FilterSolutions Help

ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
https://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

Release 2024 R2
July 2024

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001:2015 companies.

Copyright and Trademark Information

© 2002-2024 ANSYS, Inc. Unauthorized use, distribution or duplication is prohibited.

ANSYS, Ansys Workbench, AUTODYN, CFX, FLUENT and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries located in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license. All other brand, product, service and feature names or trademarks are the property of their respective owners. FLEXlm and FLEXnet are trademarks of Flexera Software LLC.

Disclaimer Notice

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFIDENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS. The software products and documentation are furnished by ANSYS, Inc., its subsidiaries, or affiliates under a software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, compliance with exporting laws, warranties, disclaimers, limitations of liability, and remedies, and other provisions. The software products and documentation may be used, disclosed, transferred, or copied only in accordance with the terms and conditions of that software license agreement.

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001: 2015 companies.

U.S. Government Rights

For U.S. Government users, except as specifically granted by the ANSYS, Inc. software license agreement, the use, duplication, or disclosure by the United States Government is subject to restrictions stated in the ANSYS, Inc. software license agreement and FAR 12.212 (for non-DOD licenses).

Third-Party Software

See the legal information in the product help files for the complete Legal Notice for Ansys proprietary software and third-party software. If you are unable to access the Legal Notice, please contact ANSYS, Inc.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Table of Contents

Table of Contents **Contents-1**

1 - Welcome to FilterSolutions Help **1-1**

Resources 1-1

Help Topic Conventions 1-2

Using the Online Help Search Function 1-2

FilterSolutions Terminology Table 1-4

Engineering Notation 1-12

Legal Notices 1-12

System Requirements 1-13

Obtaining Information about the Software and Release 1-13

2 - FilterQuick **2-1**

FilterQuick Overview 2-1

FilterQuick Lumped Topologies 2-3

FilterQuick Distributed Topologies 2-6

FilterQuick Active Topologies 2-11

FilterQuick Switched Capacitor Topologies 2-15

FilterQuick Digital Topologies 2-18

FilterQuick Overview Video 2-21

3 - Automated Parameter Synthesis **3-1**

Automated Parameter Synthesis 3-1

Constricted Equiripple Pass Bands and Stop Bands 3-1

User Selection 3-2

Implementation 3-3

Constricted Ripple Video Tutorial 3-5

Single Point Ripple 3-5

Half Band Ripple 3-9

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Bessel and Gaussian Stop Bands ... 3-12
Minimum Inductor Zigzags and High/Low 2 ... 3-15
Manual Tx Zero Placement ... 3-16
Positioning Zeros With the Mouse ... 3-18
4 - Descriptions of Filters ... 4-1
Low Pass, High Pass, Band Pass, and Band Stop ... 4-1
Pass Band and Stop Band Definitions ... 4-2
Frequency Asymmetrical Band Pass Filters ... 4-5
Parametric Filters ... 4-6
Source Resistor Restrictions and Compensation ... 4-7
Setting the Stop Bands ... 4-7
Gaussian Filters ... 4-8
Transitional Filters ... 4-8
Synthesis Options ... 4-8
Stop Bands and Transmission Zeros ... 4-9
Bessel Filters ... 4-9
Linear Phase Filters with Equiripple Group Delay ... 4-9
Butterworth Filters ... 4-11
Asymmetrical Band Pass filters ... 4-12
Legendre Filters ... 4-13
Inverse Legendre Filters ... 4-13
Chebyshev Type I Filters ... 4-14
Asymmetrical Band Pass filters ... 4-14
Stop Band Zeros ... 4-14
Constricted Equiripple ... 4-15
Chebyshev Type II Filters ... 4-15
Asymmetrical Band Pass Filters ... 4-16
Hourglass Filters ... 4-16

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Elliptic Filters ... 4-16
Constricted Equiripple Passband ... 4-17
Frequency Asymmetrical Elliptic Band Pass Filters. ... 4-17
Custom Filters ... 4-18
Prototypes ... 4-18
Actual, S Transform ... 4-18
Actual, Mag Freq ... 4-18
G Values ... 4-18
Digital ... 4-19
Formats ... 4-19
G Values ... 4-21
Raised Cosine Filters ... 4-21
Raised Cosine Impulse Response ... 4-23
Data Transmission Filters ... 4-24
Matched Filters ... 4-25
Continuous Approximations of Matched Filters ... 4-25
Impulse Error ... 4-26
Delay Filters ... 4-27
Ripple Period ... 4-28
Ripple Magnitude ... 4-28
5 - Using FilterSolutions ... 5-1
The Control Panel ... 5-1
Control Panel Group Box Definitions ... 5-1
Filter Attributes Selections Definitions ... 5-2
Multi-Band Definitions Spread Sheet ... 5-13
Custom Filters ... 5-17
Prototypes ... 5-17
Actual, S Transform ... 5-18

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Actual, Mag Freq 5-18
G Values 5-18
Digital 5-18
Formats 5-18
G Values 5-20
Adding Stop Band and Off-Axis Zeros 5-21
Lumped Selections Definitions 5-23
Distributed Selections Definitions 5-31
Active Selections Definitions 5-42
Switched Capacitor Selections Definitions 5-47
Digital Selections Definitions 5-49
Main Menu Selections Definitions 5-51
Setting the Q (Lumped filters only) 5-56
Setting Series and Parallel Parasitic Resistance (Lumped filters only) 5-56
Saving on a common network server (Floating licenses only) 5-57
Options Menu 5-61
Engineering Notation 5-65
Transfer Functions 5-66
Continuous Transfer Function Display 5-66
Discrete Transfer Function Display 5-68
Transfer Function Vectors 5-70
Graphical Displays 5-75
Pole Zero Plot Display 5-75
Pole Zero Plot Modifications 5-78
Time Display 5-79
Frequency Display 5-81
Reflection and Impedance Displays 5-84
Smith Charts and Polar Plots 5-86

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Smith Charts 5-87
Supported Displays 5-87
Cursor Data 5-87
Change Graph Limits 5-89
Circuit Displays 5-90
Lumped Schematic Display 5-90
Distributed Schematic Display 5-96
Active and Switched Capacitor Circuit Display 5-100
Editing Schematics Video 5-103
6 - Delay Equalizers 6-1
About Delay Equalizers 6-1
All Pass Pole Zero Placement 6-4
Automatic Pole Zero Placement 6-8
Band Pass Filter Delay Equalizers 6-8
Digital and Transmission Line Equalizers 6-9
Off Axis Quadruplet and Dual Zeros 6-9
Seeding The Off-Axis Zeros 6-9
Band Pass Filters 6-10
Adjusting the Zero Locations 6-10
Synthesizing The Filter 6-12
Group Delay and Phase Deviation 6-13
Group Delay 6-15
Phase Deviation 6-16
Phase Deviation Regression 6-16
6-17
Custom Regression Limits 6-19
Bridged Element Delay Equalization Video 6-20
7 - Arithmetic Frequency Symmetry 7-1

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Arithmetic Symmetry Definition and Purpose 7-1
Gaussian Example 7-1
Arithmetic Group Delay Equalization 7-3
Arithmetic Bessel and Gaussian Wide Band Equalization 7-5
Asymmetric Improvements 7-6
8 - User Data Analysis **8-1**
User Data Analysis 8-1
Sampling 8-2
Digital Filters 8-2
Output Data 8-2
User Data Files 8-3
9 - Parts Libraries **9-1**
Parts Libraries 9-1
Customizing Libraries 9-5
User Library Directories 9-7
Designing With Manufacturer's Parts 9-7
10 - Zmatch Impedance Matching **10-1**
General Information 10-1
Zmatch Impedance Matching 10-1
Using Zmatch 10-1
The Control Panel 10-1
Tuning Matching Circuits 10-2
Lumped Impedance Matching 10-5
Concepts and Theories 10-5
Frequency Matching Networks 10-6
Reflection Coefficient 10-6
Reactive Elements 10-7
Inductors 10-7

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Capacitors 10-7
Single Point Impedance Matching 10-8
Single Element Match 10-8
Double Element Match 10-8
Triple Element Match 10-9
Multiple Point Impedance Matching 10-9
Broadband Impedance Matching 10-10
Distributed Impedance Matching 10-10
Concepts and Theories 10-10
Frequency Matching Networks 10-11
Reflection Coefficient 10-11
Transmission Line Stubs 10-12
Transmission Line Stub Impedance 10-12
Transmission Line segment impedance 10-12
Single Point Impedance Matching 10-13
Double Stub Match 10-14
Multiple Point Impedance Matching 10-15
Broadband Impedance Matching 10-16
11 - Multi-Band Filters 11-1
Low Pass Multi-Band 11-1
High Pass Multi-Band 11-2
Band Pass Multi-Band 11-2
Band Stop Multi-Band 11-3
Trap Topology 11-4
12 - Lumped Filters 12-1
Passive LC Features 12-1
Circuit Synthesis and Analysis 12-2
Terminations 12-2

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Singly and Doubly Terminated Filters 12-3
Doubly and Equally Terminated Filters: 12-4
Singly Terminated Filters: 12-4
Impedance Matching 12-4
Reactive Element Parasitic Models 12-6
Inductors 12-6
Capacitors 12-7
Node Capacitance and Lead Inductance 12-7
Node Capacitance and Lead Inductance Models 12-8
About Diplexers 12-9
Contiguous Diplexers 12-9
Noncontiguous Diplexers 12-10
Triplexers 12-10
Band Pass Diplexers 12-11
Coupled Resonator Band Pass Filters 12-16
Swap End LC for More Symmetric Response 12-19
Coupled Resonator Example: 12-20
Tubular Band Pass Filters 12-22
Outer Shunt Inductors 12-23
Low Pass Approximation Tubulars 12-26
Improving Frequency Symmetry 12-27
Cross Coupled Resonator Filters 12-29
Design Flexibility 12-30
Output 12-30
Required Criteria: 12-30
Finite Q Analysis 12-31
Sensitivity and Monte Carlo Analysis 12-31
Optional Views 12-31

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Quadruplet Coupled Resonator Band Pass Filters 12-34
FilterQuick 12-34
Coupling Matrix Differences 12-34
Bridged Tubular and Resonator Filters 12-36
Design Flexibility 12-37
Required Criteria: 12-37
Minimum Inductor Zigzag Filters 12-38
Even Order Zigzags 12-38
Odd Order Zigzags 12-38
Setting the Source Resistor 12-38
Equal Inductors 12-39
Negative Element Values 12-39
Zigzag Filter Example 12-39
Matching Unequally Terminated Filters 12-41
Automatic Matching of Unequal Terminated Band Pass Filters 12-43
Matching Minimum Zigzag Filters 12-44
Low Pass, High Pass, and Band Stop Filters 12-45
Bridged T Topologies 12-45
Equal Inductor Band Pass Filters 12-46
All Pole Filters 12-46
Stop Band Filters ("Parametric" Filters) 12-47
Cascaded High and Low Pass Topologies 12-49
Source Resistor 12-49
All Pole Filters 12-50
Stop Band Filters 12-50
Asymmetrical Band Pass Filters 12-50
Amplitude Equalizers 12-52
Shunt Amplitude Equalizers 12-52

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Series Amplitude Equalizers ... 12-53
Constant Resistance Amplitude Equalizers ... 12-54
Effect of Amplitude Equalization ... 12-56
Shunt Node Capacitance Compensation ... 12-57
Even Order Modification ... 12-58
Description of the Problem ... 12-58
All Pass Passive Circuit Elements ... 12-60
Real All Pass Circuit Elements ... 12-60
Complex All Pass Circuit Elements ... 12-60
Balanced Passive Circuits ... 12-63
Balanced All Pass Sections ... 12-64
Working With Lumped Filters ... 12-66
Modify Lumped Circuit Elements ... 12-66
Part Tolerances ... 12-74
S-Parameter and Spice Model Imports ... 12-75
Update to Nearest Matching S-Parameter or Spice Model File ... 12-78
Automatic Design to Nearest Matched S-Parameter or Spice Model File ... 12-78
Parts Library Menu ... 12-79
S-Parameter and Spice Model Exports ... 12-79
Inductor and Capacitor Tuning ... 12-80
Add or Delete Circuit Elements ... 12-82
Splitting Elements ... 12-84
Balancing individual elements ... 12-84
Norton, PI, and T Transformations ... 12-85
To perform a Norton Transformation: ... 12-86
Negative Element Values ... 12-87
Pi to T and T to PI Transformation ... 12-87
Norton Theory ... 12-87

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Impedance Matching 12-88
LC Transformations 12-89
Richards Transforms 12-91
Kuroda Transforms 12-93
Setting the Q (Lumped filters only) 12-97
Setting Series and Parallel Parasitic Resistance (Lumped filters only) 12-97
Saving on a common network server (Floating licenses only) 12-98
Complex Terminations 12-102
Element Tuning 12-102
Impedance Compensate 12-103
Filter Display 12-103
Lumped Netlists 12-103
Finite Q 12-104
Coupled Coils 12-104
Dummy Resistors 12-104
Sensitivity 12-105
Monte Carlo Analysis 12-105
Selecting element tolerance 12-106
Example 12-106
Element Sensitivity Tables 12-108
Sensitivity Definitions 12-108
Frequency Selection 12-109
Specific element sensitivity analysis 12-109
Element Sensitivity Plots 12-109
Example: 12-110
Modelithics Exports to Electronic Desktop 12-110
Modelithics in FS and EDT 12-110
Loading Modelithics into Circuits 12-111

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selecting Modelithics Models 12-111
Selecting Multiple Families 12-112
Defining the Substrate 12-113
Circuits Schematics and Layouts 12-114
Tuned/Optimized Layouts 12-115
Automated Discrete Design with Modelithics Tutorial 12-116
About designing with FilterSolutions and Modelithics: 12-116
AEDT Export Model Selection: 12-117
AEDT Substrate Definition 12-117
Initial and Updated Interconnect Geometry 12-119
Multiple Family Model Selection 12-120
About AEDT Optimization 12-120
AEDT Schematic Selection 12-120
AEDT Optimization Process 12-121
Automated Discrete Optimization 12-127
Automated Discrete Optimization 12-127
Modelithics 12-128
Steps for Automated Discrete Optimization: 12-128
Ideal Design Synthesis 12-128
Optional Preoptimizing With Nonlinear Programming Quadratic Lagrangian or Other Optimizer 12-129
Raw Modelithics Export to Ansys Circuits 12-130
Optional Manual Tuning 12-130
Discrete Optimize with Modelithics Parts 12-131
Full parts and Interconnect Optimization 12-132
Retrieving Partially Optimized Variables 12-133
Exporting to Other Software 12-133
Modelithics in FS and EDT 12-134

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Loading Modelithics into Circuits 12-134
Selecting Modelithics Models 12-134
Selecting Multiple Families 12-135
Defining the Substrate 12-136
Circuits Schematics and Layouts 12-137
Tuned/Optimized Layouts 12-138
Touchstone Files 12-139
Port Definitions: 12-139
Tips: 12-140
13 - Distributed Filters 13-1
Transmission Line Concepts 13-1
Transmission Line Equations 13-1
Striplines and Microstrips 13-3
RLGC, Microstrips, Striplines, and Suspended Substrates 13-3
Coupled Lines 13-4
Losses 13-4
Attenuation 13-4
Substrate Geometry Definitions 13-4
Inductor Approximations 13-6
Tx Segment Zo 13-7
Capacitor Approximations 13-8
Tx Segment Zo 13-9
Radial and Delta Stubs 13-10
Radial Stub Geometry 13-10
Delta Stub Geometry 13-11
Parallel Resonator Approximations 13-11
Coupled Transmission Lines 13-13
Series Resonator Approximations 13-14

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Stepped Impedance Filters 13-15
Spaced Stub Filters 13-16
Examples: 13-17
Open and Short Terminations 13-19
Open End Models 13-19
Shorted End Models 13-20
Transmission Line Prewarping 13-20
Prewarping Exceptions 13-21
Prewarping Equations 13-21
Transmission Signal Propagation 13-22
Lumped Circuit Approximations 13-22
Lumped Element Replacements 13-22
LC Resonator Replacements 13-23
Example 1 13-23
Lumped Element Models 13-25
Inductors 13-25
Capacitors 13-26
Coupled Resonator and Multiconductor Filters 13-26
Lumped Coupled Resonator Simulations 13-27
Shunt Stub Resonators 13-29
Open End Stubs 13-30
Narrow Gaps: 13-31
Equal Width Conductors 13-33
Multiconductor Tx Line Models 13-33
RLGC: 13-33
Striplines: 13-33
Microstrips and Suspended Substrates: 13-34
Optimization: 13-34

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Analysis Calculations: 13-34
Hairpin Resonator Filters 13-34
Hairpin Options 13-34
Tuning 13-35
AEDT Tuning and Optimizing 13-35
Miniaturized Hairpin Resonator Filters 13-36
Miniaturized Hairpin Options 13-37
Manufacturing Geometry Limitations 13-37
Tuning 13-37
AEDT Tuning and Optimizing 13-37
Ring Resonators Resonator Filters 13-38
Ring Resonator Options 13-38
Manufacturing Geometry Limitations 13-39
Tuning 13-39
Third Party Tuning and Optimizing 13-39
Series Quadruplet Resonators 13-40
Designing Cascaded Quadruplet Resonator Filters 13-41
Single Point Ripple Stop Bands 13-41
Cascaded Quadruplet Topology 13-41
EM and Circuits Analysis 13-42
EM Considerations 13-42
Interdigital Filters 13-43
Wideband Interdigital Filters 13-43
Tapped Interdigital Filters. 13-43
Pinned Interdigital Filters 13-43
Interdigital Options 13-43
Manufacturing Geometry Limitations 13-44
Tuning 13-44

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

AEDT Tuning and Optimizing 13-44
Combline Filters 13-46
Quarter Length Frequency 13-46
Taps 13-47
Combline Options 13-47
Equal Width Approximation 13-47
Manufacturing Geometry Limitations 13-47
Tuning 13-47
AEDT Tuning and Optimizing 13-47
Taps and Pins 13-49
Distributed Line Notch Filters 13-50
Wide Notch Filters 13-50
Narrow Notch Filters 13-50
Equal Line Widths 13-50
Open Stub Grounds 13-50
Working With Distributed Filters 13-51
Simulating Transmission Lines 13-51
Optimizing 13-52
Finalizing 13-52
Distributed Filter Materials 13-52
Series Overlay Capacitors 13-52
Overlay Concept 13-52
Overlay Geometry 13-53
Overlay Controls 13-53
Optimizing the Frequency Response 13-55
Example: 13-55
Setting Geometry Limits 13-57
Adjusting Line Lengths to Maintain Equivalent Impedance 13-57

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Controlling Stub Widths 13-60
Layouts and Schematic Views 13-61
Schematic view: 13-61
Layout Geometry Definitions 13-62
Multiconductors 13-62
Coupled Resonators 13-66
Ladder Structures 13-67
Radial and Delta Stub Geometry 13-68
Suspended Substrate Overlay Capacitors 13-69
Modify Distributed Filters 13-69
Monte Carlo Analysis 13-74
Multiconductor Filters 13-76
S-Parameter and Spice Model Imports 13-76
Update to Nearest Matching S-Parameter or Spice Model File 13-78
S-Parameter and Spice Model Exports 13-79
Alter Transmission Line Designs 13-80
Adding All Pass Sections 13-81
Segments Between Stubs 13-82
Tuning Planar Filters 13-82
Third Party Tuning 13-84
Composite Lumped/Line Filters 13-84
Coupled Resonator Band Pass Filters 13-85
Transmission Line Netlists 13-88
Ideal Lines 13-88
Striplines and Microstrips 13-88
Real Lines 13-89
Coupled Lines 13-89
Lumped Elements 13-89

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Dummy Resistors 13-89
Example: 13-89
Exporting to Ansys Electronic Desktop 13-90
Ansys Nuhertz Export/Import Panel 13-91
Ansys Iron Python Script 13-94
Direct Export 13-95
IronPython Script Export 13-95
Retrieving Design Variables 13-97
Optimizing in FilterSolutions and HFSS 13-99
Optimizing in FilterSolutions 13-99
Exporting to HFSS 3D Layout 13-101
HFSS 3D Layout Stackup 13-101
Setting up the Substrate In FilterSolutions 13-103
Symmetric Substrates 13-105
Stack Up Vias 13-105
Exporting to HFSS 3DL Schematics 13-105
Schematic Elements and Layout 13-106
HFSS 3D Layout Port Configurations 13-107
Lumped Ports 13-107
Wave Ports 13-108
Exporting to HFSS 13-109
HFSS Schematics 13-109
Schematic Elements 13-109
HFSS Port Configuration 13-111
Lumped Ports 13-111
Wave Ports 13-112
Exporting to Ansys Planar Circuits 13-113
Planar Ansys Circuits Stack Up 13-113

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting up the Substrate In FilterSolutions 13-113
Advanced Panel 13-115
Circuits Stack Up 13-115
Planar Circuits Schematics 13-115
Schematic Elements and Layout 13-116
Ansys Circuits Optimization 13-117
Optimizing in FilterSolutions 13-117
Automated Port Tuning 13-119
General Information 13-119
Automated Port Tuning Description 13-120
General 13-120
Resonator Length Updates 13-120
Resonator Separation Adjustments 13-121
Tuning Port Error 13-122
Optimization Goals 13-122
Reconstructing the Planar Geometry 13-123
Final Design Optimizations. 13-123
Automated Port Tuning Goals 13-123
Ansys Circuits Optometrics 13-123
FilterSolutions Goals Table 13-124
Reconstructing the Filter Design 13-125
Reconstructing the Resonator lengths 13-125
Adjusting the Resonator Separation 13-126
Ring Resonator Gap Tuning 13-128
Ansys Desktop Automated Port Tuning 13-129
Horizontal Series Ports 13-129
Deembedding the Tuning Ports 13-130
Desktop Automated Port Tuning Entries 13-131

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Circuits/Sonnet Automated Port Tuning 13-133
Exporting Sonnet Tuning Port format to AEDT 13-134
Automated Port Tuning Tutorial 13-135
Introduction 13-135
Sonnet Tuning Port Projects 13-151
Sonnet Tuning Port Project 13-152
Sonnet Automated Port Tuning Tutorial 13-153
Introduction 13-153
Parameter Optimization with Sonnet Tutorial 13-170
Exporting to Other Software 13-182
Modelithics in FS and EDT 13-182
Loading Modelithics into Circuits 13-182
Selecting Modelithics Models 13-183
Selecting Multiple Families 13-184
Defining the Substrate 13-184
Circuits Schematics and Layouts 13-186
Tuned/Optimized Layouts 13-187
3D Data 13-188
DXF AutoCAD 13-189
Exported Features 13-189
DXF Shapes and VIAs 13-189
Geometry Dimension Units 13-190
Reference Planes and BoxBoundaries 13-190
Touchstone Files 13-190
Port Definitions: 13-190
Tips: 13-191
Illustrations: 13-191
Sonnet Exports 13-192

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Exporting to Sonnet 13-193
Tips: 13-196
Modelithics® in Sonnet 13-196
Editing Modelithics in Sonnet 13-197
14 - Active Filters **14-1**
Thomas Biquads 14-1
Advantages and Disadvantages 14-3
Thomas Biquad Design Equations 14-3
Thomas Biquad With Third Additional Pole 14-4
Akerberg-Mossberg Description 14-4
Akerberg-Mossberg Biquad 14-4
Miller Integrator 14-5
Akerberg-Mossberg Design Equations 14-6
Akerberg-Mossberg With Third Additional Pole 14-6
Sallen & Key 14-6
Sallen & Key Single Amplifier Biquads 14-7
Sallen & Key Equal Element Stages 14-9
Low Pass Stage: 14-9
High Pass Stage: 14-9
Twin T Notch Configuration 14-10
Twin T Stage 14-10
Sallen & Key Fourth Order Stages 14-10
Disadvantages 14-11
Multiple Feedback (MFB) 14-11
MFB Single Amplifier Biquads 14-12
Positive Feedback Enhancement 14-13
MFB Alternate Notch Configuration 14-16
MFB Fourth Order Stages 14-17

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Disadvantages 14-18
Leap Frog Active Filters 14-18
Leap Frog Active Filters 14-18
Synthesis of Filters 14-19
Leap Frog Transmission Zeros 14-20
Low pass and high pass leap frog transmission zeros: 14-21
Band Pass Transmission Zeros: 14-23
Parallel Active Filters 14-23
Parallel Active Filters 14-23
Parallel Stage Gain Settings 14-24
Example 14-24
GIC Stages 14-25
Low Pass Unity Gain: 14-26
High Pass Unity Gain: 14-26
Band Pass Unity Gain: 14-27
Even Gain Notch: 14-27
Low Pass Notch: 14-27
High Pass Notch: 14-28
All Pass: 14-29
Third Order GICs 14-29
GIC Ladder Filters 14-30
Options 14-30
Limitations 14-30
Impedance scaling 14-30
Inductor Simulation: 14-31
D Device Simulation: 14-31
M Device Simulation: 14-32
Bridged GIC Ladder Filters 14-32

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Required Criteria: 14-33
Active Stages 14-33
Stage Parameters 14-33
Wo and Fo 14-33
Q 14-33
Wn and Fn 14-34
Gains 14-34
Active All Pass Stages 14-34
First Order Stages 14-35
Second Order Stages 14-35
Differential Stages 14-35
Differential Op Amps 14-36
Band Pass Architectures 14-37
Active Band Pass Architectures 14-37
Which Architecture to Use: 14-38
Illustration: 14-38
Opamp and Element Parasitic Effects 14-39
Real Opamps 14-40
Resistor Parasitic Capacitance 14-41
Updates to Parasitic Shunt Capacitance 14-41
Display of Parasitic Shunt Capacitance 14-41
Parasitic Shunt Capacitance model 14-41
Active Capacitor Losses 14-42
Sensitivity 14-43
Monte Carlo Analysis 14-43
Selecting element tolerance 14-43
Example 14-43
Element Sensitivity Tables 14-45

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sensitivity Definitions 14-45
Frequency Selection 14-45
Specific element sensitivity analysis 14-45
Element Sensitivity Plots 14-45
Example: 14-46
Modify Active Filters 14-46
Capacitor and Resistor Selections 14-47
Cap Compute 14-47
Caps Select, Resis Select 14-47
Recommended Design Sequence 14-48
Modify Active Filter Stages 14-48
Modify Stage Control Panel 14-49
S-Parameter and Spice Model Imports 14-52
Update to Nearest Matching S-Parameter or Spice Model File 14-54
S-Parameter and Spice Model Exports 14-54
Active Stage Gains 14-55
Band Pass Filters 14-55
Band Stop Filters 14-56
User Entered Capacitors 14-57
Modify Active Circuit Elements 14-60
Setting the Q (Lumped filters only) 14-68
Setting Series and Parallel Parasitic Resistance (Lumped filters only) 14-68
Saving on a common network server (Floating licenses only) 14-69
Net Lists 14-73
Active Netlists 14-73
Component Indexing 14-73
Op Amps 14-73
Exporting to Other Software 14-75

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CKT Spice Files 14-75
Touchstone Files 14-76
Port Definitions: 14-77
Tips: 14-77
Illustrations: 14-78
15 - Switched Capacitor Filters **15-1**
Purpose 15-1
Purpose and Definitions 15-1
Component Definitions 15-1
Positive Switched Capacitors 15-2
Switch Models 15-2
Z Transform 15-3
Impedance 15-3
Negative Switched Capacitors 15-3
Switch Models 15-4
Z Transform 15-4
Impedance 15-4
Unswitched Capacitors 15-4
Z Transform 15-5
Impedance 15-5
Z Transforms 15-5
IIR Z Transform 15-5
IIR(Infinite Impulse response) 15-5
Bilinear 15-6
Bilinear with Prewarping 15-6
Matched Z 15-6
Impulse Invariant 15-6
Step Invariant 15-6

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modified Impulse Invariant 15-6
Modified Step Invariant 15-7
FIR Z Transform 15-7
FIR (Finite Impulse Response) 15-7
User Entered Z Transform 15-7
Which Z Transform to Use 15-7
Biquad Architectures 15-7
Double Op Amp Biquads 15-8
Negative Single Opamp Stages 15-10
Positive Single Opamp Stages 15-11
Cascade Switched Capacitor Biquads 15-12
First Order Cascaded Terms 15-12
Second Order Cascaded Terms 15-13
Cascade vs. Parallel Biquads 15-13
Parallel Switched Capacitor Biquads 15-13
First Order Cascaded Terms 15-13
Second Order Cascaded Terms 15-13
Final Stage 15-14
Parallel vs. Cascade Biquads 15-14
Band Pass Architectures 15-14
Switched Capacitor Band Pass Architectures 15-14
Integrated Band Pass Filters 15-14
High/Low Stage Band Pass Filters 15-15
Op Amps 15-15
Ideal Opamps 15-15
Modify Switched Capacitor Filters 15-15
Modify Switched Capacitor Elements 15-17
Change Control Panel 15-19

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Monte Carlo Analysis 15-21
Net Lists 15-22
Switched Capacitor netlists 15-22
Component Indexing 15-22
Switched Capacitors 15-23
Op Amps 15-23
Dummy Resistors 15-23
Exporting to Other Software 15-24
Touchstone Files 15-24
Port Definitions 15-24
Tips: 15-24
Illustrations: 15-25
16 - Digital Filters 16-1
IIR Filters 16-1
Digital Transformations 16-1
Derivations 16-1
Bilinear Transformation: 16-2
Impulse Invariant Transformation: 16-2
Modified Impulse and Step Invariant Transformations 16-3
Illustrations 16-3
Prewarping 16-7
To Correct Warping of the Frequency Axis 16-7
Prewarping Illustration 16-8
Multi-Band Filters 16-11
FIR Approximations of Analog Filters 16-12
FIR Filters 16-13
FIR General Information 16-13
FIR Window Filters 16-14

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Information 16-14
Types of Windows 16-14
Illustration of FIR Construction 16-16
FIR Optimization Options 16-17
FIR Optimization Option 1 16-18
FIR Optimization Option 2 16-18
Band Stop Filters 16-18
Example: 16-18
Iterative FIR Solutions 16-19
FIR Communications Filters 16-20
Raised Cosine Filter 16-20
Root Raised Cosine Filter 16-21
Band Pass Raised and Root Raised Cosine Filters 16-22
Cosine Filters 16-22
Sine Filters 16-23
FIR Matched and Delay Filters 16-24
Matched Filters 16-24
Delay Filters 16-25
FIR Approximations of Analog Filters 16-26
Z Transform Output 16-26
Z Transform Vectors 16-26
Button Selections 16-28
Z Transform Digital Formats 16-29
Decimal 16-29
Signed Floating Point 16-29
Unsigned Floating Point 16-29
Signed Fixed Point 16-29
Unsigned Fixed Point 16-30

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Justification 16-30
Standard Cascade, and Parallel Forms 16-31
Standard, Cascade, and Parallel Forms 16-31
Finite Precision Analysis 16-32
Finite Precision Analysis 16-32
General Information 16-32
Finite Precision Error Example 16-33
Pole Zero Plots 16-34
Digital Filter Construction 16-35
Digital Filter Construction 16-35
Deriving Digital Filters From Z Transforms 16-35
The Standard Form 16-36
Cascade Form 16-37
The Parallel Form: 16-38
Sin(x)/x Sampling Compensation 16-39
Sin(x)/x Sampling Compensation 16-40
Graphical Comparison 16-42
Generate Automatic C Code 16-44
C Autocode 16-45
General Information 16-45
Generating Code 16-45
C code Options 16-45
Calling C Code 16-47
Digital Overview Video 16-50
Index **Index-1**

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 - Welcome to FilterSolutions Help

FilterSolutions is a comprehensive PC based filter synthesis and analysis software package. FilterSolutions provides passive (lumped and distributed), active, and switched capacitor circuit synthesis and modification analysis with parasitic effects, and digital filter IIR and FIR design with C code generation, and finite precision analysis.

