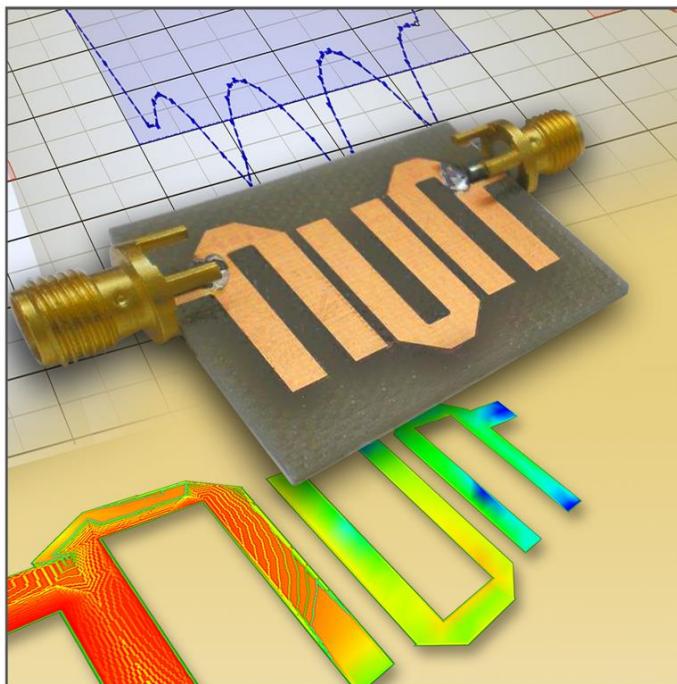


Microwave and RF Engineering

A Simulation Approach with
Keysight Genesys Software

Chapter 4: Resonant Circuits and Filters



Ali A. Behagi

Stephen D. Turner

Microwave and RF Engineering

A Simulation Approach with Keysight Genesys Software

ISBN 978-09835460-3-0

Copyright © 2015 by Ali A. Behagi

**Published in USA
BT Microwave LLC
State College, PA 16803**

All rights reserved. Printed and bound in the United States of America. No part of this book may be reproduced, stored in a retrieval system, or transmitted in any form or by any means, electronic, photocopying, recording, or otherwise, without prior permission in writing from the publishers.

Textbook Table of Contents

Foreword	xv
Preface	xvii
Chapter 1 RF and Microwave Concepts and Components	1
1.1 Introduction	1
1.2 Straight Wire, Flat Ribbon, and Skin Depth	3
1.2.1 Calculation of Straight Wire Inductance	3
1.2.2 Analysis of Straight Wire Inductor in Genesys	5
1.2.3 Skin Depth in Conductors	8
1.2.4 Wire Resistance at Different Frequencies	10
1.3 Physical Resistors	12
1.3.1 Chip Resistors	14
1.4 Physical Inductors	16
1.4.1 Air Core inductors	18
1.4.2 Modeling the Air Core Inductor in Genesys	22
1.4.3 Inductor Q Factor	27
1.4.4 Chip Inductors	28
1.4.5 Chip Inductor Simulation in Genesys	29
1.4.6 Magnetic Core Inductors	32
1.5 Physical Capacitors	40
1.5.1 Single Layer Capacitor	41
1.5.2 Multilayer Capacitors	43
1.5.3 Capacitor Q Factor	44
References and Further Reading	49
Problems	49
Chapter 2 Transmission Lines	53
2.1 Introduction	53
2.2 Plane Waves	53
2.2.1 Plane Waves in a Lossless Medium	53

2.2.2	Plane Waves in a Good Conductor	55
2.3	Lumped Element Representation of Transmission Lines	56
2.4	Transmission Line Equations and Parameters	57
2.4.1	Definition of Attenuation and Phase Constant	59
2.4.2	Definition of Transmission Line Characteristic Impedance	59
2.4.3	Definition of Transmission Line Reflection Coefficient	59
2.4.4	Definition of Voltage Standing Wave Ratio, VSWR	60
2.4.5	Definition of Return Loss	61
2.4.6	Lossless Transmission Line Parameters	61
2.4.7	Lossless Transmission Line Terminations	62
2.4.8	Simulating Reflection Coefficient and VSWR in Genesys	64
2.4.9	Return Loss, VSWR, and Reflection Coefficient Conversion	64
2.5	RF and Microwave Transmission Media	67
2.5.1	Free Space Characteristic Impedance and Velocity of Propagation	67
2.5.2	Physical Transmission Lines	68
2.6	Coaxial Transmission Line	70
2.6.1	Coaxial Transmission Lines in Genesys	73
2.6.2	Using the RG8 Coaxial Cable Model in Genesys	74
2.7	Microstrip Transmission Lines	76
2.7.1	Microstrip Transmission Lines in Genesys	78
2.8	Stripline Transmission Lines	80
2.9	Waveguide Transmission Lines	82
2.9.1	Waveguide Transmission Lines in Genesys	86
2.10	Group Delay in Transmission Lines	89
2.10.1	Comparing Group Delay of Various Transmission lines	89
2.11	Transmission Line Components	91
2.11.1	Short-Circuited Transmission Line	91
2.11.2	Modeling Short-Circuited Microstrip Lines	92
2.11.3	Open-Circuited Transmission Line	94
2.11.4	Modeling Open-Circuited Microstrip Lines	95
2.11.5	Distributed Inductive and Capacitive Elements	96
2.11.6	Distributed Microstrip Inductance and Capacitance	97
2.11.7	Step Discontinuities	98

2.11.8 Microstrip Bias Feed Networks	99
2.11.9 Distributed Bias Feed	100
2.12 Coupled Transmission Lines	102
2.12.1 Directional Coupler	105
2.12.2 Microstrip Directional Coupler Design	107
References and Further Reading	110
Problems	110
Chapter 3 Network Parameters and the Smith Chart	113
3.1 Introduction	113
3.1.1 Z Parameters	113
3.1.2 Y Parameters	114
3.1.3 h Parameters	115
3.1.4 ABCD Parameters	116
3.2 Development of Network S-Parameters	117
3.3 Using S Parameter Files in Genesys	120
3.3.1 Scalar Representation of the S Parameters	123
3.4 Development of the Smith Chart	124
3.4.1 Normalized Impedance on the Smith Chart	126
3.4.2 Admittance on the Smith Chart	128
3.5 Lumped Element Movements on the Smith Chart	130
3.5.1 Adding a Series Reactance to an Impedance	130
3.5.2 Adding a Shunt Reactance to an Impedance	132
3.6 VSWR Circles on the Smith Chart	134
3.7 Adding a Transmission Line in Series with an Impedance	137
3.8 Adding a Transmission Line in Parallel with an Impedance	139
3.8.1 Short Circuit Transmission Lines	140
3.8.2 Open Circuit Transmission Lines	141
3.9 Open and Short Circuit Shunt Transmission Lines	141

References and Further Reading	144
Problems	144
Chapter 4 Resonant Circuits and Filters	147
4.1 Introduction	147
4.2 Resonant Circuits	147
4.2.1 Series Resonant Circuits	147
4.2.2 Parallel Resonant Circuits	149
4.2.3 Resonant Circuit Loss	150
4.2.4 Loaded Q and External Q	151
4.3 Lumped Element Parallel Resonator Design	153
4.3.1 Effect of Load Resistance on Bandwidth and Q_L	154
4.4 Lumped Element Resonator Decoupling	155
4.4.1 Tapped Capacitor Resonator	156
4.4.2 Tapped Inductor Resonator	157
4.5 Practical Microwave Resonators	158
4.5.1 Transmission Line Resonators	159
4.5.2 Microstrip Resonator Example	162
4.5.3 Genesys Model of the Microstrip Resonator	164
4.6 Resonator Series Reactance Coupling	166
4.6.1 One Port Microwave Resonator Analysis	168
4.6.2 Q_o Measurement of the Microstrip Resonator	171
4.7 Filter Design at RF and Microwave Frequency	176
4.7.1 Filter Topology	176
4.7.2 Filter Order	177
4.7.3 Filter Type	179
4.7.4 Filter Return Loss and Passband Ripple	181
4.8 Lumped Element Filter Design	184
4.8.1 Low Pass Filter Design Example	184
4.8.2 Physical Model of the Low Pass Filter in Genesys	186
4.8.3 High Pass Filter Design Example	188
4.8.4 Physical Model of the High Pass Filter in Genesys	189
4.8.5 Tuning the High Pass Filter Response	190
4.8.6 S Parameter File Tuning with VBScript	191
4.9 Distributed Filter Design	196

4.9.1	Microstrip Stepped Impedance Low Pass Filter Design	196
4.9.2	Lumped Element to Distributed Element Conversion	196
4.9.3	Electromagnetic Modeling of the Stepped Impedance Filter	201
4.9.4	Reentrant Modes	205
4.9.5	Microstrip Coupled Line Filter Design	206
4.9.6	Electromagnetic Analysis of the Edge Coupled Filter	208
4.9.7	Enclosure Effects	211
	References and Further Reading	213
	Problems	214
Chapter 5 Power Transfer and Impedance Matching		217
5.1	Introduction	217
5.2	Power Transfer Basics	217
5.2.1	Maximum Power Transfer Conditions	218
5.2.2	Maximum Power Transfer with Purely Resistive Source and Load Impedance	220
5.2.3	Maximum Power Transfer Validation in Genesys	222
5.2.4	Maximum Power Transfer with Complex Load Impedance	224
5.3	Analytical Design of Impedance Matching Networks	225
5.3.1	Matching a Complex Load to Complex Source Impedance	227
5.3.2	Matching a Complex Load to a Real Source Impedance	234
5.3.3	Matching a Real Load to a Real Source Impedance	242
5.4	Introduction to Broadband Matching Networks	247
5.4.1	Analytical Design of Broadband Matching Networks	247
5.4.2	Broadband Impedance Matching Using N-Cascaded L-Networks	253
5.4.3	Derivation of Equations for Q and the number of L-Networks	257
5.5	Designing with Q-Curves on the Smith Chart	259
5.5.1	Q-Curve Matching Example	261
5.6	Limitations of Broadband Matching	264
5.6.1	Example of Fano's Limit Calculation	265

5.7	Matching Network Synthesis	266
5.7.1	Filter Characteristics of the L-networks	266
5.7.2	L-Network Impedance Matching Utility	267
5.7.3	Network Matching Synthesis Utility in Genesys	270
5.7.4	Effect of Finite Q on the Matching Networks	272
	References and Further Reading	275
	Problems	275
Chapter 6 Analysis and Design of Distributed Matching Networks		277
6.1	Introduction	277
6.2	Quarter-Wave Matching Networks	277
6.2.1	Analysis of Quarter-Wave Matching Networks	278
6.2.2	Analytical Design of Quarter-Wave Matching Networks	281
6.3	Quarter-Wave Network Matching Bandwidth	286
6.3.1	Effect of Load Impedance on Matching Bandwidth	286
6.3.2	Quarter-Wave Network Matching Bandwidth and Power Loss in Genesys	290
6.4	Single-Stub Matching Networks	292
6.4.1	Analytical Design of Series Transmission Line	293
6.4.2	Analytical Design of Shunt Transmission Line	295
6.4.3	Single-Stub Matching Design Example	296
6.4.4	Automated Calculation of Line and Stub Lengths	298
6.4.5	Development of Single-Stub Matching Utility	299
6.5	Graphical Design of Single-Stub Matching Networks	301
6.5.1	Smith Chart Design Using an Open Circuit Stub	301
6.5.2	Smith Chart Design Using a Short Circuit Stub	303
6.6	Design of Cascaded Single-Stub Matching Networks	304
6.7	Broadband Quarter-Wave Matching Network Design	307
	References and Further Reading	318
	Problems	319

Chapter 7 Single Stage Amplifier Design	321
7.1 Introduction	321
7.2 Maximum Gain Amplifier Design	322
7.2.1 Transistor Stability Considerations	323
7.2.2 Stabilizing the Device in Genesys	325
7.2.3 Finding Simultaneous Match Reflection Coefficients and Impedances	328
7.3 Analytical and Graphical Impedance Matching Techniques	328
7.3.1 Analytical Design of the Input Matching Network	329
7.3.2 Synthesis Based Input Matching Networks	331
7.3.3 Synthesis Based Output Matching Networks	333
7.3.4 Ideal Model of the Maximum Gain Amplifier	334
7.4 Physical Model of the Amplifier	336
7.4.1 Transistor Artwork Replacement	337
7.4.2 Amplifier Physical Design and Layout	339
7.4.3 Optimization of the Amplifier Response	343
7.4.4 Optimization Setup Procedure	344
7.5 Specific Gain Amplifier Design	347
7.5.1 Specific Gain Match	347
7.5.2 Specific Gain Design Example	351
7.5.3 Graphical Impedance Matching Circuit Design	356
7.5.4 Assembly and Simulation of the Specific Gain Amplifier	358
7.6 Low Noise Amplifier Design	360
7.6.1 Noise Circles	362
7.6.2 LNA Design Example	365
7.6.3 Analytical Design of the LNA Input Matching Network	367
7.6.4 Analytical Design of the LNA Output Matching Network	368
7.6.5 Linear Simulation of the Low Noise Amplifier	371
7.6.6 Amplifier Noise Temperature	373
7.7 Power Amplifier Design	376
7.7.1 Data Sheet Large Signal Impedance	377
7.7.2 Power Amplifier Matching Network Design	379
7.7.3 Input Matching Network Design	379

7.7.4	Output Matching Network Design	382
	References and Further Reading	386
	Problems	387

Chapter 8 Multi-Stage Amplifier Design and Yield Analysis 391

8.1	Introduction	391
8.2	Two-Stage Amplifier Design	391
8.2.1	First Stage Matching Network Design	392
8.2.2	Design of the Amplifier Input Matching Network	393
8.2.3	Second Stage Matching Network Design	394
8.2.4	Inter-Stage Matching Network Design	395
8.2.5	Second Stage Output Matching Network	396
8.3	Two-Stage Amplifier Simulation	396
8.4	Parameter Sweeps	398
8.5	Monte Carlo and Sensitivity Analysis	400
8.6	Yield Analysis	405
8.6.1	Design Centering	407
8.7	Low Noise Amplifier Cascade	408
8.7.1	Cascaded Gain and Noise Figure	408
8.7.2	Impedance Match and the Friis Formula	410
8.7.3	Reducing the Effect of Source Impedance Variation	412
8.8	Summary	413
	References and Further Reading	414
	Problems	414
Appendix		417
Appendix A	Straight Wire Parameters for Solid Copper Wire	417
Appendix B.1	Γ_i Line Generation	418
Appendix B.2	Q_L Lines on the Smith Chart	420

Appendix B.3 Ideal Q Circle on the Smith Chart	422
Appendix B.4 Q_0 Measurement on the Smith Chart	424
Appendix C VBScript file listing for the Matching Utility of Chapter 5	425
Appendix D VBScript file listing for the Line and Stub Matching Utility of Chapter 6	434
Index	439
About the Authors	445

Foreword

Unlike many traditional books on RF and microwave engineering written mainly for the classroom, this book adopts a practical, hands-on approach to quickly introduce students and engineers unfamiliar with this topic to this subject matter. The authors make extensive use of Electronic Design Automation (EDA) tools to illustrate the foundational principles of RF and microwave engineering. The use of EDA methodologies in the book closely parallels the latest tools and techniques commonly used in industry to accelerate the design of RF/microwave systems and components to meet demanding specifications and ensure high yields.

This book provides readers a solid understanding of RF and microwave concepts such as Smith chart, S-parameters, transmission lines, impedance matching, resonators, filters and amplifiers. More importantly, it details how to use EDA tools to synthesize, simulate, tune, and optimize these essential components into a design flow as practiced in the industry. For explanatory purposes, the authors made the judicious choice of an easy-to-use and fully featured EDA tool that is also very affordable. This ensures that the skills learned in this book can be easily and immediately put into practice without the barriers of having to acquire costly and complex EDA tools.

Genesys from Keysight Technologies is that tool and it was chosen not only for its low cost, but because it provides the ideal combination of capabilities; in circuit synthesis, simulation and optimization; MATLAB scripting, RF system design, and electromagnetic and statistical analysis. The tool is a mature, well trusted solution that has successfully proven itself in the design of state-of-the-art RF and microwave test instrumentation and been time-tested by a large following of users worldwide for over 20 years.

The investment in learning the foundational RF/microwave skills and EDA techniques taught in this book provides engineers and students with valuable knowledge that will remain relevant and sought-after for a long time to come.

I wish such a book had been available when I first started my career as a microwave component designer. Without a doubt it would have made gaining RF and microwave insights much quicker than the countless hours I invested using the “cut-and-try” method on the bench.

How-Siang Yap
Keysight EEs of EDA Genesys Planning and Marketing
1400 Fountaingrove Parkway
Santa Rosa, CA 95403, USA

Preface

Microwave Engineering can be a fascinating and fulfilling career path. It is also an extremely vast subject with topics ranging from semiconductor physics to electromagnetic theory. Unlike many texts on the subject this book does not attempt to cover every aspect of Microwave Engineering in a single volume. This text book is the first volume of a two-part series that covers the subject from a computer aided design standpoint. The first volume covers introductory topics which are appropriate to be addressed by linear simulation methods. This includes topics such as lumped element components, transmission lines, impedance matching, and basic linear amplifier design. The second volume focuses on subject matter that is better learned through non-linear computer simulation. This includes topics such as oscillators, mixers, and power amplifier design.

Almost all subject matter covered in the text is accompanied by examples that are solved using the Genesys linear simulation software by Agilent. University students will find this a potent learning tool. Practicing engineers will find the book very useful as a reference guide to quickly setup designs using the Genesys software. The authors thoroughly cover the basics as well as introducing CAD techniques that may not be familiar to some engineers. This includes subjects such as the frequent use of the Genesys equation editor and Visual Basic scripting capability. There are also topics that are not usually covered such as techniques to evaluate the Q factor of one port resonators and yield analysis of microwave circuits.

The organization of the book is as follows: Chapter 1 presents a general explanation of RF and microwave concepts and components. Engineering students will be surprised to find out that resistors, inductors, and capacitors at high frequencies are no longer ideal elements but rather a network of circuit elements. For example, a capacitor at one frequency may in fact behave as an inductor at another frequency. In chapter 2 the transmission line theory is developed and several important parameters are defined. It is shown how to simulate and measure these parameters using Genesys software. Popular types of transmission lines are introduced and their parameters are examined. In Chapter 3 network parameters and the application of Smith chart as a graphical tool in dealing with impedance behavior and reflection coefficient are discussed. Description

of RF and microwave networks in terms of their scattering parameters, known as S- Parameters, is introduced. The subject of lumped and distributed resonant circuits and filters are discussed in Chapter 4. Using the Genesys software a robust technique is developed for the evaluation of Q factor form the S- Parameters of a resonant circuit. An introduction to the vast subject of filter synthesis and the electromagnetic simulation of distributed filters are also treated in this chapter. In Chapter 5 the condition for maximum power transfer and the lumped element impedance matching are considered. The analytical equations for matching two complex impedances with lossless two-element networks are derived. Both analytical and graphical techniques are used to design narrowband and broadband matching networks. The Genesys impedance matching synthesis program is used to solve impedance matching problems. The VBScript programming techniques developed in this chapter can be used by students to generate their own synthesis applications within the Genesys software. In Chapter 6 both narrowband and broadband distributed matching networks are analytically and graphically analyzed. In Chapter 7 single-stage amplifiers are designed by utilizing four different impedance matching objectives. The first amplifier is designed for maxim gain where the input and the output are conjugately matched to the source and load impedance; the second amplifier is designed for specific gain where the input or the output is mismatched to achieve a specific gain less than its maximum; the third amplifier is a low noise amplifier where the transistor is selectively mismatched to achieve a specific Noise Figure; and the fourth amplifier is a power amplifier where the transistor is selectively mismatched to achieve a specific amount of output power. In Chapter 8 a two-stage amplifier is designed by utilizing a direct interstage matching network. Monte Carlo and Yield analysis techniques are also introduced in this chapter. Finally a brief introduction to cascade analysis is presented.

Ali A. Behagi
Stephen D. Turner
June 2015

Chapter 4: Resonant Circuits and Filters

4.1 Introduction

The first half of this chapter examines resonant circuits. Lumped element resonant circuits and the lumped equivalent networks of mechanical and distributed resonators are considered. Resonant circuits are used in many applications, such as filters, oscillators, tuners, tuned amplifiers, and microwave communication networks. The analysis of basic lumped element series and parallel RLC resonant circuits is implemented in the Genesys software. The discussion turns to microwave resonators with an analysis of the Q factor measurement of transmission line resonators. Using the Genesys software a robust technique is demonstrated for the evaluation of Q factor from the measured S parameters of a resonant circuit. The second half of the chapter is an introduction to the vast subject of filter networks. The design of lumped element filters is introduced and followed by an introduction to distributed element filters. The chapter concludes with an introduction to Electromagnetic (EM) simulation of distributed filters.

4.2 Resonant Circuits

Near resonance, RF and microwave resonant circuits can be represented either as a series or parallel RLC network.

4.2.1 Series Resonant Circuits

In this section we analyze the behavior of the resonant circuit in Genesys.

Example 4.2-1: Consider the one port resonator that is represented as a series RLC circuit of Figure 4-1. Analyze the circuit, with $R = 10 \Omega$, $L = 10 \text{ nH}$, and $C = 10 \text{ pF}$.

Solution: The plot of the resonator's input impedance in Figure 4-1 shows that the resonance frequency is about 503.3 MHz and the input impedance at resonance is 10Ω , the value of the resistor in the network.

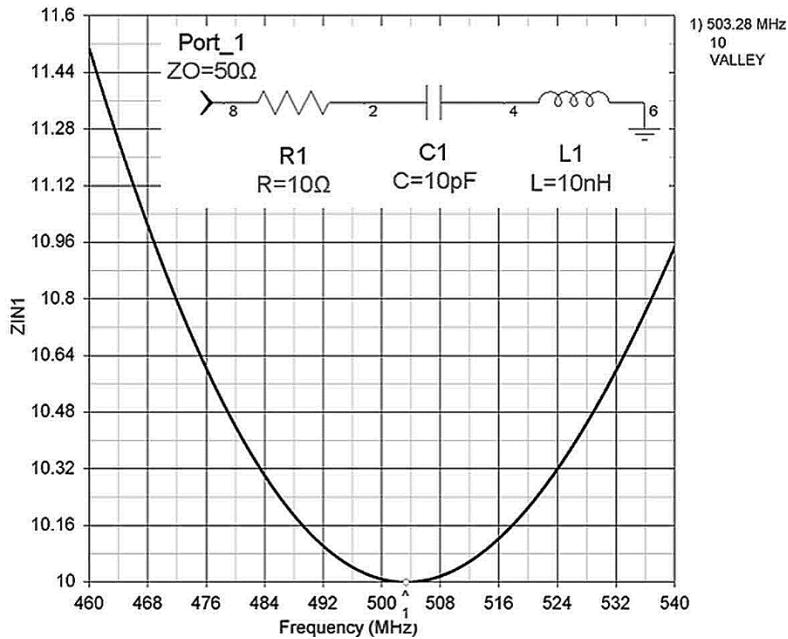


Figure 4-1 One-port series RLC resonator circuit and input impedance

The input impedance of the series RLC resonant circuit is given by,

$$Z_{in} = R + j\omega L - j \frac{1}{\omega C}$$

where, $\omega = 2\pi f$ is the angular frequency in radian per second.

If the AC current flowing in the series resonant circuit is I , then the complex power delivered to the resonator is

$$P_{in} = \frac{|I|^2}{2} Z_{in} = \frac{|I|^2}{2} \left(R + j\omega L - j \frac{1}{\omega C} \right) \quad (4-1)$$

At resonance the reactive power of the inductor is equal to the reactive power of the capacitor. Therefore, the power delivered to the resonator is equal to the power dissipated in the resistor

$$P_{in} = \frac{|I|^2 R}{2} \quad (4-2)$$

4.2.2 Parallel Resonant Circuits

Example 4.2-2: Analyze a rearrangement of the RLC components of Figure 4-1 into the parallel configuration of Figure 4-2. The schematic of Figure 4-2 represents the lumped element representation of the parallel resonant circuit.

Solution: The plot of the magnitude of the input impedance shows that the resonance frequency is still 503.3 MHz where the input impedance is $R = 10 \Omega$. Again this shows that the impedance of the inductor cancels the impedance of the capacitor at resonance. In other words, the reactance, X_L is equal to the reactance, X_C , at the resonance frequency.

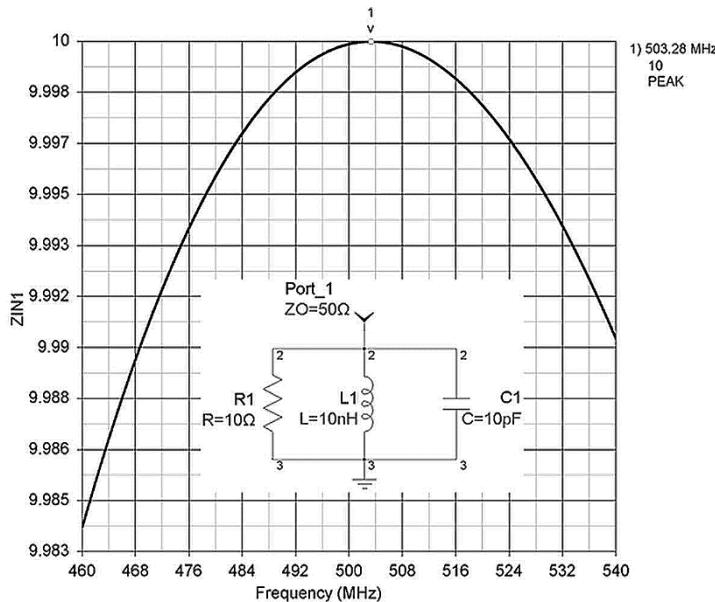


Figure 4-2 One-port parallel RLC resonant circuit and input impedance

The input admittance of the parallel resonant circuit is given by:

$$Y_{IN} = \frac{1}{R} + j\omega C - j \frac{1}{\omega L}$$

If the AC voltage across the parallel resonant circuit is V , then the complex power delivered to the resonator is:

$$P_{in} = \frac{|V|^2}{2} Y_{in} = \frac{|V|^2}{2} \left(\frac{1}{R} + j\omega C - j\frac{1}{\omega L} \right) \quad (4-3)$$

At resonance the reactive power of the inductor is equal to the reactive power of the capacitor. Therefore, the power delivered to the resonator is equal to the power dissipated in the resistor

$$P_{in} = \frac{|V|^2}{2R} \quad (4-4)$$

The resonance frequency for the parallel resonant circuit as well as the series resonant circuit is obtained by setting $\omega_0 C = \frac{1}{\omega_0 L}$ or:

$$\omega_o = 2\pi f_o = \frac{1}{\sqrt{LC}} \quad (4-5)$$

where, ω_0 is the angular frequency in radian per second and f_0 is equal to the frequency in Hertz.

4.2.3 Resonant Circuit Loss

In Figure 4-1 and 4-2 the resistor, R1, represents the loss in the resonator. It includes both the losses in the capacitor as well as the inductor. The Q factor can be shown to be a ratio of the energy stored in the inductor and capacitor to the power dissipated in the resistor as a function of frequency [6]. For the series resonant circuit of Figure 4-1 the Q factor is defined by:

$$Q_u = \frac{X}{R} = \frac{\omega_o L}{R} = \frac{1}{\omega_o RC} \quad (4-6)$$

The Q factor of the parallel resonant circuit is simply the inverse of the series resonant circuit.

$$Q_u = \frac{R}{X} = \frac{R}{\omega_o L} = \omega_o RC \quad (4-7)$$

Notice that as the resistance increases in the series resonant circuit, the Q factor decreases. Conversely as the resistance increases in the parallel resonant circuit, the Q factor increases. The Q factor is a measure of loss in the resonant circuit. Thus a higher Q corresponds to lower loss and a lower Q corresponds to a higher loss. It is usually desirable to achieve high Q factors in a resonator as it will lead to lower losses in filters or lower phase noise in oscillators. Note that the resonator Q of Equation (4-6) and (4-7) is defined as the unloaded Q of the resonator. This means that the resonator is not connected to any source or load impedance and as such is unloaded. The measurement of Q_u requires that the resonator be attached (coupled) to a signal source or load of some finite impedance. Equations (4-6) and (4-7) would then have to be modified to include the source and load resistance. We might also surmise that any reactance associated with the source or load impedance may alter the resonant frequency of the resonator. This leads to two additional definitions of Q factor; the loaded Q and external Q.

4.2.4 Loaded Q and External Q

Example 4.2-3: Analyze the parallel resonator that is attached to a 50Ω source and load as shown in Figure 4-3.

Solution: Using Equation (4-7) to define the Q factor for the circuit requires that we include the source and load resistance which is ‘loading’ the resonator. This leads to the definition of the loaded Q, Q_L , for the parallel resonator as defined by Equation (4-8).

