source.

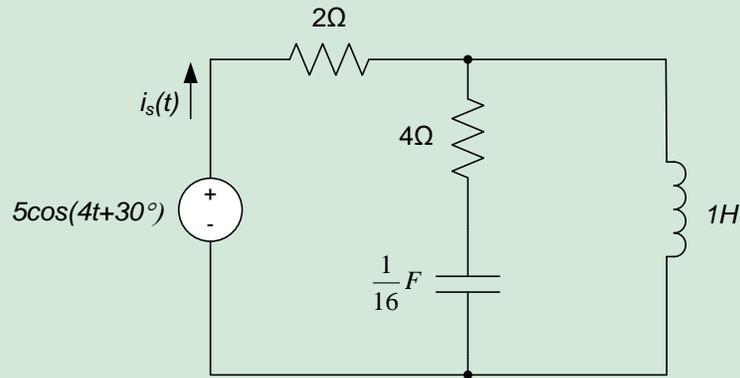


- 12.9 For the circuit shown,  $v(t) = 120 \cos(100t + 65^\circ) V$  and  $i(t) = 2 \cos(100t + 30^\circ) A$
- The complex power delivered by the source.
  - The power dissipated by the resistor.
  - The equivalent impedance seen by the source.



- 12.10 For the circuit shown, find:
- The equivalent impedance seen by the source.

- b. The complex power delivered by the source.



12.11 For the circuit shown,

- Find the complex power delivered by the source.
- Sketch a power triangle for the power delivered by the source.