This sections contains the following topics:

- FilterSolutions Terminology
- Engineering Notation
- Legal Notices

> **Note:**
>
> The FilterSolutions application is supported on the Windows operating system only. This application will not be available when the Ansys Electromagnetics Suite is installed on a Linux platform.

Resources

Short Help Videos:

See the Online Help for video tutorials on the following topics:

FilterQuick Overview Video

Constricted Ripple Video

Editing Schematics Video

Designing With Manufacturer's Parts

Bridged Element Delay Equalization Video

Digital Overview Video

FilterSolutions Tutorials:

Automated Port Tuning- Automated Port Tuning with Ansys FilterSolutions, Circuit, HFSS, and HFSS 3D Layout.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sonnet Automated Port Tuning- Automated Port Tuning with Ansys FilterSolutions, Circuit, and Sonnet.

FilterSolutions, Modelithics- Automated Design with Ansys FIlterSolutions and Modelithics

FilterSolutions, Sonnet Planar Optimization- Parameter Optimization with Ansys FilterSolutions and Sonnet.

Help Topic Conventions

The following documentation conventions are used in the help:

- Procedures are presented as numbered lists. A single bullet indicates that the procedure has only one step.
- Command font is used for:
  - Command line prompts that should be typed exactly as written.
  - Script examples.
- Bold type is used for the following:
  - Names of windows, workspaces, menu commands, and options.
    - Menu commands are often separated by angle brackets. For example, **File** > **Open**.
  - Labeled keys on the computer keyboard. For example, **Enter**.
- Italic type is used for the following:
  - Emphasis.
  - Publication titles.
- The plus sign (+) is used between keyboard keys to indicate that you should press the keys at the same time. For example, “Press **Shift**+**F1**” means to press the **Shift** key and, while holding it down, press the **F1** key also. You should always depress the modifier key or keys first (for example, Shift, Ctrl, Alt, or Ctrl+Shift), continue to hold it/them down, and then press the last key in the instruction.

Using the Online Help Search Function

When you enter words or strings to search for in the help, the search engine, by default, lists all topics in which any of the words occur. For example, if you enter "voltage source" without the quotation marks, the results show all topics that contain "voltage" or "source."

This method probably provides more hits than you want. The Search function in the help provides several methods for making searches more specific.

Performing a Basic Search

1. Type the words or string in the **Search** box.
   - The **Search** box includes a drop-down filter for specifying a product but not including its scripting guide (for example, FilterSolutions), specifying a product's scripting guide, or

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

searching across all Electronics products (for example, All Products). When you change the filter, the results dynamically reflect the selected filter.

2. Click on the topic you want in the list results.
   - If you want to view a different topic, press your browser's back button to return to the list results.
   - To turn off highlighting on the current page, click the **Remove Highlights** icon .

Searching with Quotation Marks

If you enter `"voltage source"` (with quotation marks), the results show all topics that include the phrase.

As you can see, this returns far fewer results than the basic search.

To further limit the results, you can enter additional words, such as: `"voltage source" transient solver`

> **Note:**
>
> - Searches are not case sensitive, so you can type your search in uppercase or lowercase characters.
> - You may search for any combination of letters (a-z) and numbers (0-9).
> - Punctuation marks (period, colon, semicolon, comma, hyphen) are ignored during a search.
> - When searching for a file name with an extension, group the entire string in quotation marks (for example, "filename.ext").

Using Boolean Operators

These choices affect the number of topics listed.

Operator (s)	Usage	Example(s)
AND + &	Lists all topics that contain all of the terms.	Net AND Selection Net + Selection Net & Selection
OR |	Lists all topics that contain any of the terms.	Net OR Selection Net | Selection
NEAR	Lists all topics that contain the terms near the other terms.	Net NEAR Selection
NOT	Lists all topics that	Net NOT Selection

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Operator (s)	Usage	Example(s)
! ^	contain the first term but not the second.	Net ! Selection Net ^ Selection

Use parentheses to group terms and operators. For example:

- solver AND (Circuit) NOT HFSS NEAR dynamic
- solver AND (HFSS OR Circuit) NOT (Q3D OR 2d) NEAR dynamic
- "dynamic link" ! (HFSS | circuit)

Important:

Because the characters +, &, |, !, and ^ are used as operators, you cannot search for them in the help. Doing so will result in an error.

FilterSolutions Terminology Table

Prewarp	When a continuous frequency response is transformed to a digital frequency response, aliasing occurs at half the sample rate of the digital domain. Aliasing produces errors in the digital frequency response. When Bilinear Transformation is selected and "Alias Correction" is checked in the Digital Dialog box, FilterSolutions will correct the aliasing error at the edge(s) of the pass band. `
All Pass	A transfer function or filter segment that affects only the phase and group delay of the filter without affecting the magnitude response. Useful for creating delay equalizers
Amplitude Equalizer	An RCL compensation network added to lumped passive filters with finite Q for the purpose of equalizing the magnitude frequency response

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Bilinear	A continuous to digital transformation that maps the entire continuous JW axis to the digital unit circle. Subsequent aliasing errors may be corrected by checking the Prewarp box.
Biquad	Officially, this is a complex pole pair. Filter Solutions uses this term loosely to represent any polynomial of two orders or less.
Cascade Form	A transfer function or filter implementation that is constructed as the product of two or more sections of second order or less.
Center Frequency	In a band pass or band stop filter, the geometric average of the upper and lower cutoff frequencies is the Center Frequency.
Corner Frequencies	In a band pass or band stop filter, the upper and lower cutoff frequencies are the corner frequencies.
Coupled Resonator	LC resonant pairs separated by a capacitor or inductor. Usually done for the purpose of approximating a narrow pass band in a band pass filter.
Delay Equalizer	A addition to a filter composed of all pass sections with pole and zeros that have been strategically positioned to "Flatten" the group delay of the filter pass band.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FIR Filter	Finite Impulse Response filter. Those filters whose impulse lasts for a finite duration of time before assuming a value of exactly zero. Only specially designed digital filters are FIR Filters.
Group Delay	The transport delay of a specific frequency. Calculated by the negative of the frequency derivative of the phase angle. Usually expressed in seconds.
IIR Filter	Infinite Impulse Response filter. Those filters whose impulse response asymptotically approaches zero. The digital transformations of all analog filters are IIR filters.
Impulse	For the continuous domain, this is a time curve of zero width, infinite amplitude, and a time/magnitude product of one. For the digital domain, a single pulse with an amplitude equal to the sample frequency in Hertz.
Impulse Invariant	A continuous to digital transformation that is designed to preserve the impulse response of the continuous domain.
Magnitude	The amplitude ratio of the output to the input of a filter at a specified frequency or a range of frequencies. The magnitude may be an arithmetic ratio, or it may be expressed in dB's.
Matched Filter	A filter whose time response approximates a ramp. For use in communications.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parallel Form	A transfer function or filter implementation that is constructed as the sum of two or more sections of second order or less.	
Pass Band	The portion of the frequency response where the magnitude is greater than the pass band attenuation magnitude.	
Pass Band Attenuation	The edge of the pass band, or pass bands in the case of band stop filters, is defined at some attenuation level depending on the filter according to the following schedule.	
	Gaussian Bessel	The pass band attenuation is a function of the order of the filter
	Butterworth Chebyshev II Hourglass	-3.01 dB
	Chebyshev I Elliptic	The pass band ripple magnitude
Pass Band Frequency	The frequency at edge of the Pass Band. Also known as the "Cut Off: frequency".	
Pass Band Return Loss (dB)	The pass band return loss of Chebyshev I and Elliptic filters oscillate up and down within a user-specified boundary. The higher the order of the filter, the more oscillations occur. These oscillations are known as Pass Band Ripple.	

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Note: An option exists to replace the pass band return loss with pass band ripple.
Pass Band Ripple (dB)	The pass band magnitude of Chebyshev I and Elliptic filters oscillate up and down within a user-specified boundary. The higher the order of the filter, the more oscillations occur. These oscillations are known as Pass Band Ripple. Note: An option exists to replace the pass band ripple with pass band return loss.
Pass Band Width	The difference in frequency between the two cut off frequencies in a band pass or band stop filter.
Phase Angle	The difference in phase angle between the output and the input of a filter at a specified frequency or a range of frequencies. The Phase Angle may be expressed in degrees or radians.
Prewarp	When a continuous frequency response is transformed to a digital frequency response, aliasing occurs at half the sample rate of the digital domain. This aliasing produces errors in the digital frequency response. When Bilinear Transformation is selected and "Prewarp" is checked in the Digital Dialog box, FilterSolutions will prewarp the analog frequency response such that the error in the digital frequency response is removed at the filter cutoff frequency(s).
Prototype Filter	Low pass filter with a cutoff frequency of one radian per second. Used as a design baseline for

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	more complex filters.
Q	In a complex pole pair, this is the magnitude of the pole divided by twice the real component. In a lumped passive element, this is the measure of loss. Infinite Q is ideal. Real devices have a finite Q.
Quadruplet Zeros	Refers to a set of four zeros equally spaced apart in the real and imaginary axis. Useful for delay equalization when passbands are calculated around them.
Raised Cosine	A filter magnitude response that is has the form of a raised cosine function.
Ramp	A time curve that starts at time zero, and is equal to the time in seconds.
Reflection	Passive filters (lumped and distributed) with finite source resistance reflect a portion of the signal back out onto the line where they came from. FilterSolutions computes and displays the reflection coefficient as a function of frequency for both the filter design and the user modified filters. The ideal reflection coefficient is square root function and has two solutions that differ by 180 deg, only one of which is displayed. Actual filters may be 180 deg out of phase with the ideal displayed value.
Root Raised	A filter magnitude response that has the form of a

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cosine	square root of a raised cosine function.
SAB	Single Amplifier Biquad. Refers to active filter stages that only utilize one opamp.
Source Resistor	For voltage driven circuits, this is the series resistance of the voltage source. For current driven circuits, this is the parallel resistance of the current driver.
Standard Form	A transfer function or filter implementation that is constructed as a single section. 1st and 2nd order Standard Forms may be summed or multiplied together to create a Parallel or Cascade form respectively.
Step	A time curve that starts at time zero, has an amplitude of one, and is of infinite duration.
Step Invariant	A continuous to digital transformation that is designed to preserve the step response of the continuous domain.
Stop Band	The stop band filters, Chebyshev II, Hourglass, and Elliptic, have a stop band that consists of one or more humps and a series of notches.
Stop Band Attenuation	The magnitude of the top of the humps in a stop band.
Stop Band	For a low pass or high pass filter, a horizontal line

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Frequency	drawn across the top of the hump(s) in the stop band magnitude curve will intersect a point on the curve that rises toward the pass band. The frequency where this occurs is called the stop band frequency.
Stop Band Ratio	Also known as Ws, the pass band ratio is the ratio of the pass band frequency or width and the stop band frequency or width. Always greater than 1.0. For low pass filters it is the stop band frequency over the pass band frequency. For high pass filters it is the pass band frequency over the stop band frequency. For band pass filters it is the stop band width over pass band width. For band stop filters, it is the pass band width over stop band width. The advantage of defining a stop band with the ratio rather than the frequency or width is that you may easily change from one class of filter to another or change the pass band frequency or width without adjusting the stop band frequency or width each time.
Stop Band Width	For a low band pass or band stop filter, a horizontal line drawn across the top of the hump(s) in the stop band magnitude curve will intersect two points on the curve that rises toward the pass band. The difference in frequency between these two points is called the stop band width.
Tubular	Narrow band pass topology with alternating inductors and capacitors for series elements, and all nodes connected to a grounded capacitor.
Zig Zag Filter	A band pass filter composed of "Zig Zagging" LC stages for the purpose of minimizing the number

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	of inductors in a band pass filter.

Engineering Notation

For the designer's convenience, FilterSolutions allows numbers with large or small exponents to be entered in standard engineering terms. Units may be omitted is desired. For example, a 120 pF capacitance may be entered as 1.2E-07, 120p, or 120pF. FilterSolutions will interpret any of these as 1.2E-07.

The table below provides standard engineering notation for the designer's reference.

Notation	Description	Value
f	femto	E-15
p	pico	E-12
n	nano	E-09
u	micro	E-06
m	milli	E-03
K	Kilo	E+03
M	Mega	E+06
G	Giga	E+09
T	Tera	E+12
P	Pecta	E+15

Legal Notices

Trademarks

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Trademarks

NUHERTZ®, NUHERTZ TECHNOLOGIES®, FILTER SOLUTIONS®, and FILTER QUICK(TM) are trademarks or registered trademarks of ANSYS, Inc. or its subsidiaries in the United States and other countries.

Copyright Notice

Copyright (C) 2021 ANSYS, Inc. All right reserved. Unauthorized use, distribution, or duplication is prohibited.

System Requirements

FilterSolutions supports certain versions of the Windows operating system. For supported platforms and system requirements, go to the Platform Support website and select the following document:

Ansys 2024 R2 **- Platform Support by Application / Product (PDF)**

This document covers all Ansys products. Refer to the **Electronics Applications** section.

Limitations of Linux Installations:

While the majority of the Ansys Electromagnetics Suite applications and features are supported on both Windows and Linux platforms, some, including FilterSolutions, are not supported on Linux. See the following topic for details:

Windows vs. Linux Installations

Obtaining Information about the Software and Release

To obtain information about the software and release, click **File** > **About FilterSolutions**.

The **About ANSYS® Electromagnetics Suite [release number]** window appears, listing information about the product.

To view a PDF of changes in the latest release, click **File** > **What's New**.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2 - FilterQuick

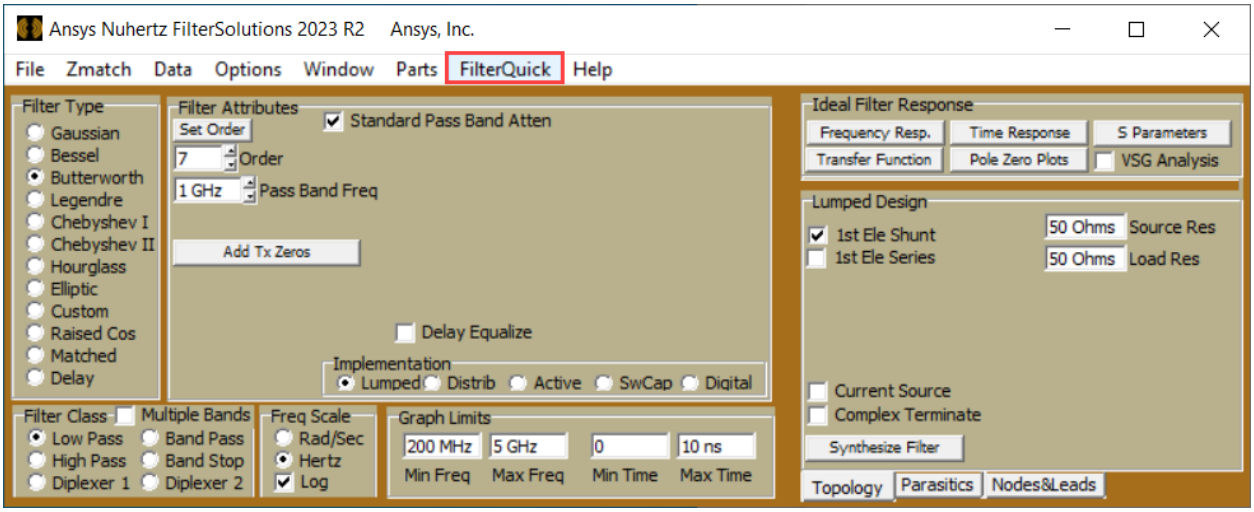
This chapter contains the following topics:

- FilterQuick Overview
- FilterQuick Topologies
- FilterQuick Distributed Topologies
- FilterQuick Active Topologies
- FilterQuick Switched Capacitor Topologies
- FilterQuick Digital Topologies
- FilterQuick Video

FilterQuick Overview

FilterQuick is designed to provide point-and-click filter design for commonly used filter topologies that do not need the advanced flexibility offered by FilterSolutions' Advanced window.

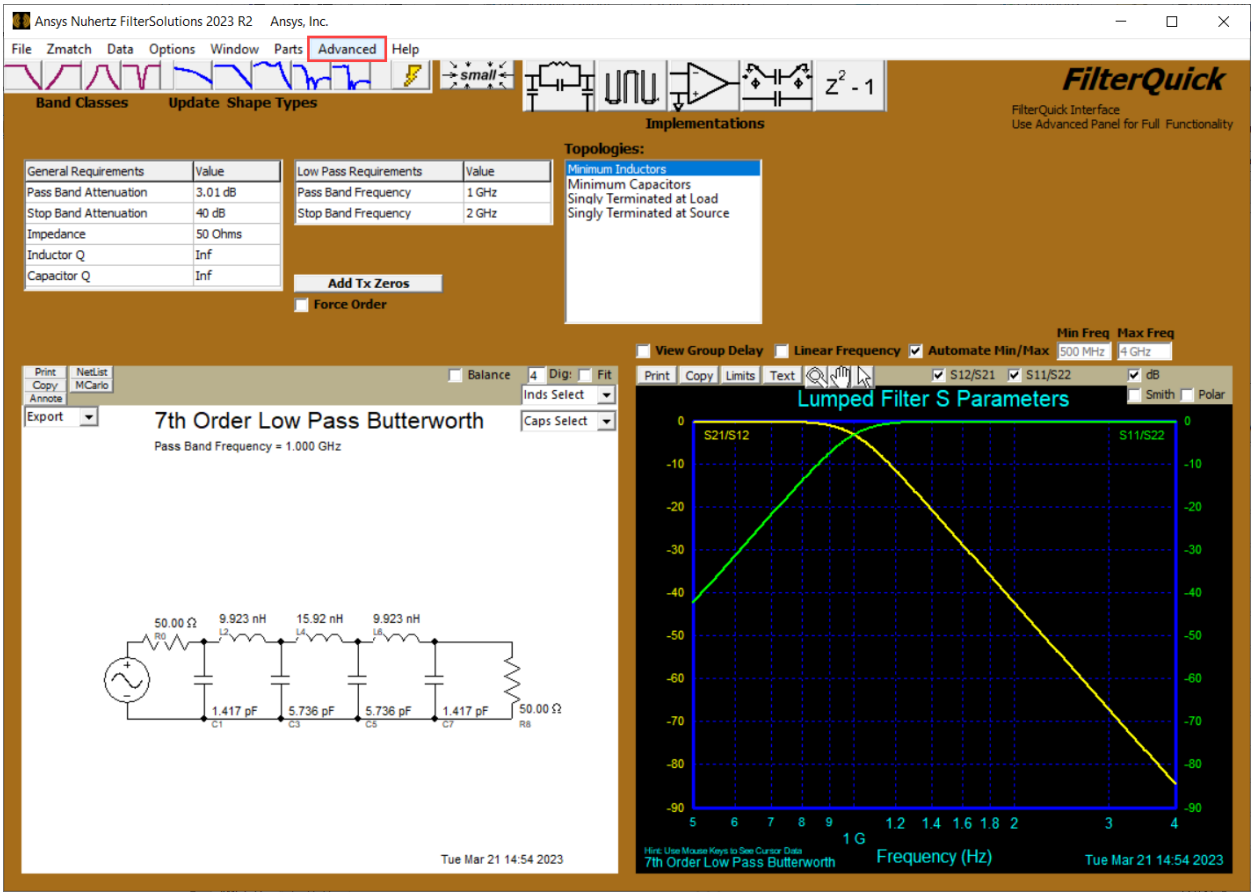
To open FilterQuick, select **FilterQuick** in the Menu.



The Menu option will change to **Advanced** while FilterQuick is active.

To discontinue FilterQuick and resume design in the standard FilterSolutions panel, click **Advanced** in the Menu.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To design a filter in FilterQuick:

1. Click a graphic that illustrates the desired shape. A pop-up description will provide the name of the shape.
2. Fill out the Requirements Forms below the graphical selections.
3. Select a graphic again.
4. Select **Update**, or select a topology on the right .

FilterQuick will instantly design and display the desired filter along with the S parameters or Frequency Response.

> **Note:** When "Singly Terminated" topology is selected, S parameters are not available because S parameters are not defined for singly terminated filters.

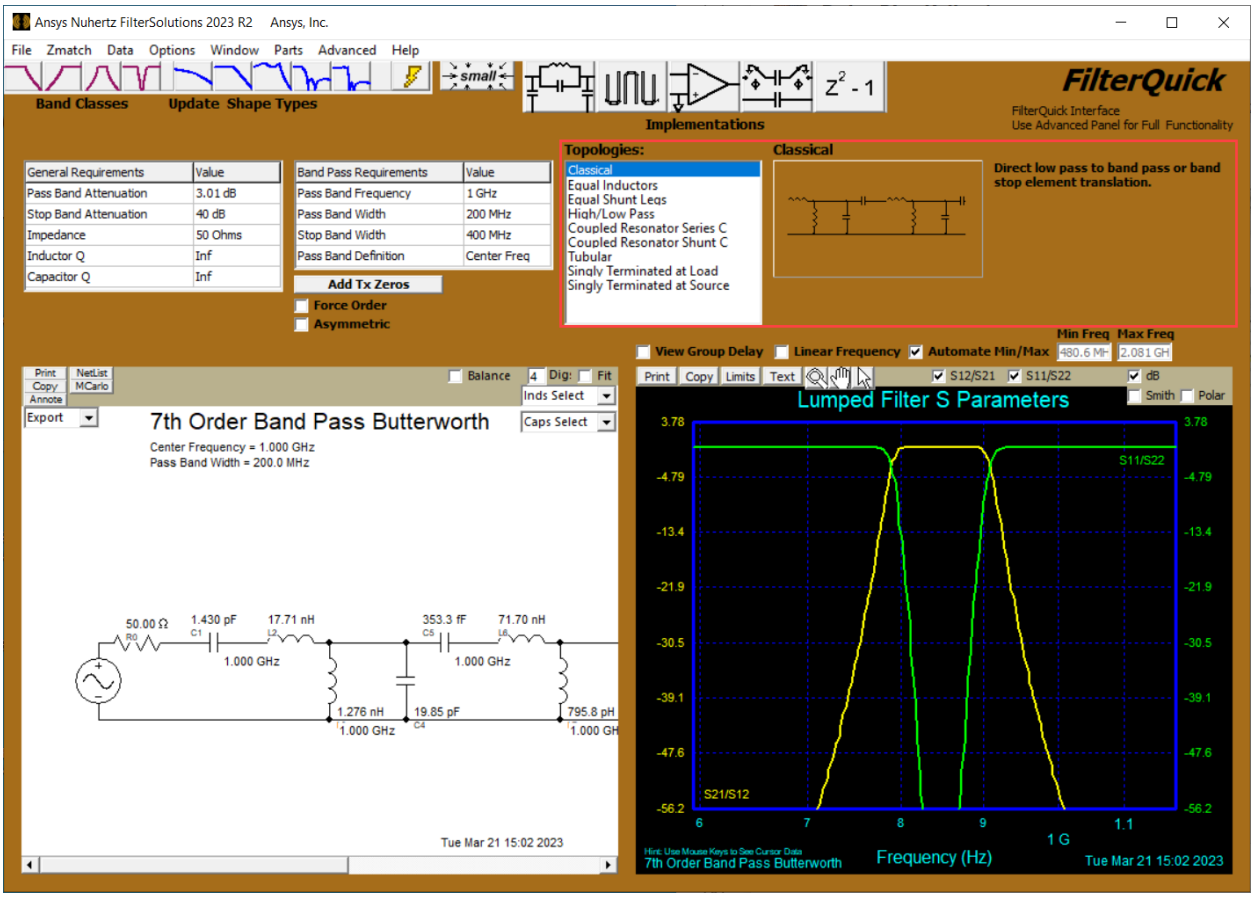
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The graphical filter displays has most of the high powered features that the standard Advanced has. Balancing, amplitude equalization, Monte Carlo analysis, manual edits, parts selections, inductor tuning, and Touchstone format S/Y/Z exports are all supported.

After entering a design requirement in the respective requirements form, filters are synthesized by selecting **Update** or by pressing the **Enter** key

FilterQuick Lumped Topologies

Scroll the mouse over the topologies options to reveal a graphical representation of the available topologies. A description is shown to the right of the topology display whenever Hints are enabled in the Options panel.