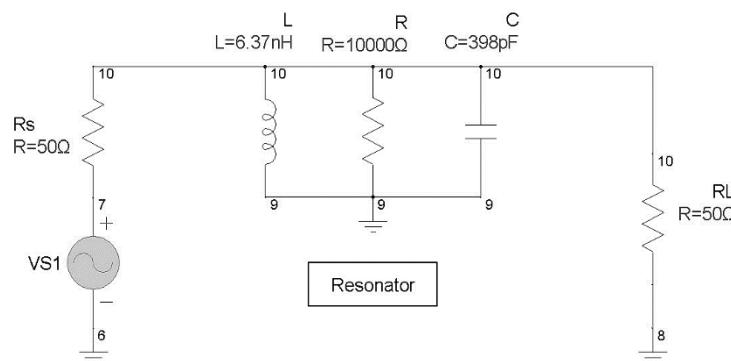


Figure 4-3 Parallel resonator with source and load impedance attached

$$Q_L = \frac{R_s + R + R_L}{\omega_o L} \quad (4-8)$$

Conversely we can define a Q factor in terms of only the external source and load resistance. This leads to the definition of the external Q, Q_E .

$$Q_E = \frac{R_s + R_L}{\omega_o L} \quad (4-9)$$

The three Q factors are related by the inverse relationship of Equation (4-10).

$$\frac{1}{Q_L} = \frac{1}{Q_E} + \frac{1}{Q_U} \quad (4-10)$$

At RF and microwave frequencies it is difficult to directly measure the Q_u of a resonator. We may be able to calculate the Q factor based on the physical properties of the individual inductors and capacitors as we have seen in chapter 1. This is usually quite difficult and the Q factor is typically measured using a Vector Network Analyzer, VNA. Therefore, the measured Q factor is usually the loaded Q, Q_L . External Q is often used with oscillator circuits that are generating a signal. In this case the oscillator's load impedance is varied so that the external Q can be measured. The loaded Q of the network is then related to the fractional bandwidth by Equation (4-11) [7].

$$Q_L = \frac{f_o}{BW} \quad (4-11)$$

where, BW is the 3 dB bandwidth in Hertz and f_0 is equal to the resonant frequency in Hertz.

4.3 Lumped Element Parallel Resonator Design

Example 4.3-1: In this example we design a lumped element parallel resonator at a frequency of 100 MHz. The resonator is intended to operate between a source resistance of 100 Ω and a load resistance of 400 Ω .

Solution: Best accuracy would be obtained by using S parameter files or Modelithics models for the inductor and capacitor. However a good first order model can be obtained by using the Genesys inductor and capacitor models that include the component Q factor. These models save us the work of calculating the equivalent resistive part of the inductor and capacitor model. Use the Q factors shown in the schematic of Figure 4-4.

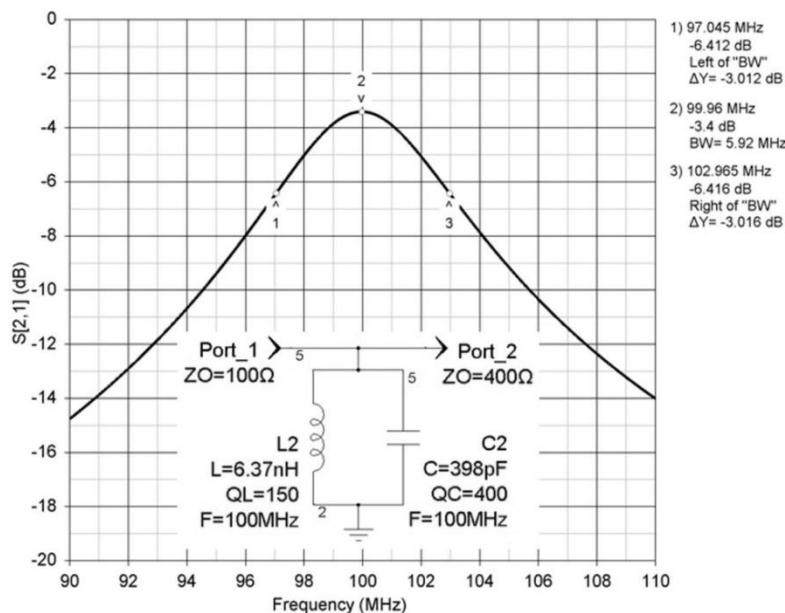


Figure 4-4 Resonator using inductor and capacitor with assigned Q values

Note that markers have been placed on the plot of the insertion loss, S_{21} that gives a direct readout of the 3 dB bandwidth. The loaded Q, Q_L can be calculated using Equation (4-11).

$$Q_L = \frac{f_o}{BW} = \frac{99.955 \text{ MHz}}{5.923 \text{ MHz}} = 16.87$$

The designer must use caution when sweeping resonant circuits in Genesys. Particularly high Q band pass networks require a large number of discrete frequency steps in order to achieve the necessary resolution required to accurately measure the 3dB bandwidth. In this example the Linear Analysis is set up to sweep the circuit from 90 MHz to 110 MHz using 2000 points.

Place a marker anywhere on the trace and double click to open the Marker Properties window. Enter any name for the marker, select Bandwidth (Tracks Peak), and make sure that -3.01dB is entered as the relative offset. As Figure 4-4 shows the 3 dB bandwidth is automatically calculated as 5.923 MHz. The frequency peak and bandwidth label next to marker 2 is then used to calculate Q_L .

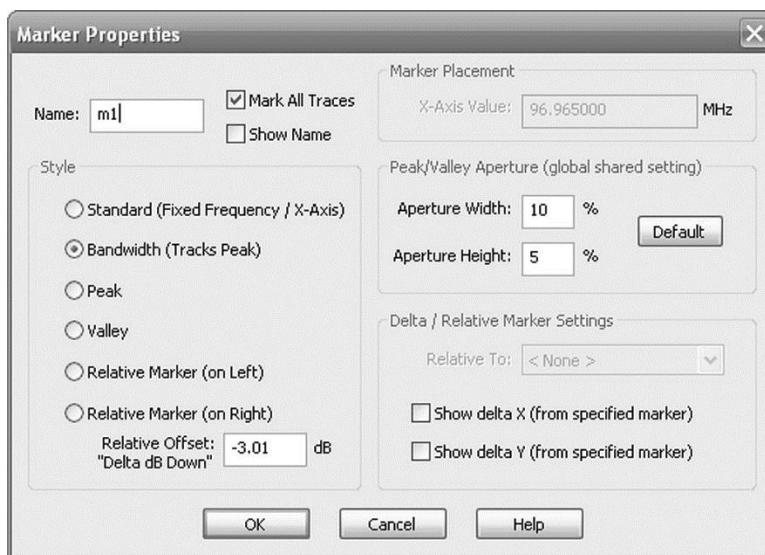


Figure 4-5 Bandwidth marker settings for measurement of 3dB bandwidth

4.3.1 Effect of Load Resistance on Bandwidth and Q_L

In RF circuits and systems the impedances encountered are often quite low, ranging from 1Ω to 50Ω . It may not be practical to have a source impedance of 100Ω and a load impedance of 400Ω .

Example 4.3-2: Using the previous example, change the load resistance from 400Ω to 50Ω and re-examine the circuit's 3 dB bandwidth and Q_L .

Solution: The 3 dB bandwidth is now 12.926 MHz resulting in a loaded Q factor of 7.73. The loaded Q factor has decreased by nearly half of the original value. We have increased the bandwidth or de-Q'd the resonator. This can also be thought of as tighter coupling of the resonator to the load.

$$Q_L = \frac{f_o}{BW} = \frac{99.965\text{MHz}}{12.926\text{MHz}} = 7.73$$

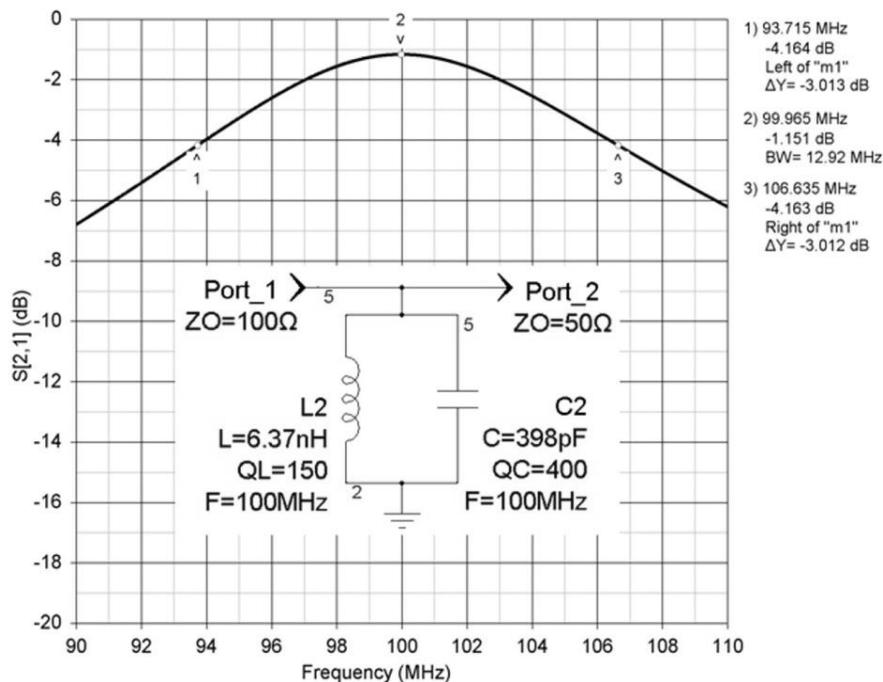


Figure 4-6 A parallel resonance circuit showing the 3dB bandwidth and insertion loss

4.4 Lumped Element Resonator Decoupling

To maintain the high Q of the resonator when attached to a load such as 50Ω , it is necessary to transform the low impedance to high impedance presented to the load. The 50Ω impedance can be transformed to the higher impedance of the parallel resonator thereby resulting in less loading of the resonator impedance. This is referred to as loosely coupling the resonator to the load. The tapped-capacitor and tapped-inductor networks can be used to accomplish this Q transformation in lumped element circuits.

4.4.1 Tapped Capacitor Resonator

Example 4.4-1: Consider rearranging the parallel LC network of Figure 4-6 with the tapped capacitor network shown in Figure 4-7. Re-examine the circuit's 3 dB bandwidth and Q_L .

Solution: The new capacitor values for C_1 and C_2 can be found by the simultaneous solution of the following equations [2].

$$C_T = \frac{C_1 \cdot C_2}{C_1 + C_2} \quad (4-12)$$

$$R_{L1} = R_L \left(1 + \frac{C_1}{C_2} \right)^2 \quad (4-13)$$

R_{L1} is the higher, transformed, load resistance. In this example substitute $R_{L1} = 400 \Omega$, the original load resistance value. C_T is simply the original capacitance of 398 pF. The capacitor values are found to be: $C_1 = 1126.23$ pF and $C_2 = 616.1$ pF. The new resonator circuit is shown in Figure 4-7. Sweeping the circuit we see that the response has returned to the original performance of Figure 4-6. The 3 dB bandwidth has returned to 5.923 MHz making the Q_L equal to:

$$Q_L = \frac{99.925}{5.923} = 16.87$$

The 50Ω load resistor has been successfully decoupled from the resonator. The tapped capacitor and inductor resonators are popular methods of decoupling RF and lower microwave frequency resonators. It is frequently seen in RF oscillator topologies such as the Colpitts oscillator in the VHF frequency range.

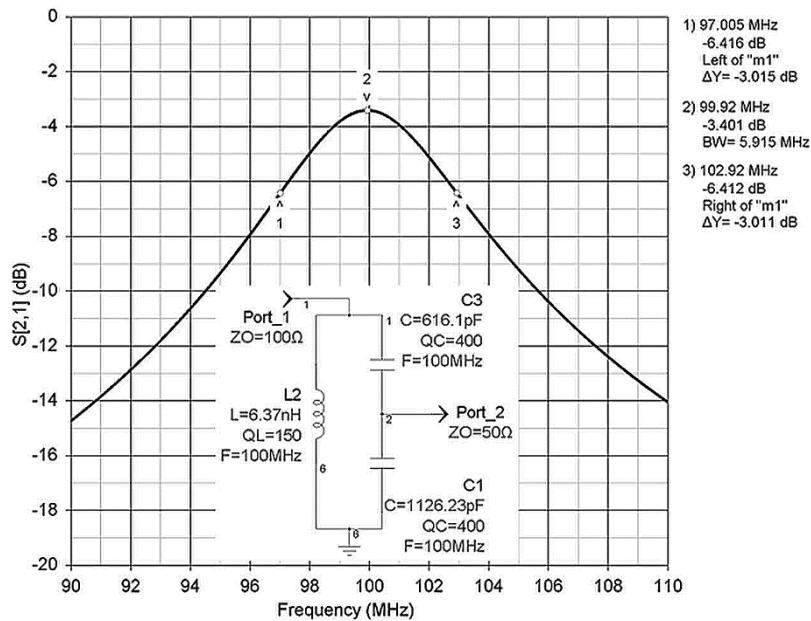


Figure 4-7 Parallel LC resonator using a tapped capacitor and response

4.4.2 Tapped Inductor Resonator

Example 4.4-2: Similarly design a tapped inductor network to decouple the 50Ω source impedance from loading the resonator.

Solution: Replace the 100Ω source impedance with a 50Ω source and use a tapped inductor network to transform the new 50Ω source to 100Ω . Modify the circuit to split the 6.37nH inductor, L_T , into two series inductors, L_1 and L_2 . The inductor values can then be calculated by solving the following equation set simultaneously^[2]. R_{S1} is the higher, transformed, source resistance. In this example substitute $R_{S1} = 100\Omega$,

$$R_{S1} = R_S \left(\frac{L_T}{L_1} \right)^2 \quad (4-14)$$

$$L_T = L_1 + L_2 \quad (4-15)$$

Solving the equation set results in values of $L_1=4.5\text{ nH}$ and $L_2=1.87\text{ nH}$. The resulting schematic and response is shown in Figure 4-8. The new response is identical to the plot of Figure 4-5. Therefore we now have a source and load resistance of 50Ω and have not reduced the Q of the resonator from what we had with the original source resistance of 100Ω and a load resistance of 400Ω .

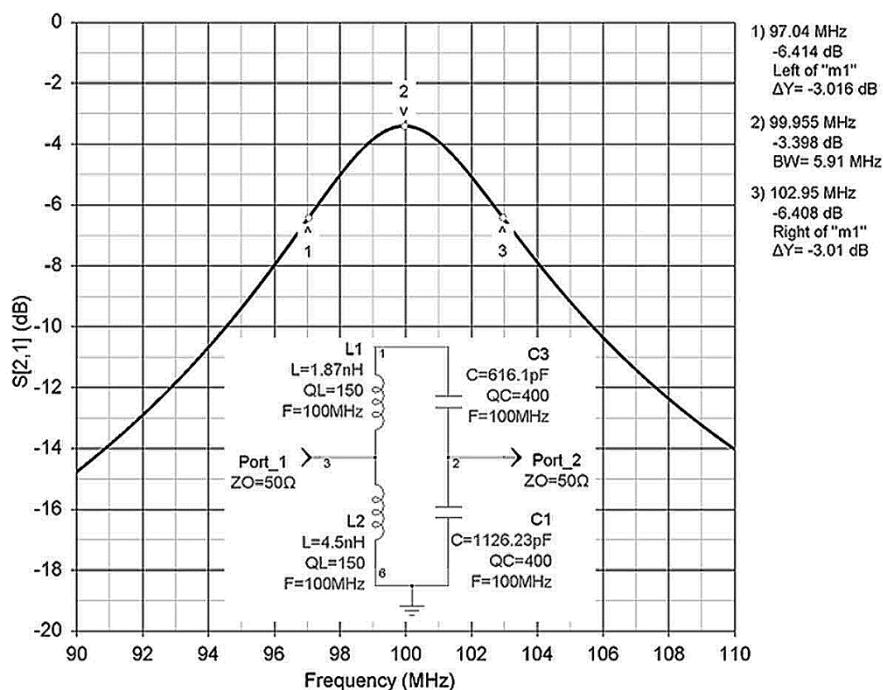


Figure 4-8 Tapped-inductor added to the parallel resonant circuit

4.5 Practical Microwave Resonators

At higher RF and microwave frequencies resonators are seldom realized with discrete lumped element RLC components. This is primarily due to the fact that the small values of inductance and capacitance are physically unrealizable. Even if the values could be physically realized we would see that the resulting Q factors would be unacceptably low for most applications. Microwave resonators are realized in a wide variety of physical forms. Resonators can be realized in all of the basic transmission line forms that were covered in Chapter 2. There are many specialized

resonators such as ceramic dielectric resonator pucks that are coupled to a microstrip transmission line as well as Yttrium Iron Garnet spheres that are loop coupled to its load. These resonators are optimized for very high Q factors and may be tunable over a range of frequencies.

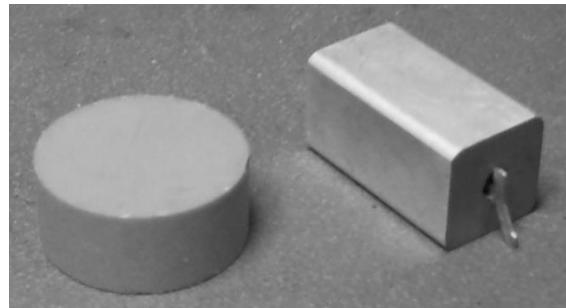


Figure 4-9 Ceramic dielectric resonator (puck) and rectangular coaxial resonator

4.5.1 Transmission Line Resonators

From Fig. 2-26 we have seen that a quarter-wave short-circuited transmission line results in a parallel resonant circuit. Similarly Figure 2-29 showed that a half-wave open circuited transmission line results in a parallel resonant circuit. Such parallel resonant circuits are often used as one port resonators. Near the resonant frequency, the one port resonator behaves as a parallel RLC network as shown in Figure 4-2. As the frequency moves further from resonance the equivalent network becomes more complex typically involving multiple parallel RLC networks. One port resonators are coupled to one another to form filter networks or directly to a transistor to form a microwave oscillator. Knowing the losses due to the physical and electrical parameters of the transmission line, one can calculate the Q_u of the transmission line resonator. The microstrip resonator Q_u is comprised of losses due to the conductor metal, the substrate dielectric, and radiation losses. The Q_u is often dominated by the conductor Q. Unfortunately it can be quite difficult to accurately determine the conductor losses in a microstrip resonator. T. C. Edwards has developed a set of simplified expressions for the conductor losses [4]. Equation (4-16) is an approximation

of the conductor losses that treats the transmission line as a perfectly smooth surface.

$$\alpha_c = 0.072 \frac{\sqrt{f}}{W_e Z_o} \lambda_g \quad \text{dB/inch} \quad (4-16)$$

where,

f = the frequency in GHz

W_e = the effective conductor width (inches)

Z_o = the characteristic impedance of the line

α_c = Conductor loss in dB/inch

λ_g = wavelength in dielectric in inches

A microstrip conductor is actually not perfectly smooth but exhibits a certain roughness. The surface roughness exists on the bottom of the microstrip conductor where it contacts the dielectric. This can be seen by magnifying the cross section of a microstrip line's contact with the dielectric material. The surface roughness is usually specified as an r.m.s. value.

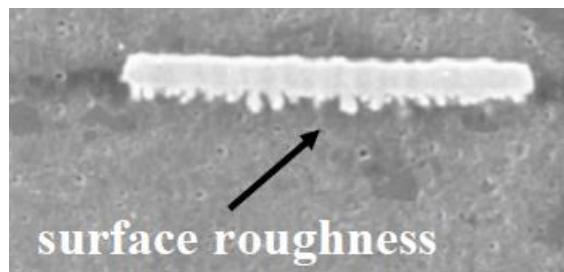


Figure 4-10 Cross section of microstrip line showing surface roughness at the conductor to dielectric interface (courtesy of Tektronix)

Edwards modified Equation (4-16) to include the effects of the surface roughness as given by Equation (4-17).

$$\alpha'_c = \alpha_c \left[1 + \frac{2}{\pi} \tan^{-1} \left(1.4 \left(\frac{\Delta}{\delta_s} \right)^2 \right) \right] \quad \text{dB/inch} \quad (4-17)$$

where, Δ is the r.m.s. surface roughness and δ_s is the conductor skin depth.

The corresponding Q factor related to the conductor is then given by:

$$Q_c = \frac{27.3\sqrt{\epsilon_{eff}}}{\alpha_c \lambda_o} \quad (4-18)$$

The dielectric loss is determined by the dielectric constant and loss tangent. It is calculated using Equation (4-19).

$$\alpha_d = 27.3 \frac{\epsilon_r (\epsilon_{eff} - 1) \tan \delta}{\sqrt{\epsilon_{eff}} (\epsilon_r - 1) \lambda_o} \text{ dB/inch} \quad (4-19)$$

where:

ϵ_r = the substrate dielectric constant

ϵ_{eff} = the effective dielectric constant

$\tan \delta$ = the loss tangent of the dielectric

λ_o = wavelength in inches

The corresponding Q factor due to the dielectric is then given by:

$$Q_d = 27.3 \frac{\sqrt{\epsilon_{eff}}}{\alpha_d \lambda_o} \quad (4-20)$$

We know that a microstrip line will also have some radiation of energy from the top side of the line. The open circuit stub will also experience some radiation effect from the open circuited end. On low dielectric constant substrates, $\epsilon_r \leq 4.0$, the radiation losses are more significant for high impedance lines. Conversely for high dielectric constant substrates, $\epsilon_r \geq 10$, low impedance lines experience more radiation loss [9]. The radiation Q factor is presented as Equation (4-21) [8].

$$Q_r = \frac{Z_o(f)}{480\pi \left(\frac{h}{\lambda_o} \right)^2 \left\{ \left(\frac{\epsilon_{eff}(f) + 1}{\epsilon_{eff}(f)} \right) - \left[\frac{(\epsilon_{eff}(f) - 1)^2}{2(\epsilon_{eff}(f))^{3/2}} \ln \left(\frac{\sqrt{\epsilon_{eff}(f)} + 1}{\sqrt{\epsilon_{eff}(f)} - 1} \right) \right] \right\}} \quad (4-21)$$

where, h = substrate thickness in cm.

Note that in Equation (4-21) the line impedance and effective dielectric constant are defined as functions of frequency. This includes the dispersion or frequency dependent effect of Z_o and ϵ_{eff} . Dispersion tends to slightly increase the ϵ_{eff} as the frequency increases. This dispersive Z_o and ϵ_{eff} are given in Equations (4-22) and (4-23).

$$\epsilon_{eff(f)} = \frac{\epsilon_r - \epsilon_{eff}}{1 + \left[(0.6 + 0.009Z_o) \left(\frac{f}{Z_o / 8\pi(h - 2t)} \right)^2 \right]} \quad (4-22)$$

where,

h = substrate thickness in mils
 t = conductor thickness in mils

$$Z_{o(f)} = Z_o \sqrt{\frac{\epsilon_{eff}}{\epsilon_{eff(f)}}} \quad (4-23)$$

Finally the resultant overall unloaded Q factor, Q_u , of the microstrip line can be determined by the reciprocal relationship of Equation (4.24).

$$\frac{1}{Q_u} = \frac{1}{Q_c} + \frac{1}{Q_d} + \frac{1}{Q_r} \quad (4-24)$$

4.5.2 Microstrip Resonator Example

Example 4.5-1: Consider a 5 GHz half wavelength open circuit microstrip resonator. The resonator is realized with a 50Ω microstrip line on Roger's RO3003 dielectric. Calculate the unloaded Q factor of the resonator. The substrate parameters are defined as:

Dielectric constant	$\epsilon_r = 3$
Substrate height	$h = 0.030$ in.
Conductor thickness	$t = .0026$ in.
Line Impedance	$Z_o = 50 \Omega$
Conductor width	$w = 0.075$ in.
Loss tangent	$\tan\delta = 0.0013$

Solution: Using the simplified expression of Equation (4-16) for a smooth microstrip line the loss and conductor Q factor is calculated as:

$$\alpha_c = 0.072 \frac{\sqrt{5}}{0.077(50)} \cdot 1.52 = 0.063 \text{ dB/inch}$$

$$Q_c = \frac{27.3\sqrt{2.41}}{(0.063)(2.36)} = 283.1$$

The dielectric loss and Q factor are then calculated from Equation (4-19) and (4-20).

$$\alpha_d = 27.3 \frac{(3)(2.41-1)(0.0013)}{\sqrt{2.41}(3-1)(2.36)} = 0.021 \text{ dB/inch}$$

$$Q_d = 27.3 \frac{\sqrt{2.41}}{(0.021)(2.36)} = 875.2$$

For simplicity the radiation Q factor will be omitted. We will model the resonator using the Genesys Linear simulator. Linear simulators often do not model the radiation effects of the microstrip line. Therefore the overall unloaded Q factors then becomes.

$$\frac{1}{Q_u} = \frac{1}{283.1} + \frac{1}{875.2} = \frac{1}{213.9}$$

$$Q_u = 213.9$$

4.5.3 Genesys Model of the Microstrip Resonator

The half wave open circuit microstrip resonator is modeled in Genesys as shown in Fig 4-11. Note that the source and load impedance has been increased to $5000\ \Omega$ to avoid loading the impedance of the parallel resonant circuit. Perform a linear sweep of the resonator using 4001 points from 4500 MHz to 5400 MHz.

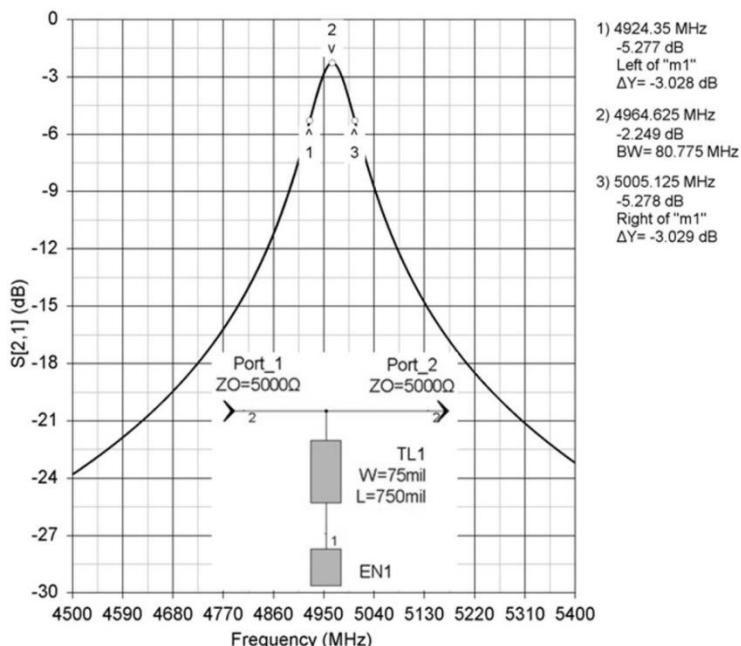


Figure 4-11 Half-wave open circuit microstrip resonator

Using the techniques of section 4.2.3 and Equation (4-10), the 3dB bandwidth is measured to determine the loaded Q of the resonator.

$$Q_L = \frac{4964.625}{80.775} = 61.5$$

The insertion loss at the resonant frequency can be used to relate the Q_L to the Q_u as shown by Equation (4-25). The Q_u as simulated by Genesys is within 20% of the value calculated using the substrate physical and electrical characteristics.

$$\text{InsertionLoss}(dB) = 20 \log \frac{Q_u}{Q_u - Q_L} \quad (4-25)$$

$$Q_u = \frac{\left(10^{\frac{IL(dB)}{20}}\right)Q_L}{\left(10^{\frac{IL(dB)}{20}}\right)-1} = \frac{\left(10^{2.249/20}\right)(61.5)}{\left(10^{2.249/20}\right)-1} = 269.5$$

It is also interesting to note that the Q_L of the resonator is related to the group delay through the two-port network.

$$Q_L = 2\pi f \left(\frac{t_d}{2} \right) \quad (4-26)$$

where:

t_d is the group delay in seconds

f is the frequency in Hertz

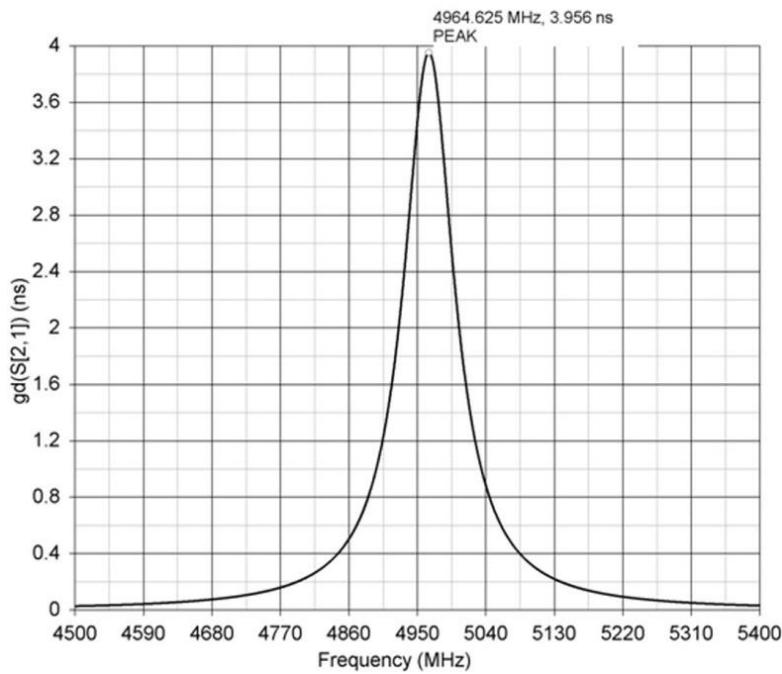


Figure 4-12 Group delay of the half-wave open circuit microstrip resonator

$$Q_L = 2\pi \left(4964.4 \cdot 10^6\right) \left(\frac{3.955 \cdot 10^{-9}}{2}\right) = 61.6$$

4.6 Resonator Series Reactance Coupling

To reduce the loading on the half wave resonator of Figure 4-11, the source and load impedances of 5000Ω were used in Genesys. In practice the resonator is typically coupled to lower impedance circuits. If we attempt to examine the resonator on a network analyzer, most modern test equipment will have 50Ω impedance levels. Such resonators are often coupled to the circuit by a highly reactive circuit element. This reactive element can be realized as a series capacitor or inductor. The resonator is then analyzed as a one port network. As the frequency is swept over a narrow frequency range around the resonant frequency, a circle is formed on the Smith Chart. This trace is known as the Q circle of the resonator [5]. Figure 4-13 shows the scalar plot and the Smith Chart plot of the resonator's input reflection coefficient, S_{11} . Three plots are shown each with a different value of coupling capacitance. We can see that as the coupling capacitance changes, the resonant frequency of the circuit also changes. The series capacitance acts to decrease the overall resonance frequency of the circuit. This new resonance frequency is known as the loaded resonance frequency, f_L . Because the series capacitance lowers the frequency, the length of the resonator was decreased to 0.727 inches to return the resonant frequency close to 5 GHz. With the coupling capacitance set at 0.0485 pF the Q circle passes through the center of the Smith Chart at the resonant frequency. This is known as critical coupling and is characterized by having the lowest return loss on the scalar plot of Figure 4-13. With the capacitance increased to 0.108 pF the resonator is more strongly coupled to the 50Ω load. The scalar plot shows that the resonance frequency is decreased. The Smith Chart shows a larger Q circle which is a characteristic of an over coupled resonator. With the coupling capacitor set to 0.025 pF the resonance frequency increases. The Smith Chart shows that the Q circle becomes much smaller thus under coupling the resonator. As Figure 4-13 shows, the value of the coupling capacitor also has an impact on the size of the Q

circle. The diameter of the Q circle is dependent on the coupling of the resonator to the $50\ \Omega$ source.