Description of Supported Lumped Topologies

Topology	Shape Type Availability	Band Class Availability	Description

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Minimum Inductors	All	Low Pass High Pass	Odd Order: Selected the ladder sequence that produces the fewest inductors. Even Order: Does nothing. No minimum inductor sequence exists.
Minimum Capacitors	All	Low Pass High Pass	Odd Order: Selected the ladder sequence that produces the fewest capacitors. Even Order: Does nothing. No minimum capacitor sequence exists.
Bridged T Traps	Chebyshev II Elliptic	Low Pass High Pass	Realizes transition zeros with Bridged T elements. Useful for element value flexibility.
Classical	All	Band Pass Band Stop	Direct low pass to band pass or band stop element translation.
Equal Inductors	Bessel Butterworth Chebyshev I	Band Pass	Odd Order: All inductors are the same value. Even Order: All inductors in the lower half and upper half of the filter are the same value.
High/Low Pass	Bessel Butterworth Chebyshev I	Band Pass	Places the low pass elements on one side of the filter, and the high pass elements on the other. Useful for wide band filters so that element value spreads are minimized on each side of the filter.
Coupled Resonators	Bessel Butterworth Chebyshev I	Band Pass	Narrow Band Approximation. Capacitors couple LC resonators together. Inductor and mutual inductance coupling is available in the schematic Coupled Resonator Selection Menu.
Tubular	Bessel Butterworth Chebyshev I	Band Pass	Narrow Band Approximation. Inductors separate PI capacitance sections. Useful for minimizing element value spread. Outer shunt inductors available in the schematic Tubular Selection Menu are useful

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

			in making the frequency response more symmetric.
Node Capacitor Grounds	Chebyshev II Elliptic	Band Pass	Provides a synthesized solution that includes a shunt capacitor on each floating node to insure a capacitor ground exits. The node parasitic capacitance may be absorbed by the synthesized shunt capacitance to completely eliminate the effects of node parasitic capacitance.
Zigzag	Chebyshev II Elliptic	Band Pass	** Minimizes Inductor Count ** Uses alternating tri-elements resonators. Useful for minimizing element value spreads and inductor counts. Even order filters require N inductors. Odd order require N-1 inductors, where N is the order.
Parametric	Chebyshev II Elliptic	Band Pass	Provides resonator separation between all series and shunt sections. Frequently useful in minimizing element value spreads.
High/Low Pass 1	Chebyshev II Elliptic	Band Pass	Places the low pass elements on one side of the filter, and the high pass elements on the other. Useful for wide band filters so that element value spreads are minimized on each side of the filter. May be slightly more accurate than High/Low 2, but requires more inductors.
High/Low Pass 2	Chebyshev II Elliptic	Band Pass	** Minimizes Inductor Count ** Places the low pass elements on one side of the filter, and the high pass elements on the other. Useful for wide band filters so that element value spreads are minimized on each side of the filter.

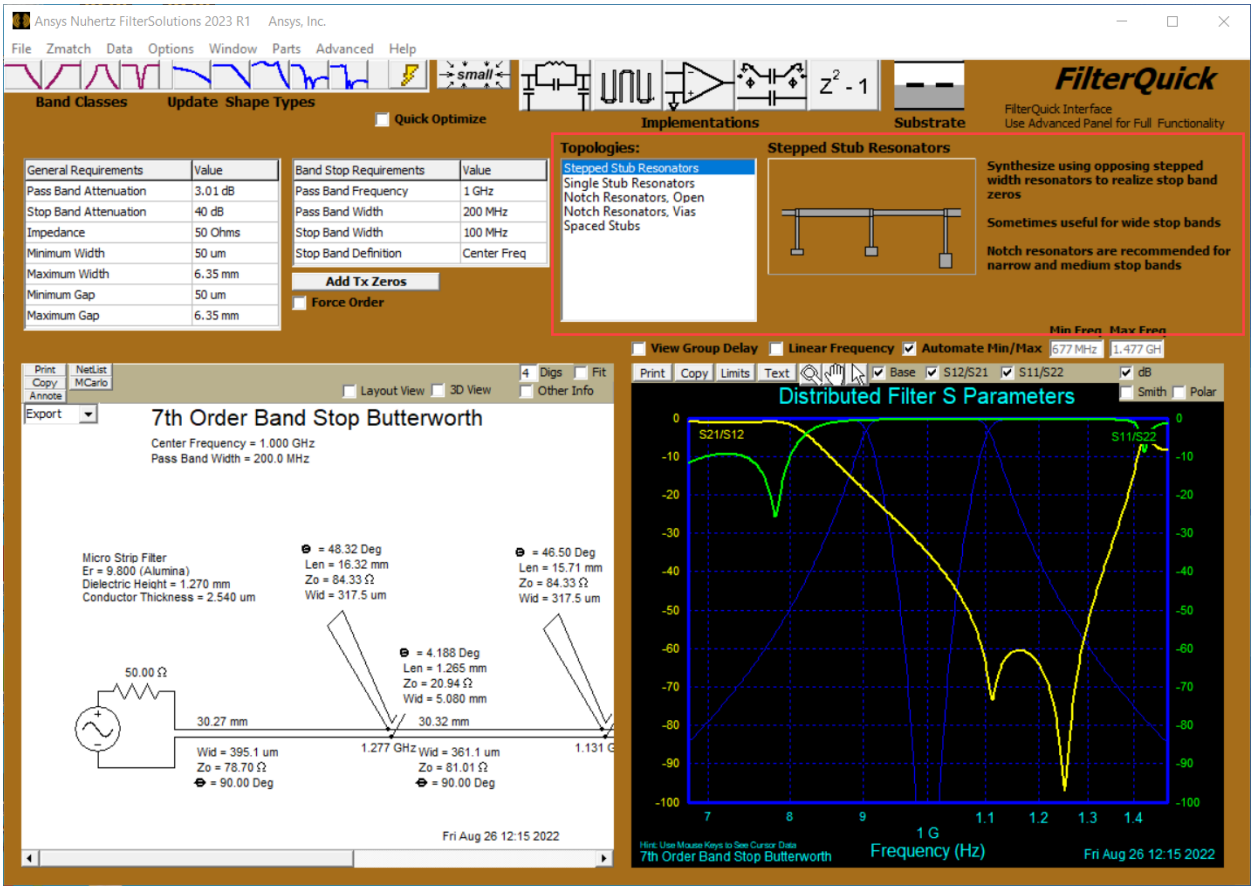
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

			Requires fewer inductors than High/Low 1, but may be slightly less accurate. Even order filters require N inductors. Odd order require N-1 inductors, where N is the order.
Cross Coupled	Chebyshev I Elliptic	Band Pass	Narrow Band Approximation. Implements a coupling matrix with cross coupled inductors. Useful in minimizing element value spreads.
Singly Terminated at Load	All	All	Synthesize with a termination at the load only. Source input is shorted. Useful for designs with sources of negligible resistance.
Singly Terminated at Source	All	All	Synthesize with a termination at the source only. Load output is open. Useful for designs with no loads or with reactive loads that may be absorbed by the last synthesized element.

FilterQuick Distributed Topologies

Scroll the mouse over the topologies options to reveal a graphical representation of the available topologies. A description is shown to the right of the topology display whenever Hints are enabled in the Options panel.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Description of Supported Distributed Topologies

Topology	Shape Type Availability	Band Class Availability	Description
Minimum Segments	Bessel Butterworth Chebyshev I	Low Pass Band Pass	Odd Order: Selects the ladder sequence that produces the fewest segments Even Order: Does nothing. No minimum segment sequence exists. Band Pass Filters: Sometimes useful for medium and wide bands
Minimum Stubs	Bessel Butterworth Chebyshev I	Low Pass Band Pass High Pass	Odd Order: Selects the ladder sequence that produces the fewest stubs Even Order: Does nothing. No minimum stub sequence exists. Band Pass Filters: Sometimes useful for

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

			medium and wide bands
Split Stubs	Bessel Butterworth Chebyshev I	Low Pass	Synthesizes using opposing rectangular stubs Useful when single rectangular stubs would be too wide
Stepped Impedance	Bessel Butterworth Chebyshev I	Low Pass	Synthesizes using alternating wide and thin line sections Useful for minimizing the physical size of the filter
Spaced Stubs	All	Low Pass High Pass Band Stop	Synthesizes using open stubs of uniform angular length and open stub resonators spaced quarter-length apart on a line of termination impedance. Useful for maintaining realizable geometry and frequency response accuracy
Radial Stubs	Bessel Butterworth Chebyshev I	Low Pass	Synthesizes using alternating radial stubs instead of rectangular stubs. Useful when rectangular stubs would be too wide in that it minimizes the width of the T-Joint and reducing the physical size when long fat sections would otherwise be required
Butterfly Stubs	Bessel Butterworth Chebyshev I	Low Pass	Synthesizes using opposing radial stubs instead of rectangular stubs. Useful when single radial stubs would be too wide
Stepped Stub Resonators	Chebyshev II Elliptic	All	Synthesize using alternating stepped resonators to realize transmission zeros

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Stepped Stub Resonators, Split	Chebyshev II Elliptic	Low Pass High Pass	Synthesize using opposing radial resonators to realize transmission zeros Useful when single radial stubs would be too wide
Single Stub Resonators	Chebyshev II Elliptic	All	Synthesize using open uniform width resonators to realize transmission zeros. Band pass and band stop synthesizes with opposing resonators
Radial Resonators	Chebyshev II Elliptic	Low Pass High Pass	Synthesize using alternating radial resonators to realize transmission zeros Useful for minimizing the discontinuity between the resonator sections and reducing the physical size when long thick sections would otherwise be required
Radial Resonators, Split	Chebyshev II Elliptic	Low Pass High Pass	Synthesize using opposing radial resonators to realize transmission zeros Useful when single radial stubs would be too wide
Minimum Capacitors	Bessel Butterworth Chebyshev I	High Pass	Odd Order: Selects the ladder sequence that produces the fewest series capacitors Even Order: Does nothing. No minimum series capacitor sequence exists.
Shunt Stub Resonators	Bessel Butterworth Chebyshev I	Band Pass	Synthesizes with shunt stub resonators Sometimes useful for medium and wide bands
Shunt Stub Resonators Equal Width	Bessel Butterworth Chebyshev I	Band Pass	Synthesizes with opposing shunt stub resonators Slight errors are introduced in the equal stub with approximation Useful for minimizing stub width and

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

			manufacturing with equal width stubs Sometimes useful for medium and wide bands
Parallel Edge Tapped	Bessel Butterworth Chebyshev I	Band Pass	Synthesizes with parallel edge coupled resonators with tapped input and output Useful for removing undesirable small gap widths in the first and last resonator
Parallel Edge Untapped	Bessel Butterworth Chebyshev I	Band Pass	Synthesizes with parallel edge coupled resonators with no taps Lack of taps may result in undesirable small gap widths in the first and last resonator
Hairpin Resonators	Bessel Butterworth Chebyshev I	Band Pass	Synthesizes using hairpin resonators with tapped inputs and outputs Useful for reducing physical size
Miniature Hairpin Resonators	Bessel Butterworth Chebyshev I Elliptic	Band Pass	Synthesizes using miniaturized hairpin resonators with tapped inputs and outputs Useful for minimizing physical size in that it requires less physical space than standard hairpins or ring resonators.
Ring Resonators	Bessel Butterworth Chebyshev I Elliptic	Band Pass	Synthesizes using miniaturized hairpin resonators with tapped inputs and outputs Useful for reducing physical size in that it requires physical space than a standard wide pin hairpin Elliptic utilizes a cross coupled configuration, and is limited to four poles.
Interdigital	Bessel Butterworth Chebyshev I	Band Pass	Synthesizes using quarter length resonators with alternating open and via shorted ends and tapped input and output. Useful for narrow and medium band filter requirements, and minimizing physical size Elliptic utilizes a cross coupled configuration, and is limited to four poles.

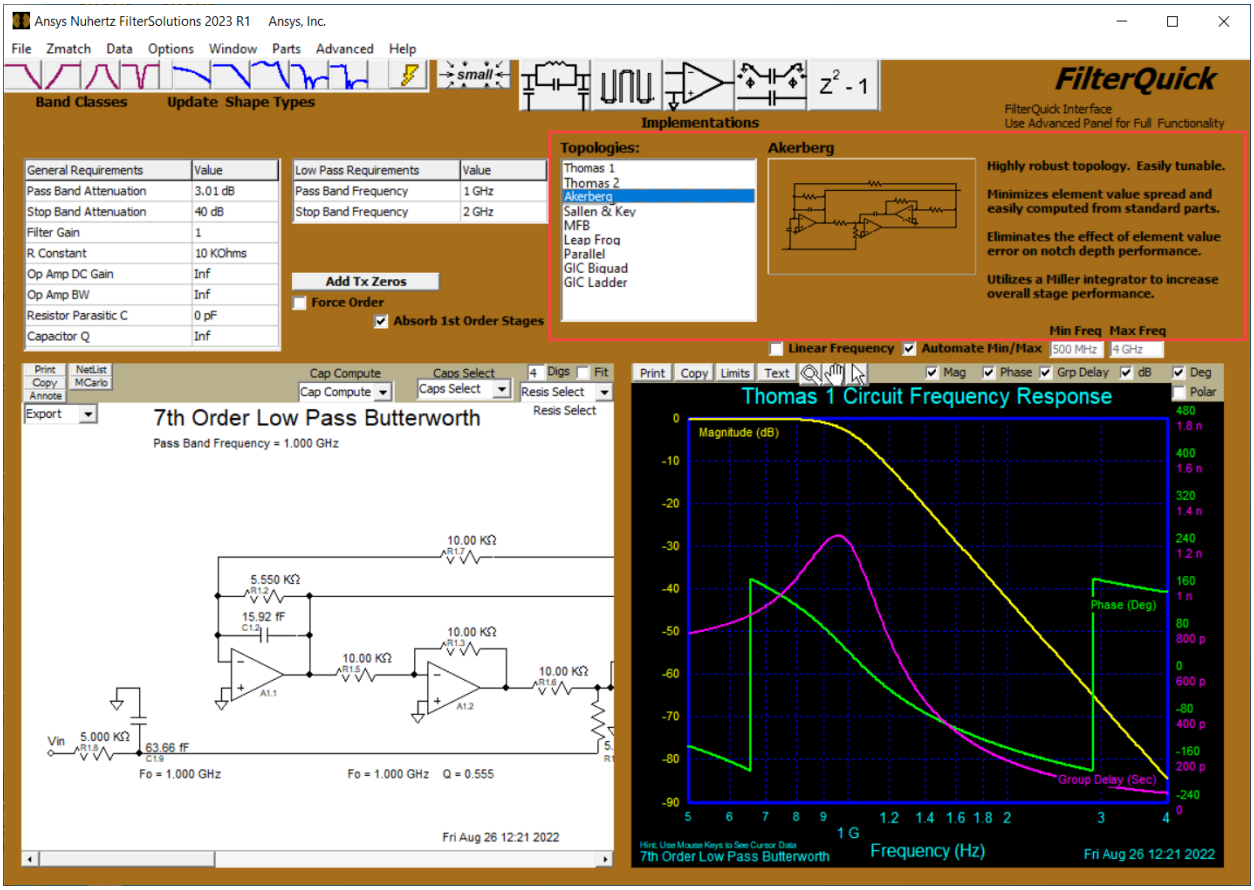
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Interdigital, Wideband	Bessel Butterworth Chebyshev I	Band Pass	Synthesizes using quarter length resonators with alternating open and via shorted ends and no taps or feed lines. Useful for wide and medium band filter requirements, and minimizing physical size
Combline	Bessel Butterworth Chebyshev I	Band Pass	Synthesizes with resonators terminated by via shorts at one end and capacitors at the other with tapped input and output Useful for capacitor tuning, minimizing physical size, and pushing out spurious frequency responses
Notch Resonators, Vias	Bessel Butterworth Chebyshev I	Band Stop	Synthesizes with parallel coupled resonators with one via short Useful for minimizing physical filter size
Notch Resonators, Open	Bessel Butterworth Chebyshev I	Band Stop	Synthesizes with parallel coupled resonators with a shorted termination simulated by a quarter length stub Useful for eliminating the need for vias, but not as accurate as the via-short solution

FilterQuick Active Topologies

Scroll the mouse over the topologies options to reveal a graphical representation of the available topologies. A description is shown to the right of the topology display whenever Hints are enabled in the Options panel.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Description of Supported Lumped Topologies

Topology	Shape Type Availability	Band Class Availability	Description
Thomas I	All	All	Highly robust topology. Easily tunable Minimizes element value spread and easily computed from standard parts.
Thomas II	All	All	Highly robust topology. Easily tunable. Minimizes element value spread and easily computed from standard parts.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

			Eliminates the effect of element value error on notch performance.
Ackerberg	All	All	Highly robust topology. Easily tunable. Minimizes element value spread and easily computed from standard parts. Eliminates the effect of element value error on notch depth performance. Utilizes a Miller integrator to increase overall stage performance.
Sallen & Key	All	All	Single opamp stage. Provides for increased robustness in the pass band with respect to element value error than MFB single amplifier stages
MFB	All	All	Single opamp stage. Provides for increased robustness with respect to opamp parasitics when positive feedback enhancement is used.
Leap Frog	All	Low Pass, High Pass, and Band Pass	Simulates an LC ladder sequence.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

			Generally the most robust topology of all, especially with respect to element value error. Eliminates the effect of element value error on notch depth performance.
Parallel	All		Sums together the outputs of multiple single amplifier biquads. Sometimes useful to minimize the compounding of opamp errors due to cascading stages.
GIC Biquad	All	All	Creates biquad stages using General Impedance Converters (GICs). Easily Tunable. Minimizes element value spread and easily computed from standard parts. Exhibits good robustness qualities for both opamp parasitics and element value error. Eliminates the effect of element value error on notch depth performance.
GIC Ladder	All	Low Pass, High Pass, and Band Pass	Simulates an LC ladder sequence using General Impedance Converters (GICs). Exhibits good robustness qualities for both

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

			opamp parasitics and element value error. Eliminates the effect of element value error on notch depth performance.

Optional Selections:

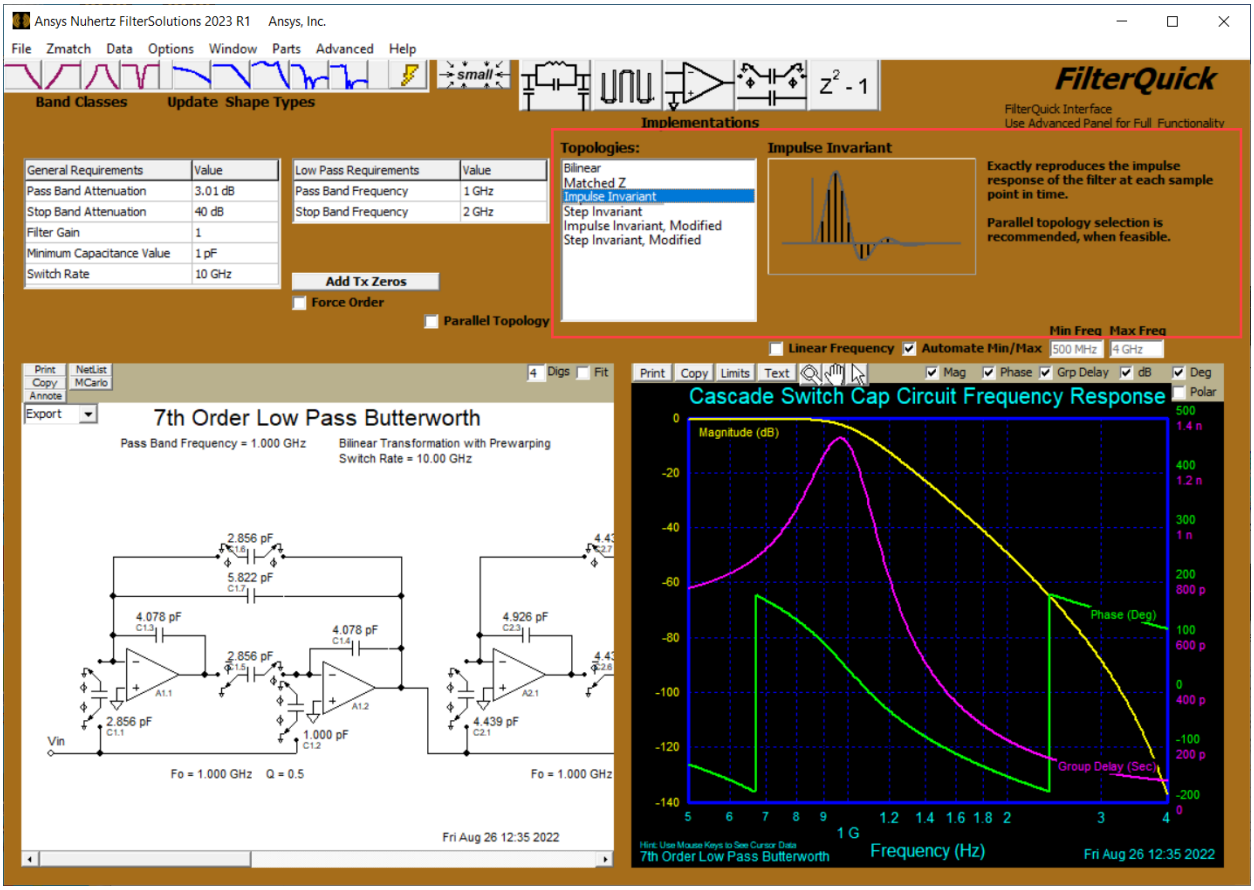
Absorb 3rd Order Stages

Check to combine three poles into a single stage whenever possible.

FilterQuick Switched Capacitor Topologies

Scroll the mouse over the topologies options to reveal a graphical representation of the available topologies. A description is shown to the right of the topology display whenever Hints are enabled in the Options panel.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Description of Supported Lumped Topologies

Topology	Shape Type Availability	Band Class Availability	Description
Bilinear	All	All	Numerical approximation to the filter S-Transform. Prewarping is applied as necessary to insure the accuracy of the pass band. Cascade topology selection is recommended when feasible.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Matched Z	All	All	Approximation to the filter S-Transform by direct pole and zero mapping. Less accurate than the Bilinear transformation, but easier to compute. Cascade topology selection is recommended when feasible.
Impulse Invariant	All	All	Exactly reproduces the impulse response of the filter at each sample point in time. Parallel topology selection is recommended, when feasible.
Step Invariant	All	All	Exactly reproduces the step response of the filter at each sample point in time. Parallel topology selection is recommended, when feasible.
Impulse Invariant, Modified	All	All	Similar to Impulse Invariant, but cascades minterms instead of summing them, thereby reducing the accuracy. Useful for switched capacitor implementations of Z-Transforms in that it frequently minimizes the part count Cascade topology selection is recommended, when feasible.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Step Invariant, Modified	All	All	Similar to Step Invariant, but cascades minterms instead of summing them, thereby reducing the accuracy. Useful for switched capacitor implementations of Z-Transforms in that it frequently minimizes the part count. Cascade topology selection is recommended, when feasible.

Optional Selections:

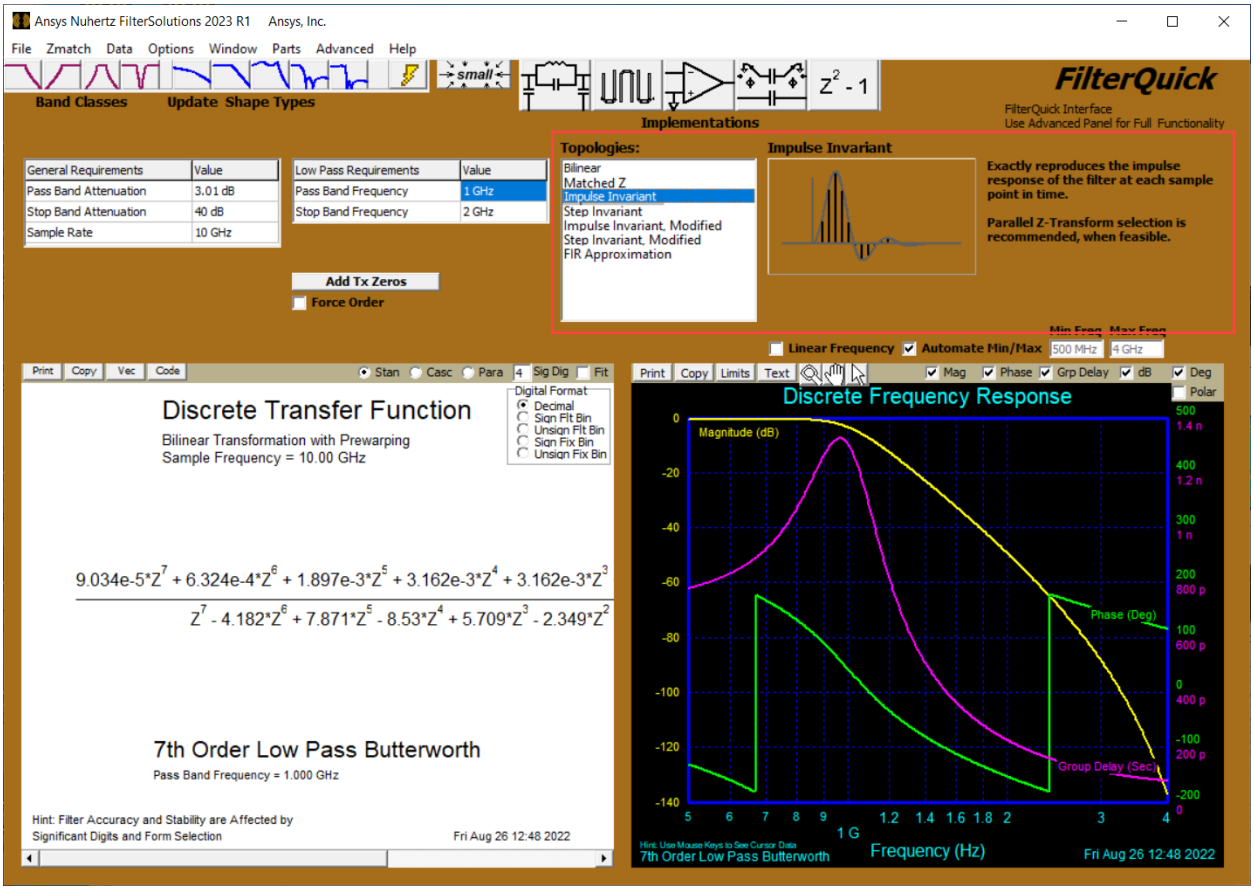
Parallel Topology

Check to display the filter in parallel biquads. Recommended for (not modified) impulse invariant and step invariant filter selections

FilterQuick Digital Topologies

Scroll the mouse over the topologies options to reveal a graphical representation of the available topologies. A description is shown to the right of the topology display whenever Hints are enabled in the Options panel.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Description of Supported Lumped Topologies

Topology	Shape Type Availability	Band Class Availability	Description
Bilinear	All	All	Numerical approximation to the filter S-Transform. Prewarping is applied as necessary to insure the accuracy of the pass band. Cascade Z-Transform implementation is recommended when feasible.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Matched Z	All	All	Approximation to the filter S-Transform by direct pole and zero mapping. Less accurate than the Bilinear transformation, but easier to compute. Cascade Z-Transform implementation is recommended when feasible.
Impulse Invariant	All	All	Exactly reproduces the impulse response of the filter at each sample point in time. Parallel Z-Transform implementation is recommended, when feasible.
Step Invariant	All	All	Exactly reproduces the step response of the filter at each sample point in time. Parallel Z-Transform implementation is recommended, when feasible.
Impulse Invariant, Modified	All	All	Similar to Impulse Invariant, but cascades minterms instead of summing them, thereby reducing the accuracy. Useful for switched capacitor implementations of Z-Transforms in that it frequently minimizes the part count Cascade Z-Transform implementation is recommended, when feasible.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Step Invariant, Modified	All	All	Similar to Step Invariant, but cascades minterms instead of summing them, thereby reducing the accuracy. Useful for switched capacitor implementations of Z-Transforms in that it frequently minimizes the part count. Cascade Z-Transform implementation is recommended, when feasible.
FIR Approximation	All	All	Reproduces the filter impulse response with FIR taps. Useful for insuring the stability of the filter under all circumstances, without exception.

FilterQuick Overview Video

See the Online Help to view the FilterQuick video.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3 - Automated Parameter Synthesis

This chapter contains the following topics:

- Automated Parameter Synthesis
- Constricted Equiripple Passbands and Stopbands
- Single Point Ripple
- Half Band Ripple
- Bessel and Gaussian Stop Bands
- Minimum Inductor Zigzags and High/Low 2
- Manual Tx Zero Placement
- Positioning Zeros With the Mouse

Automated Parameter Synthesis

Automated Parameter Design (APS) refers to the synthesis around generally placed stop band and/or reflection zeros, and automating the placement to achieve a more efficient filter attenuation response or more desirable group delay. AGPS includes:

1. Constricted Pass Band Equiripple
2. Constricted Stop Band Equiripple
3. Single Point Pass Band Ripple
4. Single Point Stop Band Ripple
5. Asymmetric Band Pass Synthesis
6. Bessel and Gaussian Stop Band Filters
7. Minimum Inductor Zigzags and High/Low Pass 2 Filters
8. User text entered manual stop band zero placement
9. Mouse cursor controlled stop band zero placement

> **Note:** Quadruplet zeros delay equalization requires user-entered-text, mouse-cursor-controlled complex zero placement, and qualifies as an advanced APS function.

Constricted Equiripple Pass Bands and Stop Bands

Chebyshev and Elliptic filters are well known for their flat equiripple passbands and/or stopbands. However, many applications require a flat equiripple effect only for frequencies

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

nearer to the cutoff frequency. In these situations, a totally flat equiripple passband and/or stopband is detrimental in that it unnecessarily reduces the rate of attenuation at the band edge and possibly requires a higher order filter.

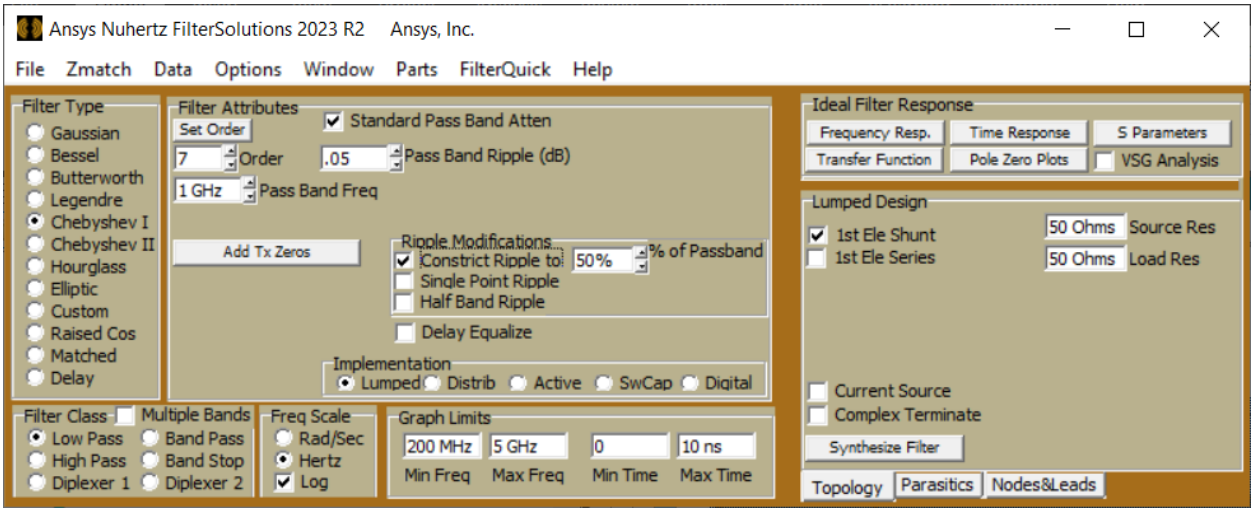
FilterSolutions removes this potential waste of synthesis quality by optionally constricting the equiripple effect to a percentage of the passband and/or stopband. For example, if the quality of a low pass passband is not an issue for the lower 50%, the equiripple may be constricted to the upper 50%. The rate of attenuation at the band edge is then increased, and the order of the filter could possibly be reduced. The same is true of the stopband.

Constricted equiripple passbands are supported for Chebyshev I and Elliptic filters. Constricted Equiripple stopbands are supported for Chebyshev II and Elliptic filters. Elliptic filters support Equiripple passband and equirippple stopbands, either individually, or both at the same time.

> **Note:** % of stop band ripple is calculated as the inverse of the stopband frequencies, where infinity becomes 0, or DC, and the initial stopband frequency becomes the cutoff frequency.

User Selection

Constrict Ripple to **% of Passband** option is available in the **Ripple Modification** group box when either **Chebyshev I**, **Chebyshev II**, or **Elliptic** filters are selected in the **Filter Type** group box. Elliptic filters have the additional option of equiripple for the passband, stopband, or both.



A scrolling function is available for ease of use. When in **Real Time Update** mode (in the **Options** menu), constant selection of the scrolling function to the right of the text entry produces a real time visual effect of Constricted Equiripple synthesis for ease of design.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

[Constricted Ripple Video Tutorial]

Implementation

Constricted ripple synthesis is performed with synthesis algorithms developed by internal Nuhertz research. Synthesis computations are fast and accurate. No optimization or tables are needed or employed.

Example:

An application requires 0.5dB passband ripple, 1.0 MHz cutoff frequency, 60dB passband attenuation at 1.14 MHz.

For an Elliptic filter, the conventional flat passband and stopband filter requires a 9th order filter. If the passband ripple is constricted to 30% of the passband and stopband, a 7th order filter will meet the filter design requirements, if the increase in ripple outside 30% is of no consequence to the application the filter is designed for.

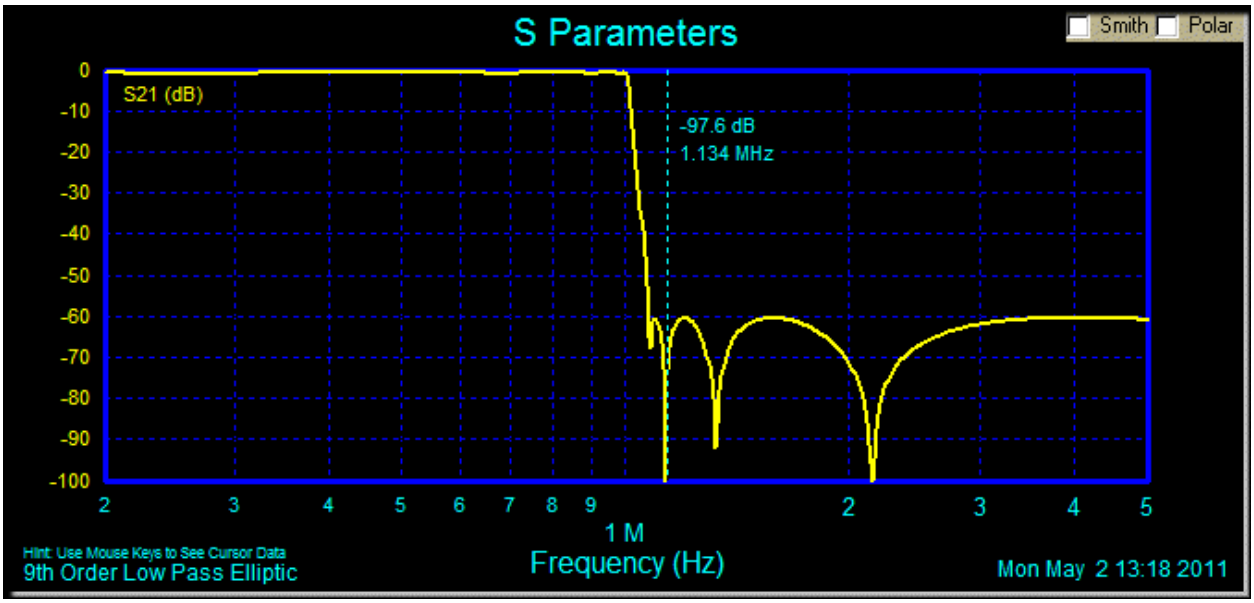


Figure 1: 9th Order Filter Elliptic With Flat Passband and Stopband meets all attenuation requirements

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

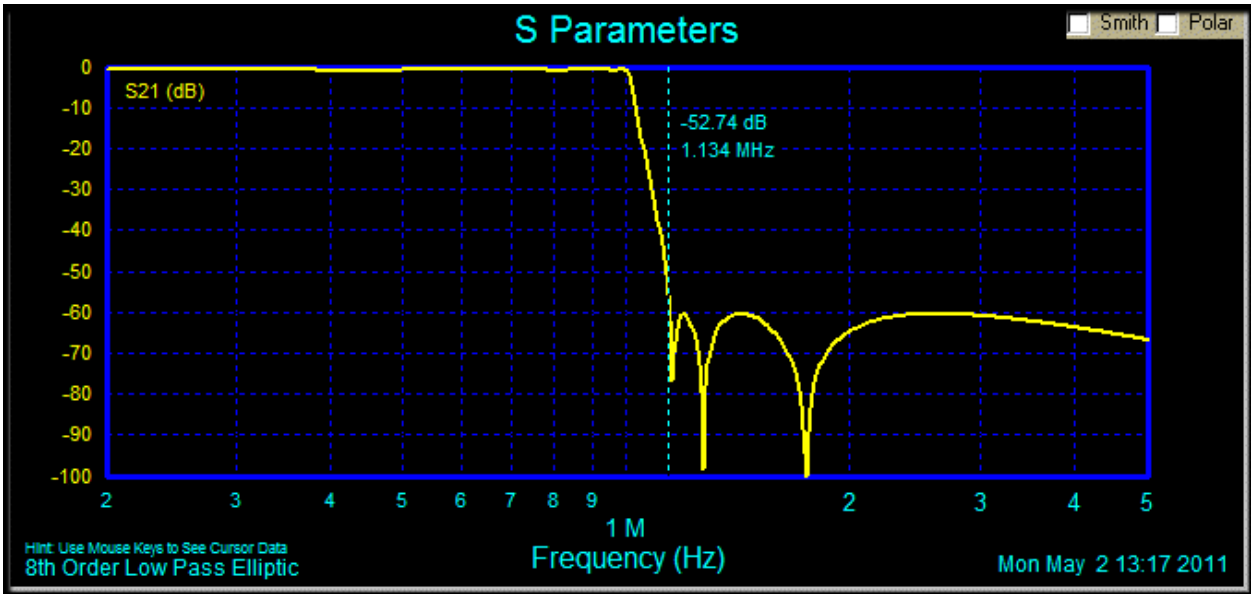


Figure 2: 8th Order Elliptic Filter With Flat Passband and Stopband FAILS to meet all attenuation requirements

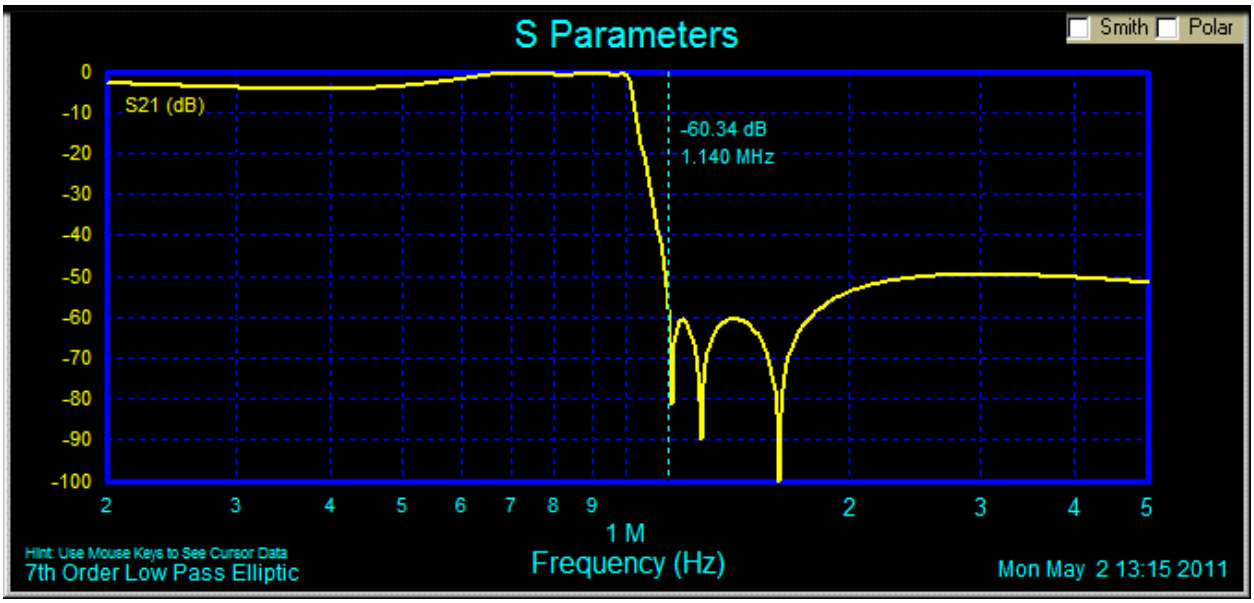
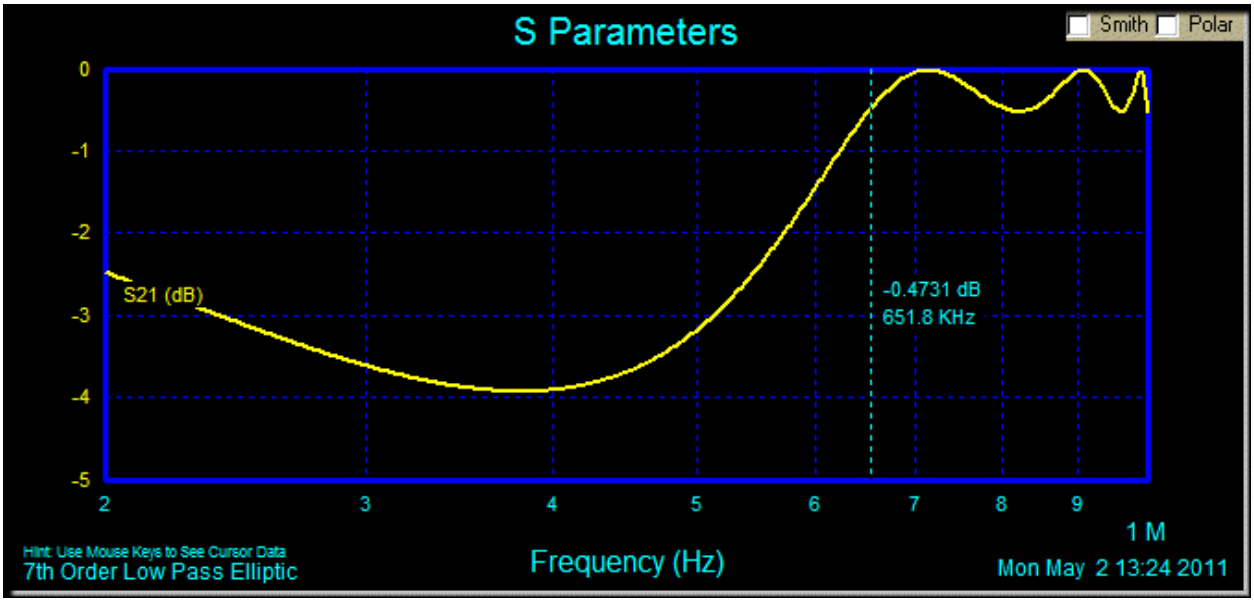


Figure 3: 7th Order Filter Elliptic with 35% Equiripple Constriction on Passband and Stopband DOES MEET all Attenuation Requirements.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Close up of 7th Order Equiripple Passband Showing 0.5 dB Equiripple in upper 35% of Passband, as Required.

Constricted Ripple Video Tutorial

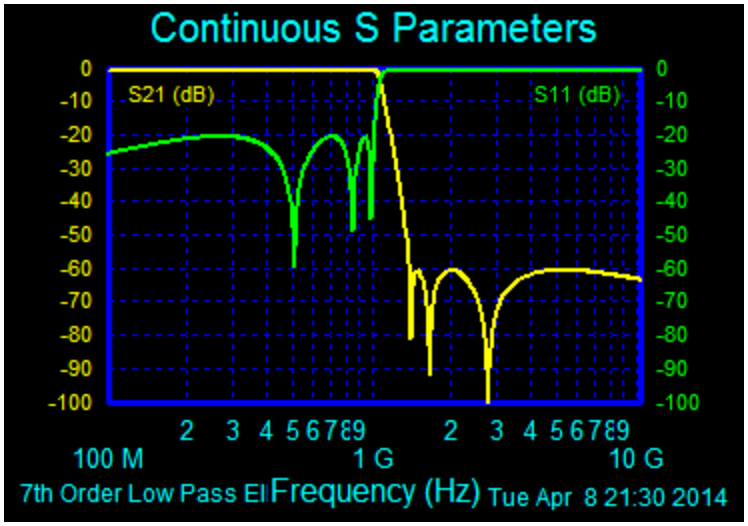
See the Online Help to view the Constricted Ripple Video.

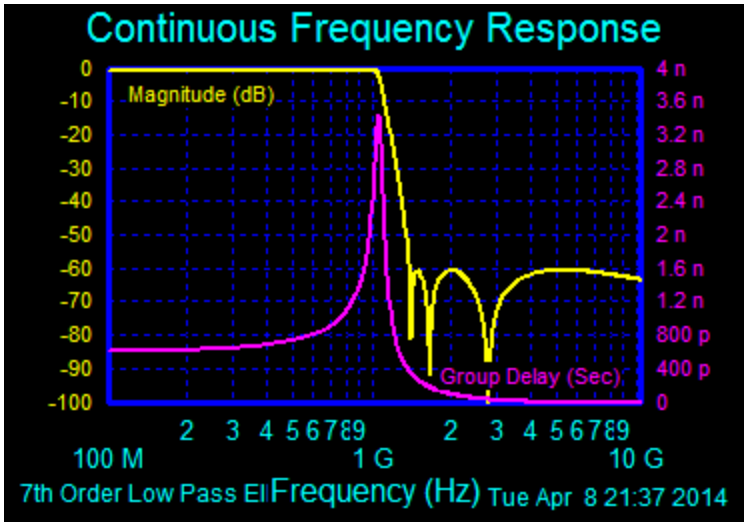
Single Point Ripple

Traditional Chebyshev and Elliptic designs employ pass band and stop band equiripple Chebyshev functions. This leads to an efficient design that maximizes cutoff attenuation, which can be further enhanced through constricted ripple designs, but has adverse effects on the group delay about the cutoff frequency, and requires a broad range of part values.

Adverse group delay effects and part value range may be reduced by confining the ripple to a single point, or Single Point Ripple. The stop band and/or pass band may have one single ripple point utilizing a finite transmission or reflection zero. For odd order filters, one or three zeros may remain at zero or infinite frequency, selectable by the user. Three zeros at zero or infinite frequency has the effect of improving attenuation at frequencies far from the cutoff frequency. For even order filters, two zeros remain at zero or infinite frequency.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





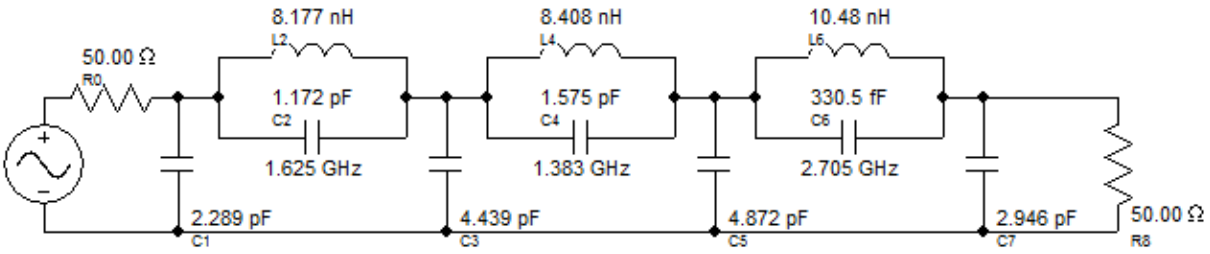
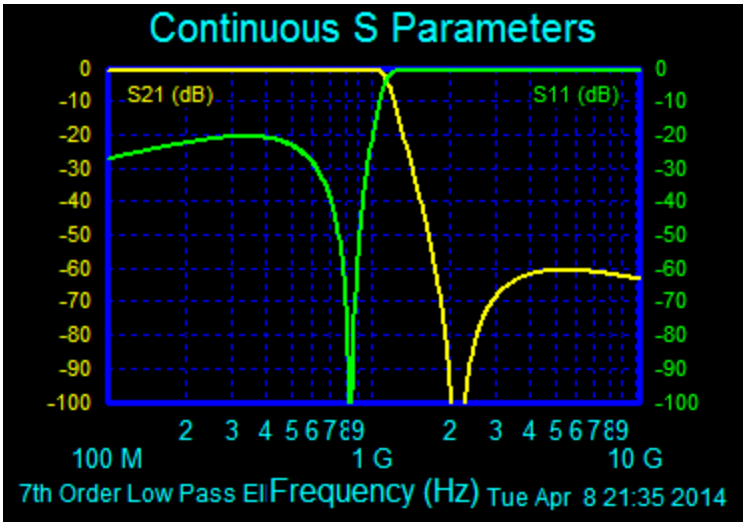
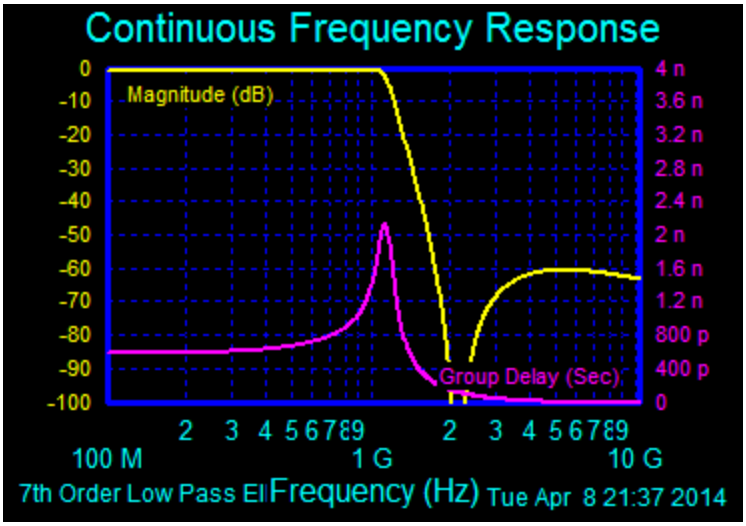


Figure 1: Traditional Elliptic Design

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





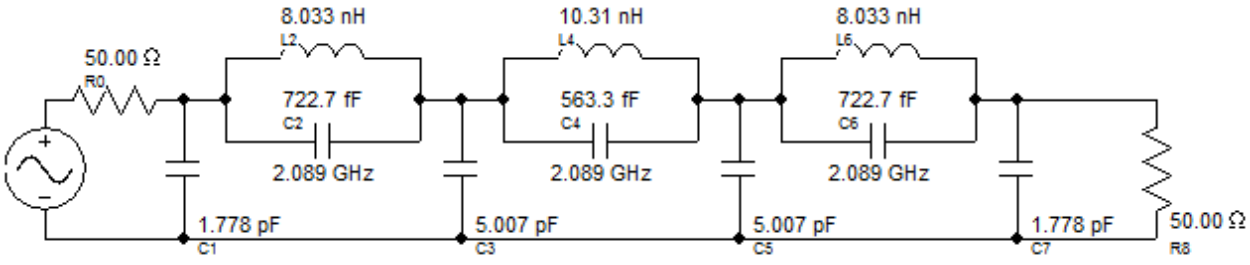
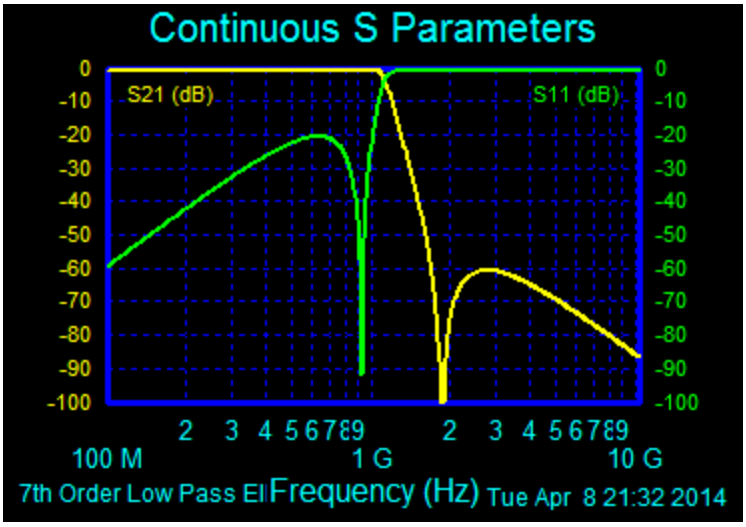
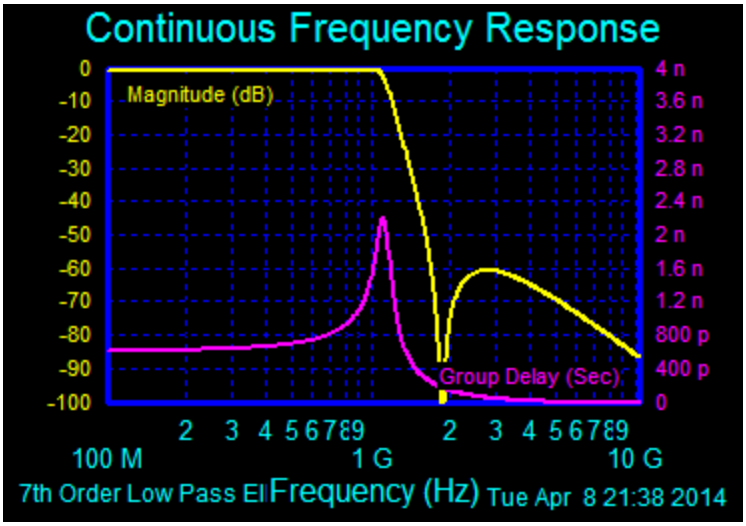


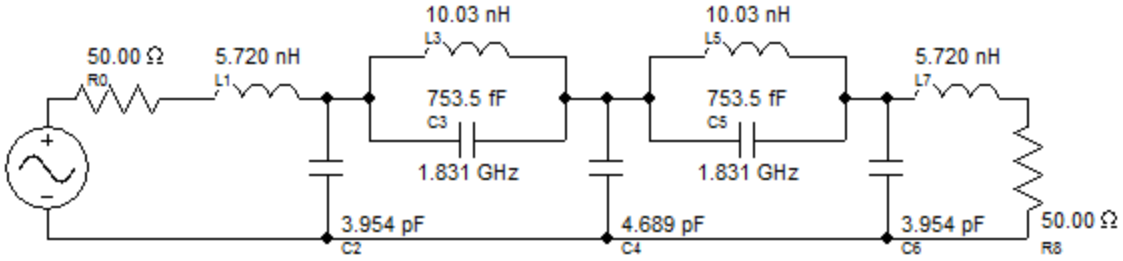
Figure 2: Elliptic Design With Pass Band and Stop Band Single Point Ripple, One Zero at Zero and Infinity

Improved Group Delay and Element Value Spread.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.







Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 2: Elliptic Design With Pass Band and Stop Band Single Point Ripple, Three Zeros at Zero and Infinity

Improved low and high pass attenuation over single zero case.