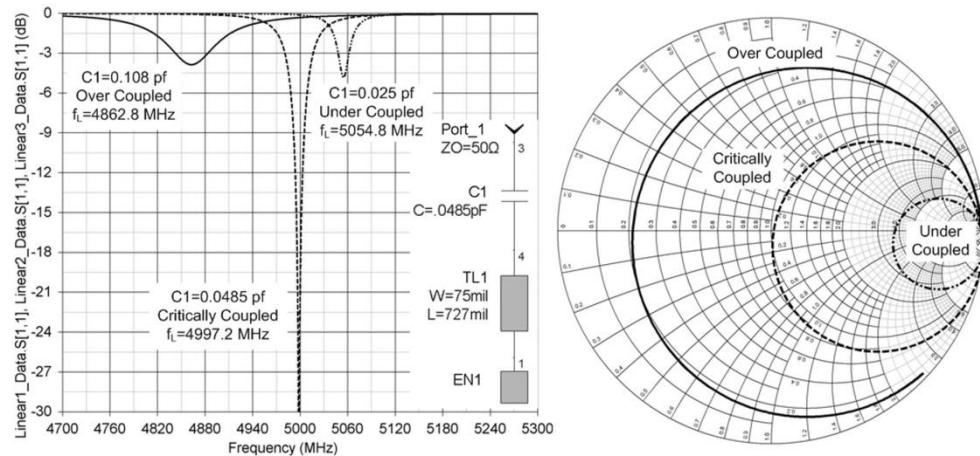


Figure 4-13 Capacitive coupled microstrip resonator and S11

4.6.1 One Port Microwave Resonator Analysis

The microstrip half wave resonator was fairly easy to model and analyze in Genesys. Many microwave resonators are not as easy to model. High Q microwave resonators are often realized as metallic cavities or dielectric resonators for which there are no native models in Genesys. The reactive coupling of the resonator to the circuit can be even more difficult to model. The coupling usually occurs by magnetic or electric coupling by a probe or loop inserted into the cavity. An E field probe coupled to a coaxial cavity resonator is shown in Figure 4-14. A dielectric resonator is coupled to a microstrip line by flux linkage in air as shown in Figure 4-15. Again there is no model in Genesys to directly model this coupling mechanism to the resonator. The designer is left to develop approximate models based on a lumped RLC equivalent models and couple the resonator to the circuit using an ideal transformer model. Linear simulation can still be of value in the design and evaluation process if we have a measured S parameter file of the resonator's reflection coefficient. Just as we have used S parameter models to represent capacitors and inductors we can also use the measured S parameters of a resonator. All modern vector network analyzers have the

ability to save an S parameter data file for any measurement that can be made by the instrument. This section will show how we can use Genesys to analyze the S parameter file of a microwave resonator.

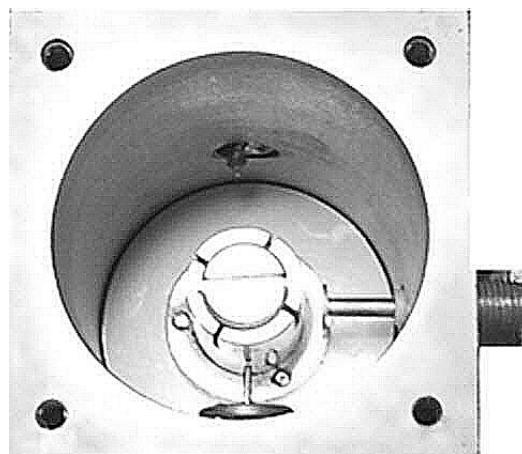


Figure 4-14 Coaxial cavity with E field probe coupled to center conductor
The coupling of microwave resonators is often characterized by a coupling coefficient k . The coupling coefficient is the ratio of the power dissipated in the load to the power dissipated in the resonator [5].

$$k = \frac{P_{load}}{P_{resonator}} = \frac{Q_o}{Q_{ext}} \quad (4-27)$$

where:

Q_o is the unloaded Q of the resonator
 Q_{ext} is the external Q of the resonator

When P_{load} is equal to $P_{resonator}$, $k = 1$ and the critical coupling case exists. Substituting the reciprocal Q factor relationship of Equation (4-10) into Equation (4-27) we can relate the coupling coefficient to the loaded Q_L and unloaded Q_o of the resonator.

$$Q_L = \frac{Q_o}{1+k} \quad (4-28)$$

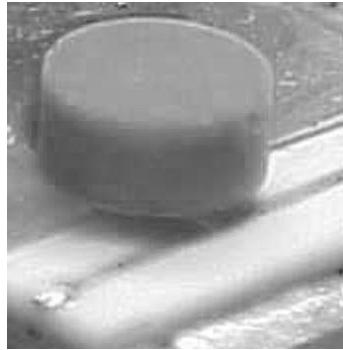


Figure 4-15 Dielectric resonator coupled to microstrip transmission line

Now the unloaded Q_o of the resonator can be calculated if the Q_L and k can be measured. Because the resonator is a one port device we cannot pass a signal through the device and measure the 3dB bandwidth as was done in section 4.5.3. Kajfez [5] has described a technique to extract the coupling coefficient k and Q_L values from the Q circle of the resonator. Consider the Q circle on the Smith Chart of Figure 4-16. A line that is projected from the center of the Smith Chart to intersect the Q circle with minimum length will intersect the circle at the loaded resonance frequency, f_L . The length of this vector is labeled as $|\Gamma_L|$. As the line projects along a path of the diameter of the circle it intersects the circle near the circumference of the Smith Chart at a point defined as $|\Gamma_d|$. The input reflection coefficient of the Q circle can be defined using the following empirical equation [5].

$$\Gamma_i = \Gamma_d \left[1 - \frac{2k}{1+k} \cdot \frac{1}{1 + j Q_L 2 \frac{\omega - \omega_L}{\omega_o}} \right] \quad (4-29)$$

Lines that are projected from Γ_d through the Q circle at the angles $\pm\phi$ are related to the loaded Q by Equation (4-30).

$$Q_L = \frac{f_L}{f_1 - f_2} \tan \phi \quad (4-30)$$

If we set $\phi = 45^\circ$ then Equation (4-30) reduces to the straightforward definition of Q_L given by (4-31).

$$Q_L = \frac{f_L}{f_1 - f_2} \quad (4-31)$$

In the previous section we saw that the diameter of the Q circle is directly related to the coupling coefficient. The diameter can be measured from:

$$|\Gamma_d| - |\Gamma_L| = d \quad (4-32)$$

The coupling coefficient is then derived from the diameter of the Q circle.

$$k = \frac{d}{2-d} \quad (4-33)$$

Finally the unloaded resonator Q_o is then calculated from Equation (4-28). We can also find the unloaded resonance frequency directly from the Q circle. Follow the reactive line on the Smith Chart that intersects the Q circle at Γ_d to the next Q circle intersection. The frequency at this Q circle intersection is the unloaded resonance frequency, f_o .

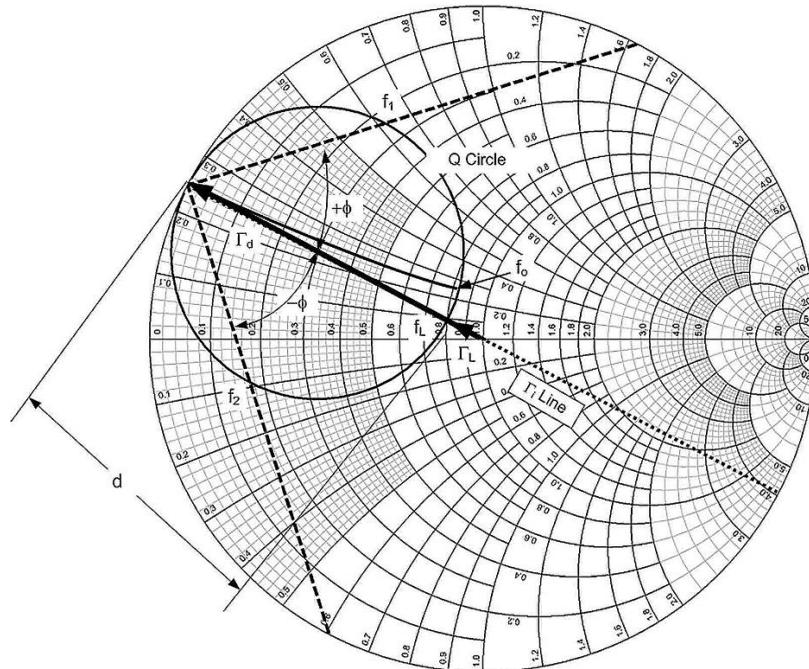


Figure 4-16 Resonator Q measurement from the resonator Q circle

4.6.2 Q_o Measurement of the Microstrip Resonator

Example 4.6-1: Use the Smith Chart technique to measure the Q_o of the half wave microstrip resonator of Figure 4-13. Solve for Q_o for all three coupling cases: under-coupled, over-coupled, and critically coupled cases.

Solution: The graphical technique requires that three overlays be placed on the Smith Chart.

- I. Γ_i Line
- II. Q_L Lines
- III. Ideal Q Circle

In the Genesys workspace a separate schematic and linear analysis is used to model a circuit that generates each of these overlays. The first schematic and analysis creates a line that passes through the center of the Smith Chart and extends to the circumference. A second schematic and analysis combination generates the line pair at an angle of $\pm 45^\circ$ from the Γ_i line. The third schematic produces an ideal Q circle. The ideal Q circle is overlaid on the S parameter data of the resonator under test. It helps to align the Γ_i line and Q_L lines. It is especially helpful when the measured data of a resonator may not form a full circle. The analysis that accompanies these designs can be simulated at any arbitrary frequency range. Only the schematic for the resonator under test needs to be swept over the actual measurement frequency range. The output from each Data Set can be plotted to the same Smith Chart so that the Γ_i line and Q_L lines are effectively an overlay on the Smith Chart. The schematics used to generate the Smith Chart overlays are given in Appendix B. Therefore the complete workspace is a collection of four schematics and linear analysis. The three overlays can be rotated around the chart by tuning the ‘angle’ variable. A ‘circle-diameter’ variable is used to vary the size of the ideal Q circle. The ‘coupling-loss’ variable moves the ideal Q circle toward the center of the chart as the resonator coupling loss increases. An Equation Editor is used to make these variables common among the three overlay schematics. The Equation Editor is also used to calculate the coupling coefficient (k), Q_L , and Q_o based on measured parameters on the Smith Chart. The procedure for using the Q measurement

overlays is summarized. Figure 4-17 shows the Genesys schematic of the microstrip resonator. The measurement process is summarized as:

1. Move the cursor over the Q circle to determine the minimum reflection coefficient, $|\Gamma_L|$. Place a marker on this point and enter the frequency, f_L , in the Equation Editor.
2. Adjust the “angle” of the Γ_i control using the slider control so that the Γ_i line intersects the $|\Gamma_L|$ marker. Note that the Q_L lines move along with the Γ_i line.
3. Iterate between the ideal Q circle diameter and coupling loss to get the best fit over the resonator’s Q circle.
4. Using the cursor measure the $|\Gamma_d|$ and $|\Gamma_L|$ and enter the values in the Equation Editor.
5. Using the cursor measure f_1 and f_2 at the intersection of the Q lines and the Q circle and enter their value in the Equation Editor.

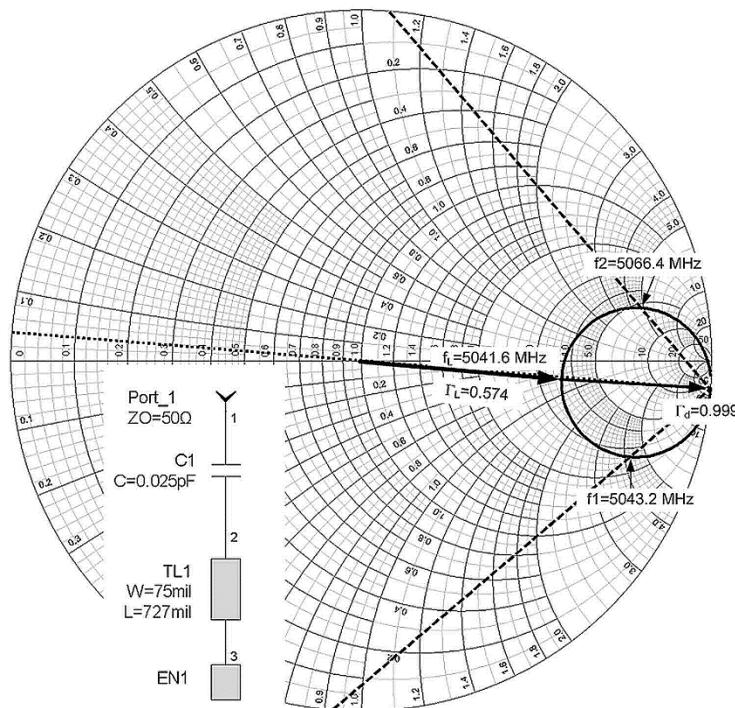


Figure 4-17 Under-coupled resonator Q circle and overlays

```

1  'Ideal Q Circle Overlay Parameters
2  angle=175.348
3  circlediameter=7440.36
4  couplingLoss=0.02
5  circleangle=-angle/2
6  'Calculation of Qo
7  'Enter measurement from Smith Chart
8  GammaD=0.999      'Enter magnitude of GammaD
9  GammaL=0.574      'Enter magnitude of GammaL
10 fL=5054.8
11 f1=5066.4
12 f2=5043.2
13 d=GammaD-GammaL  'Under and critically coupled
14 'd=GammaD+GammaL  'For over coupled
15 k=d/(2-d)
16 QL=fL/abs(f1-f2)
17 Qo=QL*(1+k)

```

Figure 4-18 Calculation of Q_L and Q_o for the under-coupled resonator

Repeat steps 1 through 5 for the critically coupled resonator of Figure 4-19.

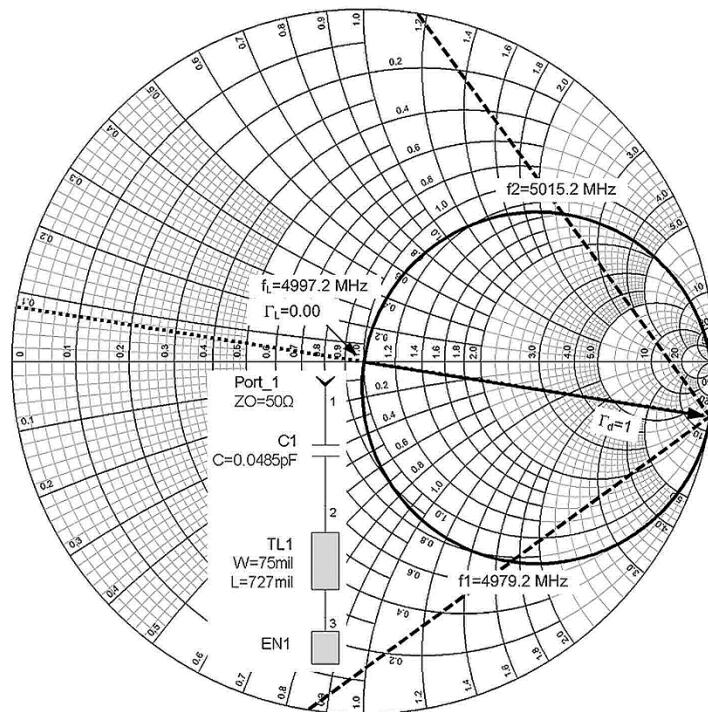


Figure 4-19 Critically-coupled Q circle and overlays for Q measurement

```

1  'Ideal Q Circle Overlay Parameters
2  angle=171.046
3  circlediameter=2025.53
4  couplingLoss=0.008
5  circleangle=-angle/2
6  'Calculation of Qo
7  'Enter measurement from Smith Chart
8  GammaD=1      'Enter magnitude of GammaD
9  GammaL=0      'Enter magnitude of GammaL
10 fL=4997.2
11 f1=5015.2
12 f2=4979.2
13 d=GammaD-GammaL    'Under and critically coupled
14 'd=GammaD+GammaL   'For over coupled
15 k=d/(2-d)
16 QL=fL/abs(f1-f2)
17 Qo=QL*(1+k)

```

Figure 4-20 Calculation of Q_L and Q_o for the critically-coupled resonator

Repeat steps 1 through 5 for the over coupled resonator of Figure 4-21.

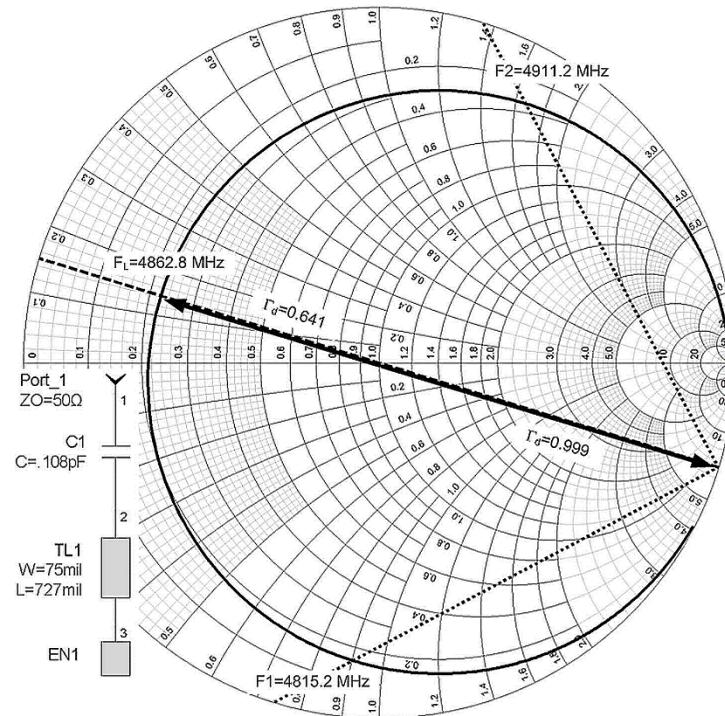


Figure 4-21 Over-coupled resonator Q circle and overlays

```

1   'Ideal Q Circle Overlay Parameters
2   angle=162.754
3   circlediameter=460.37
4   couplingLoss=0.008
5   circleangle==angle/2
6   'Calculation of Qo
7   'Enter measurement from Smith Chart
8   GammaD=0.999      'Enter magnitude of GammaD
9   GammaL=0.641      'Enter magnitude of GammaL
10  fL=4862.8
11  f1=4911.2
12  f2=4815.2
13  d=GammaD+GammaL  'Under and critically coupled
14  'd=GammaD+GammaL 'For over coupled
15  k=d/abs(2-d)
16  QL=fL/abs(f1-f2)
17  Qo=QL*(1+k)

```

Figure 4-22 Calculation of Q_L and Q_o for the over-coupled resonator

The three cases of the half wave microstrip resonator reveal the usefulness of the Smith Chart overlay technique for the measurement of reflection based one port resonator, Q_o . Even though the loaded Q 's were varied from 50.6 to 217.8 the unloaded Q_o was calculated to within 2% error.

Case	Coupling Coefficient, k	Measured Q_L	Calculated Q_o
Under Coupled	0.270	217.8	276.6
Critical Coupled	1.000	138.8	277.6
Over Coupled	4.556	50.6	281.4

Table 4-1 Comparison of Q_o calculation at various coupling coefficients

4.7 Filter Design at RF and Microwave Frequency

In Section 4.3 we have seen that it is possible to change the shape of the frequency response of a parallel resonant circuit by choosing different source and load impedance values. Likewise multiple resonators can be coupled to one another and to the source and load to achieve various frequency shaping responses. These frequency shaped networks are referred to as filters.

4.7.1 Filter Topology

The subject of filter design is a complex topic and the subject of many dedicated texts [2]. This section is intended to serve as a fundamental primer to this vast topic. It is also intended to set a foundation for successful filter design using the Genesys software. The four most popular filter types are: Low Pass, High Pass, Band Pass, and Band Stop. The basic transmission response of the filter types is shown in Figure 4-23. The filters allow RF energy to pass through their designed pass band. RF energy that is present outside of the pass band is reflected back toward the source and not transmitted to the load. The amount of energy present at the load is defined by the S21 response. The amount of energy reflected back to the source is characterized by the S11 response.

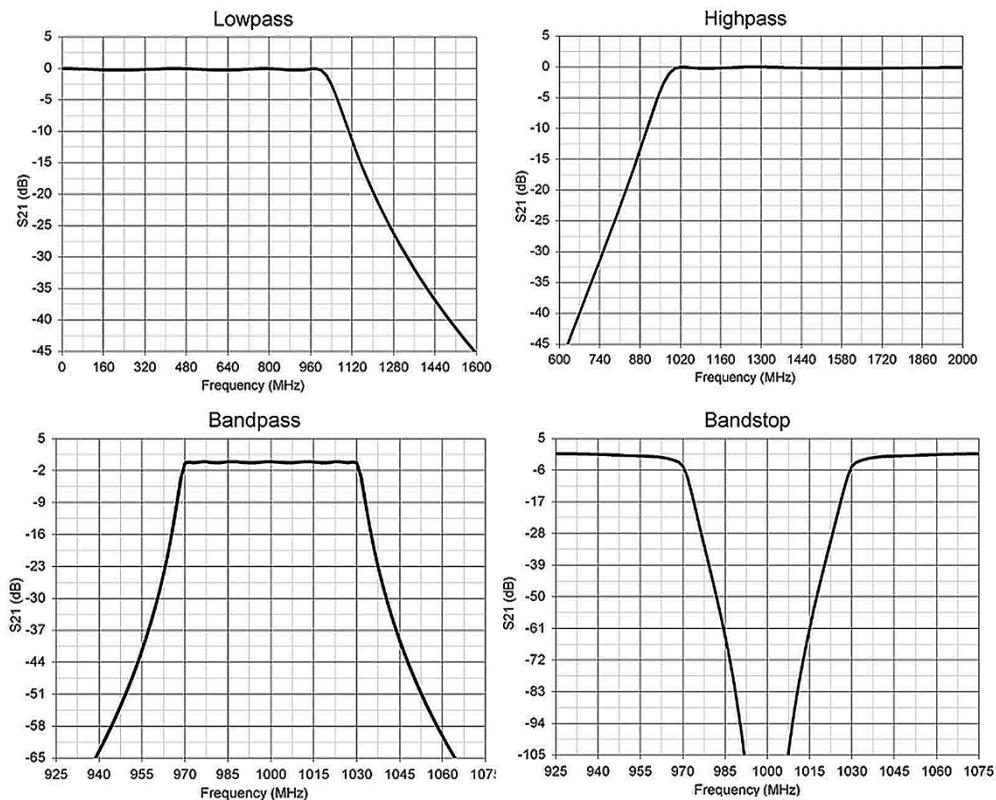


Figure 4-23 Transmission (S21) versus frequency characteristic for the basic filter types

4.7.2 Filter Order

The design process for all of the major filter types is based on determination of the filter pass band, and the attenuation in the reject band. The attenuation in the reject band that is required by a filter largely determines the slope needed in the transmission frequency response. The slope of the filter's response is related to the order of the filter. The steeper the slope or 'skirt' of the filter; the higher is the order. The term order comes from the mathematical transfer function that describes a particular filter. The highest power of s in the denominator of the filter's Laplace transfer function is the order of the filter. For the simple low pass and high pass filters presented in this chapter the filter order is the same as the number of elements in the filter. However this is not the case for general filter networks. In more complex types of lowpass and highpass filters as well as bandpass and bandstop filters the filter order will not be equal to the number of elements in the filter. In the general case the filter order is the total of the number of transmission zeros at frequencies:

- $F = 0$ (DC)
- $F = \infty$
- $0 < F < \infty$ (specific frequencies between DC and ∞)

Transmission zeros block the transfer of energy from the source to the load. In fact the order of a filter network can be solved visually by adding up the number of transmission zeros that satisfy the above criteria. Figure 4-24 shows the relationship between the filter order and slope of the response for a Low Pass filter. Each filter of Figure 4-24 has the same cutoff frequency of 1000 MHz. The third order filter has an attenuation of about 16 dB at a rejection frequency of 2000 MHz. The fifth order filter shows an attenuation of 39 dB and the seventh order filter has more than 61 dB attenuation at 2000 MHz. It is therefore clear that the order of the filter is one of the first criteria to be determined in the filter design. It is dependent on the cutoff frequency of the pass band and the amount of attenuation desired at the rejection frequency.

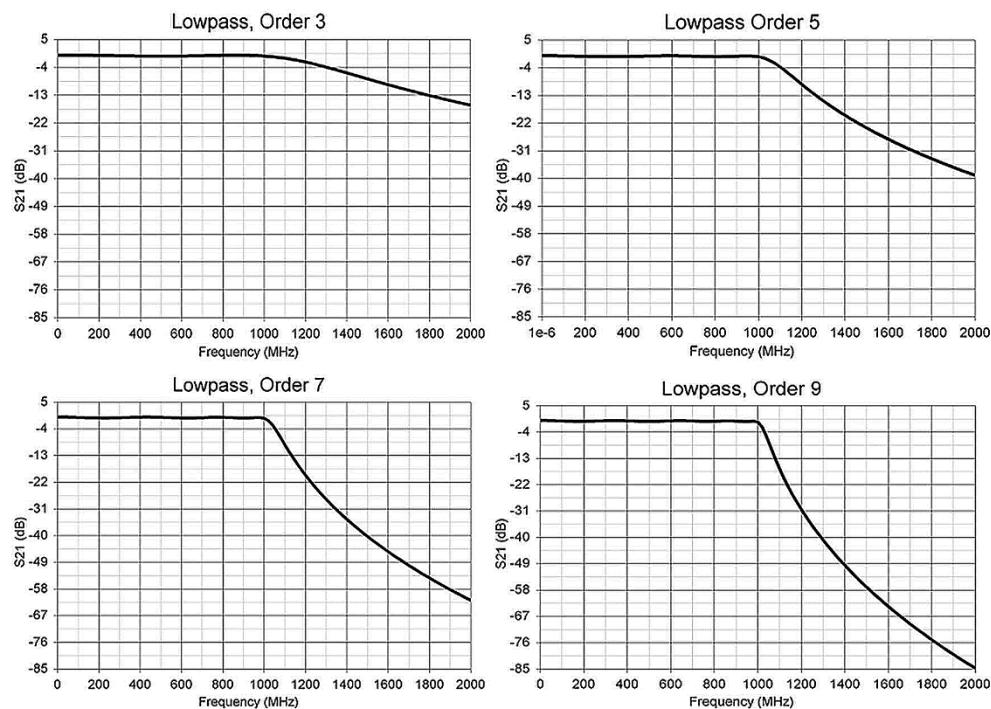


Figure 4-24 Relationship between filter order and the slope of S_{21}

4.7.3 Filter Type

The shape of the filter passband and attenuation skirt can take on different shape relationships based on the coupling among the various reactive elements in the filter. Over the years several polynomial expressions have been developed for these shape relationships. Named after their inventors, some of the more popular passive filter types include: Bessel, Butterworth, Chebyshev, and Cauer. Figure 4-25 shows the general shape relationship among these filter types for a given seventh order filter. The Bessel filter type is a low Q filter and does not exhibit a steep roll off compared to its counterparts. The benefit of the Bessel filter is its linear phase or flat group delay response. This means that the Bessel filter can pass wideband signals while introducing little distortion. The Butterworth is a medium Q filter that has the flattest pass band of the group. The Chebyshev response is a higher Q filter and has a noticeably steeper skirt moving toward the reject band.

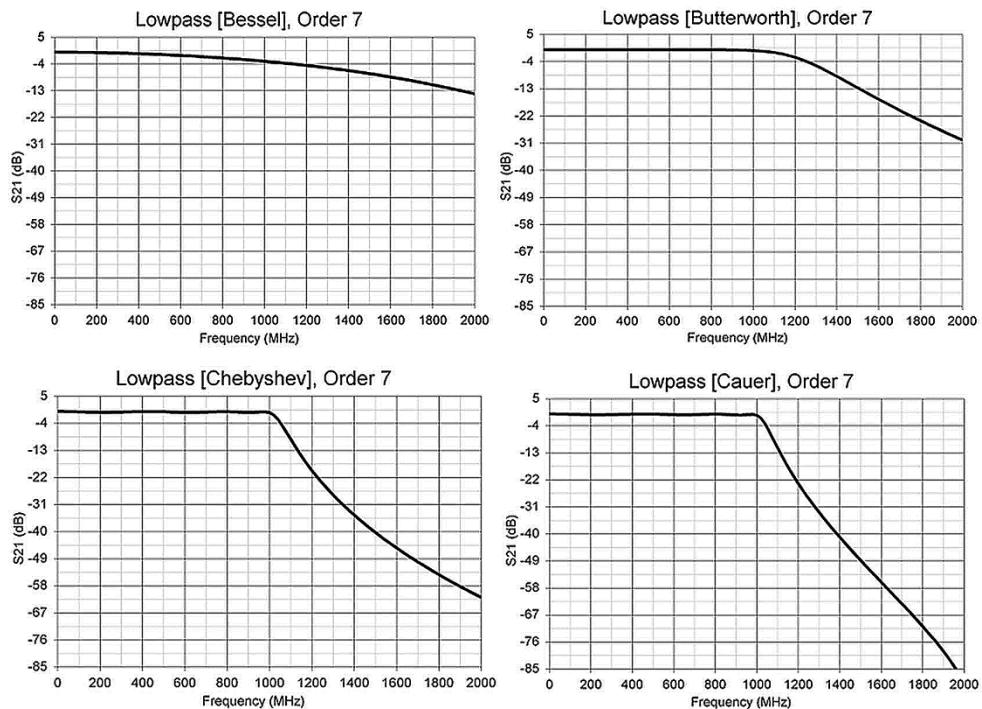


Figure 4-25 General characteristic shape of Bessel, Butterworth, Chebyshev, and Cauer filters

As a result it exhibits more transmission ripple in the pass band. The Cauer filter has the steepest slope of all of the four filter types. The Cauer filter is also known as an elliptic filter. Odd order Chebyshev and Cauer filters can be designed to have an equal source and load impedance. The even order Chebyshev and Cauer filters will have a different output impedance from the specified input impedance. Another interesting characteristic of the Cauer filter is that it has the same ripple in the rejection band as it has in the pass band. The Butterworth, Chebyshev and Cauer filters differ from the Bessel filter in their phase response. The phase response is very nonlinear across the pass band. This nonlinearity of the phase creates a varying group delay. The group delay introduces varying time delays to wideband signals which, in turn, can cause distortion to the signal. Group delay is simply the derivative, or slope, of the transmission phase and defined by Equation (4-34). Figure 4-26 shows the respective set of filter transmission characteristics with their corresponding group delay. Note the relative values of the group delay on the right hand axis.

$$\tau_g = -\frac{d\phi}{d\omega} \quad (4-34)$$

where, ϕ is the phase shift in radians and ω is in radians per second.

From the group delay plots of Figure 4-26 it is evident that the group delay peaks near the corner frequency of the filter response. The sharper cutoff characteristic results in greater group delay at the band edge.

4.7.4 Filter Return Loss and Passband Ripple

The Bessel and Butterworth filters have a smooth transition between their cutoff frequency and rejection frequency. The forward transmission, S21, is very flat vs. frequency. The Chebyshev and Cauer filters have a more abrupt transition between their cutoff and rejection frequencies. This makes these filter types very popular for many filter applications encountered in RF and microwave engineering. It is important to note however that the steeper filter skirt results as a certain amount of impedance mismatch between the source and load impedance.

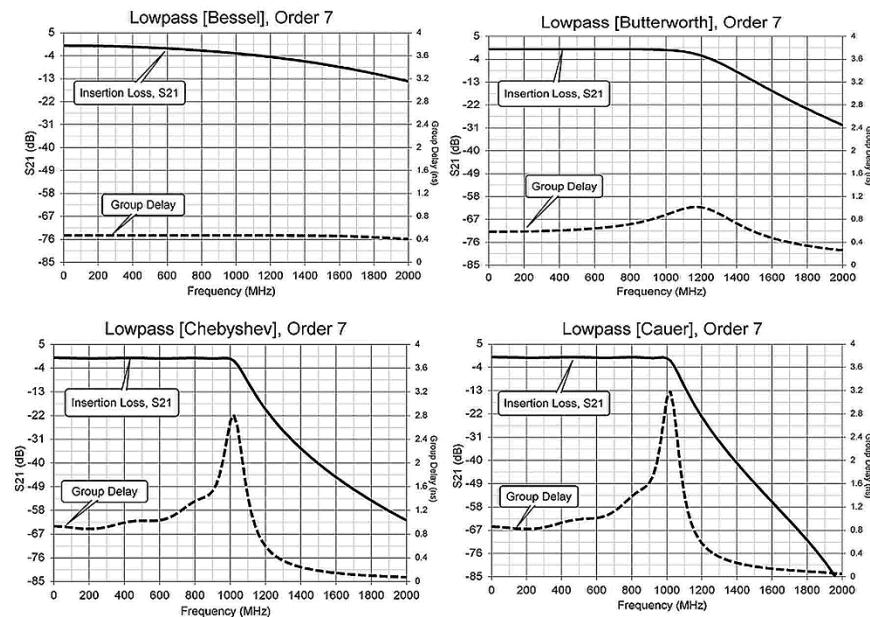


Figure 4-26 Group delay characteristic for various lowpass filter types

The Chebyshev and Cauer filter types have ripple in the forward transmission path, S21. The amount of ripple is caused by the degree of mismatch between the source and load impedance and thus the resulting return loss that is realized by these filter types. For a given Chebyshev or Cauer filter order, the roll off of the filter response is also steeper for greater values of passband ripple. The cutoff frequency of the filters that have passband ripple is then defined as the passband ripple value. For all-pole filters such as the Butterworth, the cutoff frequency is typically defined as the 3 dB rejection point. Figure 4-27 shows the passband ripple of a fifth order low pass filter for ripple values of 0.01, 0.1, 0.25, and 0.5 dB. Note that the ripple shown is produced by ideal circuit elements. In practice the finite unloaded Q or losses in the inductors and capacitors will tend to smooth out this ripple.

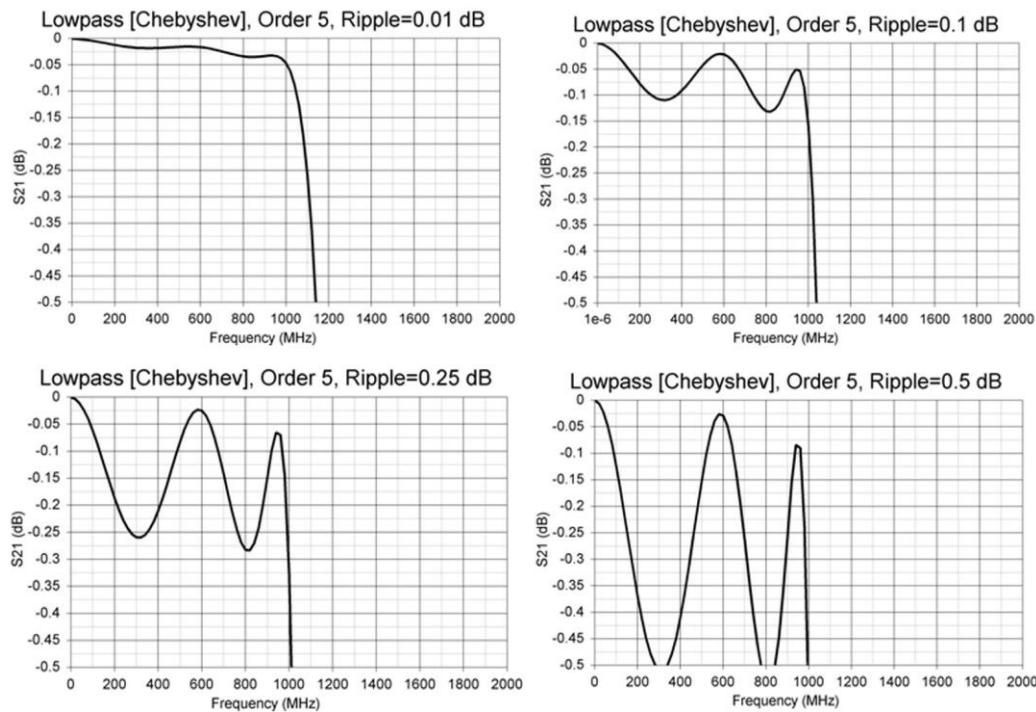


Figure 4-27 Passband ripple values in lowpass Chebyshev filter

In Chapter 2 the relationship for mismatch loss between a source and load was presented. For the Chebyshev and Cauer filters this mismatch loss is

the passband ripple. Figure 4-28 shows a solution of Equation (2-52) for selected values of VSWR.

```

1 vswr=[1.1;1.239;1.3;1.355;1.405;1.452;1.538;1.62;1.984]
2 Gamma=(vswr-1)/(vswr+1)
3 ReturnLoss_dB=20*log(Gamma)
4 MismatchLoss=-10*log(1-Gamma^2)
5 Ripple_dB=MismatchLoss

```

vswr ()	ReturnLoss_dB ()	Ripple_dB ()
1.1	26.444	0.01
1.239	19.433	0.05
1.3	17.692	0.075
1.355	16.435	0.1
1.405	15.473	0.125
1.452	14.688	0.15
1.538	13.474	0.2
1.62	12.518	0.25
1.984	9.636	0.5

Figure 4-28 Equation Editor Calculation of filter ripple versus VSWR

Figure 4-29 shows the same filters with the return loss plotted along with the insertion loss, S21. We can see that for a given filter order, there is a tradeoff between filter rejection and the amount of ripple, or return loss, that can be tolerated in the passband. In most RF and microwave filter designs the 0.01 and 0.1 dB ripple values tend to be more popular. This is due to the tradeoff between good impedance match and reasonable filter skirt slope. Figure 4-29 shows good correlation of the worst case return loss with that which is calculated in Figure 4-28. When tuning filters using modern network analyzers it is sometimes easier to see the larger changes in the return loss as opposed to the fine grain ripple as shown in Figure 4-27. For this reason it is common to tune the forward transmission of the filter by observing the level and response of the filter's return loss. Return loss is a very sensitive indicator of the filter's alignment and performance.

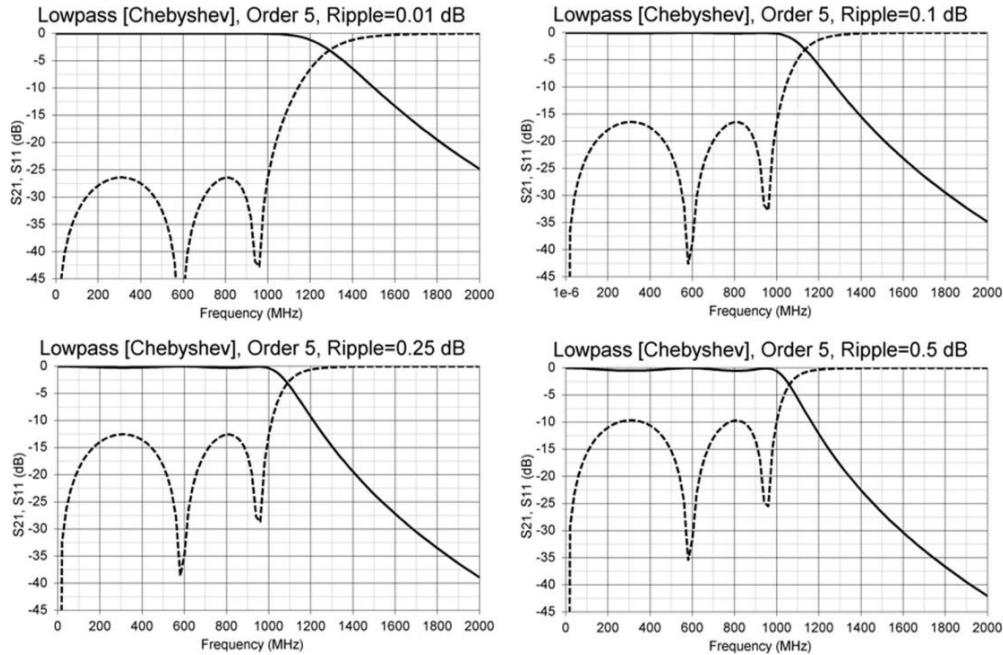


Figure 4-29 Lowpass Chebyshev filter rejection and return loss

4.8 Lumped Element Filter Design

Classical filter design is based on extracting a prototype frequency-normalized model from a myriad of tables for every filter type and order [1–2]. Fortunately these tables have been built into many filter synthesis software applications that are readily available. In this book we will examine the filter synthesis tool that is built into the Genesys software. We will work through two practical filter examples, one low pass and one high pass using the Genesys filter synthesis tool.

4.8.1 Low Pass Filter Design Example

Example 4.8-1: As a practical filter design, consider a full duplex communication link (simultaneous reception and transmission) through a satellite with the following requirements:

- The uplink signal is around 145 MHz while the downlink is at 435 MHz.

- A 20 W power amplifier is used on the uplink with 25 dB gain.
- It is necessary to provide a low pass filter on the uplink (only pass the 145 MHz uplink signal while rejecting any noise power in the 435 MHz band).
- It is necessary to provide a high pass filter on the downlink so that the 435 MHz downlink signal is received while rejecting any noise power at 145 MHz.

The transmitter and receiver antennas are on the same physical support boom so there is limited isolation between the transmitter and receiver. Even though the signals are at different frequencies, the broadband noise amplified by the power amplifier at 435 MHz will be received by the UHF antenna and sent to the sensitive receiver. Because the receiver is trying to detect very low signal levels, the received noise from the amplifier will interfere or ‘de-sense’ the received signals. Therefore it is necessary to design a 145 MHz Low Pass filter for this satellite link system. The specifications chosen for the filter design are selected as:

- Select a Chebyshev Response with 0.1 dB pass band ripple.
- Set the passband cutoff frequency (not the -3 dB frequency) at 160 MHz
- The reject requirement is at least -40 dB rejection at 435 MHz.

Solution: Use the Passive Filter synthesis utility to design the Low Pass filter. Select a Low Pass filter of the Chebyshev type. On the Topology tab select a Lowpass filter type with a Chebyshev shape. Select the minimum capacitor subtype. On the Settings tab enter the cutoff frequency of 160 MHz and pass band ripple of 0.1 dB. Also set the cutoff frequency attenuation at 0.1 dB. The Filter Settings Tab is a great place to perform “what-if” analysis. The pass band ripple, filter order, cutoff frequency, and attenuation at cutoff can all be varied while observing their impact on the filter’s characteristic. For the design example enter the parameters as shown in Fig. 4-30. The required filter order can be determined by increasing the order until the specification of -40 dBc attenuation at 435 MHz is achieved. As the filter response curve in Fig.4-30 shows, this Low Pass filter must be of fifth order to achieve the required attenuation. Along with the resulting

attenuation and return loss the synthesis program creates the filter schematic with the synthesized component values.

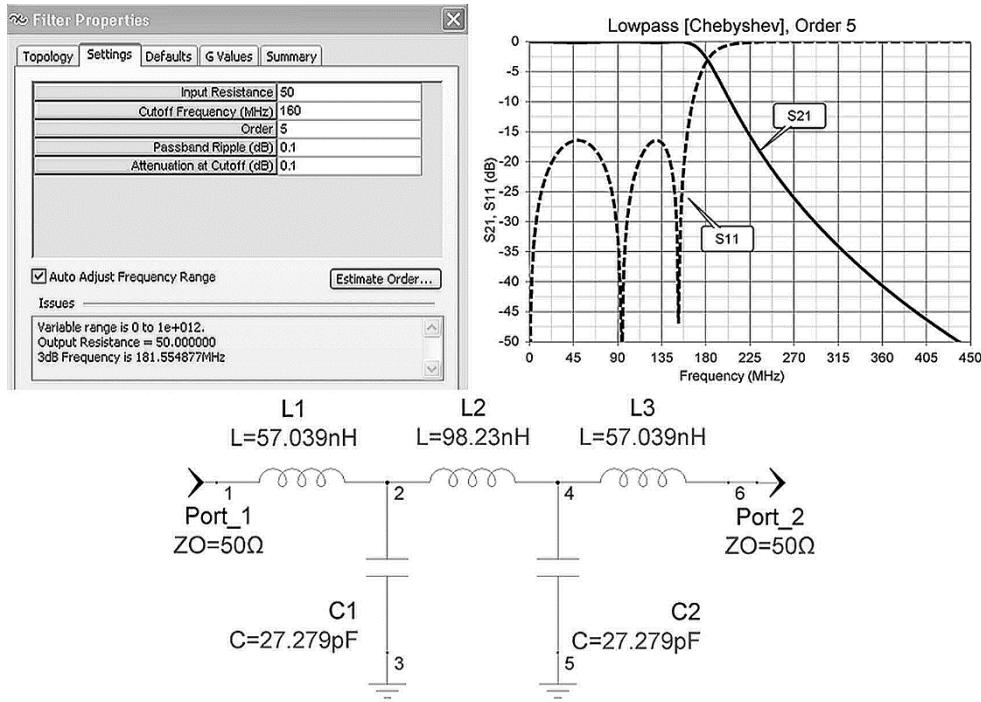


Figure 4-30 Passive filter synthesis utility: topology and settings tab

4.8.2 Physical Model of the Low Pass Filter in Genesys

It is important to realize that the synthesized filter is an ideal design in the sense that ideal (no parasitics and near infinite Q) components have been used. To obtain a good ‘real-world’ simulation of the filter we need to use component models that have finite Q and parasitics such as multilayer chip capacitors for shunt capacitors. For power handling capability, choose the 700 series chip capacitors from ATC Corporation. We will use measured S parameter files to model the shunt capacitors. The measured S parameters will account for any package parasitic effects and the finite component Q factor. Most microwave chip capacitor manufacturers will supply S parameters for their products. ATC Corporation has a useful application for

selection of chip capacitors called ATC Tech Select. This program is available for free download from the ATC web site: www.atceramics.com. Using the Tech Select program the engineer can access complete data sheets and other useful information including current and voltage handling capabilities of the various capacitors. Because the filter is passing relatively high power (20 W), we cannot use small surface mount style chip inductors. Instead we will use air wound coils to realize the series inductors. The inductors will be realized with AWG#16 wire nickel-tin plated copper wire. The wire has a diameter of 0.05 inches or 50 mils. They will be wound on a 0.141 inch diameter form. Use the techniques covered in Chapter one, Section 1.4.1 to design the inductors using the Air Wound inductor model in Genesys. The filter model is then reconstructed using the S parameter files for the shunt capacitors and the physical inductor models for the series inductors. Make sure to model the substrate and the interconnecting printed circuit board traces as microstrip lines. Also model the ground connection of the shunt capacitors as a microstrip via hole. Although these PCB parasitic effects are normally more pronounced at frequencies above 2 GHz, it is often surprising the effect that these parasitics have at lower frequencies. The final filter response and model is shown in Figure 4-32. The response shows that the attenuation specification has been achieved. Because the circuit has physical models replacing the ideal lumped element components, the engineer can be confident that the filter can be assembled and will achieve the designed response. Figure 4-31 is a photo of the prototype low pass filter circuit with SMA coaxial connectors attached to the circuit board.

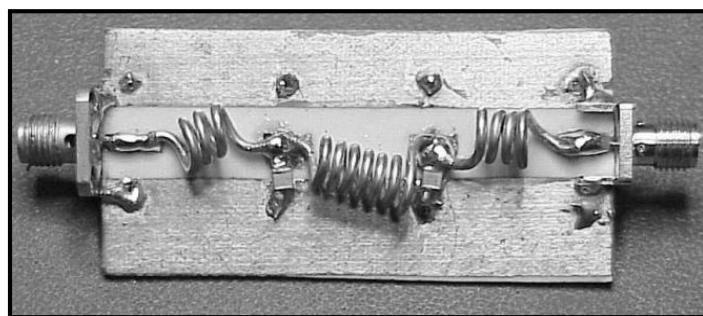


Figure 4-31 Physical prototype of the 146 MHz low pass filter

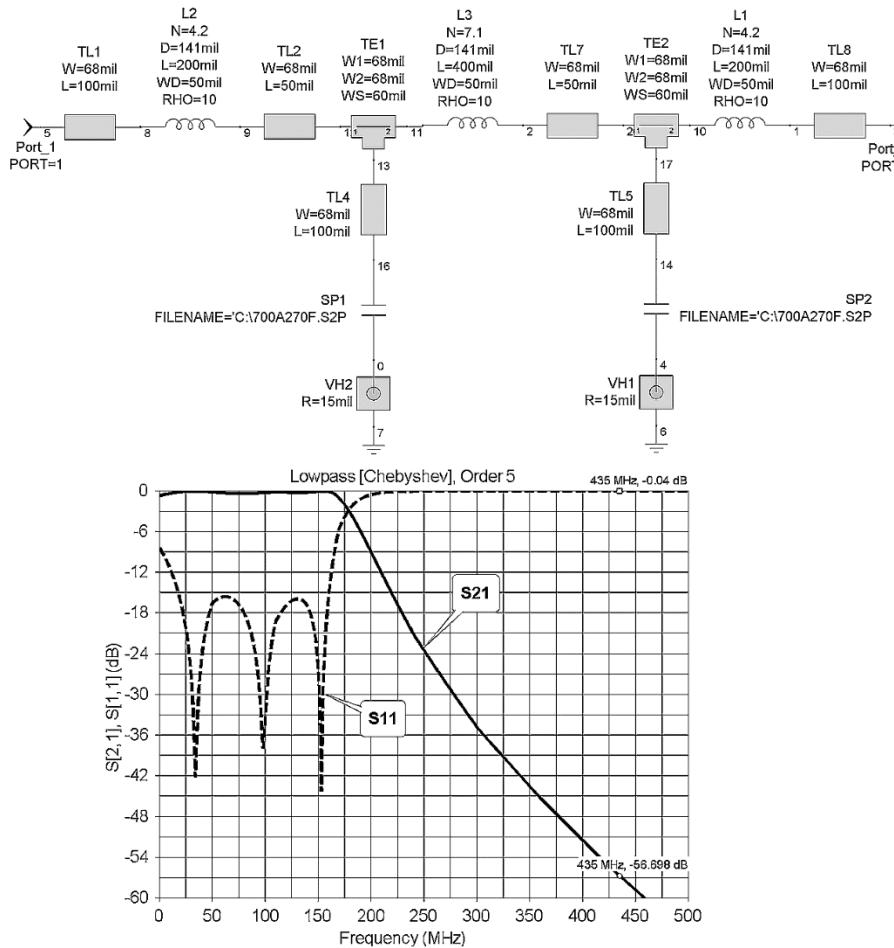


Figure 4-32 Low pass filter model with physical elements

4.8.3 High Pass Filter Design Example

Example 4.8-2: Design a high pass filter that passes frequencies in the 420 MHz to 450 MHz range. This filter could be placed in front of the preamp used in the downlink of the satellite system. This would help to keep out any of the transmit energy or noise power in the 146 MHz transmit frequency range. The High Pass Filter specifications are:

- The pass band cutoff frequency (not the -3 dB frequency) is 420 MHz.
- The filter has a Chebyshev response with 0.1 dB pass band ripple.
- The reject requirement is at least -60 dB rejection at 146 MHz.

Solution: Using the Passive Filter Synthesis tool in Genesys, vary the filter order until sufficient attenuation is achieved at 146 MHz. It is always a good practice to design for some additional rejection (margin) that exceeds the minimum requirement. The Filter synthesis session recommends a filter of 7th order.

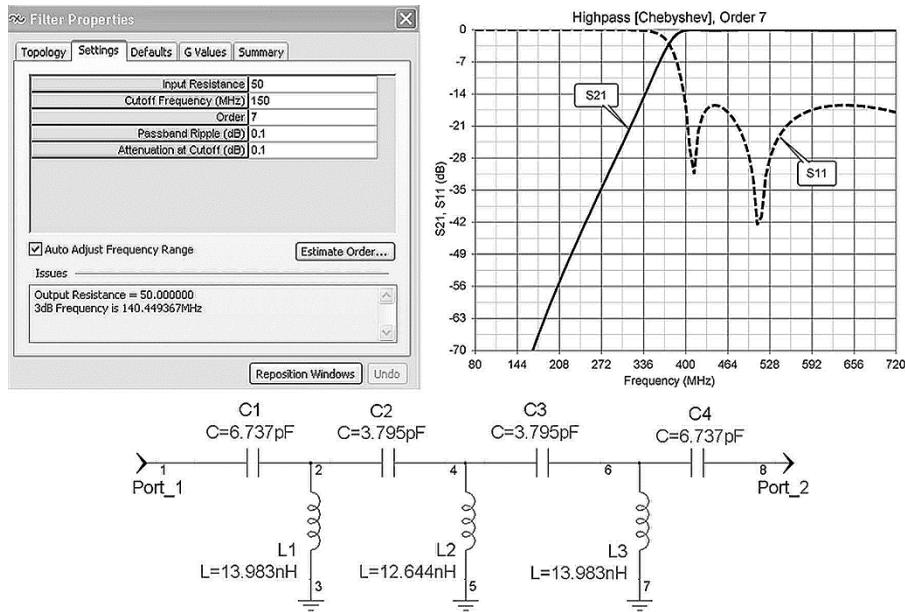


Figure 4-33 Passive filter synthesis model of the high pass filter

4.8.4 Physical Model of the High Pass Filter in Genesys

Using the same techniques as described for the Low Pass Filter design we can proceed with the High Pass Filter realization. The Passive Filter Synthesis application calculated series capacitance values of 6.7 pF and 3.8 pF. Looking through the available ATC 700 series chip capacitors, the nearest values are 6.8 pF and 3.9 pF capacitors. We will select the S parameters files for these chip capacitors to use in the final filter model. The shunt inductors are realized using the Air Wound inductor model. Figure 4-34 shows the final circuit model and filter response. From the response we can see that the required rejection specification of $S21 < -60$ dBc has been maintained. Figure 4-35 shows the completed assembly of the High Pass Filter on a printed circuit board with coaxial SMA connectors.