Half Band Ripple

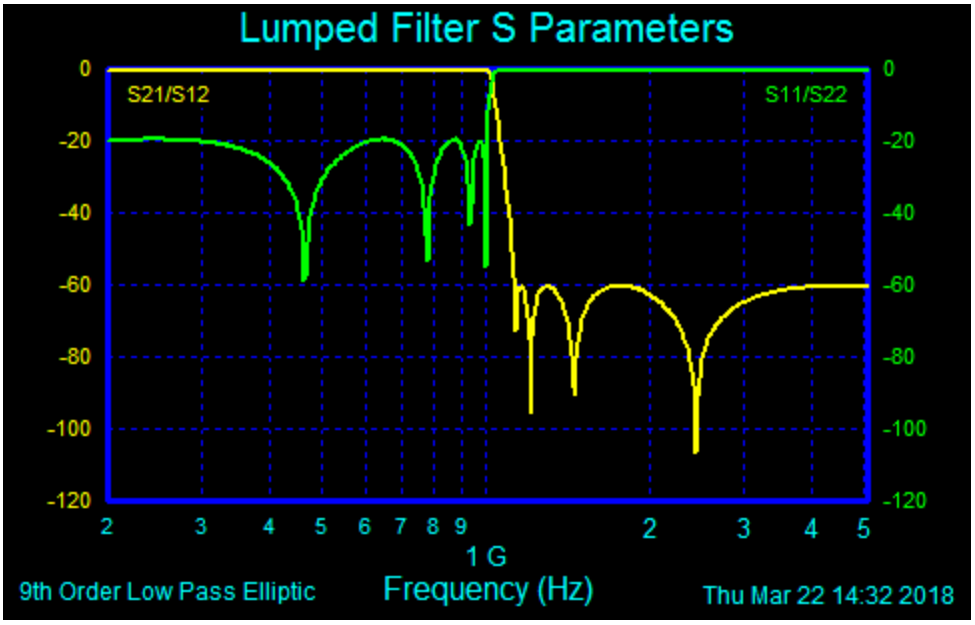
Traditional Chebyshev and Elliptic designs employ pass band and stop band equiripple Chebyshev functions with all possible zeros distributed fully across the band. At times, it may be desired to distribute only half of the zeros across the band, and leave the other half at frequencies zero or infinity. This eases the sharpness of the group delay and aids in group delay equalization, accommodates potential manufacturing restrictions, and accommodates some planar geometry requirements.

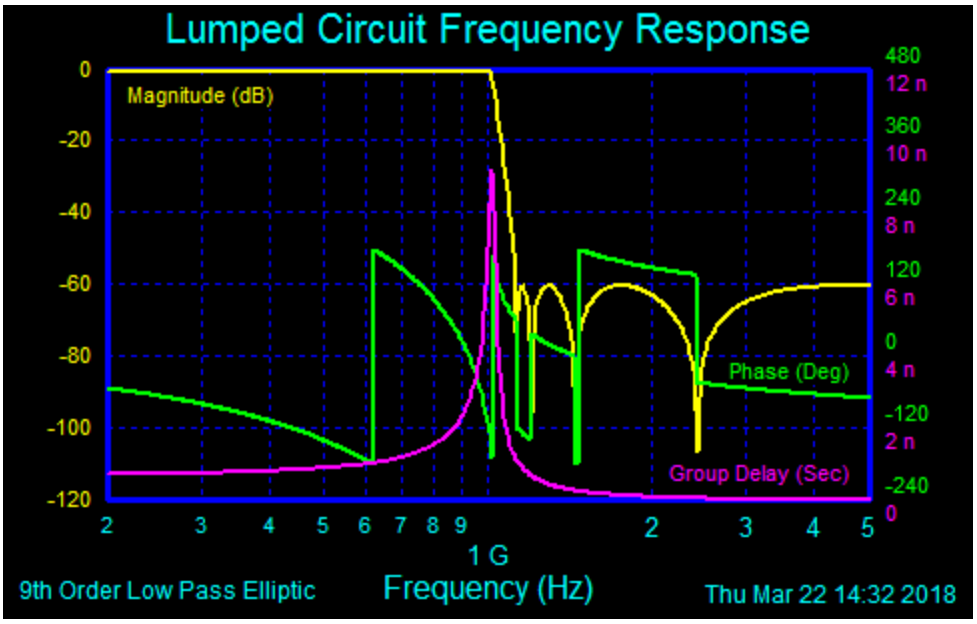
Half band ripple function uses half the zeros half the zeros to create the equiripple in the pass band and/or stop band, and leaves the other half at frequencies zero or infinity in accordance with the following schedule:

Half Band Zeros Distribution

Prototype Poles	Stop Band Zeros in the Pass Band and/or Stop Band	Traditional Equiripple Function with Even Order Modification
<5	Function Unavailable	
5 to 6	1	2
7 to 10	2	3 to 4
11 to 14	3	5 to 6
15 to 18	4	7 to 8
>18	5	9 to 10

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

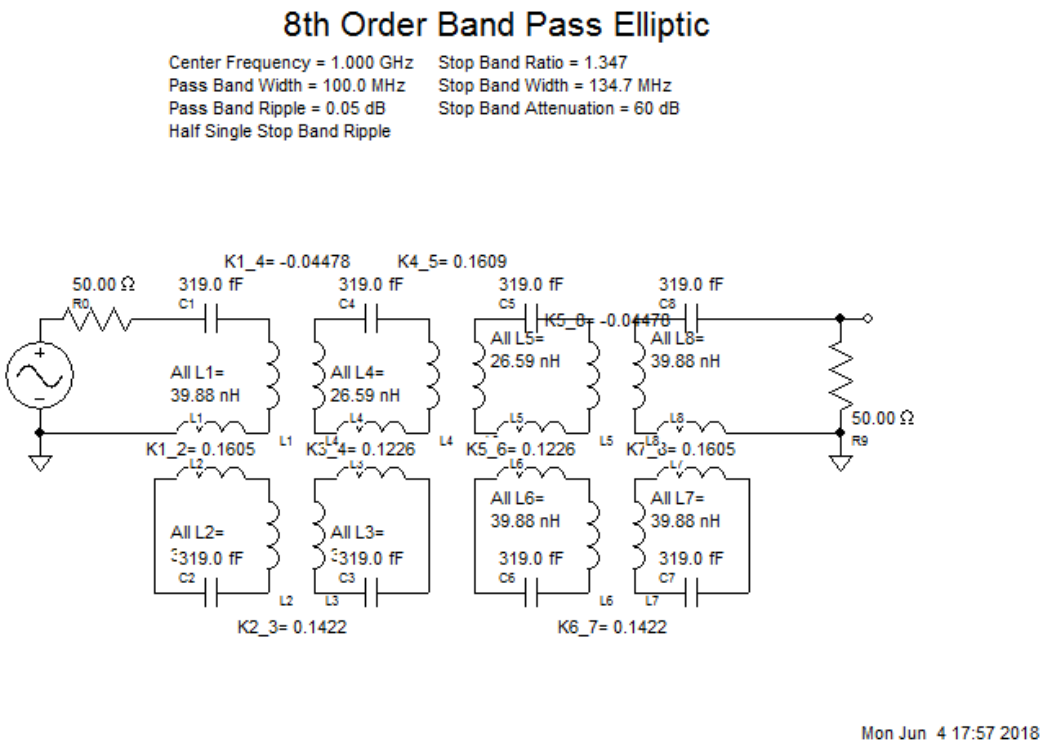
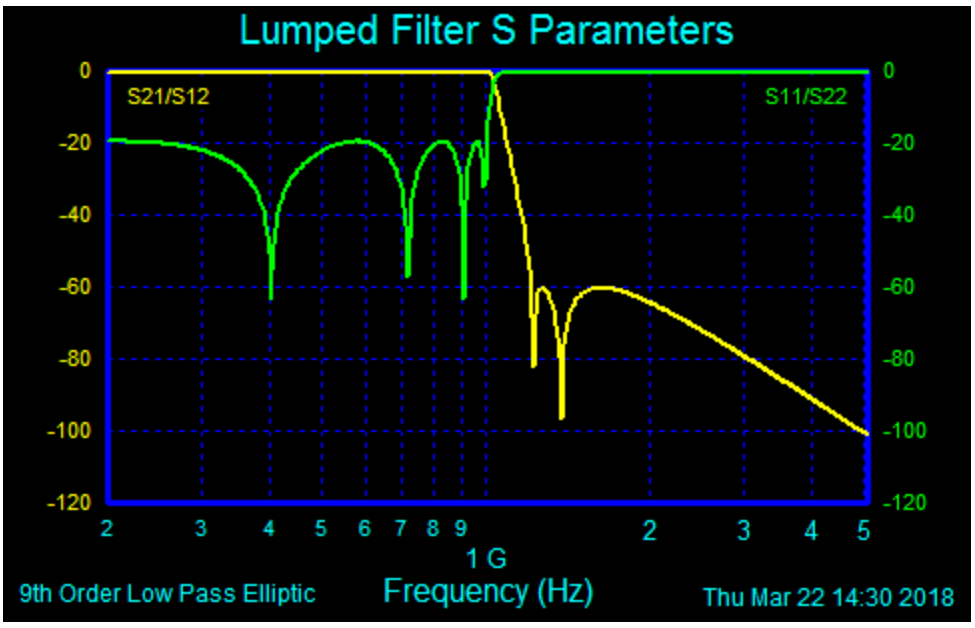
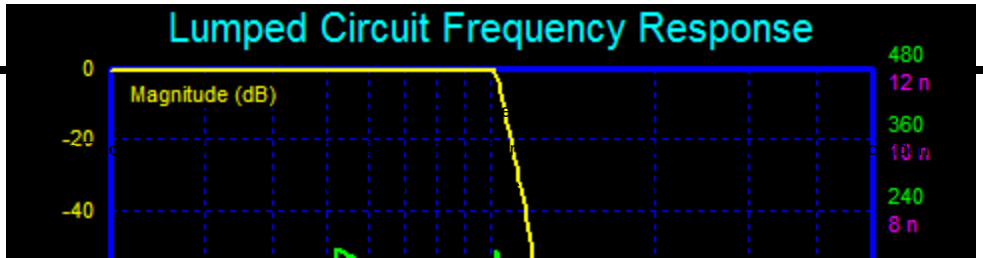


Figure 1: Traditional 9 Pole Elliptic Design





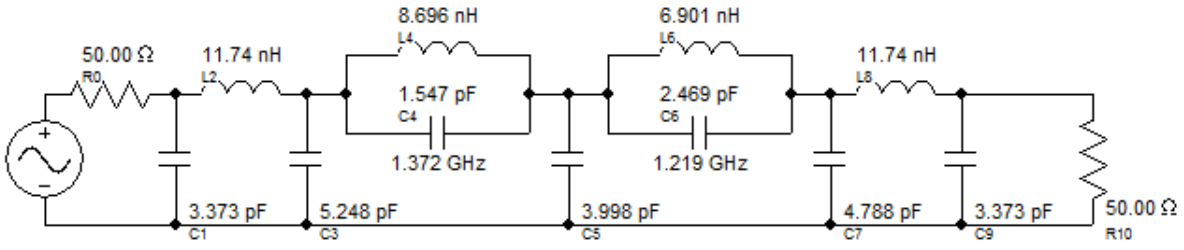


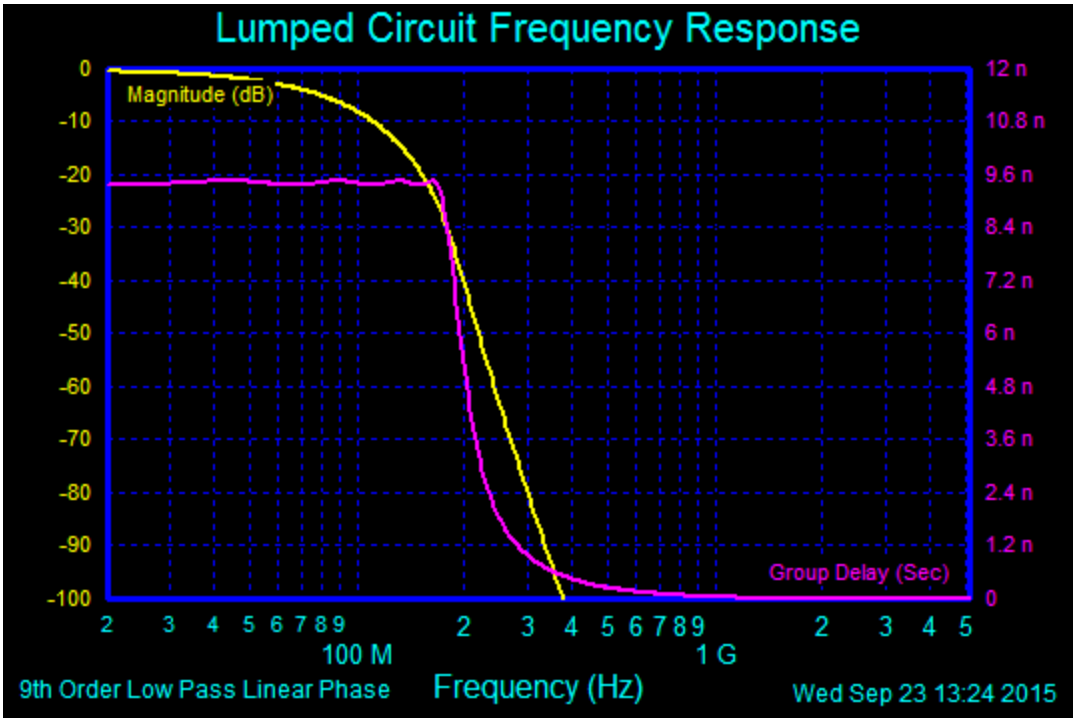
Figure 2: 9 Pole Half Band Ripple Stop Band Design

Bessel and Gaussian Stop Bands

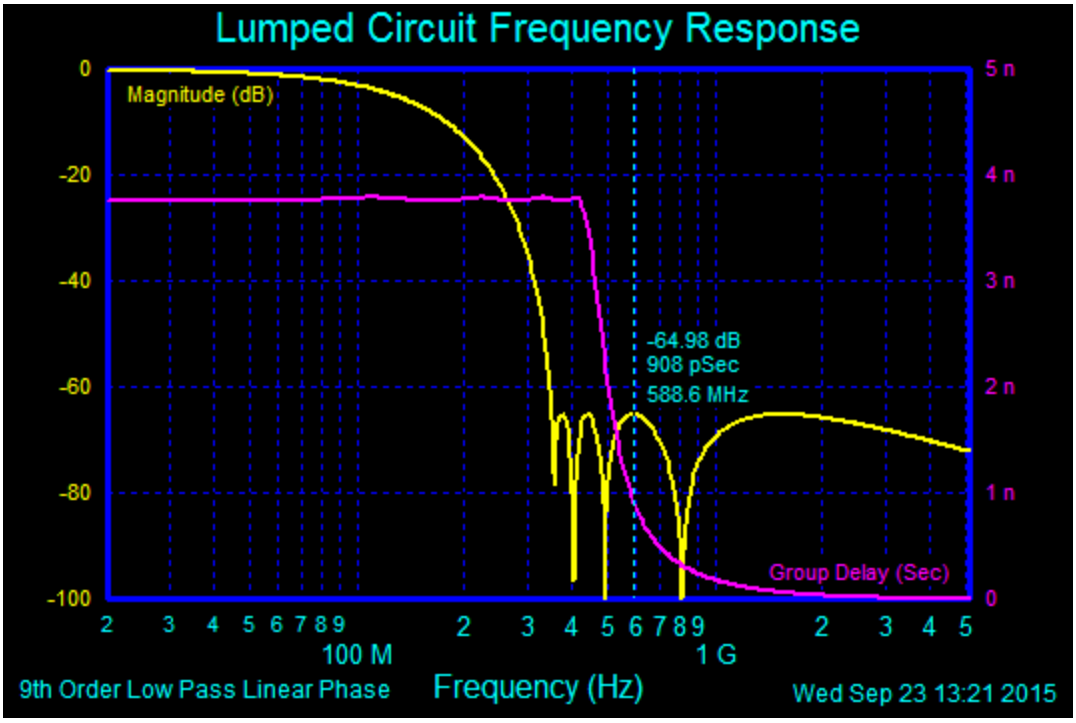
Traditional filter theory does not provide for synthesized stop band shapes in Bessel and Gaussian filters. FilterSolutions uses advanced automated parameter synthesis to perform this function, including Linear Phase and Transitional Filters. Equiripple and single point ripple attenuation requirements may be set by the user, and FilterSolutions will synthesize the filter automatically to meet the entered attenuation requirements. The effects of synthesizing a 65 dB stop band on a 1% group delay Linear phase Filter are seen below. The Single Point Ripple stop band has an advantage over the Equiripple stop band with reduced element value spread at the cost of some attenuation cutoff steepness.

Equiripple Bessel stop bands are supported in both the **FilterQuick** design panel and the **Advanced** design panel. Bessel Single Point Ripple and Gaussian and Linear Phase Single Point Ripple is supported only in the Advanced design panel.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

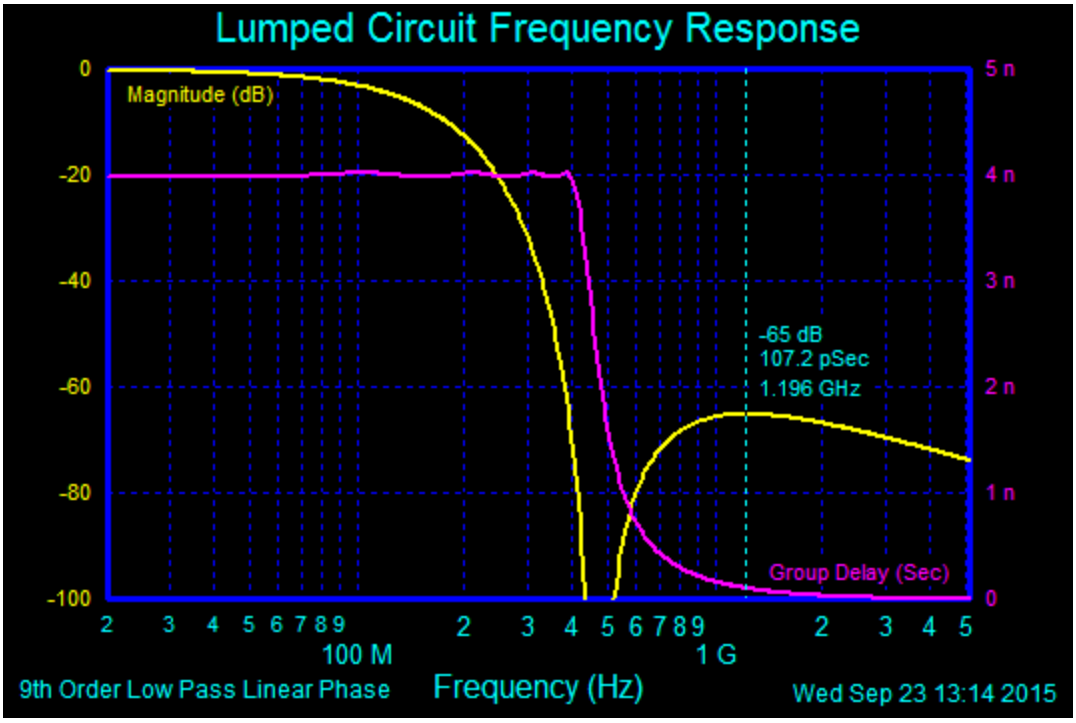


Traditional Linear Phase Filter With 1% Group Delay Ripple

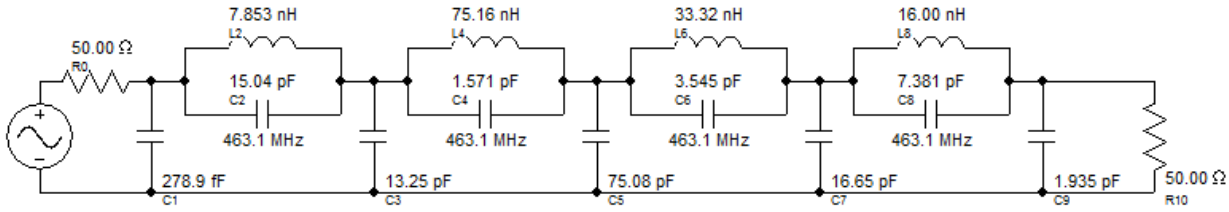


Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

65 dB Equiripple Stop Band Linear Phase Filter With 1% Group Delay Ripple



65 dB Single Point Ripple Stop Band Linear Phase Filter With 1% Group Delay Ripple



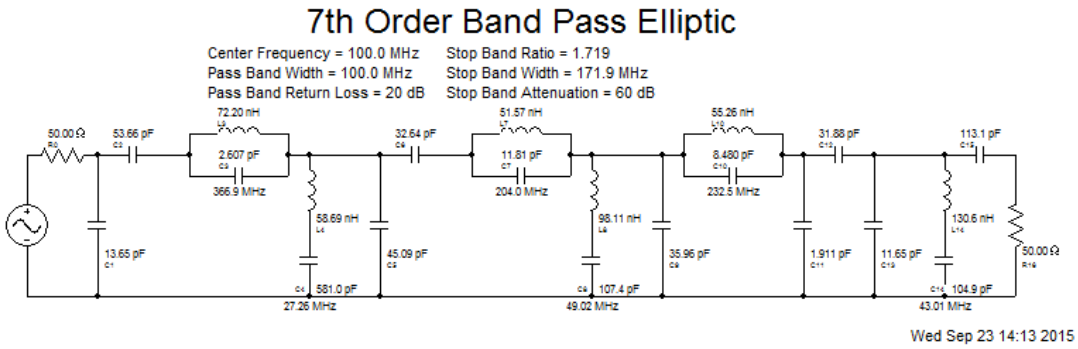
Synthesized Elements for 65 dB Single Point Ripple Stop Band Linear Phase Filter With 1% Group Delay Ripple

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

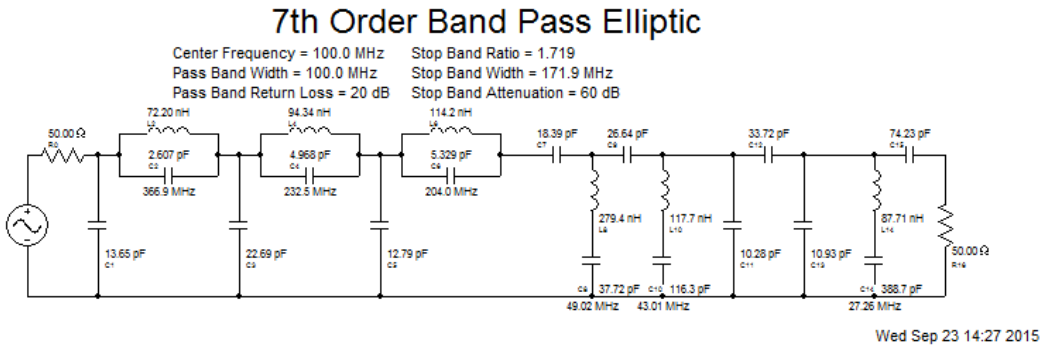
Minimum Inductor Zigzags and High/Low 2

Inductor count for band pass filters is minimized with the use odd order Zigzag filters and High/Low 2 band pass topologies. These type of useful topologies require special synthesis techniques to avoid residual error, the amount of residual error depending upon the skills of the synthesizer. FilterSolutions' Automated Parameter Synthesis provides an exact synthesis solution with no visible residual error. Design requirements of the synthesized ideal filters are met precisely.

Odd Order Minimum Inductor Zigzags also have the advantage being realized with crystal oscillators.

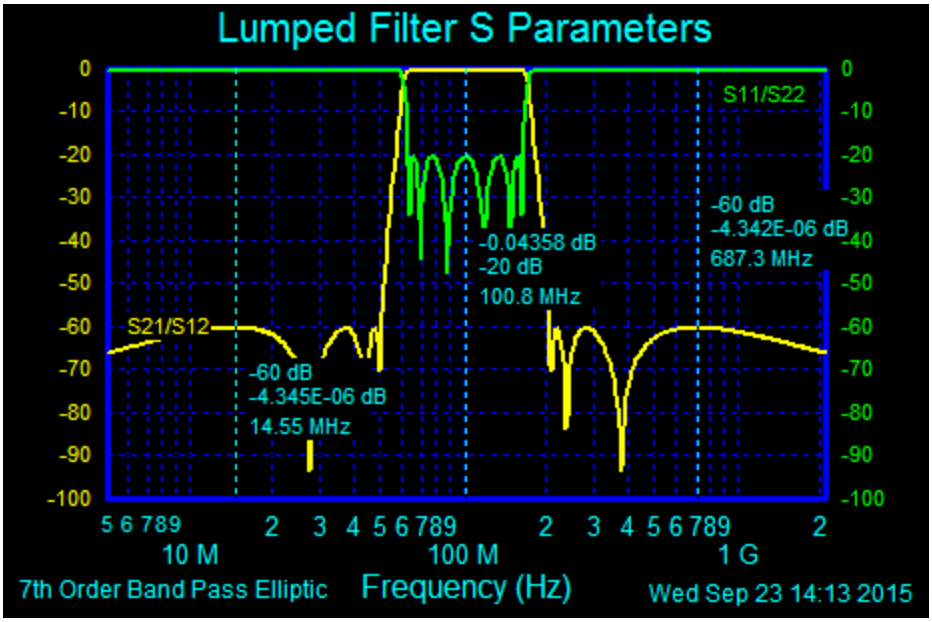


7 Pole Minimum Inductor Zigzag With 6 Inductors That Meets EXACT Design Requirements



7 Pole Minimum Inductor High/Low Pass 2 With 6 Inductors That Meets Exact Design Requirements

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



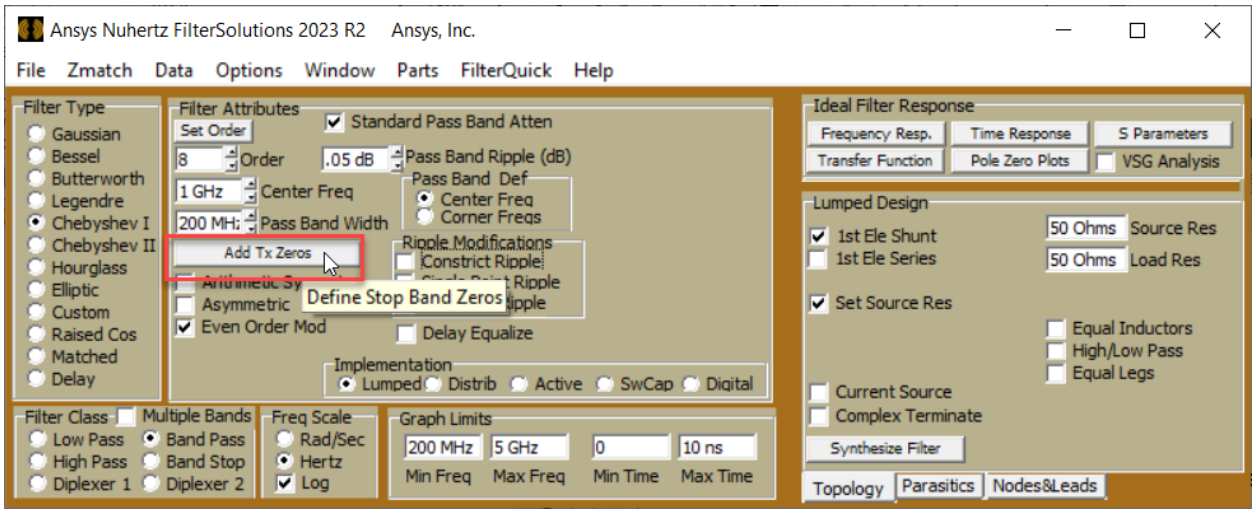
S-Parameter Response of Above Ideal Minimum Inductor Filters.