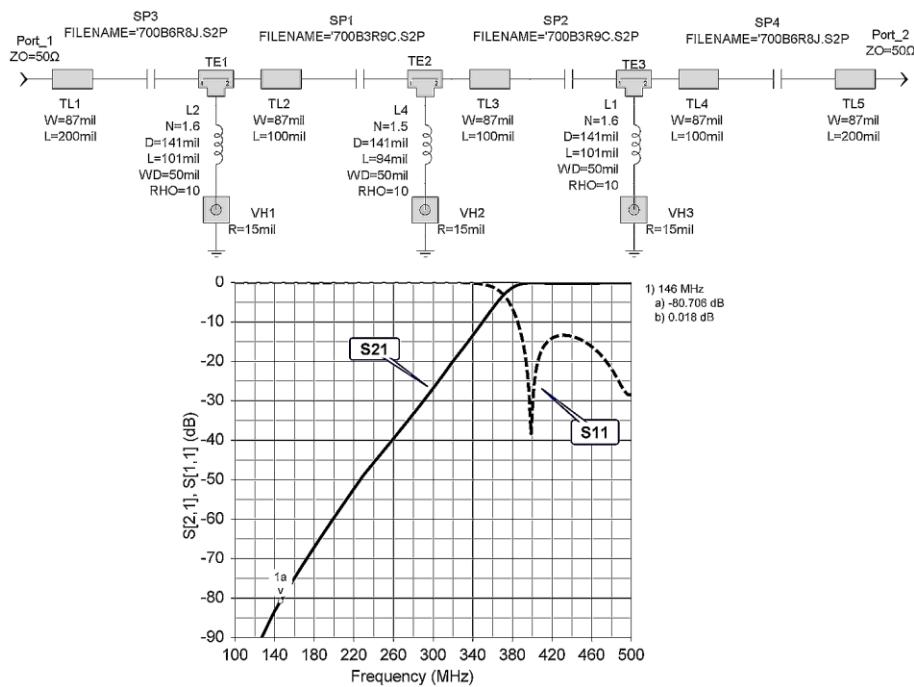


Figure 4-34 High pass filter model with physical elements

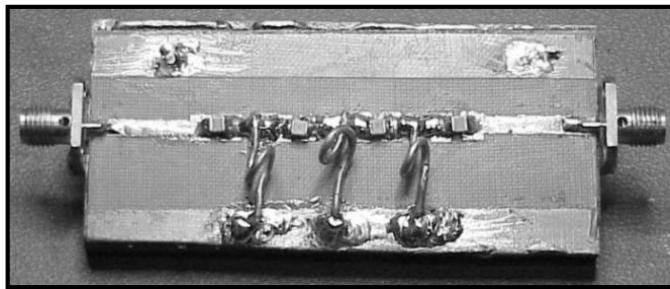


Figure 4-35 Physical prototype of the 420 MHz high pass filter

4.8.5 Tuning the High Pass Filter Response

In many cases we would like to tune the component values to change or tweak the filter's characteristic to fit the desired response. The shunt inductors in the high pass filter are very easy to tune because they are modeled with the native inductor model from the Genesys library. To tune

the inductors simply check the tune box on the properties tab of the parameter to be tuned. Enable tuning of the inductor length of all three inductors. Using the Tune Window the length of each inductor can be increased or decreased by the step size or percentage selected. Each time the value is changed the analysis is run and the graph is updated. However the capacitors cannot be tuned because their physical model is based on an S parameter file that describes the capacitor's physical model. To change to another capacitor's S parameter file, we must edit the data file and browse to select a new S parameter file. Then we must sweep the circuit to observe the new response. This process involves several steps and we lose the 'real time' sense of tuning the capacitor values and seeing the response change quickly.

4.8.6 S Parameter File Tuning with VBScript

It is helpful to use measured S parameter files of inductors and capacitors to improve the accuracy and manufacturability of filter designs. The designer should be cautious however because a manufacturer's S parameter data file may not contain enough points to be used in a high Q circuit design. There can also be some variation in a component's S parameters based on the type of substrate and the techniques in which the S parameters are measured. In these cases it may be necessary to perform one's own measurement of component S parameters rather than rely on the manufacturer. Another issue in filter design is that the S parameters are typically measured in $50\ \Omega$ systems. When combined with other highly reactive S parameter files numerical instabilities may result. One of the inconveniences of S parameter files is that it is difficult to perform real time component tuning. A new S parameter file must be loaded into the schematic and the analysis is re-simulated every time that a new component is selected. One technique around this problem is to create a method of real time tuning of S parameter files using a VBScript application. Genesys has the capability to incorporate Visual Basic Script, VBScript, files to perform various operations on the simulator. We can take advantage of this capability to create an application within the Workspace that performs the required tasks of selecting S parameter files from disk, performing the linear sweep, and plotting the results on the graph. This allows us to create a real-time tuning tool using

individual S parameter files from disk. The VBScript capability within Genesys allows all of the standard text based visual basic programming operations. To understand how to interact with objects within the Genesys Workspace use the VBBrower.exe application that is located in the Scripting folder in the Genesys Examples section. This application will show the proper syntax for accessing and setting the values and methods of the various objects in the Genesys workspace. A new schematic of the High Pass Filter that has been modified for tuning the S parameter files of the capacitors is shown in Figure 4-36.

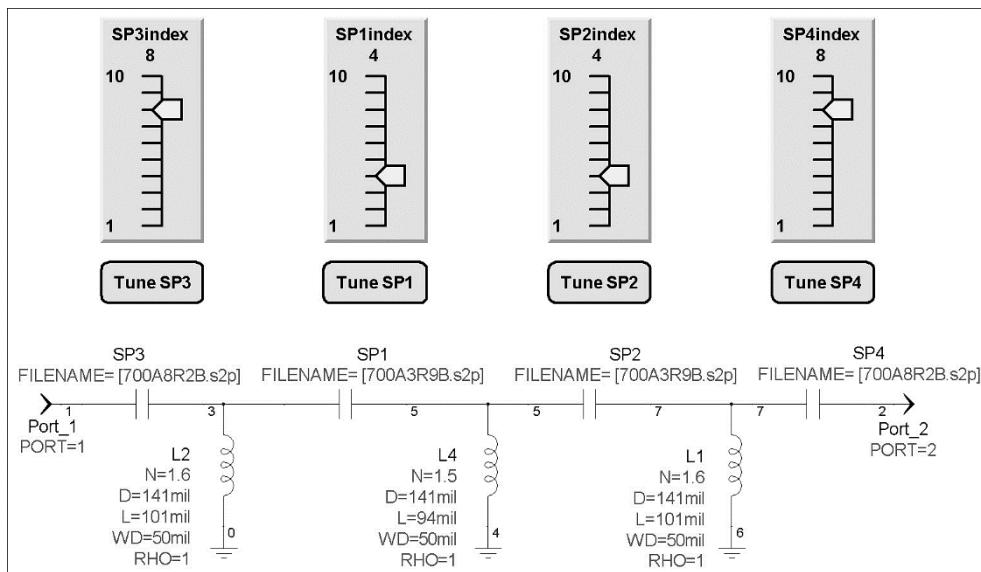


Figure 4-36 High pass filter schematic with slider controls and command buttons for tuning S parameter files

The VBScript code listing for the S parameter Tuner is shown in Figure 4-37. Lines 1 through 3 define the objects and variables used in the program. Lines 5 through 14 contain the S parameter files to be used in the tuning process. You can copy as many S parameter files as you desire to compare in your design. In this example we have chosen ATC700 chip capacitors from 2.2 pF to 12 pF. Make sure to include the full filename and path to the location of your S parameter files. This program listing enables real time tuning by selecting among ten different chip capacitor S parameter files.

The index variable, i, is used by the program to select the appropriate file from the list of S parameters.

```

1  Dim WsDoc
2  Dim My_Tuner, My_Analysis
3  Dim name(10), i, number, pos
4  '....Copy S-Parameter Path & Filename here.....
5  name(1)="C:\700A2R2C.s2p"
6  name(2)="C:\700A2R7C.s2p"
7  name(3)="C:\700A3R3B.s2p"
8  name(4)="C:\700A3R9B.s2p"
9  name(5)="C:\700A4R7C.s2p"
10 name(6)="C:\700A5R6C.s2p"
11 name(7)="C:\700A6R8J.s2p"
12 name(8)="C:\700A8R2B.s2p"
13 name(9)="C:\700A100K.s2p"
14 name(10)="C:\700A120F.s2p"
15 '
16
17 WsDoc= theApp.GetWorkspaceByIndex(0)      'define the workspace object
18 My_Analysis=WsDoc.Designs.Linear1
19 My_Analysis.ClearModelCache           'clear the memory
20 'Get index value from Equation block
21 i = WsDoc.Equation.VarBlock.SPIndex.GetValue()
22 'Load the S-Parameter file from disk
23 My_Tuner=WsDoc.Designs.Sch1.PartList.SP1.ParamSet.FILENAME.SetValue(name(i))
24 My_Tuner=WsDoc.Designs.Sch1.PartList.SP1.ParamSet.FILENAME.Set(name(i))
25 My_Analysis.ClearModelCache           'clear the memory
26 My_Analysis.RunAnalysis            'simulate the circuit
27
28 Graph=WsDoc.Designs.Graph1      'select the graph
29 Graph.OpenWindow                'Display Graph

```

Figure 4-37 VBScript code listing for S parameter file tuner

Slider controls are placed on the schematic as a means of selecting from the list of S parameter files. Variables in the equation block can be assigned to the Slider Controls. Equation block variables can also be input to the VBScript code. Therefore the Equation block is useful for exchanging variables between the program code and objects on the schematic. Define an index variable for each of the four S parameter data file elements, as shown in Figure 4-38. Make each variable tunable by placing a {=?} before the numerical value. Enter any integer value for each variable. Line 21 reads the index value from the Equation block and assigns it to the variable, i. Line 24 then uses this variable as an index to select the appropriate S parameter file to load from disk file and assign it to the S parameter data element.

```

1     SP1index=?4
2     SP2index=?4
3     SP3index=?7
4     SP4index=?7

```

Figure 4-38 Index variables for each S parameter file, SP1-SP4

Lines 25 to 29 then execute the linear simulation and graph the response. The VBScript program code can then be copied into a Command Button object so that the code can be executed from the Genesys schematic. After placing a Command Button on the schematic expose the Properties tab and copy and paste the code from the VBScript editor to the command button. Figure 4-36 shows the schematic with a Slider Control and Command button for each S parameter file. Make sure to edit lines 21 and 24 in each Command Button to change the variable name, SP1index, and the file reference, SP1, because these are unique for each S parameter files. Figure 4-39 shows the VBScript code listing inside the Properties Tab of the Command Button.

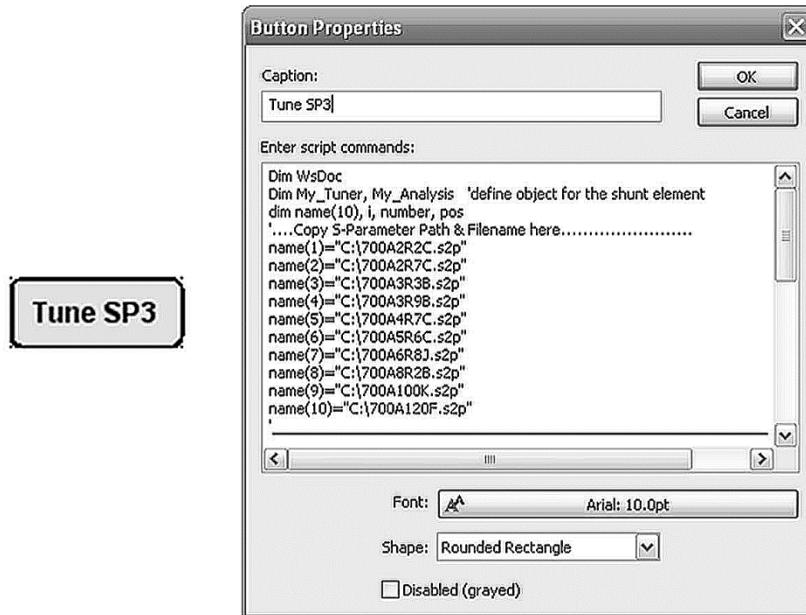


Figure 4-39 Property tab for the tune button command showing VBScript

The Slider Control Properties Tab is also shown in Figure 4-40. Using the drop-down list, select the Equation variable to assign to the Slider. Set the “Max” limit to the number of S parameter files that you have entered into the VBScript code listing. Uncheck the ‘Run simulation’ box because we will use the Command Button to run the simulation. Then select ‘Snap to Integer values’ so that the Slider Control can choose the appropriate S parameter file. Once the Slider Control has been moved, press the Command Button to sweep the circuit and plot the response. Now we can tune the S parameter files of the capacitors just as easily as we can tune the inductor model parameters.

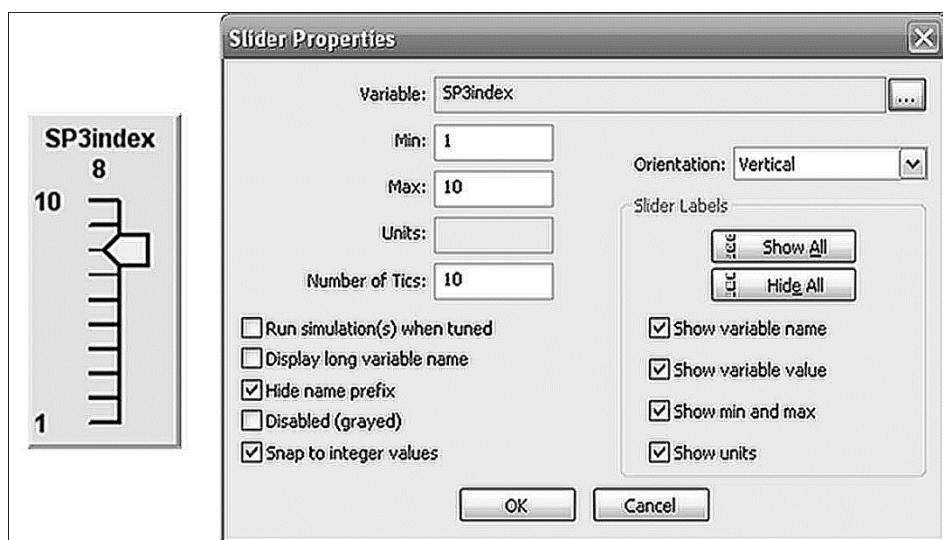


Figure 4-40 Slider control Tab for S parameter file selection

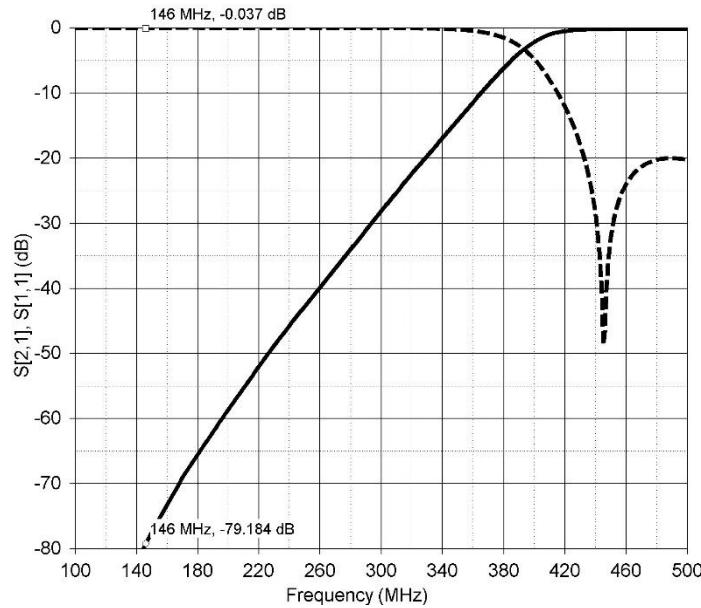


Figure 4-41 High pass filter response with tuned S parameter files

4.9 Distributed Filter Design

4.9.1 Microstrip Stepped Impedance Low Pass Filter Design

In the microwave frequency region filters can be designed using distributed transmission lines. Series inductors and shunt capacitors can be realized with microstrip transmission lines. In the next section we will explore the conversion of a lumped element low pass filter to a design that is realized entirely in microstrip.

4.9.2 Lumped Element to Distributed Element Conversion

Example 4.9-1: Consider the lumped element 2 GHz low pass filter schematic and response shown in Figure 4-42. This low pass filter has a 3 dB bandwidth of approximately 2470 MHz. Use the microstrip equivalent models of series inductance and shunt capacitance to realize the filter in

microstrip. The microstrip substrate is Rogers's 6010 material with a 0.025 dielectric thickness.

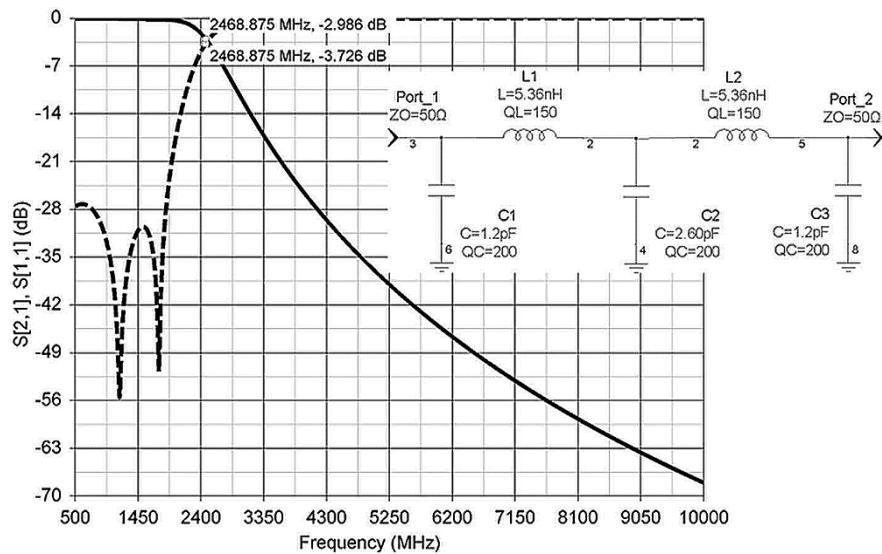
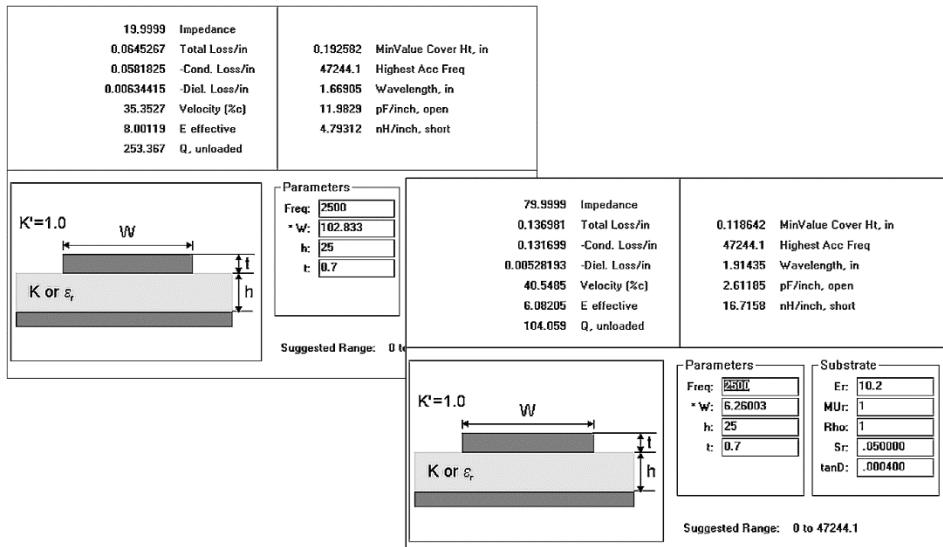


Figure 4-42 Lumped element 2.2 GHz low pass filter and frequency response

Solution: The series inductors will be realized as 80Ω transmission lines of sufficient length to act as a 5.36 nH inductor. The TLINE Transmission Line synthesis program is used to calculate the microstrip line width for an 80Ω transmission line on the Rogers 0.025 in. RO3010 material. As Figure 4-43 shows the 80Ω transmission line has a line width of 6.26 mils with an effective dielectric constant $\epsilon_r = 6.08$. To realize the required inductance value a specific length of 80Ω transmission line is required. Equation (2-69) is solved in the Equation Editor of Figure 4-44 to calculate the length of microstrip line that is required to realize the inductance values used in the filter. The length of line required for a 5.36 nH inductor is then found to be 321 mils. The shunt capacitors will be realized as 20Ω transmission lines. Using TLINE the 20Ω line width is calculated to be 102.8 mils with an effective dielectric constant $\epsilon_r = 8.00$. Equation (2-70) is solved in the Equation Editor of Figure 4-45 to find the length of 20Ω transmission line required to behave as 2.6 pF shunt capacitor is 217 mils. The line length for the 1.2 pF capacitors is then found to be 100 mils.

Figure 4-43 TLINE calculations for 20Ω and 80Ω microstrip lines

```

1 %Enter design parameters
2 F1=2.5e9
3 Lambda1=3e8/F1           %free space wavelength
4 erEFF1=6.08               %effective dielectric constant
5 LambdaG1=Lambda1/sqrt(erEFF1) %wavelength in meters
6 LambdaG1=LambdaG1/.0254      %wavelength in inches
7 L=5.36e-9
8 Z1=80
9 %Microstrip Line Length is:
10 Length1=F1*LambdaG1*L/Z1

```

Figure 4-44 Calculating the length of inductive microstrip line

```

1 %Enter frequency, imedgance and the effective dielectric constant
2 F=2.5e9
3 Z=20
4 erEFF=8.00
5 %Enter the desired shunt Capacitance
6 C=2.6e-12
7 %Calculate the following....
8 Lambda=3E8/F   %free space wavelength in meters
9 LambdaG=Lambda/sqrt(erEFF)/.0254 %wavelength in meters
10 %Microstrip Line Length is:
11 Length=LambdaG*F*Z*C %inches

```

Figure 4-45 Calculating the length of capacitive microstrip line

The length of the microstrip lines are 0.321 and 0.217 inches, respectively. Adding a short 50Ω section to the input and output, the initial low pass filter schematic and response is shown in Figure 4-46.

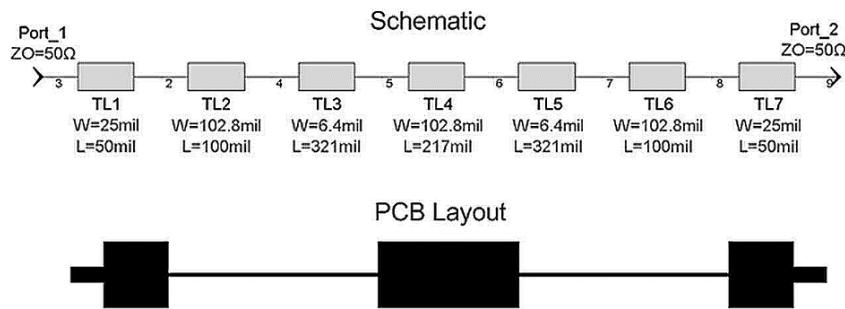


Figure 4-46 Initial schematic and PCB layout of the low pass filter

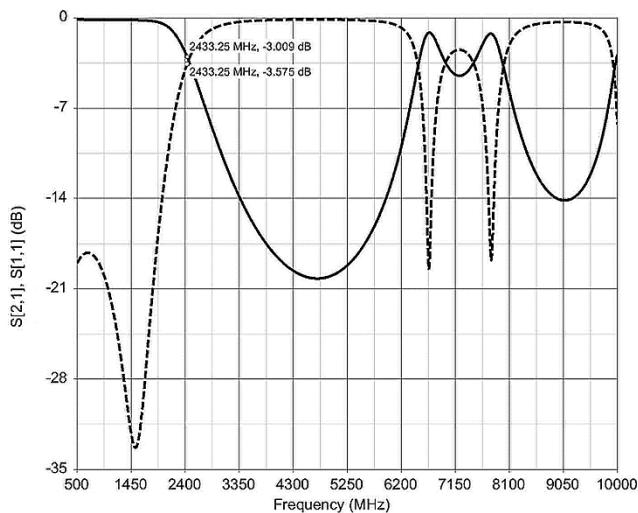


Figure 4-47 Initial Schematic and Response of the Low Pass Filter

Examine the printed circuit board, PCB, layout of the Low Pass Filter of Figure 4-46. Note the change in geometry as the impedance transitions from 50Ω to 20Ω and from 20Ω to 80Ω . These abrupt changes in geometry are known as discontinuities. Discontinuities in geometry result in fringing capacitance and parasitic inductance that will modify the frequency response of the circuit. At RF and lower microwave frequencies (up to

about 2 GHz) the effects of discontinuities are minimal and sometimes neglected [4]. As the operation frequency increases, the effects of discontinuities can significantly alter the performance of a microstrip circuit. Genesys has several model elements that can help to account for the effects of discontinuities. These include: T-junctions, cross junctions, open circuit end effects, coupling gaps, and bends. A Microstrip Step element can be placed between series lines of abruptly changing geometry to account for the step discontinuity. Place the Microstrip Step element at each impedance transition in the filter. Make sure that the narrow side and wide side are directed appropriately. The Microstrip Step element will automatically use the adjacent width in its calculation. Figure 4-48 shows the low pass filter schematic with the step elements added between transmission line sections.

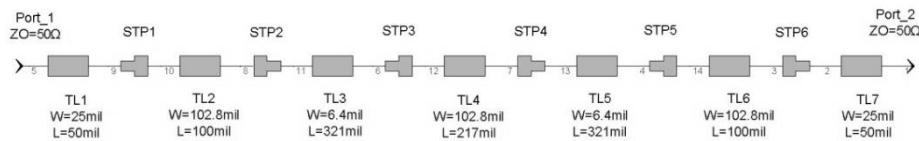


Figure 4-48 Stepped impedance filter with added “step” elements

A comparison of the initial low pass filter model and the modified model is shown in Figure 4-49. As the Figure shows there is a slight difference in the filter insertion loss, S21, particularly as the frequency increases from the cutoff at 2400 MHz.

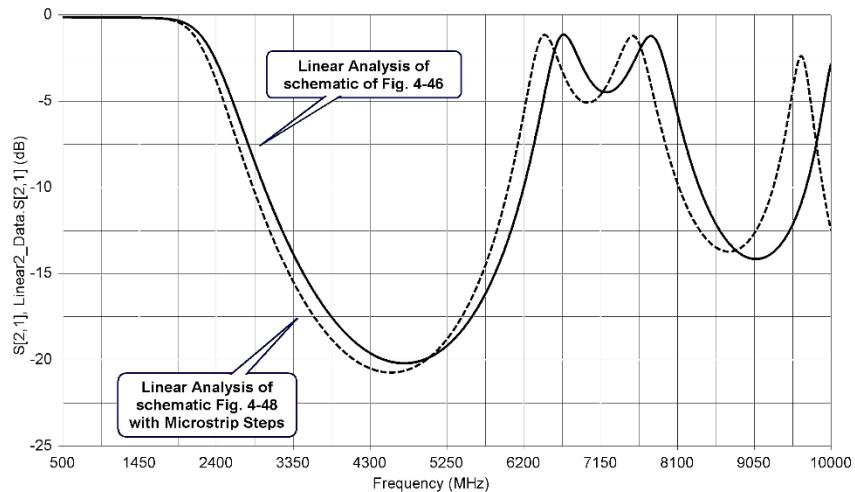


Figure 4-49 Filter cutoff frequency shift due to step discontinuities

4.9.3 Electromagnetic Modeling of the Stepped Impedance Filter

Electromagnetic, EM, modeling is a useful tool in microstrip circuit design as it offers a means of potentially more accurate simulation than linear modeling. The linear microstrip component models used to model the stepped impedance filter are based on closed form expressions developed over many years. For many designs the linear model is quite acceptable. Microstrip circuits that contain several distributed components in a dense printed circuit layout will be affected by cross coupling and enclosure effects. This is because the microstrip circuitry is quasi-TEM with some portion of the EM fields in the free-space above the dielectric material. These effects are very difficult to accurately simulate with linear modeling techniques. The Genesys software suite has a very useful electromagnetic (EM) simulation engine named Momentum. Momentum is based on the method-of-moments (MoM) numerical solution of Maxwell's equations [3]. Unlike some EM simulation software Momentum's solutions are presented in the S parameter format that is familiar to the microwave circuit designer. A dataset is created that can be graphed just like any linear simulation. The Momentum model is created from the circuit layout rather than the schematic. In this section we will create a Momentum model from the layout that was created by the linear schematic. However we could import an arbitrary PCB artwork from any CAD program. Right click on the Layout window to expose the Layout Properties. On the General Tab make sure to specify the correct units (mils) that represent the drawing. Also check the 'show EM Box' check box so that a proper enclosure is modeled for the circuit. The box represents a metal enclosure that will serve as the boundary conditions for the EM simulation. The simulator will identify any box resonances that may occur which could have an adverse effect on the circuit design. The box sides must be lined up perpendicular to the input and output ports. The box size (length and width) can be adjusted by entering the desired dimensions in the Box Width (X) and Box Height (Y) settings as shown in Figure 4-50. The box height is specified in the Layers tab as the air above the metal conductor or 250 mils. On the Layer Tab make sure that the microstrip dielectric material is defined on the Substrate

line. Once all of the Layer settings have been specified add the Momentum Analysis to the Workspace as shown in Figure 4-52.