Manual Tx Zero Placement

Text Placement:

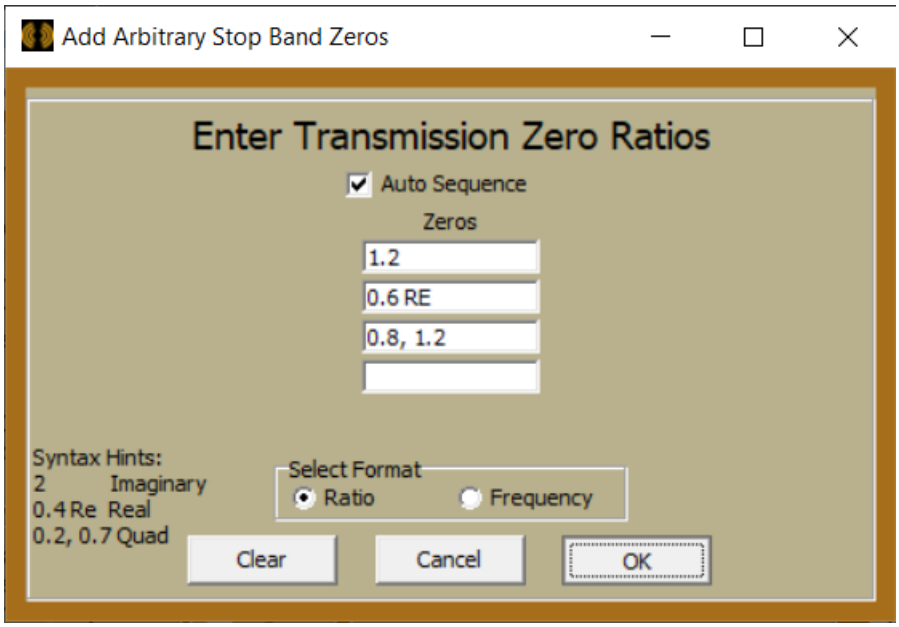
For Butterworth and Chebyshev I filters:

1. Click **Add Tx Zeros** in the **Filter Attributes** group box.



Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Fill out the pop-up dialog with desired zero positions.
   - Frequency axis zeros are entered as a simple numerical value, "1.2" or "1.2 MHz".
   - Real zeros (useful for group delay manipulation and equalization) are entered with a "**RE**" suffix, "1.2 RE" or "1.2 MHz RE"
   - Complex zeros, or quadruplet zeros, are entered as two numbers separated by a comma, "0.8, 1.2" for 0.8 + J1.2.



3. Select **Ratio** for zeros normalized to the cutoff or center frequency, otherwise, select **Frequency**.

Placement rules:

i. Frequency axis zeros may not be placed directly on the passband. Therefore, all ratio values MUST be >1.0.

ii. Complex zeros are always created in quadruplet form. Enter one of the quadruplet zeros should be entered, and the other three are created automatically.

iii. Distributed Filters support only frequency axis zeros.

Sequencing Zeros in the Ladder Network (lumped filters only)

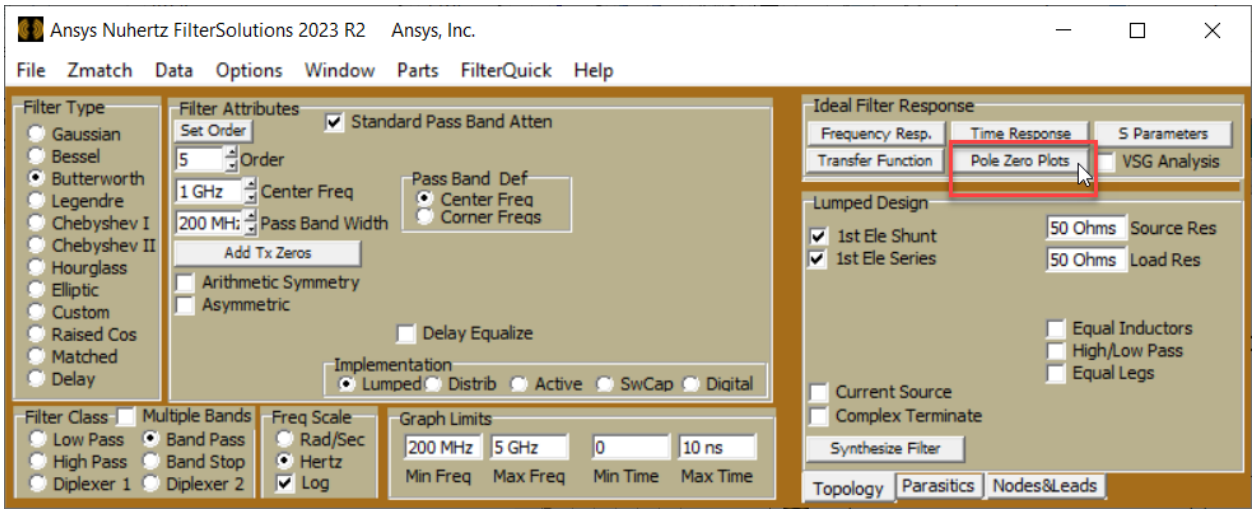
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Check "Auto Sequence" to sequence the zeros automatically. Uncheck to enter manual sequence positions. Final zeros position may differ from manual entries if the ladder network cannot synthesize the desired position(s).

Positioning Zeros With the Mouse

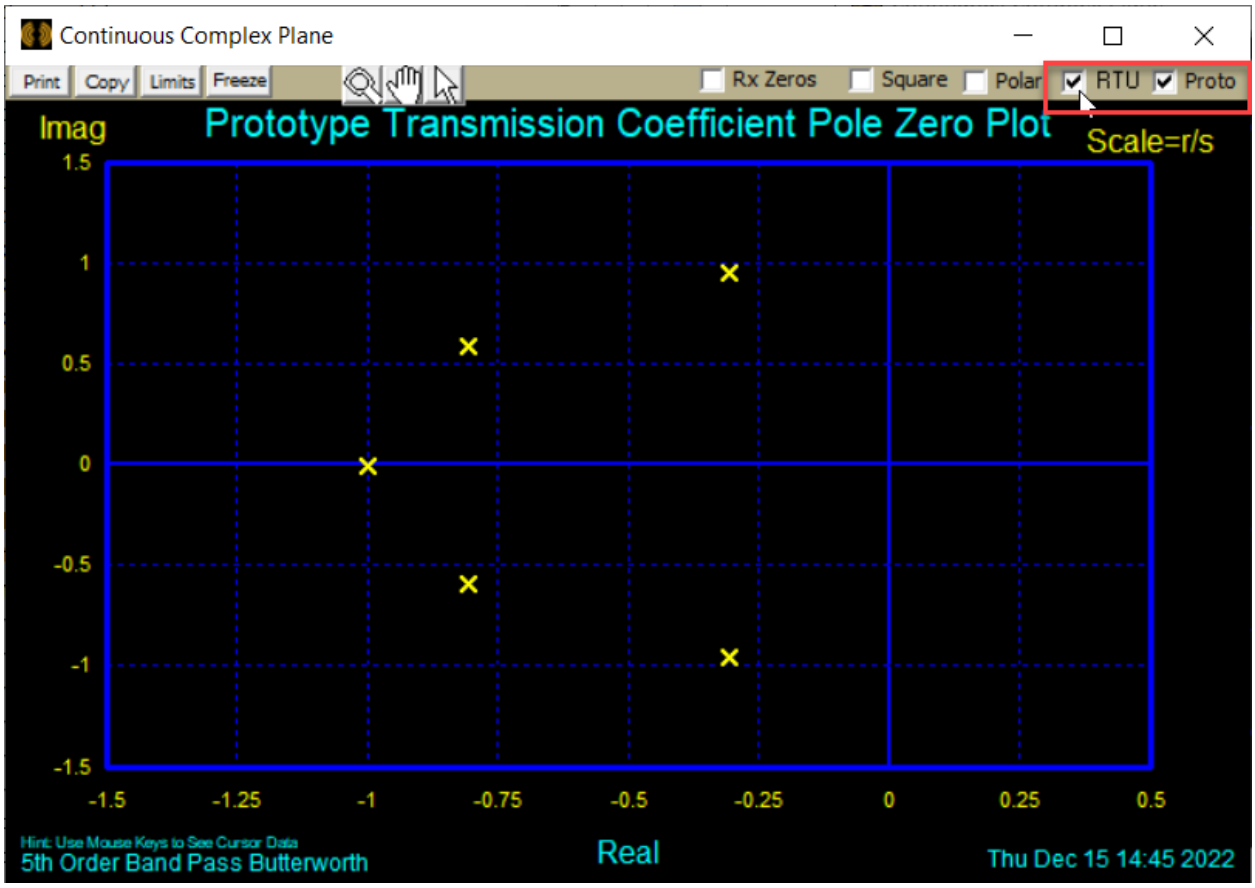
General

1. Select **Pole Zero Plots**



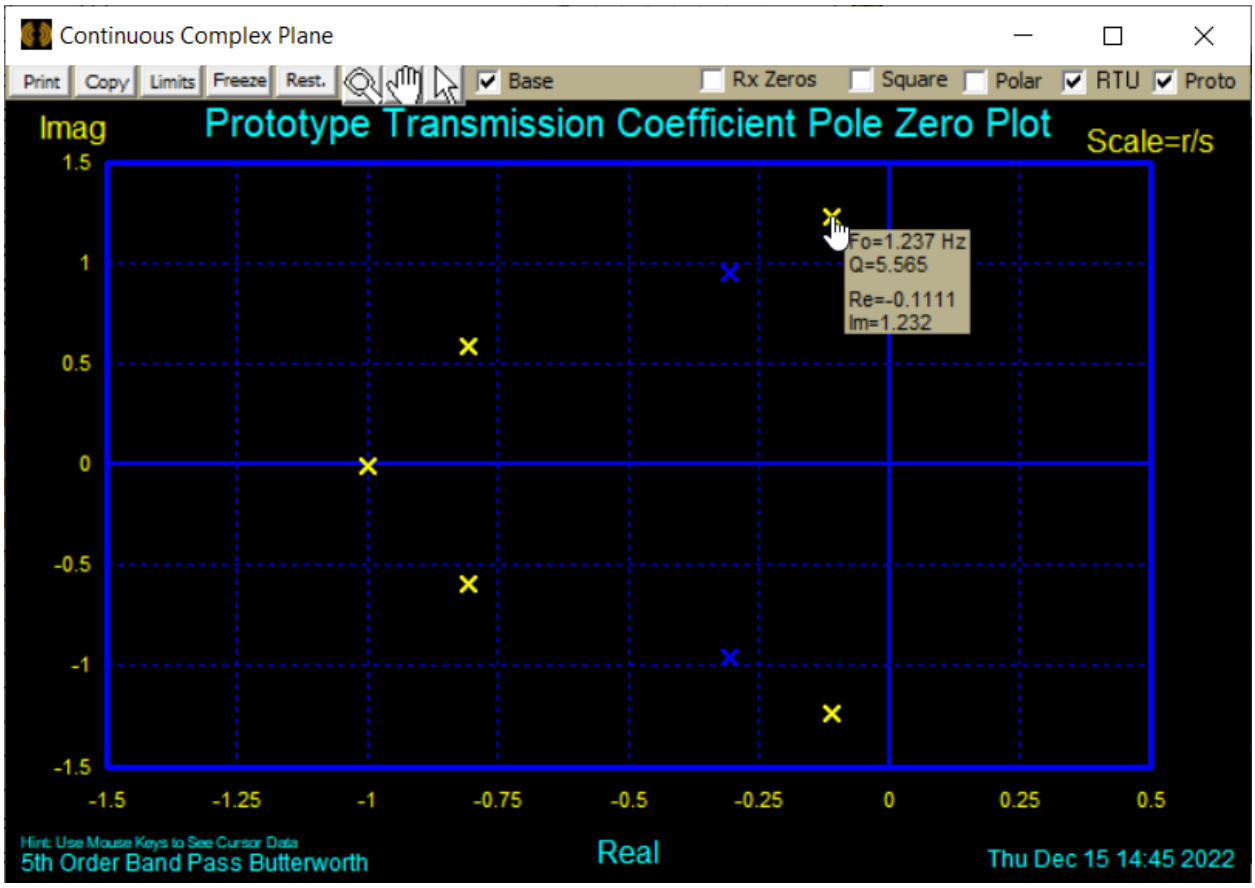
2. Check **RTU** (Real Time Update)

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Check **Restore Passband** (Available for Butterworth, Chebyshev I, Chebyshev II, and Elliptic only)
4. Select a zero on the plot, and move it around with the mouse. The pass band will be recalculated automatically. It is useful to keep an ideal analysis plot open while doing this to view the updated analysis in real time. When done, synthesize the filter with the "Synthesize Filter" button.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. Use the **Zoom** and **Drag** functions as needed to easily access the desired zeros on the display.

Saving Zeros Positions

(Available for Butterworth, Chebyshev I, Chebyshev II, and Elliptic only)

When satisfied with the positions, click **Update Zeros** to save. Zero positions will be automatically entered into the **Zeros** table accessed via **Add Tx Zeros**. Elliptic and Chebyshev II filters will revert to Butterworth and Chebyshev I filters with stop band zeros.

Band Pass Filter Zeros

Frequency Symmetrical band pass filters (and band stop filters) support prototype zeros movement only. Check **Proto** to display and manipulate the band pass prototype zeros.

Frequency asymmetrical band pass filters support direct zero manipulation. On the Main window, select **Asymmetrical** and enter the desired low pass and high pass orders to design

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the baseline filter. Frequency asymmetrical filters have no prototype, and support direct manipulation of zeros.

If a baseline filter is frequency symmetric, but there is no need or desire to maintain the symmetry, begin by designing the symmetric filter by selecting "Asymmetric", and entering the same filter order for both the low pass and high pass entries. Moving the around in this configuration will allow flexible zero placement, without concern for maintaining frequency asymmetry.

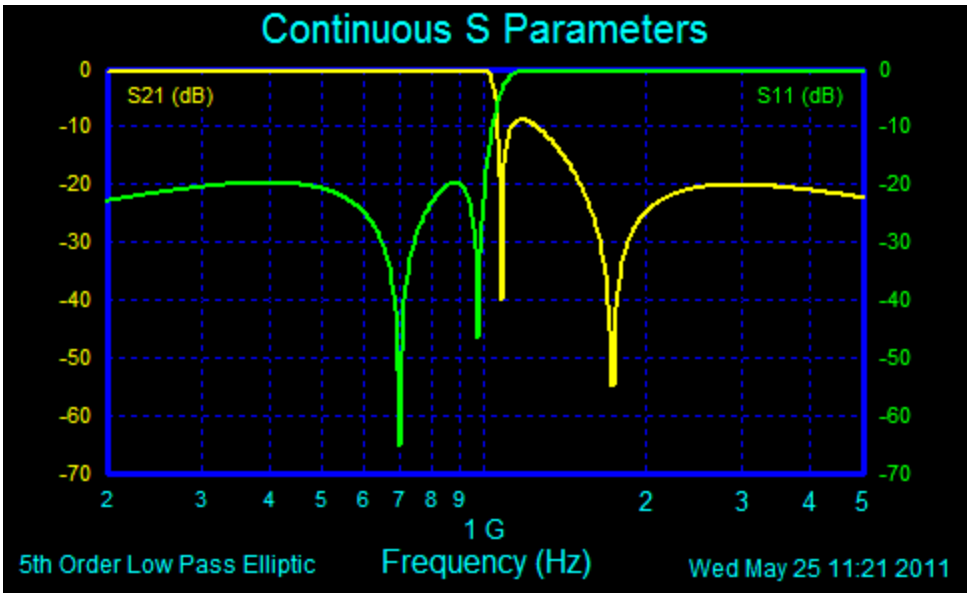
Zeros Movement Rules

i. Frequency axis zeros must always remain on the frequency axis

ii. Real axis zeros must always remain on the real axis

iii. Only one zero of a conjugate pair or quadruplet set needs to be moved. The others will update automatically.

iv. Pole positions update automatically as needed when "Restore Passband" is checked to maintain passband maximal flatness, equiripple, or constricted equiripple.



Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4 - Descriptions of Filters

This chapter contains the following topics:

- Low Pass, High Pass, Band Pass, and Band Stop
- Pass Band and Stop Band Definitions
- Asymmetrical Band Pass Filters
- Gaussian
- Bessel and Linear Phase
- Butterworth
- Legendre
- Chebyshev I
- Chebyshev II
- Hourglass
- Elliptic
- Custom
- Raised Cosine
- Matched
- Delay

Low Pass, High Pass, Band Pass, and Band Stop

FilterSolutions defines a filter class as one of the following: Low Pass, High Pass, Band Pass, Band Stop, and Diplexer. Diplexers are only defined for passive circuits with nonzero source resistance.

It is typical for filters to be derived from their prototypes, which is the low pass case with a pass band frequency of one radian per second, and then translated to the to the low pass, high pass, band pass and band stop transfer functions with the pass band frequency, center frequency, and/or bandwidth set to the desired value.

The translations for each case are:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$LP: S' = S/Wo$$

$$HP: S' = WoWo$$

$$BW: S' = \frac{Wo}{Bw} * \left[\frac{\left(\frac{S}{Wo}\right)^2 + 1}{\left(\frac{S}{Wo}\right)} \right]$$

$$BS: S' = \frac{Bw}{Wo} * \left[\frac{\left(\frac{S}{Wo}\right)}{\left(\frac{S}{Wo}\right)^2 + 1} \right]$$

Where :

S = Prototype Complex Frequency

S' = Mapped Complex Frequency

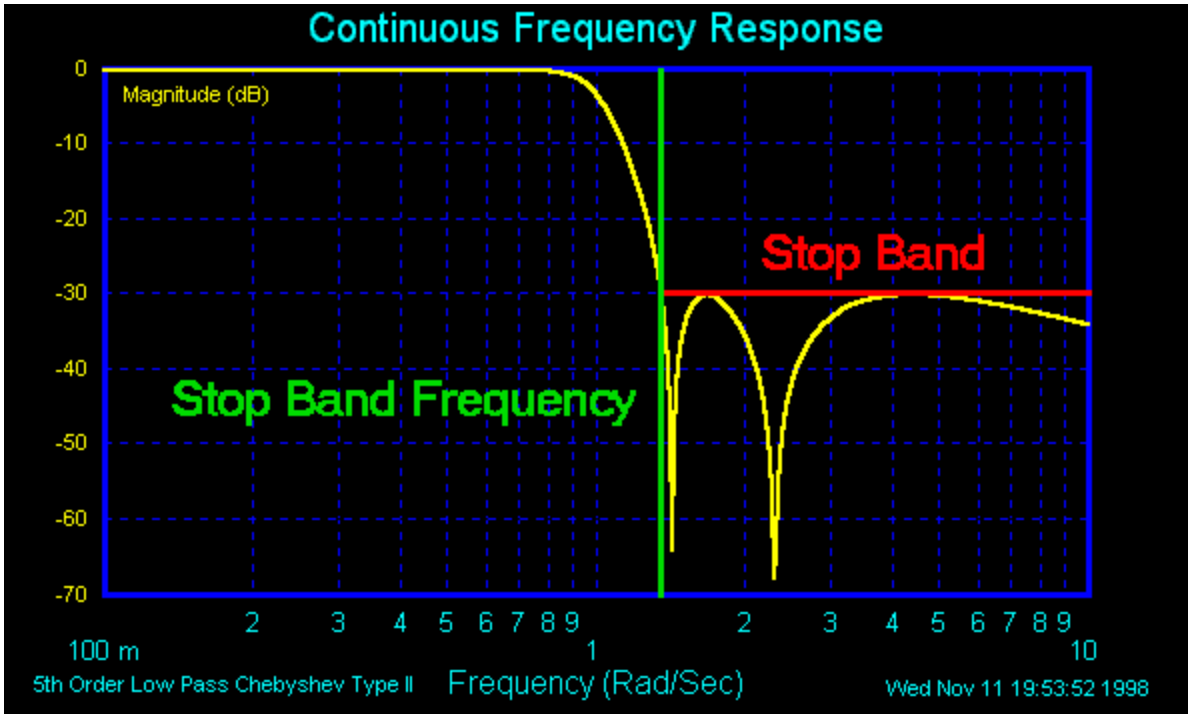
Wo = Desired Pass Band or Center Frequency

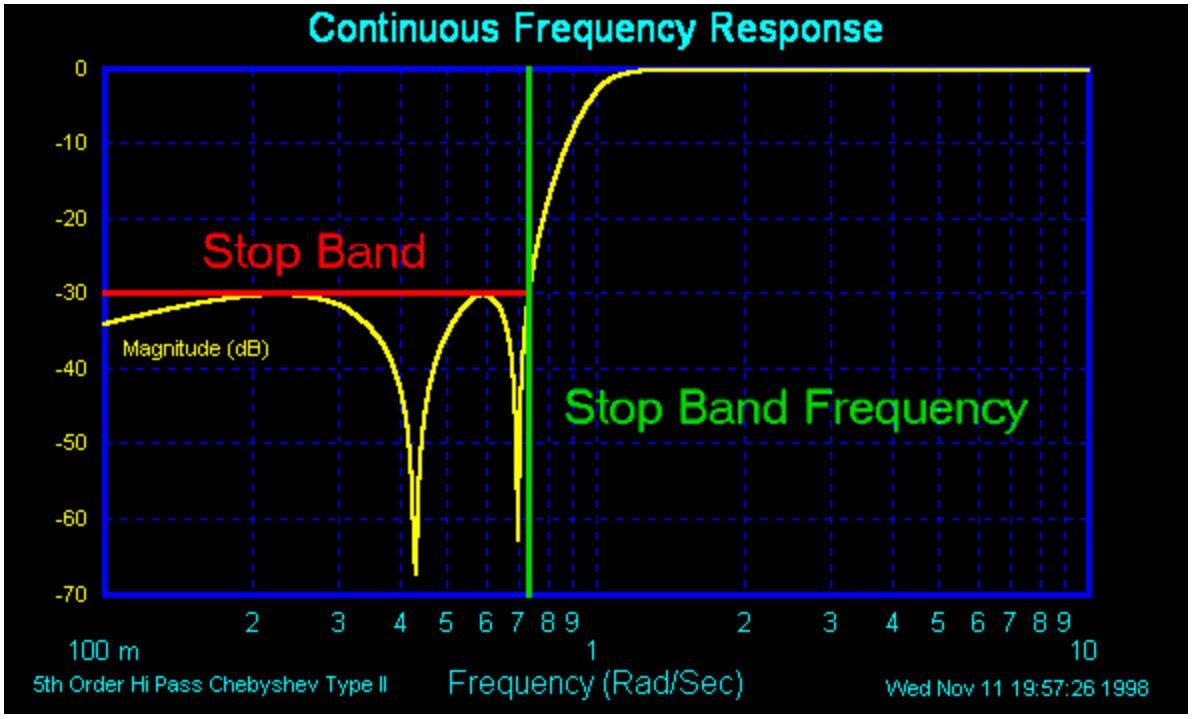
Bw = Desired Bandwidth

Pass Band and Stop Band Definitions

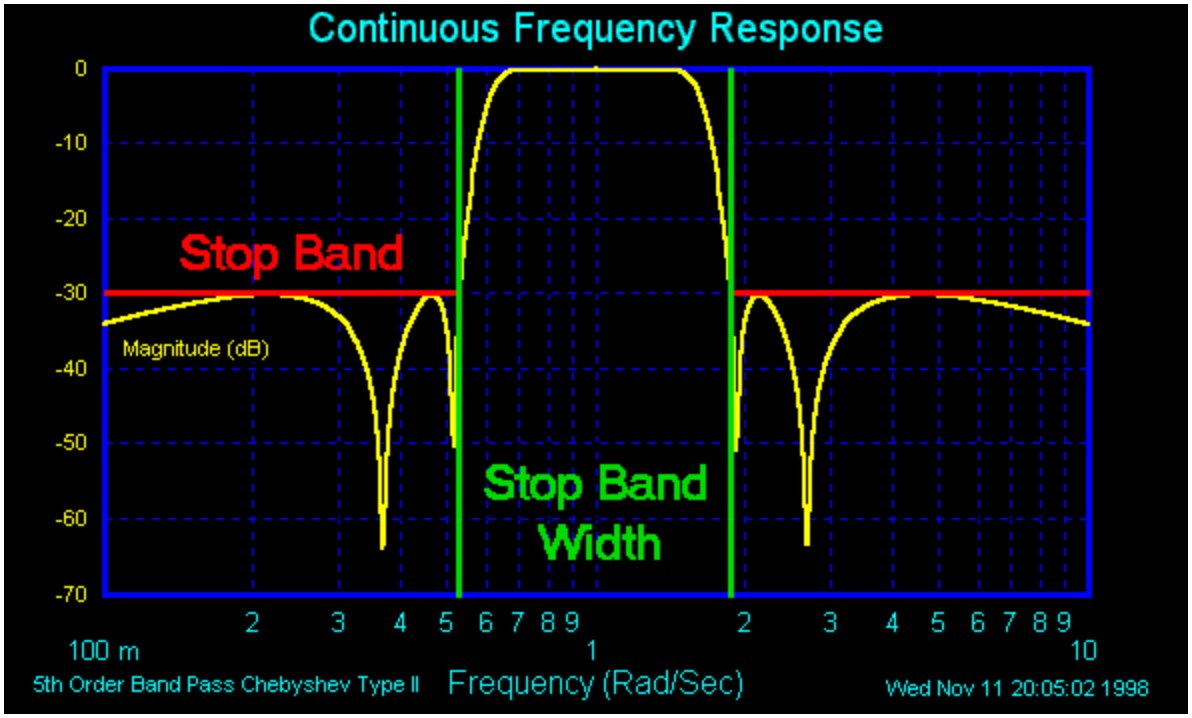
Below are pictorial definitions of Pass Bands and Stop Bands as used by FilterSolutions

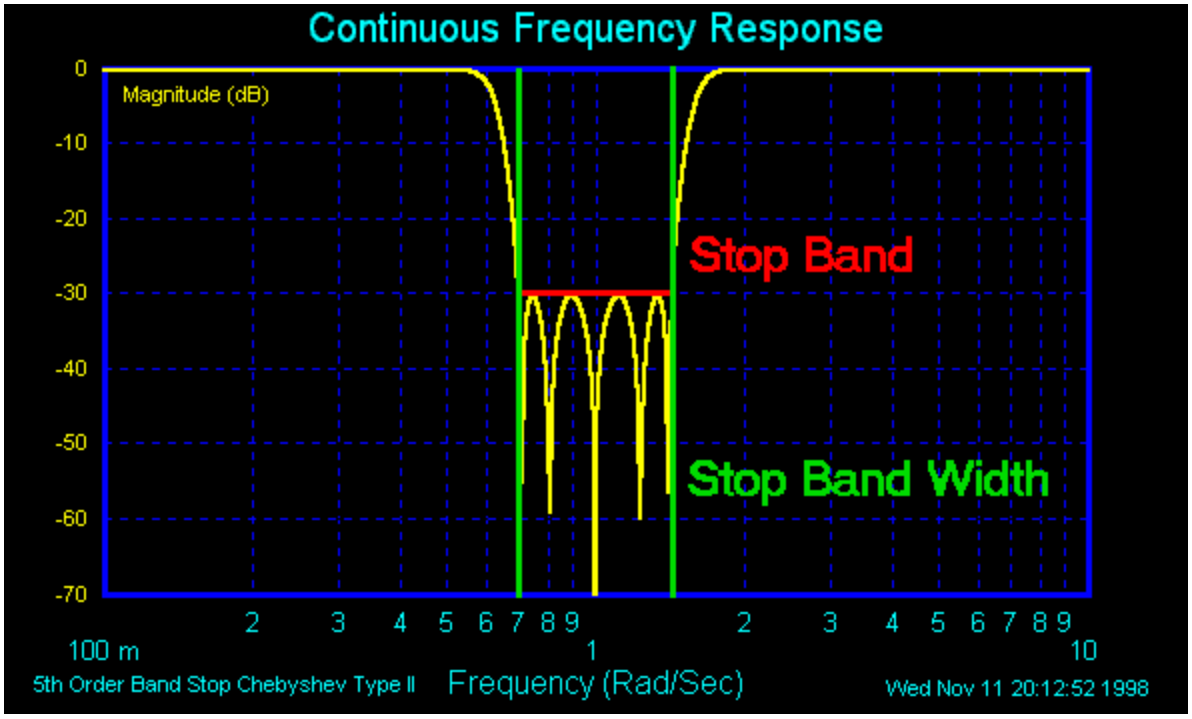
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