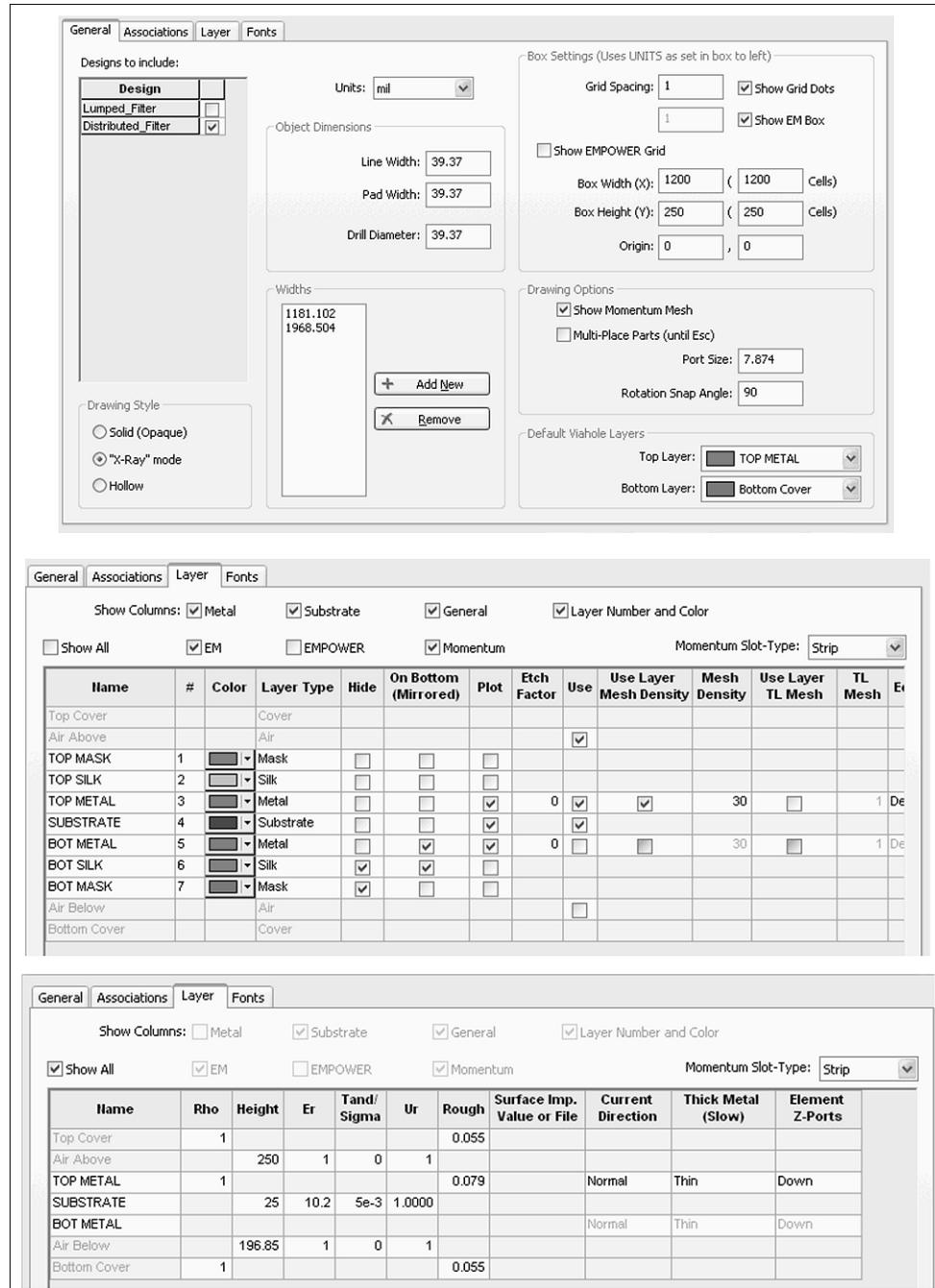


Figure 4-50 General and layer tabs of the layout properties

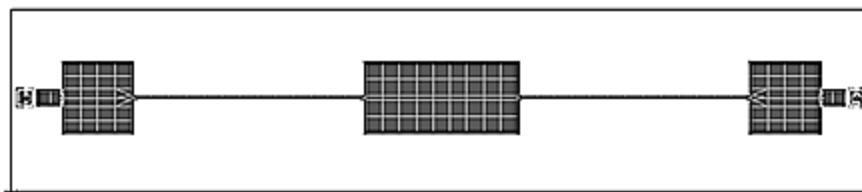


Figure 4-51 Filter layout showing box outline and conductor mesh

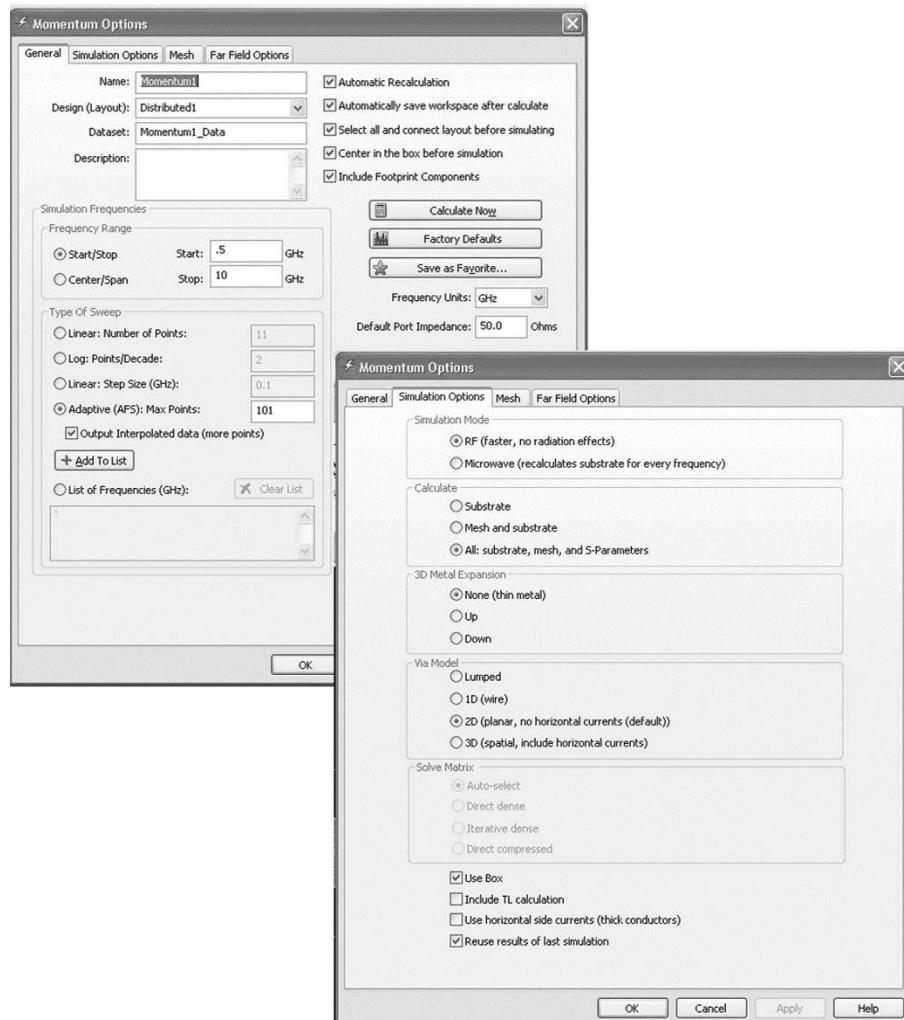


Figure 4-52 Momentum simulation options setup

On the General Tab, set the start and stop frequency for the simulation and select the adaptive sweep type. The adaptive sweep reduces frequency point interpolation error. On the Simulation Options Tab choose the RF

simulation mode. The RF simulation mode is a much faster EM simulation and is suitable for lower microwave (RF) frequencies where there is not a significant amount of coupling among transmission lines. The Microwave simulation mode is a full wave EM analysis that includes all coupling radiation within the box. Also check the ‘Calculate – All’ button so that the metal mesh and the substrate are used in the solution.

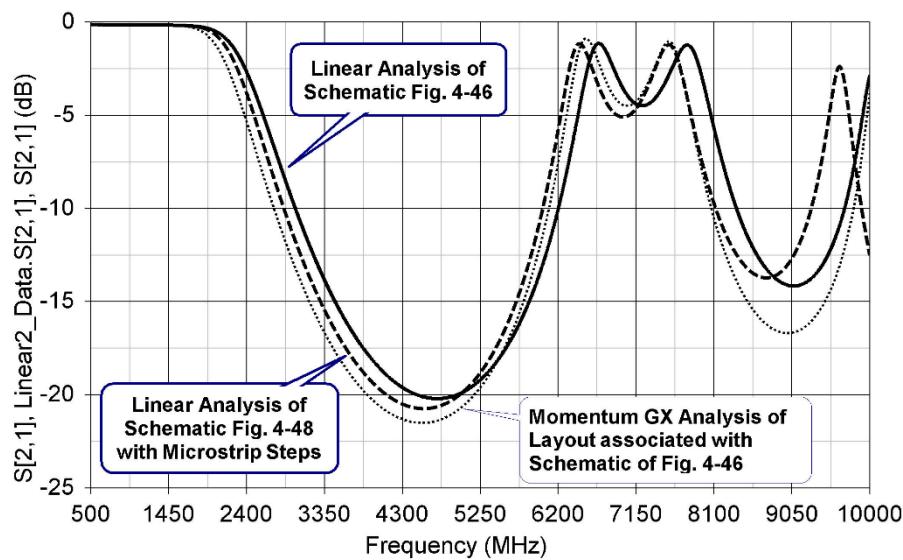


Figure 4-53 Linear simulation and momentum simulation comparison

The comparison between the Linear and Momentum simulations shows that there is some further deviation in the filter rejection as the frequency increases above 2 GHz.

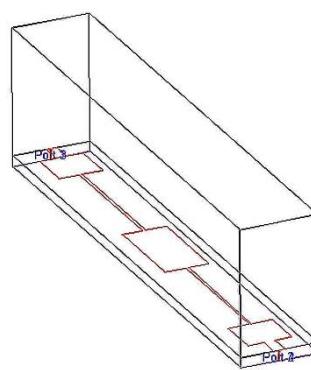


Figure 4-54 3D View of filter and prototype filter printed circuit board

4.9.4 Reentrant Modes

In Chapter 3 section 3.7 it was demonstrated that distributed transmission lines have repetitive impedance characteristics which are dependent on the physical length and frequency. In planar distributed filter design this leads to the creation of reentrant responses in the filter's passband characteristic. Reentrant modes can be seen in the filter response of Figure 4-53 at frequencies near 6667 MHz, 7826 MHz and 10 GHz. Depending on the filter design goals these modes may be harmless. If the distributed low pass filter's goal is to reject frequencies near 5 GHz only, then the circuit can be used as designed and the reentrant modes are of no consequence. If however the filter is required to have > 20 dB rejection of all frequencies from 5 GHz through 10 GHz then the reentrant modes present a problem as these frequencies will be passed by the filter.

Example 4.9-2: One technique to remove reentrant modes from a filter response is to cascade the filter with a 'clean-up' filter. For the filter design of Figure 4-53 the 'clean-up' filter is a stepped impedance low pass filter with a higher cutoff frequency. Check the reentrant response of the filter.

Solution: Figure 4-55 shows a low pass filter with a 4 GHz cutoff frequency in cascade with the 2 GHz filter.

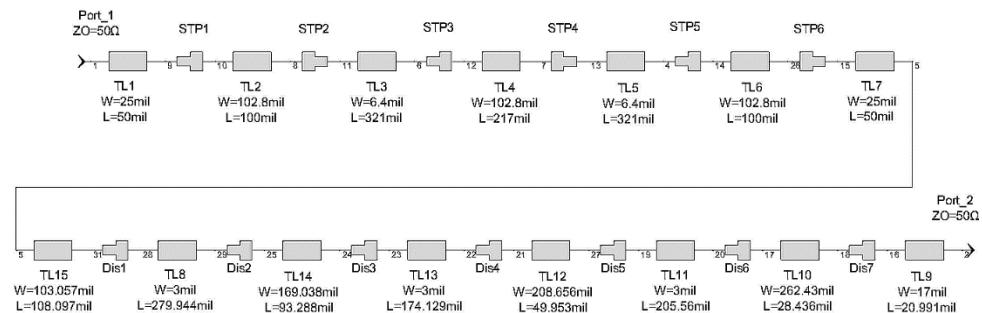


Figure 4-55 Cascaded stepped impedance low pass filter schematic

The response of the cascaded filters is shown in Figure 4-56. Compared to the response of Figure 4-53, the reentrant responses have been reduced

greater than 20 dB. The filter now has a very nice ultimate band rejection characteristic through 10 GHz.

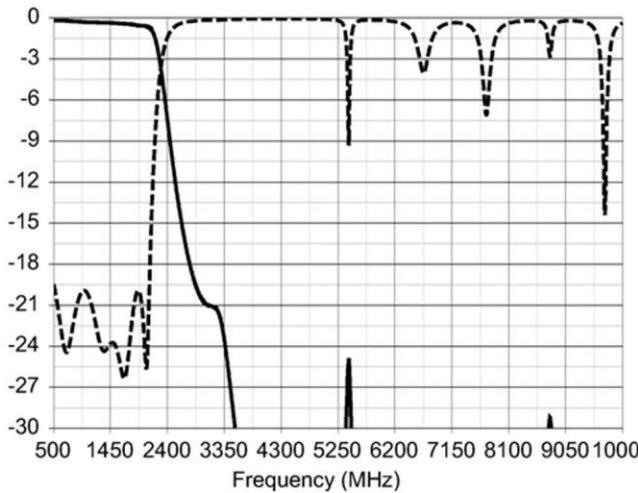


Figure 4-56 Cascaded stepped impedance low pass filter response

4.9.5 Microstrip Coupled Line Filter Design

The edge coupled microstrip line is very popular in the design of band pass filters. A cascade of half-wave resonators in which quarter wave sections are parallel edge coupled lines, are very useful for realizing narrow band, band pass filters. This type of filter can typically achieve $\leq 15\%$ fractional bandwidths [4].

Example 4.9-3: Design a band pass filter at 10.5 GHz. The filter is designed on RO3010 substrate ($\epsilon_r = 10.2$) with a dielectric thickness of 0.025 inches. The filter should have a pass band of 9.98 – 11.03 GHz. As a design goal the filter should achieve at least 20 dB rejection at 9.65 GHz. In other words the filter is required to have > 20 dB rejection at 330 MHz below the lower passband frequency.

Solution: The Microwave Filter synthesis utility in Genesys is used to design the filter network. Figure 4-57 shows the entry of the design parameters into the Topology, Settings, and Options tabs.

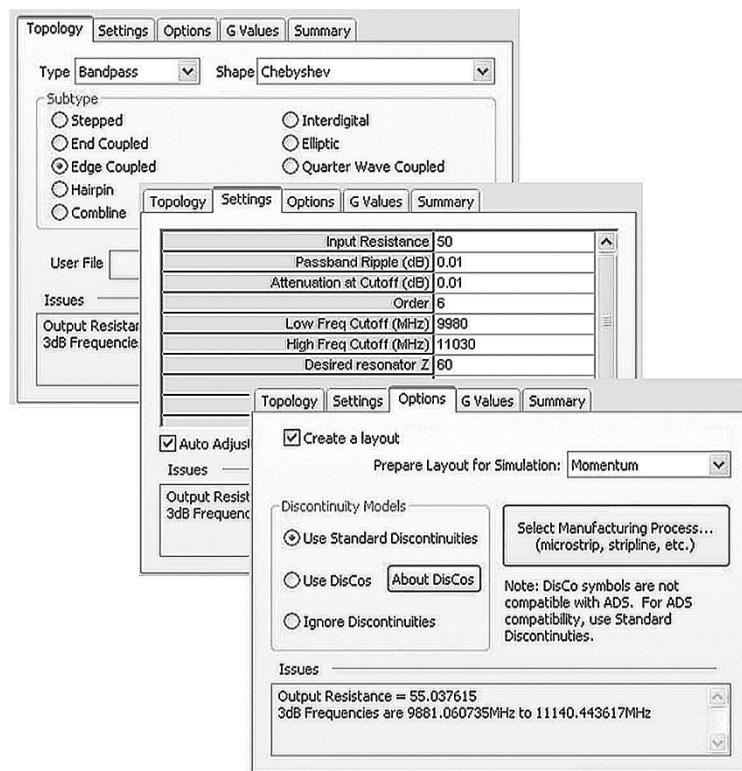


Figure 4-57 Bandpass filter synthesis selections

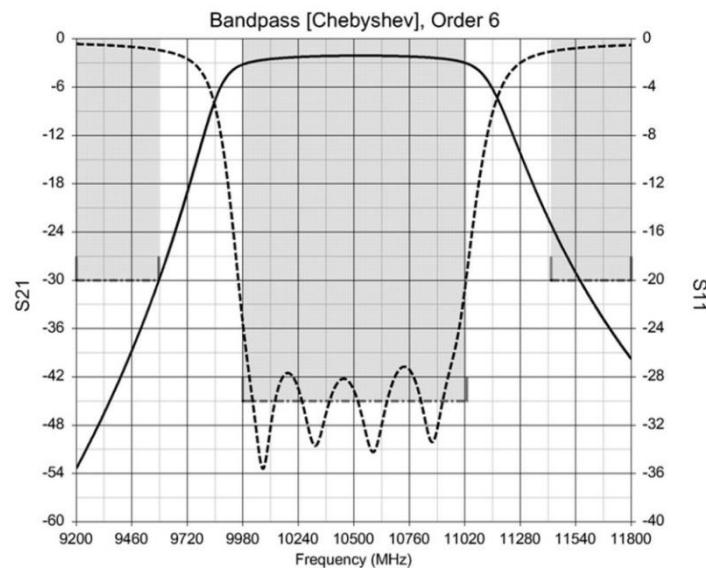


Figure 4-58 Synthesized bandpass filter response

On the Topology tab set the design for an edge coupled band pass filter with a Chebyshev shape. On the Settings tab, enter the pass band frequencies per the filter specification. Vary the order until the desired filter rejection is achieved. The Microwave Filter synthesis program shows that a 6th order filter should meet the rejection specifications. On the Options tab, select standard discontinuities and check the ‘Create a Layout’. The synthesized filter schematic is shown in Figure 4-59.

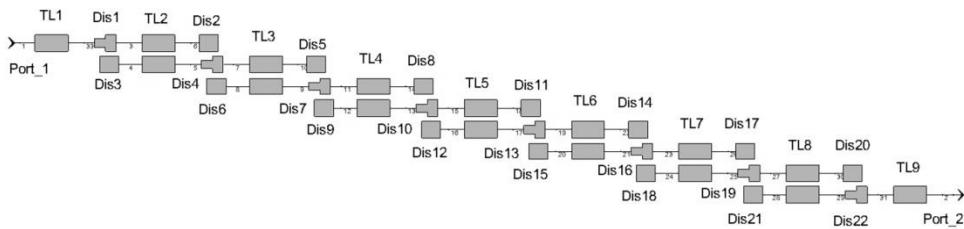


Figure 4-59 Parallel line edge coupled microstrip bandpass filter schematic

4.9.6 Electromagnetic Analysis of the Edge Coupled Filter

The designer must be cautious when using Linear Analysis techniques to design circuits with multiple edge coupled microstrip lines. We know that there is a large percentage of the parallel line coupling that occurs in the free space above the microstrip substrate and the conductors. This can lead to considerable error when relying on the linear simulation results. The coupled line models used by the linear simulator are based on closed form expressions derived from coupling measurements on parallel lines with loosely defined boundary conditions.

Example 4.9-4: Use the microwave mode in the Momentum software to simulate the edge coupled filter.

Solution: Using the circuit model created in the last section examine the layout of the filter. On the Layout-Properties window, make sure that the “Show EM Box” is selected. Then adjust the size of the box to fit the filter. Set the filter box width (Y) to 400 mils and the box height (X dimension) to 760 mils. The EM box will show up as a red rectangle on the layout

window. Next edit the Layer tab. Check the “Show All” box. Scroll across to make sure that the substrate thickness, dielectric constant, and loss tangent parameters are correct. Check “Use” boxes as appropriate. Set the initial box height (Z dimension) at 300 mils.

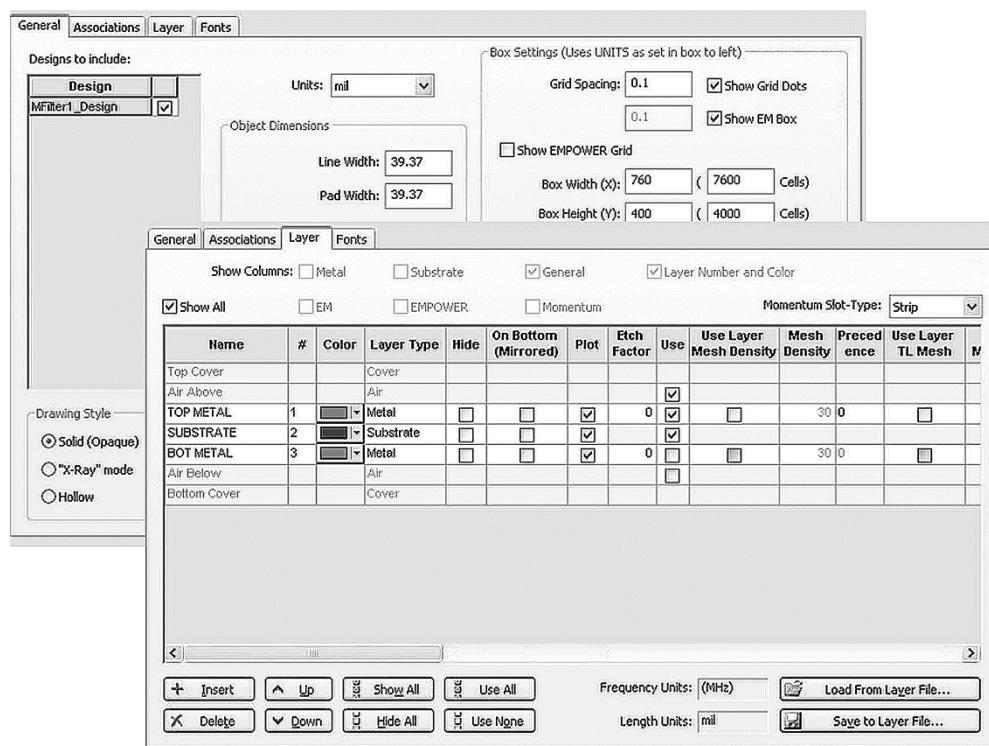


Figure 4-60 Layer setup for momentum analysis of edge coupled filter

Add a Momentum GX Analysis to the workspace and set the parameters as shown in Figure 4-61. Set the analysis frequency range of 9 to 12 GHz with an Adaptive sweep type. On the Simulation Options tab, make sure that the “use box” check box has been selected. This forces a 3D mesh of the entire filter and box. This time we must select the microwave simulation mode so that the box radiation effects are properly modeled.

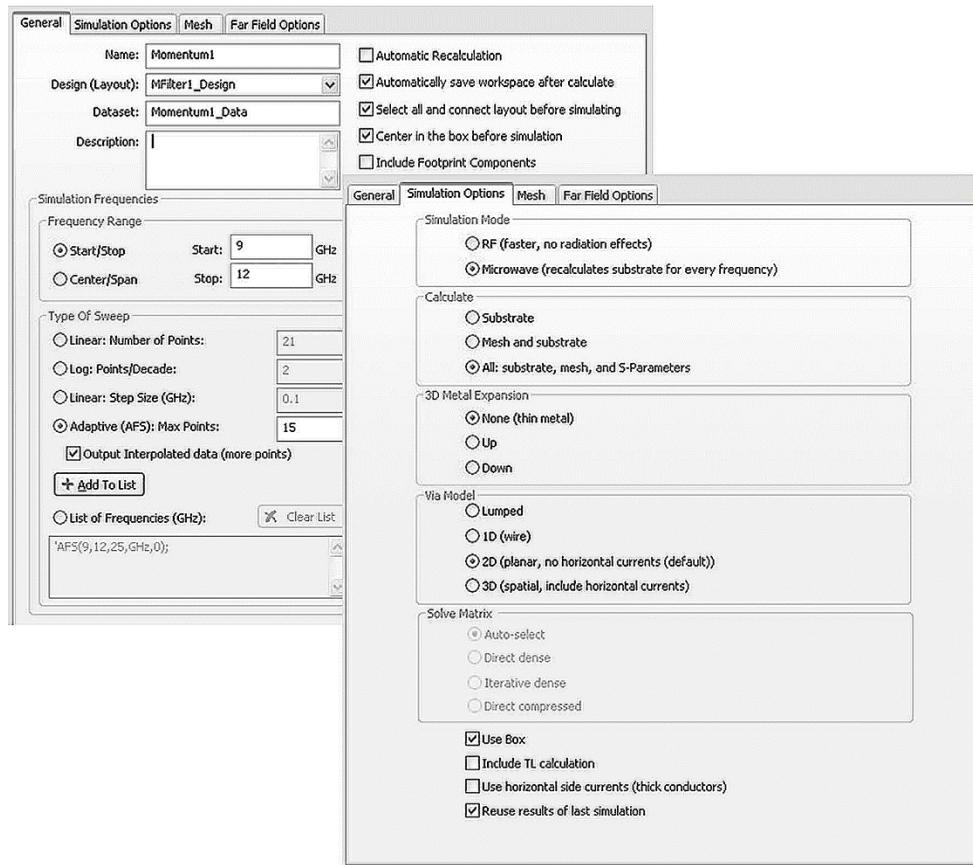


Figure 4-61 Momentum GX general setup

Perform a Momentum simulation on the filter and observe the response. The edge-coupled filter is particularly sensitive to the sidewalls providing the proper boundary conditions for the coupled energy between the parallel-coupled lines. Begin the tuning process by first reducing the box height to 250 mils. Keep the cover height at 250 mils. A 3D picture of the filter is shown in Figure 4-64 to have a good appreciation of the physical geometry of the filter. Figure 4-62 shows the comparison between the Linear model and the Momentum EM model. We see that the actual pass band is shifted up in frequency slightly while the bandwidth has been reduced. Varying the box dimensions shows that the filter skirts are heavily dependent on the box around the filter.

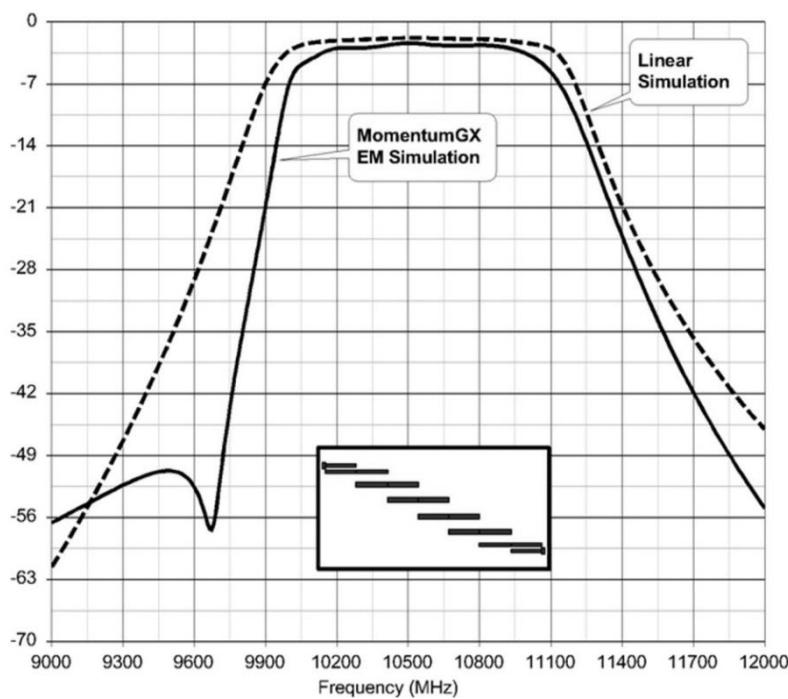


Figure 4-62 Comparison between the linear and the Momentum EM model

4.9.7 Enclosure Effects

Another issue with the box or enclosure design is the possibility of cavity-like resonances that can occur in the physical surroundings of a microwave circuit. Resonances within the metallic enclosure can allow a parallel path for microwave radiation that could bypass the resonators of the filter. This can cause the filter skirts to have significantly less attenuation than the design predicts. In active circuits such as amplifiers the box resonance can result in problems with oscillation.

Example 4.9-5: Analyze the effects of placing the edge coupled filter in a wider 900 mil box as shown in Figure 4-63.

Solution: When the initial EM model is setup the Momentum simulation status window will issue a warning about any box resonances that may exist. As Figure 4-63 shows this box has a dominant resonance at 10.28 GHz. As the resulting filter response of Figure 4-63 shows the box

resonance has an extremely detrimental effect on the low side pass band response. Very little filter rejection is achieved on the low side of the filter pass band. EM simulation is a powerful tool that the engineer should consider for accurate microwave circuit design. By modeling radiation effects accurately a better representation of resonator Q factor is realized. This is particularly important in the design of microwave filter networks.

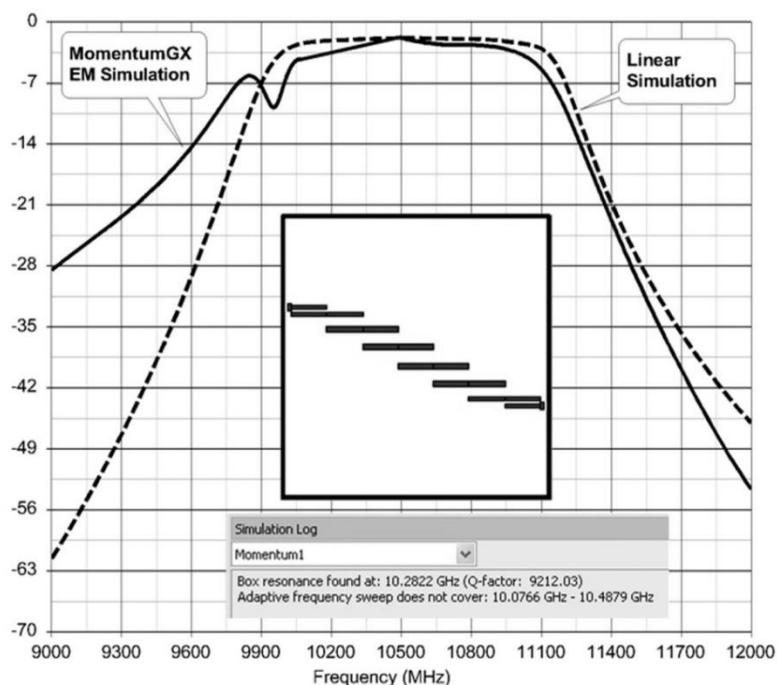


Figure 4-63 Effect of box resonance on edge coupled filter passband

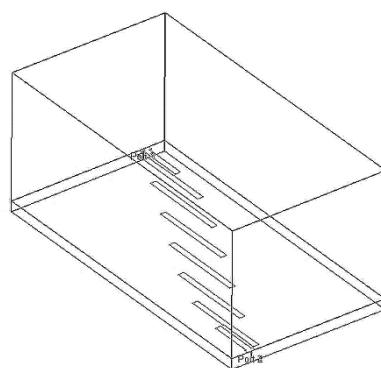


Figure 4-64 3D View of the edge coupled filter showing layer stack

References and Further Reading

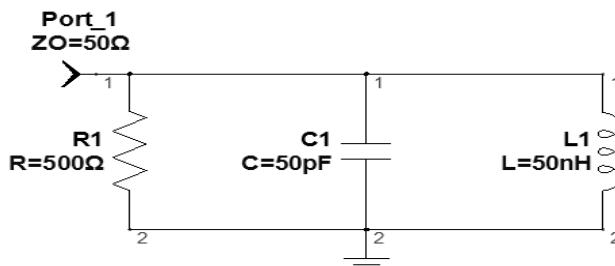
- [1] Ali A. Behagi and Stephen D. Turner, *Microwave and RF Engineering*, BT Microwave LLC, State College, PA, 2011
- [2] RF Circuit Design, Second Edition, Christopher Bowick, Elsevier 2008
- [3] Keysight Technologies, *Genesys 2014.03, Users Guide*, www.keysight.com
- [4] Foundations for Microstrip Circuit Design, T.C. Edwards, John Wiley & Sons, New York, 1981
- [5] Q Factor, Darko Kajfez, Vector Fields, Oxford Mississippi, 1994
- [6] Q Factor Measurement with Network Analyzer, Darko Kajfez and Eugene Hwan, IEEE Transactions on Microwave Theory and Techniques, Vol. MTT-32, No. 7, July 1984.
- [7] High Frequency Techniques, Joseph F. White, John Wiley & Sons, Inc., 2005
- [8] Soft Substrates Conquer Hard Designs, James D. Woermke, Microwaves, January 1982.
- [9] Principles of Microstrip Design, Alam Tam, RF Design, June 1988.
- [10] David M. Pozar, *Microwave Engineering*, Fourth Edition, John Wiley & Sons, New York, 2012

Problems

- 4-1. Consider the one port resonator that is represented as a series RLC circuit as shown. Analyze the circuit, with $R = 5 \Omega$, $L = 5 \text{ nH}$, and $C = 5 \text{ pF}$. Plot the magnitude of the resonator input impedance and measure the resonance frequency.



- 4-2. Consider the one port resonator that is represented as a parallel RLC circuit as shown. Analyze the circuit, with $R = 500 \Omega$, $L = 50 \text{ nH}$, and $C = 50 \text{ pF}$. Plot the magnitude of the resonator input impedance and measure the resonance frequency.



- 4-3. Design a Butterworth lowpass filter having a passband of 2 GHz with an attenuation 20 dB at 4 GHz. Plot the insertion loss versus frequency from 0 to 5 GHz. The system impedance is 50Ω .
- 4-4. Design a 5th order Chebyshev highpass filter having 0.2 dB equal ripples in the passband and cutoff frequency of 2 GHz. The system impedance is 75Ω . Plot the insertion loss versus frequency from 0 to 5 GHz.
- 4-5. In a full duplex communication link, the uplink signal is around 200 MHz while the downlink is at 500 MHz. A 25 Watt power amplifier is used on the uplink with 20 dB gain.

- (a) Design a low pass filter on the uplink to pass the 200 MHz uplink signal while rejecting any noise power in the 500 MHz band. Design the passband cutoff frequency is at 220 MHz, therefore, the filter should have a Chebyshev Response with 0.1 dB pass band ripple. The reject requirement is at least -40 dB rejections at 500 MHz.
- (b) Design a High Pass Filter that passes frequencies in the 480 MHz to 520 MHz range. The High Pass Filter specifications are: The passband cutoff frequency is 480 MHz, therefore, the filter should have a Chebyshev response with 0.1 dB pass band ripple. The reject requirement is at least -60 dB losses at 190 MHz.
- 4-6. Design a 75Ω transmission line of sufficient length to act as a 10 nH inductor. Use the TLINE Transmission Line Synthesis Program to calculate the microstrip line width on the Rogers 0.025 inch RO3010 material.
- 4-7. Using the microwave filter synthesis tool, design a stepped impedance low pass filter on RO3003 material that is 0.010 inches thick. Use a Chebyshev response with a 0.01dB ripple and a cutoff frequency of 4 GHz. Determine the worst case in band return loss and the rejection at 6 GHz.
- 4-8. For the filter design of Problem 4-7 create an EM simulation using Momentum. Compare the EM simulation to a linear simulation. Comment on the rejection comparison at 6 GHz.
- 4-9. For the filter design of Problem 4-7 determine the frequencies at which reentrant modes exist up through 20 GHz.
- 4-10. Design a half wave microstrip resonator at 10 GHz using RO3003 substrate that is 0.020 thick. Initially design the resonator with a 50 W line impedance. Select a coupling capacitor to critically couple the resonator to the 50Ω source. Then determine the resonator line impedance that results in the highest unloaded Q_o .

- 4-11. Use the microwave filter synthesis tool to design a parallel edge coupled filter on RO3003 substrate that is 0.010 thick. Use a Chebyshev characteristic with 0.10 dB ripple. Design the passband to cover 10.7 GHz to 12.2 GHz. Determine the filter order required to achieve 30 dB rejection at 9 GHz.
- 4-12. For the filter design of Problem 4-10, create an EM simulation using Momentum. Compare the linear and EM simulations. Determine the minimum box width (EM Box height, Y) that creates a box resonance frequency. What is the frequency of the box resonance?

Appendix A

Straight Wire Parameters for Solid Copper Wire

Wire Size (AWG)	Diameter in Mils	Resistance Ohms/1000 ft.	Area in circular Mils	Suggested Maximum Current Handling, Amperes ¹
0000	460.0	0.049	211600	1000
000	409.6	0.062	167800	839
00	364.8	0.078	133100	665
0	324.9	0.098	105500	527
1	289.3	0.124	83690	418
2	257.6	0.156	66360	332
3	229.4	0.197	52620	263
4	204.3	0.249	41740	208
5	181.9	0.313	33090	165
6	162.0	0.395	26240	131
7	144.3	0.498	20820	104
8	128.5	0.628	16510	83
9	114.4	0.793	13090	65
10	101.9	0.999	10380	52
11	90.7	1.26	8230	41
12	80.8	1.56	6530	32
13	72.0	2.00	5180	26
14	64.1	2.52	4110	20
15	57.1	3.18	3260	16
16	50.8	4.02	2580	13
17	45.3	5.05	2050	10
18	40.3	6.39	1620	8.0
19	35.9	8.05	1290	6.0
20	32.0	10.1	1020	5.0
21	28.5	12.8	812	4.0
22	25.3	16.2	640	3.0
23	22.6	20.3	511	2.5
24	20.1	25.7	404	2.0
25	17.9	32.4	320	1.6
26	15.9	41.0	253	1.2
27	14.2	51.4	202	1.0
28	12.6	65.3	159	0.80
29	11.3	81.2	123	0.61
30	10.0	104.0	100	0.50
31	8.9	131	79.2	0.40
32	8.0	162	64.0	0.32
33	7.1	206	50.4	0.25
34	6.3	261	39.7	0.19
35	5.6	331	31.4	0.16
36	5.0	415	25.0	0.12
37	4.5	512	20.2	0.10
38	4.0	648	16.0	0.08
39	3.5	847	12.2	0.06
40	3.1	1080	9.61	0.05
41	2.8	1320	7.84	0.04
42	2.5	1660	6.25	0.03
43	2.2	2140	4.84	0.024
44	2.0	2590	4.00	0.020
45	1.76	3350	3.10	0.016
46	1.57	4210	2.46	0.012
47	1.40	5190	1.96	0.010
48	1.24	6750	1.54	0.008
49	1.11	8420	1.23	0.006
50	0.99	10600	0.98	0.005

Current Handling based on 1 Amp/200 Circular Mils-no insulation and free air conditions.
 Insulated and stranded Copper wire must be de-rated from the values in the Table.

Workspace Schematics for the Q_o Measurement Smith Chart Overlay

Appendix B contains the collection of Genesys Workspace schematics that are used to produce the Smith Chart overlays used for resonator Q_o measurement of Section 4.6.2.

B.1 Γ_i Line Generation

The Γ_i Line is produced by using the manual S-Parameter model of Figure B1-1. A Parameter Sweep is used to vary the magnitude of S11 from -1 to +1. This will produce a straight line across the Smith Chart. The angle of S11 is made a variable in the Equation Editor. This will allow the ‘angle’ variable to be shared with the overlays described in Appendix B-2 and B-3. The angle of the Γ_i can be rotated around the Smith Chart using the Tune control. The resulting line on the Smith Chart is shown in Figure B1-3.

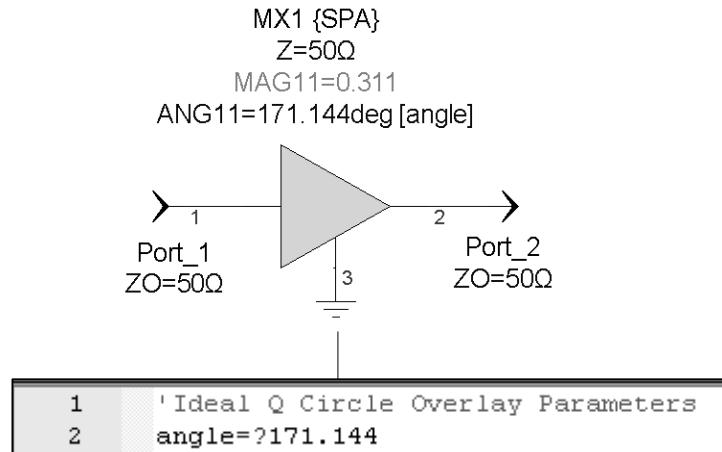


Figure B1-1 Manual S-Parameter Model and Variable Defined in the Equation Editor

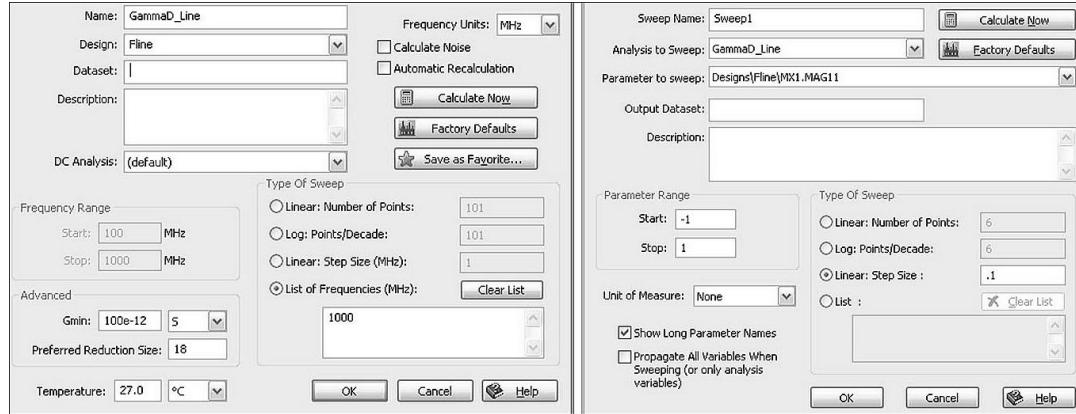


Figure B1-2 Linear Analysis and Parameter Sweep used for the Γ_i Line Generation Schematic

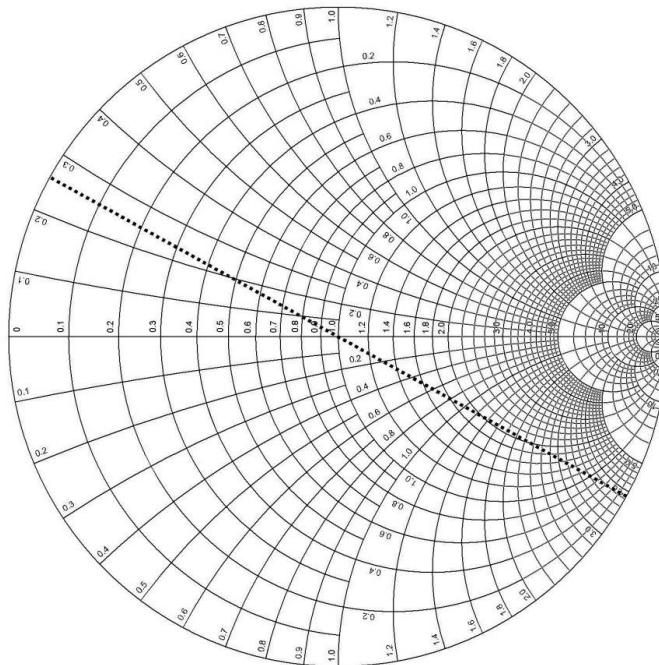


Figure B1-3 Γ_i Line drawn across Smith Chart

Appendix B.2 Q_L Lines on the Smith Chart

The Q_L line Smith Chart overlay is used to measure the loaded Q, Q_L, from the S11 measurement of a resonator. The lines are drawn at an angle of $\pm 45^\circ$ from the Γ_i line of Appendix A.1. This will satisfy the Q_L measurement described in Section 4.6.2. The schematic of Figure B2-1 uses the accompanying two-port S-Parameter file to generate the lines. One line is produced from S11 while the other line is produced by S22 of the circuit. A phase shift model is used on the input and output of the S-Parameter file to rotate the intersection of the lines around the Smith Chart in sync with the Γ_i line. An attenuator element is used on the input and output so that the intersection of the lines can be moved inward to the center of the Smith Chart. This allows proper alignment with Γ_d for lossy coupling cases. The attenuation and phase shift values are defined by the same variable and defined in the Equation Editor.

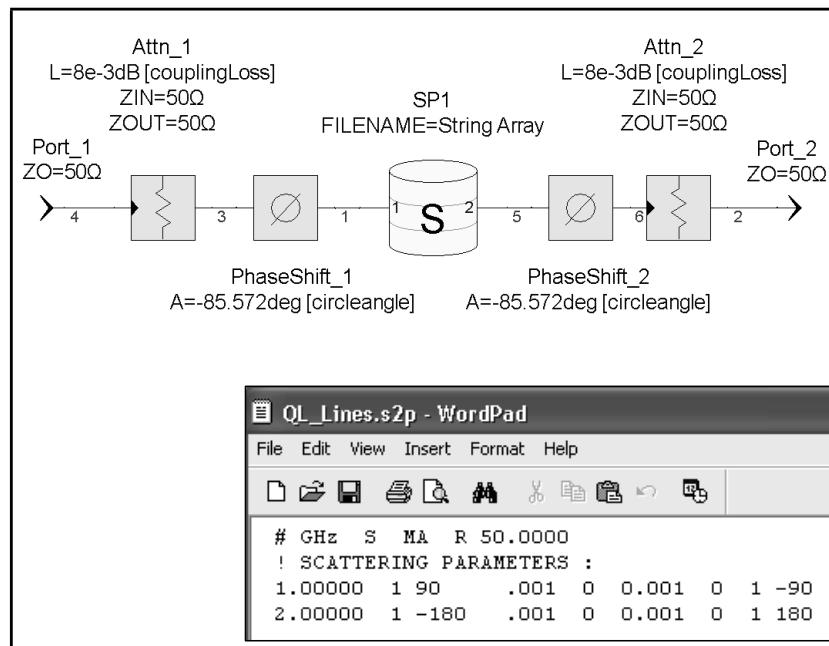
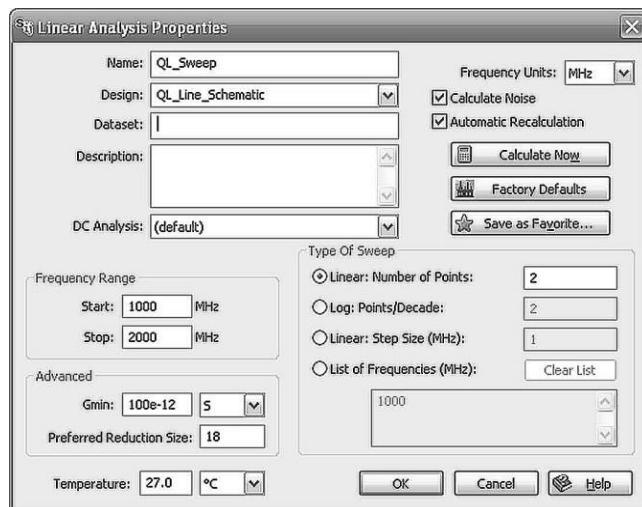
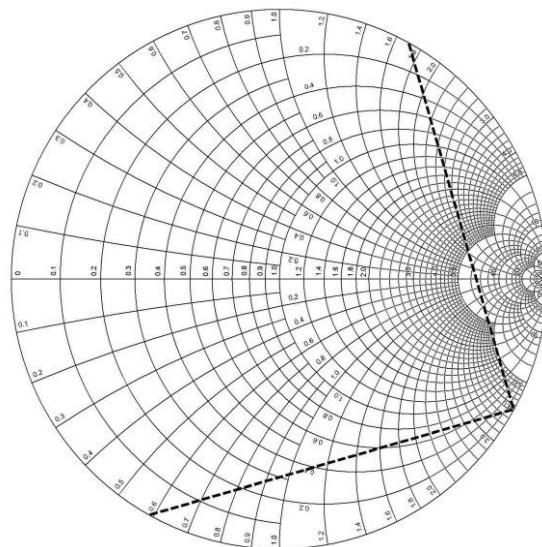


Figure B2-1 Schematic used for Generation of Q_L Lines on the Smith Chart

```

1   ' Ideal Q Circle Overlay Parameters
2   angle=?171.144
3   circlediameter=?1988.72
4   couplingLoss=?0.008
5   circleangle=-angle/2

```

Figure B2-2 Equation Editor with Variables required by the Smith Chart Overlay**Figure B2-3** Linear Analysis used by the Q_L Line Schematic**Figure B2-4** Q_L Lines Produced on the Smith Chart

Appendix B.3 Ideal Q Circle on the Smith Chart

The schematic of Figure B3-1 is used to generate the ideal Q circle Smith Chart overlay. A parallel RLC resonator is coupled to the load by an ideal transformer model. The turn ratio of the transformer is defined by variable ‘circlediameter’ because the coupling determines the size of the Q circle on the Smith Chart. A phase shift element and attenuator are used to move the circle inward toward the center of the chart for lossy coupling cases.

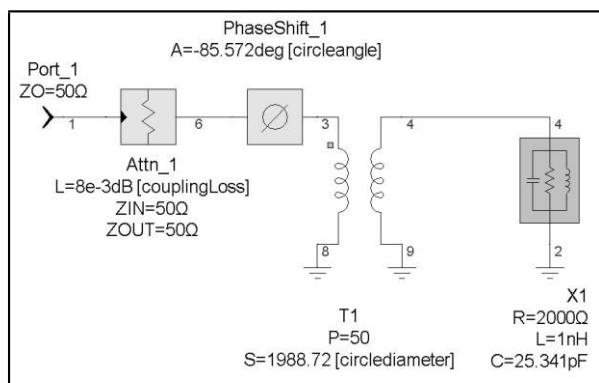


Figure B3-1 Schematic used for Ideal Q Circle Drawn on the Smith Chart

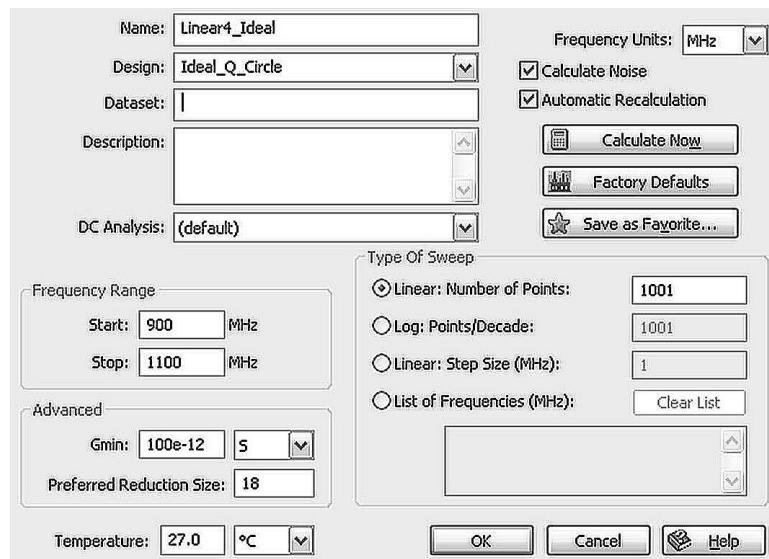


Figure B3-2 Linear Analysis used for the Ideal Q Circle Generation
Copyright © 2015 by Ali A. Behagi

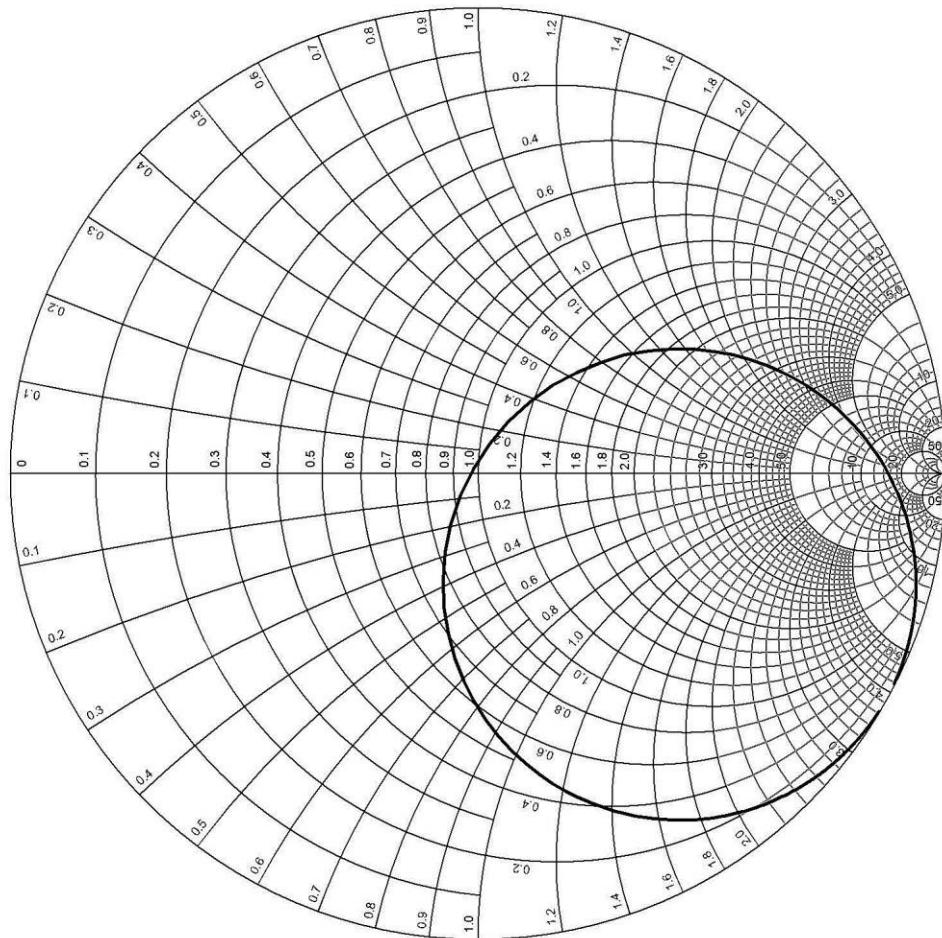


Figure B3-3 Ideal Q Circle Overlay Drawn on the Smith Chart

Appendix B.4 Q_o Measurement on the Smith Chart

The complete set of measurements required for Q_o measurement are shown in Figure B4-1. Each output is identified on the Smith Chart of Figure B4-2.

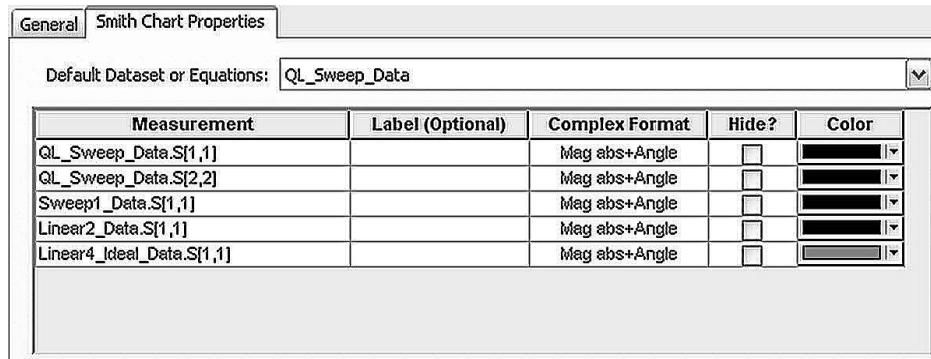


Figure B4-1 Output of Four Datasets on Single Smith Chart

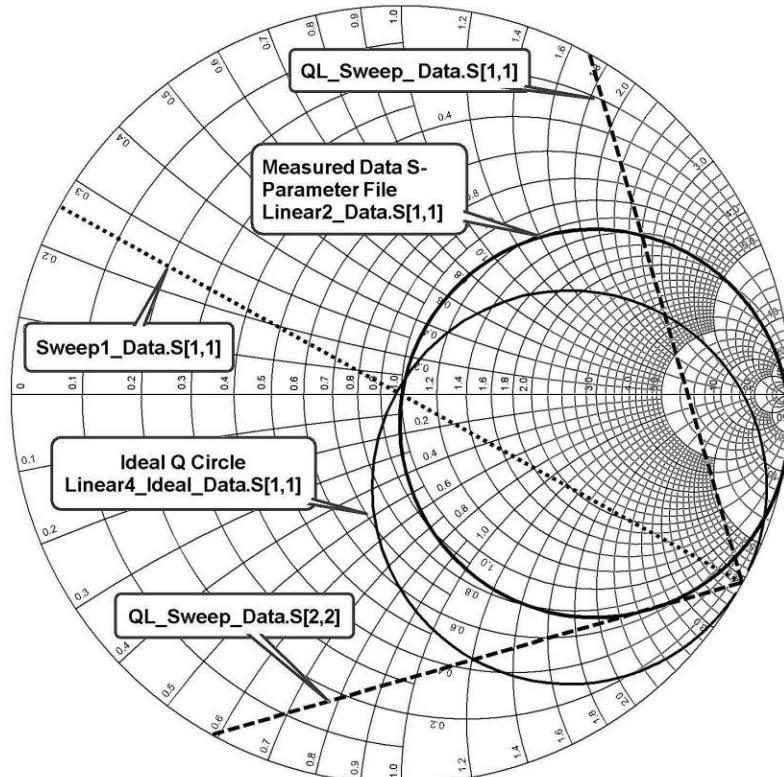


Figure B4-2 Datasets identified on the Smith Chart

Appendix C

VBScript file listing for the Matching Utility of Chapter 5.

```

Dim WsDoc
Dim GL, BL, L2, L1, L3, RS, XS,z, A, A1, B,L, X, C, C1,C2, wo,T
Dim RL, XL, F, complex, pos, i, r, xi, Ls, Cs, Cp, Lp,nsolutions
WsDoc= theApp.GetWorkspaceByIndex(0)
Sim=WsDoc.Designs.Linear1
F = WsDoc.Designs.Sch1.PartList.CWSource_3.ParamSet.F.GetValue()
PARSE THE REAL AND IMAGINARY PART OF THE IMPEDANCE

complex = WsDoc.Designs.Sch1.PartList.Port_1.ParamSet.ZO.GetData()
z=len(complex)
pos=instr(complex,"j")
If pos=0 then
XS=0
RS = WsDoc.Designs.Sch1.PartList.Port_1.ParamSet.ZO.GetData()
else XS=mid(complex,pos-1)
XS=replace(XS,"j","")
RS=left(complex,z-((z-(pos-1))+1))
end if

PARSE THE REAL AND IMAGINARY PARTS OF THE PORT 2
IMPEDANCE

complex = WsDoc.Designs.Sch1.PartList.Port_2.ParamSet.ZO.GetData()
z=len(complex)
pos=instr(complex,"j")
If pos=0 then
XL=0
RL = WsDoc.Designs.Sch1.PartList.Port_2.ParamSet.ZO.GetData()
else XL=mid(complex,pos-1)
XL=replace(XL,"j","");
RL=left(complex,z-((z-(pos-1))+1))
end if

```

My_L_Network=WsDoc.Designs.Sch1.PartList.L1.ParamSet.L.SetValue(XL)

r=RL/RS

xi=XL/RS

```
wo=(2*3.14*F)
i=WsDoc.Equation.VarBlock.Solution.GetValue()
```

' First Condition

A1=((RS^2)+(XS^2))-(RL*RS)