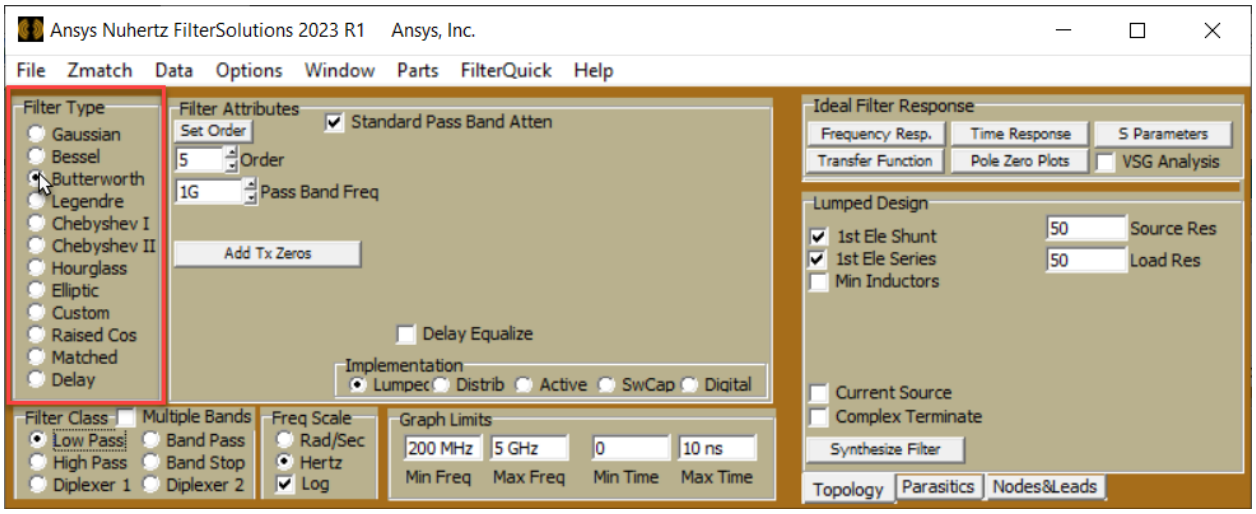


Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

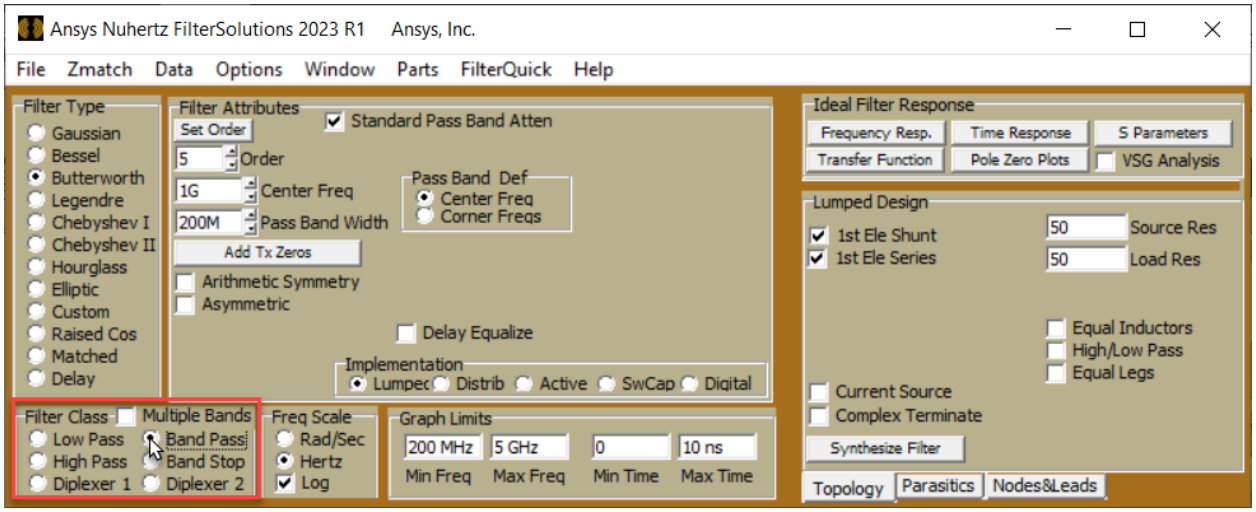
Frequency Asymmetrical Band Pass Filters

Asymmetrical filters are band pass filters are created using a different orders for the high pass and low pass parts of the filters. In the case of Chebyshev I1, Chebyshev II, Hourglass and Elliptic filters, the low pass and high pass orders must both be odd in order to implement with passive or tx line filters.

1. Select a filter type in the **Filter Type** group box.

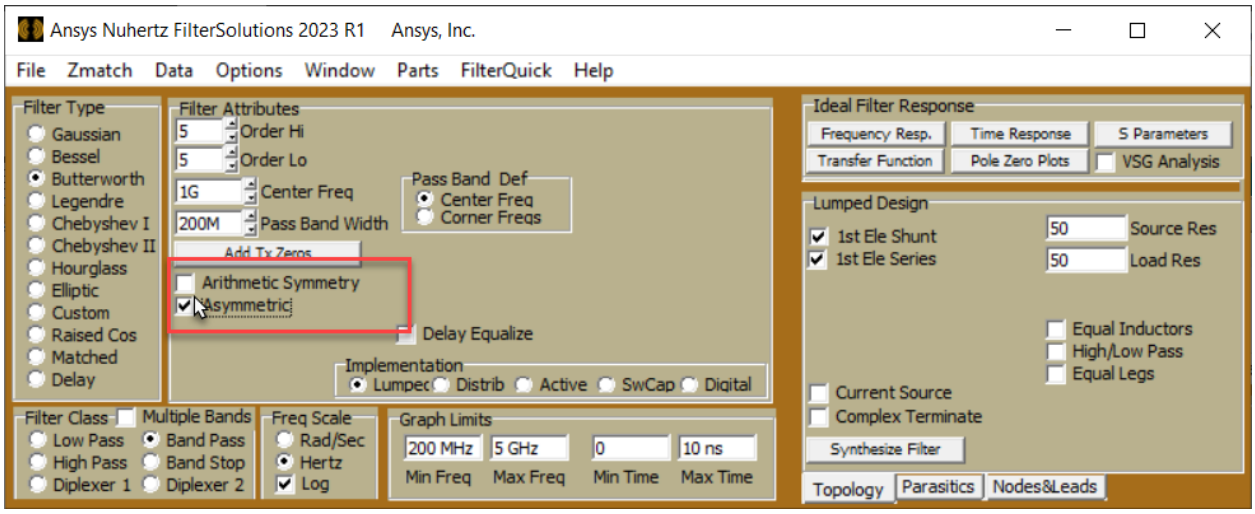


2. Select **Band Pass** in the **Filter Type** group box.

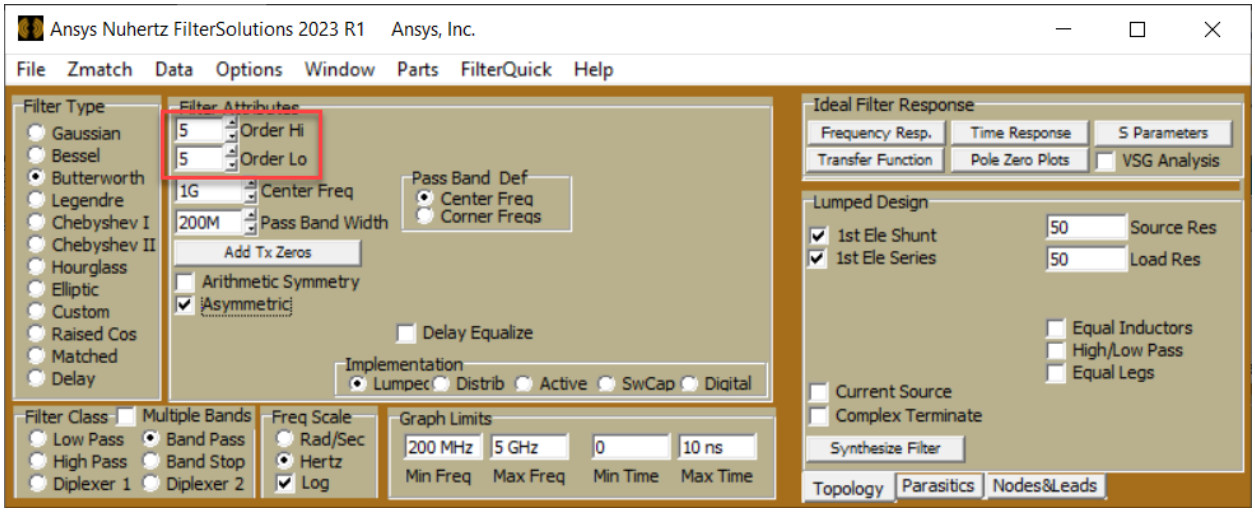


3. Select **Asymmetric** in the **Filter Attributes** group box.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Enter the desired low pass and high pass orders into the respective **Order Lo** and **Order Hi** text boxes.



Parametric Filters

Frequency asymmetrical band pass filters with an odd total order (one side even and one side odd) are synthesized through a processes known as parametric synthesis. By necessity, such even/odd order filters synthesized though a parametric process have a reflection zero on the real frequency axis. Butterworth , Chebyshev I, Chebyshev II, and Elliptic filters support a parametric synthesis. Other filter types are not supported in their parametric form at this time

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Source Resistor Restrictions and Compensation

The source resistor specification for passive and distributed filters has the effect of setting the reflection coefficient that would be a matching equivalent to that found in a symmetrical filters. The result is that the calculated source resistance is not the same as the specified source resistance. The **Set Source Res** switch in the Lumped control panel on the right will, if displayed, reset the source resistance using one or more Norton transformations and preserve the reflection coefficient, but will increase the component count of the filter in that Norton transforms require additional components. The **Set Src Res** switch found directly below the asymmetric switch will attempt to recompute the filter to achieve the desired source resistance without adding any new components, but will result in a reflection coefficient mismatch.

Setting the Stop Bands

Stop band filters of Chebyshev II, Hourglass, and Elliptic may have their high pass and low pass leg stop bands set individually by selecting the respective "Low Stop Ratio" and "High Stop Ratio" text entries. In the case of Elliptic filter, the stop band attenuation may be selected independently for both high pass and low pass legs.

Figure 1 is an example of a frequency asymmetrical Elliptic filter, 5th order low pass, 11th order high pass filter with the low pass and high pass legs set to 40dB and 80dB of attenuation respectively.

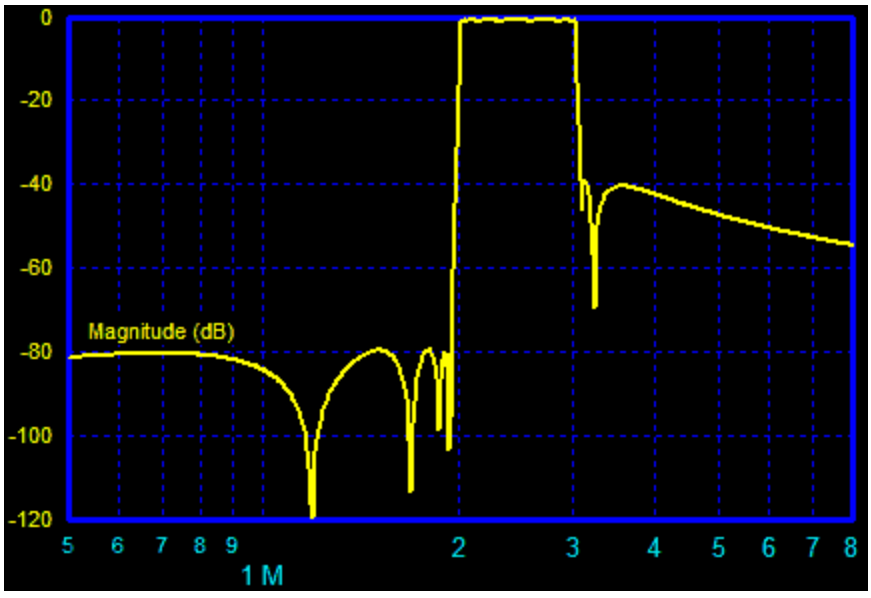


Figure 1: Frequency Asymmetric Elliptic Filter

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Gaussian Filters

The Gaussian Filter is the filter type that results in the most gradual pass band roll-off and the lowest group delay. As the name states, the Gaussian Filter is derived from the same basic equations used to derive the Gaussian Distribution. The significant characteristic of the Gaussian Filter is that the step response contains no overshoot at all.

FilterSolutions normalizes the Gaussian filter such that the prototype high frequency attenuation matches the Butterworth filter. The pass band attenuation of the Gaussian filter increases with the order of the filter when this normalization is applied. However, FilterSolutions allows the user the option of selecting the desired pass band attenuation in dB's.

Derivation

The transfer function of the prototype Gaussian filter is one over the square root of the Maclaurin series expansion of $e^{(s^2)}$ using the same number of terms as the order of the filter and using left half plane poles only for the square root.

The prototype transfer function is normalized so that the high frequency attenuation matches the Butterworth filter.

Transitional Filters

A transition option is available for Gaussian filters of order 3 or more. If a transition is selected, the attenuation of the Gaussian filter greatly increases for frequencies higher than the frequency that exists at the transition dB selection point. Gaussian Transition Filters are created with numerical methods that minimize the magnitude RMS error between the realizable Gaussian Transition filter and the ideal Gaussian Transition filter.

All Gaussian Transitional filters use -3 dB as the attenuation cutoff frequency. The steep roll off frequency is selectable by the user.

Synthesis Options

Specific element values of synthesized filters are no unique. Three different options are provided in the Advanced panel for design flexibility. The FilterQuick design panel using the option that most likely results in all positive element values.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Stop Bands and Transmission Zeros

Gaussian and Transitional filters may be synthesized with stop bands, both equiripple and single point ripple, using advanced synthesis techniques, and with arbitrarily placed transmission zeros. Stopband synthesis may lead to undesirable negative element values, especially at small attenuations. Increasing the stop band attenuation will eventually make all values positive. The attenuation threshold of negative element values changes with different synthesis option selections, so the first attempt to obtain an all-positive element value solution is to try different synthesis options. Element values are more likely to be positive when fewer transmission zeros are used, so creating a stopband manually with transmission zeros placement may also help. Selecting an attenuation requirement that maximizes stop band attenuation also helps mitigate the negative element value problem.

Bessel Filters

The Bessel Filter's distinguishing characteristic is the near constant group delay throughout the pass band of the low pass filter.

FilterSolutions normalizes the Bessel filter such that the prototype high frequency attenuation matches the Butterworth filter. The pass band attenuation of the Bessel filter increases with the order of the filter when this normalization is applied. However, FilterSolutions allows the user the option of selecting the desired pass band attenuation in dB's.

Derivation:

The continuous transfer function of the prototype Bessel filter is K/F(S) where F(S) is the recursive Bessel relation:

$$Fn(S) = (2n - 1)\, F_{n-1}(S) + S^2 F_{n-2}(S)$$

Where:

$$F0(S) = 1$$

$$F1(S) = 1 + S$$

Linear Phase Filters with Equiripple Group Delay

Select **Equiripple Delay** to generate Linear Phase filters with an equiripple group delay. Linear Phase filters are a group of filters that includes Bessel Filters. The synthesis process for

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

equiripple group delay is identical to Delay filters, except the numerator of the prototype transfer function is a constant. The low pass group delay of Linear Phase equiripple filters remains nearly constant at higher frequencies than the Bessel Filters.

The group delay is designed to be accurate for frequencies up to approximately (N-1) /(T*PI) Hz or 2*(N-1)/T Rad/Sec where N is the order of the filter, and T is the design transport delay. This figures varies as the ripple period varies.

Ripple Period

FilterSolutions allows one to fine tune the Bessel Filter by changing the ripple period. With an equiripple period of 2.0, the group delay of the filter is set to a forced value approximately every R/(T*2*PI) Hz or R/T Rad/Sec where R is the ripple period and T is the transport delay. Increasing the ripple period has the effect of extending the upper frequency limit of the group delay at the cost of increasing the error of the group delay. The period of each individual ripple is modified slightly to obtain the equiripple effect of the group delay.

The allowable values of the ripple period are 0 to 2.8. Setting the ripple period to zero produces a classical Bessel group delay response.

Ripple Magnitude

Several precomputed group delay ripple magnitudes may be selected in lieu of selecting the ripple period. The advantage of using a precomputed filter is that the group delays are far more accurate equiripples.

With Stop Bands

Low pass Linear Phase Filters with stop bands are essentially Matched Filters that have been optimized for frequency response. The prototype is derived directly from the Delay Filter prototype:

$$F_{\substack{Linear\\Phase}}(S) = \frac{\left(1 - F_{Delay}(S)\right)}{S}$$

Synthesis Options

Specific element values of synthesized filters are no unique. Three different options are provided in the **Advanced** panel for design flexibility. The **FilterQuick** design panel using the option that most likely results in all positive element values.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Stop Bands and Transmission Zeros

Bessel and Linear Phase filters may be synthesized with stop bands, both equiripple and single point ripple, using advanced synthesis techniques, and with arbitrarily placed transmission zeros. Stopband synthesis may lead to undesirable negative element values, especially at small attenuations. Increasing the stop band attenuation will eventually make all values positive. The attenuation threshold of negative element values changes with different synthesis option selections, so the first attempt to obtain an all-positive element value solution is to try different synthesis options. Element values are more likely to be positive when fewer transmission zeros are used, so creating a stopband manually with transmission zeros placement may also help. Selecting an attenuation requirement that maximizes stop band attenuation also helps mitigate the negative element value problem.

Butterworth Filters

The Butterworth Filter is the filter type that results in the flattest pass band and contains a moderate group delay.

A standard Butterworth Filter's pass band attenuation is -3.01dB. However, FilterSolutions allows the user the option of selecting any pass band attenuation in dB's that will define the filters cut off frequency.

FilterSolutions also offers the user the option of placing user-defined zeros in the stop band. Such a filter with stop band zeros is no long a true Butterworth Filter, but is still in the Maximally Flat filter family.

Derivation:

The characteristic equation for the Butterworth filter is

$$K(S) = es^N$$

Where:

e = Arithmetic Pass Band Attenuation

N = Order of the Filter

The addition of stop band zeros, generally referred to as a "Maximally Flat" filter, modifies the characteristic equations to be:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$K(s) = \frac{es^N}{\prod_{i=1}^{N_p}(s - Z_i)}$$

Where:

e = Arithmetic Pass Band Attenuation

N = Total Order of the Filter

Z_i = Finite Stop Band Zeros (Complex)

N_p = Number of Complex Stop Band Zeros

Asymmetrical Band Pass filters

In the case of asymmetrical band pass filters, the low pass prototype cannot be used. The characteristic equation is then,

$$K(s) = \frac{e(s^2 + 1)^{N/2}}{\prod_{i=1}^{N_p}(s - Z_i)}$$

Where:

e = Arithmetic Pass Band Attenuation

N = Total Order of the Filter

Z_i = Finite Stop Band Zeros (Complex)

N_p = Number of Complex Stop Band Zeros

Note that the band width of the functions is essentially set by the variable "e". Iterations are generally required to set the band width to the desired width at the specified attenuation.

Using half plane poles only, he transfer function is:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$G(S) = \sqrt{\frac{1}{1 + K(S)K(-S)}}$$

If zeros are assigned to the filter, the zeros are placed in polynomial form in the denominator of the characteristic equation.

Stop band zeros may be placed anywhere on the JW axis outside the stop band or anywhere on the real axis. To enter real zeros, add the suffix "re" to the frequency value to indicate real.

Legendre Filters

The Legendre filter is an all pole filter that produces the steepest cutoff frequency without producing any passband ripples of oscillating slopes. The pass band curve slope varies between downward and zero only, never upwards. Legendre filters are sometimes referred to as Monotonic Class L filters.

Legendre filters are useful in applications which require a steep cutoff at the pass band edge without pass band ripples, or in cases where a Chebyshev I filter produces too much group delay at the pass band edge.

Derivation

The filter is characteristic function is derived from the Legendre polynomials in which the filter is named after.

The transfer function is:

$$G(S) = \sqrt{\frac{1}{1 + e^2 L(S^2)}}$$

Where L(S) is a squared characteristic polynomial derived from the Legendre polynomials.

The -3dB bandwidth is set with the variable *e*. There are no provisions for stop band zeros or an asymmetrical band pass with Legendre filters.

Inverse Legendre Filters

The Legendre function may apply to the stop band too, producing the most rapid monotonic attenuation in the stop band near the cutoff frequency. However, this requires complex stop

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

band zeros, making the function unrealizable with ideal inductors and capacitors. FilterSolutions supports Inverse Legendre filters for active, switched capacitor and digital filters.

Chebyshev Type I Filters

The Chebyshev Type I Filter is the filter type that results in the sharpest pass band cut off and contains the largest group delay. The most notable feature of this filter is the ripple in the pass band magnitude.

A standard Chebyshev Type I Filter's pass band attenuation is defined to be the same value as the pass band ripple amplitude. However, FilterSolutions allows the user the option of selecting any pass band attenuation in dB's that will define the filters cut off frequency.

FilterSolutions also offers the user the option of placing user-defined zeros in the stop band.

Derivation:

The characteristic equation of the Chebyshev I filter is:

$$K(S) = eCS(N)$$

Where:

e = Arithmetic Ripple Magnitude

CS(N) = Nth Order Chebyshev Polynomial

Stop band zeros may be placed anywhere on the JW axis outside the stop band or anywhere on the real axis. To enter real zeros, add the suffix "re" to the frequency value to indicate real. A more generalized expression is used to create Chebyshev filters with stop band zeros

Asymmetrical Band Pass filters

In the case of asymmetrical band pass filters, the low pass prototype cannot be used. The characteristic equation is replaced with a more generalized equiripple function.

Stop Band Zeros

If zeros are assigned to the filter stop band, the zeros are placed in polynomial form in the denominator of the K(S) characteristic equation, and the Chebyshev polynomial in the K(S)

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

numerator is replaced with a more general equiripple equation.

Constricted Equiripple

The performance of Chebyshev I filters (and also Elliptic filters) may sometimes be improved by constricting the equiripple to a percentage of the passband near to the cutoff frequency. FilterSolutions provides for a "Constrict Equiripple" function to accomplish this. If the passband performance away from the cutoff frequency is not an issue, it is sometimes possible to reduce the required order of the filter by constricting the ripple.

See Constricted Equiripple Passbands in the Help for more information and examples.

Chebyshev Type II Filters

The Chebyshev Type II Filter, also known as the Inverse Chebyshev Filter, contains a Butterworth style, or maximally flat, pass band, a moderate group delay, and an equiripple stop band.

Like the Butterworth Filter, the pass band attenuation of the Chebyshev Type II Filter is defined to be -3.01 dB. However, FilterSolutions allows the user the option of selecting any pass band attenuation in dB's that will define the filters cut off frequency.

Derivation:

The characteristic equation of the Chebyshev I filter is:

$$K(S) = \frac{es^N}{CrS(N)}$$

Where:

e = Arithmetic Pass Band Attenuation

N = Order of the Filter

CrS(N) = Nth order Chebyshev Polynomial Revolved About the Stop Band Ratio.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Asymmetrical Band Pass Filters

If it is desired to create asymmetrical band pass filters, the steps are to create a Chebyshev I asymmetrical filter, then revolve the K(s) numerator roots about the respective corner frequencies. This process creates a flat stop band, but slightly corrupts the pass band. The filter pass band is fixed using numerical iterations at the expense of stop band quality.

Using left half plane poles only, the transfer function is:

$$G(S) = \sqrt{\frac{1}{1 + K(S)K(-S)}}$$

Hourglass Filters

The Hourglass filter's distinguishing trait is that the reflection zero frequencies are exactly the inverse of the transmission zero frequencies. The Hourglass Filter is similar to the Chebyshev Type II Filter, but has a sharper cut off, higher group delay, and greater stop band attenuation. The pass band also contains a slight equiripple characteristic, which makes it a special case of the Elliptic filter.

Like the Chebyshev Type II Filter, the pass band attenuation of the Chebyshev Type II Filter is defined to be -3.01 dB. However, FilterSolutions allows the user the option of selecting any pass band attenuation in dB's that will define the filters cut off frequency.

Derivation:

The Hourglass filter derivation involves many steps of iteration that produces the defining traits.

The Hourglass Filter was first derived and documented by Dr. Byron Bennett of Montana State University, and is documented in *IEEE Transactions on Circuits and Systems, December 1988, volume 12 page 1469.*

Elliptic Filters

The Elliptic Filter contains a Chebyshev Type I style equiripple pass band, an equipped stop band, a sharp cut off, high group delay, and greatest stop band attenuation.

Like the Chebyshev Type I Filter, the Elliptic pass band attenuation is defined to be the same value as the pass band ripple amplitude. However, FilterSolutions allows the user the option of selecting any pass band attenuation in dB's that will define the filters cut off frequency.

Derivation:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The equations shown in the Chebyshev I and II filters are used iteratively until a K(S) is established that maintains an equiripple in both the pass band and the stop band. The transfer function is then

$$G(S) = \sqrt{\frac{1}{1 + K(S)K(-S)}}$$

using left half plane poles only.

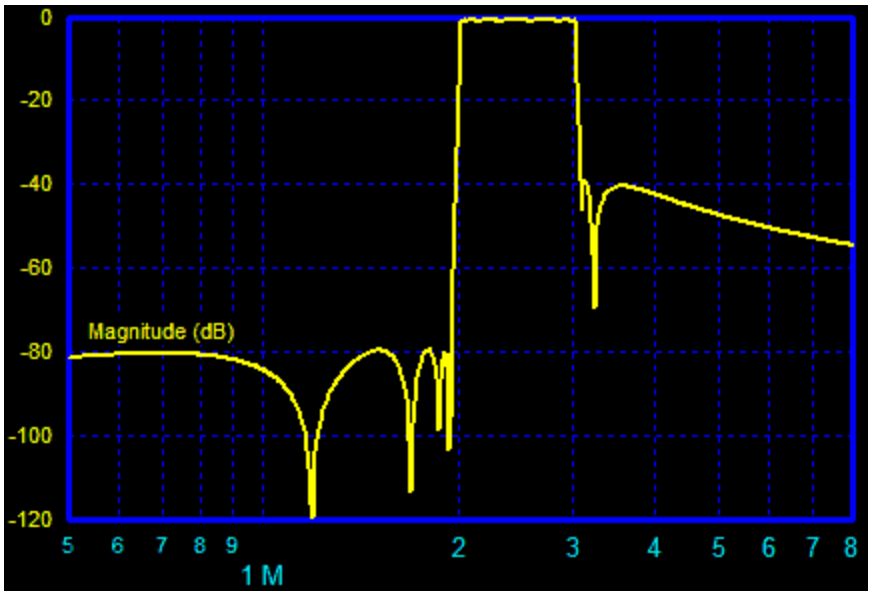
Constricted Equiripple Passband

The performance of Elliptic filters (and also Chebyshev I filters) may sometimes be improved by constricting the equiripple to a percentage of the passband near to the cutoff frequency. FilterSolutions provides for a "Constrict Equiripple" function to accomplish this. If the passband performance away from the cutoff frequency is not an issue, it is sometimes possible to reduce the required order of the filter by constricting the ripple.

See Constricted Equiripple Passbands in the Help for more information and examples.

Frequency Asymmetrical Elliptic Band Pass Filters.