' Second Condition

A=((RL^2)+(XL^2))-(RL*RS)

If A1>0 and A>0 Then

nsolutions=4

WsDoc.Designs.Sch1.Schematic.Page.Annotate2.Max.Set(nsolutions)

Select Case i

case 1

B = ((RL * XS) + Sqr(RS * RL * (RS ^ 2 + XS ^ 2 - RS * RL))) / (RL * (RS ^ 2 + XS ^ 2))

X = (RL * XS - RS * XL) / RS + (RS - RL) / (B * RS)

If B>0 Then 'B is a shunt C

B=B/wo

ShuntC

elseif B<0 then 'B is a shunt L

B=-1/(wo*B)

ShuntL

else

end if

If X>0 then 'X is a series L

X=X/wo

SeriesL_X2

elseif X<0 then 'X is a series C

X=-1/(wo*X)

SeriesC_X2

else

end if

PlotResponse

'

case 2

$$B = ((RL * XS) - Sqr(RS * RL * (RS^2 + XS^2 - RS * RL))) / (RL * (RS^2 + XS^2))$$

$$X = (RL * XS - RS * XL) / RS + (RS - RL) / (B * RS)$$

If B>0 Then 'B is a shunt C

B=B/wo

ShuntC

elseif B<0 then 'B is a shunt L

$$B=-1/(wo*B)$$

ShuntL

else

end if

If X>0 then 'X is a series L

X=X/wo

SeriesL_X2

elseif X<0 then 'X is a series C

$$X=-1/(wo*X)$$

SeriesC_X2

else

end if

PlotResponse

'

case 3

$$B = ((RS * XL) + Sqr(RS * RL * (RL^2 + XL^2 - RS * RL))) / (RS * (RL^2 + XL^2))$$

$$X = (RS * XL - RL * XS) / RL + (RL - RS) / (B * RL)$$

If B>0 Then 'B is a shunt C

B=B/wo

ShuntC

elseif B<0 then 'B is a shunt L

$$B=-1/(wo*B)$$

```

ShuntL
else
end if

If X>0 then          'X is a series L
X=X/wo
SeriesL_X3
elseif X<0 then      'X is a series C
X=-1/(wo*X)
SeriesC_X3
else
end if
PlotResponse

```

case 4

```

B = ((RS * XL) - Sqr(RS * RL * (RL ^ 2 + XL ^ 2 - RS * RL))) / (RS * (RL ^ 2 +
XL ^ 2))
X = (RS * XL - RL * XS) / RL + (RL - RS) / (B * RL)
If B>0 Then          'B is a shunt C
B=B/wo
ShuntC
elseif B<0 then      'B is a shunt L
B=-1/(wo*B)
ShuntL
else
end if

If X>0 then          'X is a series L
X=X/wo
SeriesL_X3
elseif X<0 then      'X is a series C
X=-1/(wo*X)
SeriesC_X3
else
end if
PlotResponse
End select
'
```

ElseIf A1>0 and A<0 then
nsolutions=2

WsDoc.Designs.Sch1.Schematic.Page.Annotate2.Max.Set(nsolutions)
Select Case i

case 1

$$B = ((RL * XS) + Sqr(RS * RL * (RS^2 + XS^2 - RS * RL))) / (RL * (RS^2 + XS^2))$$

$$X = (RL * XS - RS * XL) / RS + (RS - RL) / (B * RS)$$

If B>0 Then 'B is a shunt C

B=B/wo

ShuntC

elseif B<0 then 'B is a shunt L

$$B=-1/(wo*B)$$

ShuntL

else

end if

If X>0 then 'X is a series L

X=X/wo

SeriesL_X2

elseif X<0 then 'X is a series C

$$X=-1/(wo*X)$$

SeriesC_X2

else

end if

PlotResponse

case 2

$$B = ((RL * XS) - Sqr(RS * RL * (RS^2 + XS^2 - RS * RL))) / (RL * (RS^2 + XS^2))$$

$$X = (RL * XS - RS * XL) / RS + (RS - RL) / (B * RS)$$

If B>0 Then 'B is a shunt C

B=B/wo

ShuntC

elseif B<0 then 'B is a shunt L

$$B=-1/(wo*B)$$

ShuntL

else

end if

If X>0 then 'X is a series L

```

X=X/wo
SeriesL_X2
elseif X<0 then          'X is a series C
X=-1/(wo*X)
SeriesC_X2
else
end if
PlotResponse
end select
'
```

```

ElseIf A1<0 then
nsolutions=2
WsDoc.Designs.Sch1.Schematic.Page.Annotate2.Max.Set(nsolutions)
Select case i
```

case 1

```

B = ((RS * XL) + Sqr(RS * RL * (RL ^ 2 + XL ^ 2 - RS * RL))) / (RS * (RL ^ 2
+ XL ^ 2))
X = (RS * XL - RL * XS) / RL + (RL - RS) / (B * RL)
If B>0 Then          'B is a shunt C
B=B/wo
```

```

ShuntC
elseif B<0 then          'B is a shunt L
B=-1/(wo*B)
ShuntL
else
end if
```

```

If X>0 then          'X is a series L
X=X/wo
SeriesL_X3
elseif X<0 then          'X is a series C
X=-1/(wo*X)
SeriesC_X3
else
end if
PlotResponse
```

case 2

$$B = ((RS * XL) - Sqr(RS * RL * (RL^2 + XL^2 - RS * RL))) / (RS * (RL^2 + XL^2))$$

$$X = (RS * XL - RL * XS) / RL + (RL - RS) / (B * RL)$$

If B>0 Then 'B is a shunt C

B=B/wo

ShuntC

elseif B<0 then 'B is a shunt L

$$B=-1/(wo*B)$$

ShuntL

else

end if

If X>0 then 'X is a series L

$$X=X/wo$$

SeriesL_X3

elseif X<0 then 'X is a series C

$$X=-1/(wo*X)$$

SeriesC_X3

else

end if

PlotResponse

'

end select

Else

End if

' Define Network Configurations _____

Sub ShuntC

My_L_Network=WsDoc.Designs.Sch1.PartList.X1.ChangeSymbol("CAPACITOR")

My_L_Network=WsDoc.Designs.Sch1.PartList.X1.ChangeModel("CAP")

My_L_Network=WsDoc.Designs.Sch1.PartList.X1.ParamSet.C.SetValue(B)

end sub

Sub ShuntL

My_L_Network=WsDoc.Designs.Sch1.PartList.X1.ChangeSymbol("INDUCTOR")

```

My_L_Network=WsDoc.Designs.Sch1.PartList.X1.ChangeModel("IND")
My_L_Network=WsDoc.Designs.Sch1.PartList.X1.ParamSet.L.SetValue(B)
end sub

Sub SeriesL_X2
L2=.0001*10^-20
My_L_Network=WsDoc.Designs.Sch1.PartList.X3.ChangeSymbol("Ferrite")
My_L_Network=WsDoc.Designs.Sch1.PartList.X3.ChangeModel("IND")
My_L_Network=WsDoc.Designs.Sch1.PartList.X3.ParamSet.L.SetValue(L2)
My_L_Network=WsDoc.Designs.Sch1.PartList.X2.ChangeSymbol("INDUCTOR")
")
My_L_Network=WsDoc.Designs.Sch1.PartList.X2.ChangeModel("IND")
My_L_Network=WsDoc.Designs.Sch1.PartList.X2.ParamSet.L.SetValue(X)
end sub

Sub SeriesC_X2
L2=.0001*10^-20
My_L_Network=WsDoc.Designs.Sch1.PartList.X3.ChangeSymbol("Ferrite")
My_L_Network=WsDoc.Designs.Sch1.PartList.X3.ChangeModel("IND")
My_L_Network=WsDoc.Designs.Sch1.PartList.X3.ParamSet.L.SetValue(L2)
My_L_Network=WsDoc.Designs.Sch1.PartList.X2.ChangeSymbol("CAPACITO
R")
My_L_Network=WsDoc.Designs.Sch1.PartList.X2.ChangeModel("CAP")
My_L_Network=WsDoc.Designs.Sch1.PartList.X2.ParamSet.C.SetValue(X)
end sub

sub SeriesL_X3
L3=.0001*10^-20
My_L_Network=WsDoc.Designs.Sch1.PartList.X3.ChangeSymbol("INDUCTOR")
")
My_L_Network=WsDoc.Designs.Sch1.PartList.X3.ChangeModel("IND")
My_L_Network=WsDoc.Designs.Sch1.PartList.X3.ParamSet.L.SetValue(X)
My_L_Network=WsDoc.Designs.Sch1.PartList.X2.ChangeSymbol("Ferrite")
My_L_Network=WsDoc.Designs.Sch1.PartList.X2.ChangeModel("IND")
My_L_Network=WsDoc.Designs.Sch1.PartList.X2.ParamSet.L.SetValue(L3)
end sub

sub SeriesC_X3
L1=.0001*10^-20
My_L_Network=WsDoc.Designs.Sch1.PartList.X3.ChangeSymbol("CAPACITO
R")
My_L_Network=WsDoc.Designs.Sch1.PartList.X3.ChangeModel("CAP")

```

```
My_L_Network=WsDoc.Designs.Sch1.PartList.X3.ParamSet.C.SetValue(X)
My_L_Network=WsDoc.Designs.Sch1.PartList.X2.ChangeSymbol("Ferrite")
My_L_Network=WsDoc.Designs.Sch1.PartList.X2.ChangeModel("IND")
My_L_Network=WsDoc.Designs.Sch1.PartList.X2.ParamSet.L.SetValue(L1)
end sub
```

```
Sub PlotResponse
Sim.ClearModelCache
Sim.RunAnalysis
Graph=WsDoc.Designs.Graph1
Graph.OpenWindow
End Sub.
```

Appendix D

VBScript file listing for the Line and Stub Matching Utility of Chapter 6.

```
Dim WsDoc
Dim RS, d1,d2,pi, serieslinelength, B1, B2,so1,so2,ss1,ss2,
OpenShuntStub,ShortedShuntStub
Dim RL, XL, F, complex, pos, i, r, x, t1, t2 ,nsolutions, z1,Length,e
pi=3.141592653589793
WsDoc= theApp.GetWorkspaceByIndex(0)

F = WsDoc.Designs.Sch1.PartList.CWSource_3.ParamSet.F.GetValue()
My_LineStub=WsDoc.Designs.Sch1.PartList.TL1.ParamSet.F.SetValue(F)
My_LineStub=WsDoc.Designs.Sch1.PartList.TL2.ParamSet.F.SetValue(F)
RS = WsDoc.Designs.Sch1.PartList.Port_1.ParamSet.ZO.GetData()
complex = WsDoc.Designs.Sch1.PartList.Port_2.ParamSet.ZO.GetData()
pos=instr(complex,"j")
complex=left(complex,pos-1)
z=len(complex)
```

' Find the 1st space position

```
space_position=instr(complex," ")
RL=left(complex,(space_position))
XL=Mid(complex,space_position+1,((z-space_position)+1))
```

' Convert image string to number

```
s=len(XL)
sign=mid(XL,1,1)
XL=mid(XL,3,(s-2))
If sign= "+" then
XL=XL
else
XL=XL*-1
end if
```

' Calculation of Series Line Length

```
r=RL/RS
x=XL/RS
t1=(x+sqr(r*(r^2+x^2-(2*r)+1)))/(r-1)
t2=(x-sqr(r*(r^2+x^2-(2*r)+1)))/(r-1)
```

' Calculate the electrical lengths

```
If t1>0 then
d1=(360/(2*pi))*atn(t1)
else
d1=360*(pi+atn(t1))/(2*pi)
end if
If t2>0 then
d2=(360/(2*pi))*atn(t2)
else
d2=360*(pi+atn(t2))/(2*pi)
end if
If d1<d2 then
SeriesLineLength=d1
else
SeriesLineLength=d2
end if
```

' Calculation of Shunt Line Length

```
B1=(x*t1^2+(r^2+x^2-1)*t1-x)/(RS*(r^2+x^2+t1^2+2*x*t1))
B2=(x*t2^2+(r^2+x^2-1)*t2-x)/(RS*(r^2+x^2+t2^2+2*x*t2))
```

' Open circuit stubs

```
so1=-360*(atn(RS*B1))
so2=-360*atn(RS*B2)/(2*pi)
```

```
If so1<0 then
OpenShuntStub=so2
else
OpenShuntStub=so1
end if
```

' Shorted shunt stubs

```
ss1=(360*atn(1/(RS*B1)))/(2*pi)
ss2=(360*atn(1/(RS*B2)))/(2*pi)
```

```
If ss1<0 then
ShortedShuntStub=ss2
else
ShortedShuntStub=ss1
end if
```

' Determine valid solutions

```
nsolutions=2
WsDoc.Designs.Sch1.Schematic.Page.Annotate2.Max.Set(nsolutions)
i=WsDoc.Equation.VarBlock.Solution.GetValue()
Select Case i
```

```
case 1
If ss1>0 then
serieslinelength=d1
SeriesLine
ShortedShuntStub=ss1
ShortedCircuitStub
elseif ss1<0 then
serieslinelength=d2
SeriesLine
ShortedShuntStub=ss2
ShortedCircuitStub
end if
Plotresponse
```

case 2

```
If so1>0 then
serieslinelength=d1
SeriesLine
OpenShuntStub=so1
OpenCircuitStub
elseif so1<0 then
```

```
serieslinelength=d2
SeriesLine
OpenShuntStub=so2
OpenCircuitStub
end if
Plotresponse
End Select
```

' Define Transmission Line Configurations

```
Sub OpenCircuitStub
Length=so2
Length=Length*(pi/180)
My_LineStub=WsDoc.Designs.Sch1.PartList.TL2.ParamSet.L.SetValue(Length)
z1=100e6
My_LineStub=WsDoc.Designs.Sch1.PartList.Port_4.ParamSet.ZO.SetValue(z1)
My_LineStub=WsDoc.Designs.Sch1.PartList.Port_4.ChangeSymbol("TRL_END")
")
End sub
```

```
Sub ShortedCircuitStub ShortedShuntStub=ShortedShuntStub*(pi/180)
My_LineStub=WsDoc.Designs.Sch1.PartList.TL2.ParamSet.L.SetValue(Shorted
ShuntStub)
z1=.0001
My_LineStub=WsDoc.Designs.Sch1.PartList.Port_4.ParamSet.ZO.SetValue(z1)
My_LineStub=WsDoc.Designs.Sch1.PartList.Port_4.ChangeSymbol("GROUND")
")
End sub
```

```
Sub SeriesLine serieslinelength=serieslinelength*(pi/180)
My_LineStub=WsDoc.Designs.Sch1.PartList.TL1.ParamSet.L.SetValue(serieslin
elength)
End sub
```

' Simulate and Plot Response of Matching Network

```
Sub PlotResponse
Sim=WsDoc.Designs.Linear1
Sim.RunAnalysis
Sim.ClearModelCache
```

```
Graph=WsDoc.Designs.Graph1
Graph.OpenWindow
End Sub.
```

Index

A

- Admittance circles, 128–129, 142, 301
- Amplifier design, 217, 234, 321–389, 391–416
 - low noise, 321–322, 348–350, 360–362, 365–366, 371–375, 385, 389, 391, 408–413
 - maximum gain, 321–322, 324, 334–336, 347, 353, 360, 366, 372, 385, 387–389, 391–392, 414–415
 - power, 184, 213–214, 321–322, 376–385, 391
 - specific gain, 321, 347–360, 366, 385, 388, 391
- Amplifier noise figure, 371
- Amplifier noise temperature, 373–375
- Amplifier types, 321–389
- Attenuation constant, 53, 56, 59, 62

B

- Band pass filter, 205–207
- Bandwidth limitation of matching networks, 264–266
- Broadband matching networks, 217, 247–259, 307–308, 311–318

C

- Capacitors, aluminum, 41 ceramic, 41, 43–44
 - chip, 41, 43–48, 51, 80, 185–186, 188–189, 191–192, 341, 398
 - multilayer, 41, 43–48
 - polystyrene, 41
 - silver mica, 41
 - single layer, 41–43, 51
 - tantalum, 41
 - thin film, 43
- Capacitor tolerance, 401, 405–406
- Cascaded gain, 408–410, 412–413, 415
- Cascaded L-networks, 248, 250–251, 253–259
- Cascaded matching networks, 253, 307, 310
- Coaxial transmission line. *See* Transmission lines

- Conductor Q factor, 163
- Coupled line filter, 205–207
- Coupling coefficient, 168–171, 175
- Critically coupled resonator, 173–174

D

- Design centering, 407
- Dielectric loss, 80, 161, 163
- Directional coupler, 53, 80, 91, 105–109, 112, 377 coupling, 105 directivity, 105 insertion loss, 105 isolation, 105
- Distributed bias feed, 100–102, 111

E

- Edge coupled filter, 207–211, 215
- Effective area, 11
- Effective capacitance, 47–48
- Effective dielectric constant, 76, 78, 81, 93, 97, 161–162, 196
- Electric field, 1, 41, 53, 67–68, 77, 83, 103
- Electromagnetic spectrum, 1–3
- Electromagnetic waves, 1, 49, 53
- Enclosure effects, 200, 210–211
- Equal-Q L-networks, 248, 253–254, 257–259, 276
- External Q, 151–152, 168, 260, 263, 281

F

- Filter attenuation, 177–178, 184–186, 188, 210, 213
- Filter order, 177–178, 181–182, 184, 188, 215
- Filter return loss, 180–183
- Filter ripple, 182, 412
- Filter synthesis utility, 184–185, 205, 270
- Filter topology, 176
- Filter type, 176–181, 183–184
- Flat ribbon inductance, 11–12
- Fractional bandwidth, 152, 205, 230–233, 239–242, 246–247, 251–253, 256–257, 276, 279–282, 284, 286, 289–292, 296, 304, 307–308, 314,

318–320, 336, 416. *See also* Quarter-wave matching network
 Free space, 1, 4, 9, 54–55, 67–69, 82, 89, 200, 207
 Friis formula, 408–411, 416

G

Genesys artwork replacement, 337
 GMAX, 325–328, 335, 348–349, 351, 353, 359, 365–366, 387–388, 394, 397, 415

H

High pass filter, 84, 87, 177, 184, 187–191, 195, 214, 266
 High pass matching L-networks, 267

I

Impedance matching,
 complex load to complex source, 227–234
 complex load to real source, 234–242
 real load to real source, 242–247
 Incident power. *See* Normalized waveforms
 Inductors,
 air core, 17–28, 32, 36, 50
 chip, 17, 28–31, 51, 80, 186, 398
 coil, 16, 18–24, 32–33, 50, 186
 cylindrical, 8, 16, 18, 33
 magnetic core, 18, 32–40
 rectangular, 16
 rod, 32–33
 toroidal, 17, 32–34, 36, 37–40, 51
 Input reflection coefficient, 64, 110–111, 120, 166, 169, 279–280, 283, 285–286, 290, 367
 Insertion loss, 74, 86–87, 101, 105, 108, 111, 124, 153, 155, 164, 182, 199, 213, 222, 225, 231–233, 238, 240, 249–250, 270, 273, 330, 341, 394, 396, 411–412
 Intermediate resistors, 248–249, 252, 254, 264, 305, 308–309, 315
 Interstage matching network, 225, 392, 394–396, 415

L

L-network matching utility, 217
 L-Network Synthesis Utility, 329
 Loaded Q, 151–154, 164, 168–169, 175, 257, 260, 264–265, 276, 280–285
 Lossless transmission line, 61–63, 67, 293
 characteristic impedance, 61–63, 293
 parameters, 61–62
 propagation constant, 61–62
 Low pass Chebyshev filter, 182–184
 Low pass filter, 177, 181–188, 195–196, 198–199, 204–205, 214
 Low pass matching L-networks, 266

M

Magnetic fields, 1, 3, 16, 32–33, 53, 55, 67–68, 76–77, 83–84
 Magnetic flux, 9, 16, 21, 32–33
 Magnetic flux density, 32
 Magnetization intensity, 32
 Maximum gain amplifier. *See* Amplifier design
 Maximum power transfer, 217–225, 227, 234–235, 242–243, 275, 278
 Microstrip bias feed, 99
 Microstrip resonator, 102, 159, 162–167, 171–175, 214
 Microstrip stepped impedance, 195
 Microstrip transmission lines, 70, 76–80
 Microwave resonators, 147, 158–159, 167–175. *See also* One port microwave resonators
 Mismatch loss, 65–66, 75, 182
 Monte Carlo analysis, 400–405

N

Network parameters, 113–146
 ABCD parameters, 113, 116–117, 120
 h parameters, 113, 115, 122
 Y parameters, 113–115, 122
 Z parameters, 113–114, 122
 Normalized impedance, 125–128, 137, 378
 Normalized waveforms, 118 incident power, 61–62, 106, 118 reflected power, 65–66, 105–106, 118

Number of L-networks, 253, 257–259

O

One port microwave resonators, 167–170

Over coupled resonators, 166, 174–175

P

Parallel resonant circuits, 26, 94, 139, 149–151, 158–159, 164, 175

Parameter sweeps, 23, 65, 91, 94, 134–136, 261, 287, 398–400, 402

Parasitic capacitance, 13, 16, 25

Permeability, 9–10, 32–36, 54, 56, 67–68

free space, 9
relative, 9

Permittivity, 54, 67–69

Plane waves, 53–56

equations, 53–56

in a good conductor, 55–56

in a lossless medium, 53–55

phase velocity, 54

Power gain circles, 353–354, 376, 388

Q

Q factor, 17–18, 27–29, 31, 33–39, 44–47, 51, 147, 150–154, 158–159, 161–163, 168, 185, 211, 247, 251–254, 257–260, 263, 272–273, 276–277, 280–285, 308, 310–311, 319–320.

See also Radiation Q factor

Q₀ measurement on the Smith Chart, 171–175

Q-curves, 259–264

Quarter-wave matching networks, 277–292, 307–320

characteristic impedance, 278–279, 282, 284, 286, 288

fractional bandwidth, 279–282, 284, 286, 289–292

loaded Q factor, 280–282, 284

matching bandwidth, 277, 280, 286–292

power loss, 277, 285–286, 290–292, 319

Quarter-wave transformer, 63, 110–111, 277, 281, 288, 307–308, 311–312, 314, 316

R

Radiation Q factor, 161, 163

Reactance power of capacitor, 26, 125, 149, 224

Reactance power of inductor, 148

Rectangular waveguide, 82–85, 111

characteristic impedance, 84

cutoff frequency, 84

field pattern, 83

intrinsic impedance, 84

propagation mode, 82–84

TE mode, 83

TM mode, 83

wavelength, 84

Reentrant modes, 204–205, 214

Reflected power. *See* Normalized waveforms

Resistivity, 5, 9–10, 12, 17, 71, 80, 86

Resistors,

chip, 14–16, 28–29, 44, 50

leaded, 12–14

thick film, 14–15

Resonant circuits, 26, 147–215. *See also*

Parallel resonant circuits, Series resonant circuits

Resonator decoupling, 155–158

Resonator Q circle, 170, 172, 174

Resonator series reactance, 166–175

Return loss, 53, 61, 64–66, 75, 101, 105–

106, 110–111, 124, 145, 166, 180–

183, 185, 214, 230–233, 238–242,

246–251, 253, 256, 264–266, 270–

273, 280–286, 289–292, 296, 304,

307–308, 310–311, 314, 318–320,

322–323, 330, 334–336, 343–345,

359, 363, 365–366, 372, 384–385,

389, 391, 394, 396–397, 400, 402,

410, 416. *See also* Transmission lines

S

Scalar S-Parameters, 123–124

input return loss, 124

insertion gain, 124

insertion loss, 124

isolation, 124
 output return loss, 124
 reverse gain, 124
Scattering matrix, 117, 119
Self inductance, 3
Self resonant frequency, 24, 26, 29, 31, 36, 39, 45, 47–48, 50–51
Sensitivity analysis, 400–404
Series resonant circuits, 48, 96, 147–148, 150–151
Single-stub matching networks, 277, 292–307, 311–314, 316–318, 367, 370–371
 analytical design, 292–298
 calculation of line length, 297–299
 calculation of stub length, 297–299
 graphical design, 301–304
Single-stub matching utility, 299–301
Skin depth, 3, 8–11, 50, 56, 160
Skin effects, 8, 10, 17, 36, 38, 71
S parameters, 28–29, 31, 44, 117–124, 127, 134–135, 145, 147, 153, 167–168, 171, 185–186, 188, 200, 222, 272, 325, 335, 339–341, 351, 359, 362–363, 365, 376, 380, 382–384, 387–389
 file tuning, 190–195
 forward transmission, 120
 input reflection coefficient, 120
 output reflection coefficient, 120
 reverse transmission, 120
Stability circle, 350–354, 357, 363–366, 369–370, 388–389
 input, 350
 output, 350
Stability factor, 324–326, 351, 353, 387
Stability measure, 324–326, 387
Stability parameters, 325–327, 365, 384, 387, 389
Step discontinuities, 98, 199
Straight wire inductance, 3–8
Stripline transmission lines, 70, 80–82
Surface roughness, 160

T

Tapped capacitor resonator, 156–157
Tapped inductor resonator, 157–158
TEM wave propagation, 68–69
Transistor stability, 323–328

Transmission line components, 91–102
 distributed capacitive, 69, 96–98
 distributed inductive, 69, 96–98
 half wavelength, 63, 295
 open circuited, 92, 94–96, 101–102, 141–143, 159, 292, 301–302
 quarter wavelength, 63, 92
 short circuited, 62, 91–94, 139–143, 159, 266, 292, 300, 303–304
Transmission line resonator, 159–166
Transmission lines, 10, 53–112, 137–143, 145, 147, 158–162, 169, 195–196, 199, 203–204, 214, 234, 260, 277, 281, 284, 288, 292–296, 298, 300–304, 319. *See also* Lossless transmission line
 attenuation constant, 53, 56, 59, 62
 broadside coupled, 102–103
 characteristic impedance, 53, 59, 61–63, 67–68, 71, 73, 77, 81, 84, 104
 coaxial, 50, 53, 68, 70–75, 80, 85–86, 89, 111, 159, 186, 189
 coupled, 53, 102–109, 159, 167, 169, 205–207
 distributed capacitive, 69, 96–98
 distributed inductive, 69, 96–98
 edge coupled, 102–104, 107–109, 205–207
 end coupled, 102–103
 equations, 57–66
 equivalent model, 53
 group delay, 53, 89–90
Microstrip, 53, 68, 70, 76–80, 85, 89, 91–102, 104–105, 107–108, 111, 159–163, 167, 169, 186, 195–198, 205–207, 214, 292, 344, 372. *See also* Microstrip transmission lines
 parameters, 53, 57–66
 phase constant, 53, 59, 62
 reflection coefficient, 53, 59–61, 64–66, 110–111
 return loss, 53, 61, 64–66, 75, 101, 105–106, 110–111
 stripline, 68, 70, 80–82. *See also* Stripline transmission lines
 terminations, 62–63, 94
 two parallel wire, 56
 voltage standing wave ratio, 53, 60–61

U

Under-coupled resonator, 172–173
Unloaded Q, 151, 162–163, 168–169,
175, 181, 214, 264–265

V

VBScript code, 191–194, 268–269, 299–
300
listing, 191–194
slider control, 192–194, 268–269, 299
Velocity of light, 1, 55
Voltage standing wave ratio (VSWR),
53, 60–61, 64–66, 75, 105–106, 109–
112, 134–137, 144–145, 182, 225,
265

W

Wavelength, 1–2, 49, 53, 55, 57, 63, 68–
69, 84, 92–93, 97, 111, 117, 138–139,
141, 143, 145, 160–162, 217, 277,
295–296, 337

Y

Yield analysis, 391–416

About the Authors

Ali Behagi and Stephen Turner bring a unique perspective to this subject material. Their long association of over 25 years blends a strong academic and industrial background to the text.

Ali A. Behagi received the Ph.D. degree in electrical engineering from the University of Southern California and the MS degree in electrical engineering from the University of Michigan. He has several years of industrial experience with Hughes Aircraft and Beckman Instruments. Dr. Behagi joined Penn State University as an associate professor of electrical engineering in 1986. He has devoted over 20 years to teaching microwave engineering courses and directing university research projects. While at Penn State he received a National Science Foundation grant to establish a microwave and RF engineering lab and the Agilent ADS software grant to use in teaching high frequency circuit design courses and laboratory experiments. After retirement from Penn State he has been active as an educational consultant. Dr. Behagi is a Keysight Certified Expert, a Senior Member of the Institute of Electrical and Electronics Engineers (IEEE), and the Microwave Theory and Techniques Society.

Stephen D. Turner has over 30 years of experience in the microwave industry with focus on amplifier, oscillator, and system design. He is an avid microwave CAD enthusiast going back to Touchstone v1.0. Having familiarity with all of the major commercial microwave CAD products, these days he is a proponent and user of the Keysight/Agilent Advanced Design System (ADS) and Genesys. He received the Master of Engineering degree from Penn State University and a BS degree from the University of Pittsburgh. He is a registered professional engineer in the state of Pennsylvania and member of the Institute of Electrical and Electronics Engineers and the Microwave Theory and Techniques Society.