Band pass Elliptic filters may have the high pass and low legs set independently. Figure 1 illustrates a frequency asymmetrical Elliptic filter, 5th order low pass, 11th order high pass filter with the low pass and high pass legs set to 40dB and 80dB of attenuation respectively.



Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1: Frequency Asymmetric Elliptic Filter

Custom Filters

FilterSolutions provides the capability to create the filter based upon an existing Laplace Transform by selecting **Custom** in the Filter Type group box. Before selecting "Custom", have the implementation method already selected. If the Laplace Transform is a prototype, have the desired cutoff frequency and bandwidth already selected as well. Staged filters may be created by placing desired poles and zeros into the Custom Filter form entries.

Prototypes

A prototype transform is a low pass filter with a cutoff frequency of 1 radian per second. If **Proto S** is selected in the Custom Box, FilterSolutions will transform to the desired high pass, band pass or band stop filter, and provide frequency scaling prior to generating the passive or active filter. The user must enter pass band frequency, band width, and filter class.

Actual, S Transform

If **Act S** is selected along with **S Trans** in the lower left, FilterSolutions will set the magnitude of the filter, and generate a digital or analog circuit filter based upon exactly what was entered. It is important to note that not all transfer functions are realizable in passive circuit form, especially when the load and source resistors are equal. FilterSolutions will not generate band pass or band stop passive filters from actual Laplace transforms.

Actual, Mag Freq

If **Act S is** selected along with **Mag Freq** in the lower left, FilterSolutions will use an RMS error optimization algorithm to generate a digital or analog filter. Minimum and maximum transfer function stage Q may be entered as well as the number of transmission zeros and frequency limits for transmission zeros. Only low pass and high pass filters are permitted. FilterSolutions automatically detects low pass and high pass entries.

G Values

G values are the low pass, all pole prototype element values. These are used frequency as the starting point in microwave filter design. Select **Gvals** to enter a string of G values. G value

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

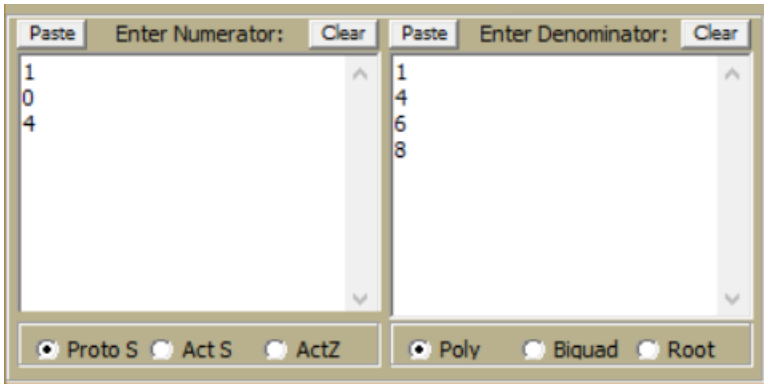
strings must be terminated, that is, they must include the source resistor (Go) and the load resistance (G(n+1)). For equally terminated filters, Go and (G(n+1)) are both one (1.000).

Digital

If **Act Z** is selected, FilterSolutions will assume that a Z transform has been entered and will output frequency and time traces and pole zero plots for the entered Z transform.

Formats

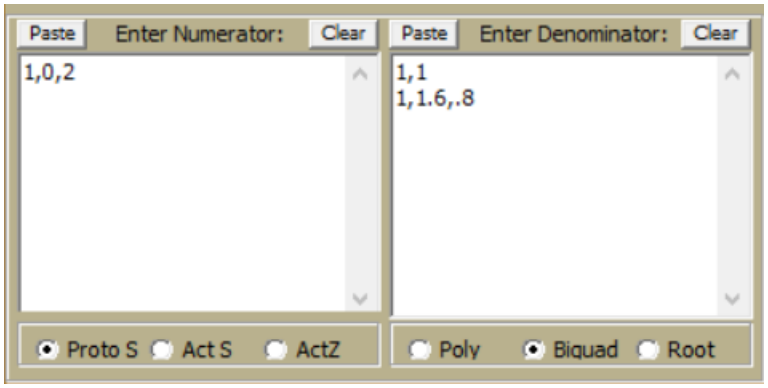
When entering transforms in polynomial form, enter the polynomial in descending powers.:



$$\frac{S^2 + 4}{S^3 + 4S^2 + 6S + 8}$$

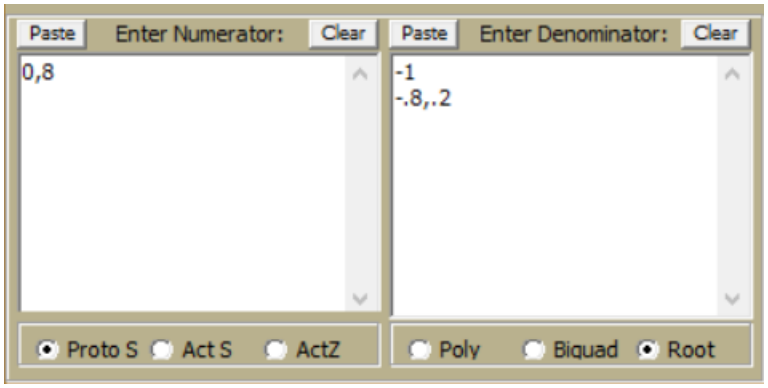
When entering biquads, use commas to separate biquad terms. Enter one biquad per line. The term "Biquad" as applied to this custom panel refers to any polynomial or order 2 or less.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



$$\frac{S^2+2}{(S+1)(S^2+1.6S+0.8)}$$

When entering roots, use commas to separate real and imaginary part. Enter only one root per conjugate pair.



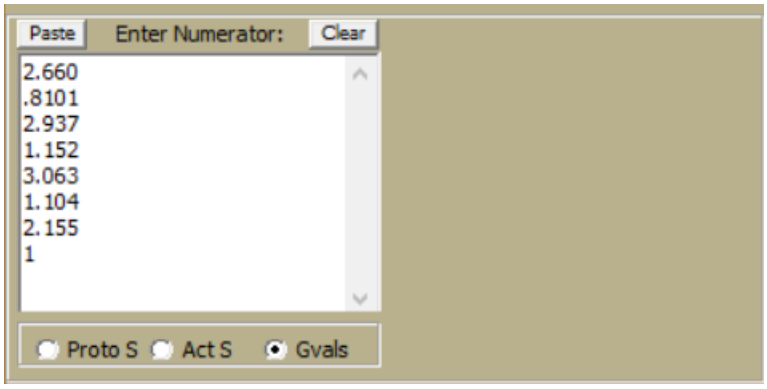
$$\frac{S^2 \pm J8}{(S+1)(S^2+0.8 \pm J0.2)}$$

Since roots do not contain any gain information, FilterSolutions will attempt to estimate the gain, but this estimated gain may not be what is expected by the user. If not satisfied with the gain from transfer functions generated from roots, it will be necessary for you to use the biquad or poly selection instead.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

G Values

When entering G values, start with Go at the top, and G(n+1) At the bottom. Below are the entries for a 6^{th} order Chebyshev with 1dB of passband ripple. Note that Go is the normalized source resistance of 2.660.



Raised Cosine Filters

The Raised Cosine Filter is for use in communications. The distinguishing characteristic of a Raised Cosine Filter is the symmetry of the magnitude frequency response on either side of the pass band cutoff frequency and a constant group delay throughout the pass band and into the stop band until 15 to 20 dB of attenuation is achieved. The magnitude frequency response of the filter is ideally a raised cosine function. The accuracy of the raised cosine function increases as the order of the filter increases. FIR Raised Cosine filters are more accurate than analog or IIR Raised Cosine filters.

FilterSolutions analog and IIR Raised Cosine filters are produced by minimizing the square of the error between the ideal filter and the actual and then delay equalizing out to 20 dB of attenuation

In FilterSolutions, the definition of the order of an analog or IIR Raised Cosine filter does not include the delay equalization part of the filter.

The ideal raised cosine filter frequency response consists of unity gain at low frequencies, a raised cosine function in the middle, and total attenuation at thigh frequencies. The width of the middle frequencies are defined by the roll off factor constant Alpha, (0<Alpha<1). In FilterSolutions, the pass band frequency is defined as the 50% signal attenuation point.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The selection of Alpha is based on the filter requirements. Real filters with low Alphas produce more ISI due to filtering error, but use less bandwidth. Real filters with high Alphas produce less ISI, but use more bandwidth.

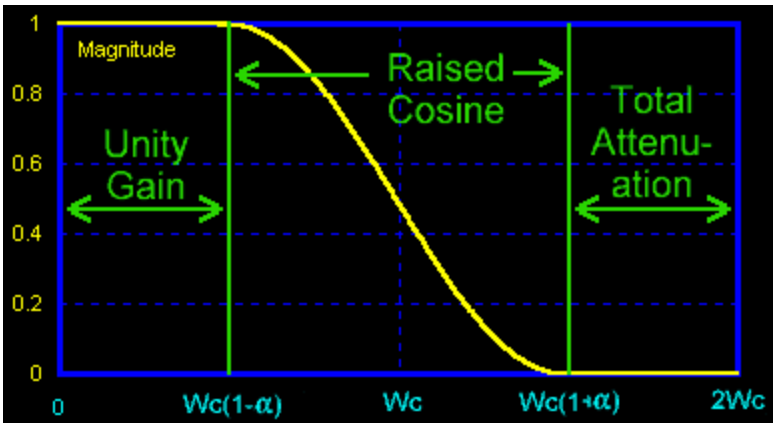
When the pass band frequency of a raised cosine filter is set to half the data rate, then the impulse response Nyquist's first criteria is satisfied in that the impulse response is zero for T = NTs, where N is an integer, and T is the data period.

Mathematically, the frequency response may be written as:

$$F(\omega) = \begin{cases} 1 & \text{For } \omega < \omega_c(1-\alpha) \\ 0 & \text{For } \omega > \omega_c(1+\alpha) \\ \dfrac{1 + COS\left(\dfrac{\pi(\omega - \omega_c(1-\alpha))}{2\alpha\omega_c}\right)}{2} & \text{For } \omega(1-\alpha) < \omega_c < \omega(1+\alpha) \end{cases}$$

Raised Cosine Filter Frequency Response

The ideal raised cosine filter frequency response is shown below:



Ideal Raised Cosine Frequency Response

The FIR Raised Cosine filter is synthesized by encoding its impulse response directly into the Z transform numerator:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$h(t) = \frac{4\alpha}{\pi\sqrt{T}} \frac{\cos\left(\frac{(1+\alpha)\pi t}{T}\right) + \frac{T}{4\alpha t}\sin\left(\frac{(1-\alpha)\pi t}{T}\right)}{1-\left(\frac{4\alpha t}{T}\right)^2}$$

Raised Cosine Impulse Response

Root Raised Cosine Filter

The ideal root raised cosine filter frequency response consists of unity gain at low frequencies, the square root of raised cosine function in the middle, and total attenuation at the high frequencies. The width of the middle frequencies are defined by the roll off factor constant Alpha, (0<Alpha<1). In FilterSolutions, the pass band frequency is defined as the .707 half power point.

The root raised cosine filter is generally used in dual series pairs, so that the total filtering effect is that of a raised cosine filter. The root raised cosine filter is generally used in series pairs, so that the total filtering effect is that of a raised cosine filter. The advantage is that if the transmit side filter is stimulated by an impulse, then the receive side filter is forced to filter an input pulse shape that is identical to its own impulse response, thereby setting up a matched filter and maximizing signal to noise ratio while at the same time minimizing ISI.

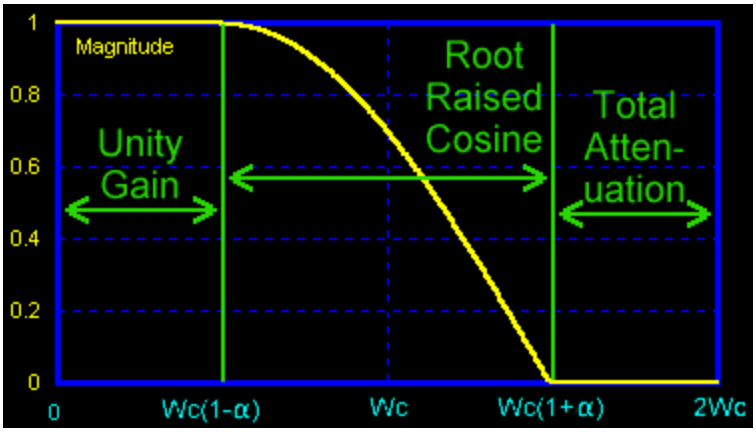
Mathematically, the frequency response may be written as:

$$F(\omega) = \begin{cases} 1 & \text{For } \omega < \omega_c(1-\alpha) \\ 0 & \text{For } \omega > \omega_c(1+\alpha) \\ \sqrt{\frac{1+COS\left(\frac{\pi(\omega-\omega_c(1-\alpha))}{2\alpha\omega_c}\right)}{2}} & \text{For } \omega(1-\alpha) < \omega_c < \omega(1+\alpha) \end{cases}$$

Root Raised Cosine Filter Frequency Response

The ideal root raised cosine filter frequency response is shown below:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ideal Root Raised Cosine Frequency Response

The FIR Raised Cosine filter is synthesized by encoding its impulse response directly into the Z transform numerator:

Ideal Root Raised Cosine Impulse Response

Data Transmission Filters

Data Transmission Filters are filters designed to minimize ISI without the need for a constant group delay. These filters are more efficient than analog raised cosine filters in that they do not require delay equalization components. However, only postcursor ISI is eliminated, that is ISI that occurs after the peak of the impulse response. Data Transmission Filters are created in FilterSolutions by minimizing the sum of the squared ISI error with numerical methods. This derivation is not unique. Other solutions for Data Transmission Filters are known to exist, such as that found in The CRC Handbook of Electrical Filters. The solutions offered by FilterSolutions has the advantage of offering the user flexibility in designing for accuracy vs. bandwidth by selecting different Alpha values.

An inspection of the data transmission filter impulse response shows that precursor ISI may be generally eliminated by doubling the pass band frequency. However, this has the undesirable effect of doubling the required bandwidth of the data channel or lessening the filtering quality.

Like Raised Cosine filters, the presence of element value errors increases ISI more in the low alpha filters than the high alpha filters, and high alpha filters require more bandwidth than low alpha filters.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

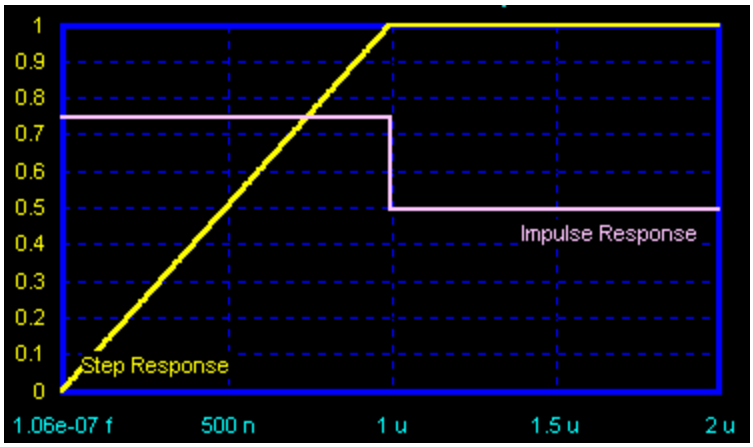
Matched Filters

The Matched Filter is for use in communications. The distinguishing characteristic of a Matched Filter is the step response approximating a ramp, and the impulse response approximates a pulse. The purpose of the Matched Filter is to maximize the signal to noise ratio at the sampling point of a rectangular pulse and thereby minimize the probability of undetected errors received from a signal.

To achieve maximum signal to noise ratio, a Matched filter impulse response must be the time inverse of the shape of the pulse being sampled. Filter Solutions Matched filters are therefore only matched to rectangular shaped pulses.

Filter Solutions allows one to define the Matched Filter by setting the rise time of the ramp. The proper use of the matched filter is to set the rise time to be equal to the pulse width of the pulses in a bit stream.

The ideal matched filter step and impulse response are shown below. It is realizable with FIR filters. FIR Matched filters are also known as Moving Average filters.



Ideal Matched Filter Time Response

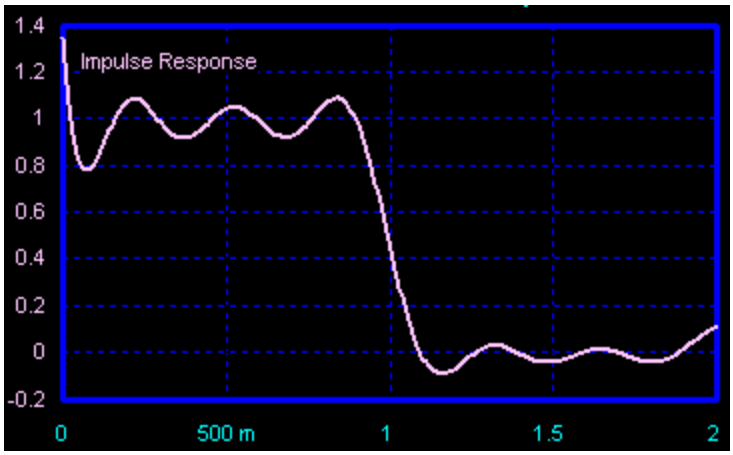
Continuous Approximations of Matched Filters

Since ideal continuous and IIR matched FilterSolutions are not realizable, they must be approximated. FilterSolutions uses an approximate solution that optimizes the time response of the filter with the constraint that the transfer function zeros remain on the JW axis. Specifically, the integration of the square of the error between the filter impulse response and the ideal impulse response (a square pulse) is minimized under the mentioned restraint conditions. The purpose of the JW zeros constraint is to allow the filter to be realized with passive elements.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Excessively large doubly terminated and most singly terminated matched filters may produce negative passive element values and, therefore, are still not realizable with passive elements.

An example of a seventh order with minimum impulse error is shown below.



Continuous Matched Filter Approximation

Impulse Error

As expected the impulse error generally goes down as the order of the filter increases, but not always. Large odd order filters are more efficient than smaller even order filters.

The table below shown the integration of the square of the impulse error from zero to five pulse widths of a matched filter with a one second rise time.

Order	Integration of the square of the impulse error
1	0.1932
2	0.1309
3	0.05975
4	0.05850

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5	0.03397
6	0.03626
7	0.02354
8	0.02609
9	0.01795
10	0.02029
11	0.01449
12	0.01629
13	0.01221
14	0.01368
15	0.01064
16	0.01166
17	0.009349
18	0.01021
19	0.008332
20	0.009057

Delay Filters

The Delay Filter simulates a transport delay frequency response. The frequency response magnitude of an ideal transport delay filter is unity for all frequencies, and the frequency response group delay is equal to the duration of the transport delay for all frequencies.

It is frequently necessary to account for transport delay in controls applications, and it is occasionally useful to delay a signal for timing purposes. FilterSolutions provides a Pade approximation of an ideal transport delay frequency response consisting of a series of all pass stages with a controlled group delay. The group delay is designed to be accurate for frequencies up to approximately (N-1) /(T*PI) Hz or 2*(N-1)/T Rad/Sec where N is the order of the filter, and T is the design transport delay

FilterSolutions allows one to define the Delay Filter by setting the delay time of the filter.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important: It is important to remember that the delay filter implemented by FilterSolutions optimizes the frequency response of the filter, not the time response.

Ripple Period

FilterSolutions allows one to fine tune the Delay Filter by changing the ripple period. The group delay of the filter is set to a forced value approximately every R/(T*2*PI) Hz or R/T Rad/Sec where R is the ripple period and T is the design transport delay. Increasing the ripple period has the effect of extending the upper frequency limit of the group delay at the cost of increasing the error of the group delay. The period of each individual ripple is modified slightly to obtain the equiripple effect of the group delay.

The allowable values of the ripple period are 0 to 2.8. Setting the ripple period to zero has the effect of producing a Bessel filter style maximally flat group delay.

Ripple Magnitude

Several precomputed group delay ripple magnitudes may be selected in lieu of selecting the ripple period. The advantage of using a precomputed filter is that the group delays are far more accurate equiripples.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5 - Using FilterSolutions

This chapter contains the following topics:

- The Control Panel
- Transfer Functions
- Graphical Displays
- Circuit Displays

The Control Panel

This chapter contains the following topics:

- Control Selections Box
- Filter Attributes
- Multi-Band Definitions Spread
- Custom Filters
- Adding Stop Band and Off-Axis Zeros
- Lumped Selections
- Distributed Selections
- Active Selections
- Switched Capacitor Selections
- Digital Selections
- Main Menu
- Standard Parts Lists
- Options Selections
- Entering Numbers

Control Panel Group Box Definitions

Graph Limits	Select the frequency and time graph limits for the graphical plots
Filter Type	Select one of seven filter types or custom filters. Not displayed for FIR selections.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Filter Class	Select one of four filter classes or Diplexers
Freq Scale	Select all frequency values to be entered in radians per second or Hertz. Default is radians per second

Filter Attributes Selections Definitions

Alpha	Displayed for Raised Cosine and Root Raised Cosine filters. Constant to determine the sharpness of cutoff of the filter. 0 is a perfect square cutoff, 1 is a full raised cosine or root raised cosine. Analog and IIR filters offer only discrete selections of Alpha.
Asymmetrical	Displayed for band pass and diplexer 1 filters only. Check to produce frequency asymmetrical band pass filters or diplexer legs of different order.
Add Tx Zeros	For Butterworth and Chebyshev I filters, this button is used to add user defined transmission zeros to the stop band, or dual or quadruplet zeros to alter the phase and group delay. In effect, one creates the stop band or delay equalization. Butterworth filters become "Maximally flat" filters when zeros are placed in the stop band. When zeros are installed, the caption of this button becomes "Have Stop Band Zeros", "Have Off-Axis Zeros", or :Have Mixed Zeros". This button is displayed for Butterworth and Chebyshev I filters types only.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Band Stop Width	Displayed for band pass/band stop diplexers only. Enter the desired width of the band stop leg of the diplexer.
Band Width 1/2	Displayed for band pass diplexers only. Enter the arithmetic band width of the lower and upper frequency band pass leg s respectively.
Center Freq	Enter the center frequency (geometric mean) for band pass and band stop filters. The text changes to "Pass Band Freq" for low pass and high pass filters. The text changes to "Lower Corner Frequency" when "Corner Freqs" is selected in the Pass Band Def box.
Center Freq 1/2	Displayed for band pass band pass diplexers 2 and triplexers 2 only. For band pass diplexers, enter the geometric mean of the two band pass legs. For triplexers, enter the geometric mean of the two frequency gaps.
Compute Src	"Compute Source", Displayed for even order asymmetrical Chebyshev I lumped and distributed filters only. Check to compute the source resistance that is as nearly equal as possible to the load. (Classic unmodified even order Chebyshev I filters may not have equal terminations.)
Constrict Ripple	Displayed for Chebyshev I, Chebyshev II, and Elliptic Filters.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Check to constrict the equiripple to a percentage of the pass band and/or stop band. This is useful for increasing the performance of the filter about the cutoff frequency.
Constrict Ripple % of Pass/Stop Band	Displayed for Chebyshev I, Chebyshev II, and Elliptic Filters. Enter the desired percentage of pass band and/or stop band to constrict the equiripple to. For Elliptic filters, select the pass band, stop band, or both to be equiripple constricted.
Data Transmission Filter 2	Displayed only when analog or IIR Raised Cosine filters are selected. Check to compute Data Transmission Filters, which are a derivative and modification of Raised Cosine Filters. Data Transmission filters eliminate only postcursor ISI
Define Filter	Depress this button to turn off the Multi-Band filters input definition spread sheet.
Delay Equalize	Displayed for low pass and band pass filters only. Check to equalize the group delay in the pass band of low pass or band pass filters
Diplexer Type	Displayed for Diplexer 1 and 2 selection only. For Diplexer 1, select between low/high

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	pass diplexers and band pass 1 or 2 diplexers. Band Pass 1 diplexers are defined by the inner and outer corner frequency band widths. Band Pass 2 diplexers are defined by the center frequency and bandwidths of the two band pass legs. The center frequency is the geometric mean of the respective corner frequencies. For Diplexer 2, select between band pass/band stop diplexers, Triplexer 1, or Triplexer 2. Triplexer 1 is defined by the inner and outer corner frequency band widths. Triplexer 2 is defined by the center frequency and bandwidths of the two band stop gaps. The center frequencies are the geometric mean of the gap corners.
Equalize To	Displayed for Delay Equalization and Passband unchecked only. Enter the attenuation of the passband which is desired to delay equalize out to.
Equalization Order	Displayed only when Delay Equalize is checked. Enter the order of the desired delay equalizer
Equiripple Delay	Displayed for Bessel and Delay Filters only. Check to convert the group delay to an equiripple delay as opposed to a maximally flat delay.
Even Order Mod	Check his box to modify Chebyshev I, Chebyshev II, Hourglass and Elliptic filter responses to allow these filters to be

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	constructed with RCL circuits that have equal or nearly equal source and load resistances. Not displayed for Gaussian, Bessel or Butterworth filters, for zero ohm source resistance, or for odd order filters. The option is also available for active and digital filters.
High Stop Atten	Displayed for frequency asymmetrical band filters with stop bands only. Enter the stop band ratio of the High pass leg.
High Stop Ratio	Displayed for frequency asymmetrical Elliptic only. Enter the stop band ratio of the high pass leg.
Implementation	Select the implementation to be Lumped, Distributed, Active, Switched Capacitor, or Digital
In Band	Displayed for Elliptic Single Point Ripple filters only. Select which band(s) to design single point ripple for.
Inner Band Width	Displayed for band pass diplexers only. Enter the desired width between the inner band edges of the two band pass legs.
Low Stop Atten	Displayed for frequency asymmetrical band filters with stop bands only. Enter the stop band ratio of the low pass leg.
Low Stop Ratio	Displayed for frequency asymmetrical

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Elliptic only. Enter the stop band ratio of the low pass leg.
Lower Corner Freq	Enter the lower corner frequency of a band pass or band stop filter when "Corner Freqs" is selected in the Pass Band Def box.
Mod Pass	Displayed for Elliptic and Hourglass filters when Even Order Mod is selected. Modify the pass band for even order filters.
Mod Stop	Displayed for Elliptic and Hourglass filters when Even Order Mod is selected. Modify the stop band for even order filters.
Multi-Band Def	Depress this button to display the Multi-Band filter input definitions spread sheet.
Nonfin	Displayed for Single Point Ripple filters only. Number of nonfinite zeros (zero or infinity) to be placed in the pass band and/or stop band of Single Point Ripple designed filters. Three nonfinite zeros has the effect of increasing attenuation away from the cutoff frequency.
Order	Enter the order of the filter. Band pass and band stop filter refer to the order of a side only. The text for this box becomes "Taps" when FIR filters are selected.
Order Hi	Displayed for asymmetrical band pass filters only. Enter the desired order of the high pass portion of the filter.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Order Lo	Displayed for asymmetrical band pass filters only. Enter the desired order of the low pass portion of the filter.
Outer Band Width	Displayed for band pass diplexers and triplexers only. Enter the desired width of the outer band edges for band pass diplexers, or the width of the low pass and high pass legs of the triplexer diplexer. The outer band width is the geometric mean of the outer corner frequencies.
Passband	Displayed for Delay Equalization only. Check to delay equalize only the passband of the filter.
Pass Band Atten	Enter the desired pass band attenuation in dB's. Displayed only when Standard Attenuation is not checked
Pass Band Def	Define the pass band with center frequency and bandwidth or by upper and lower corner frequencies. Displayed only for band pass and band stop filters.
Pass Band Freq	Enter the pass band frequency for low pass and high pass filters. The text changes to "Center Freq" or "Lower Corner Freq" for band pass and band stop filters

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Pass Band Return Loss (dB)	Enter the desired pass band return loss in dB for Chebyshev I and Elliptic filters. Note: Pass Band Ripple vs. Pass Band Return Loss is an option selectable in the Options Panel.
Pass Band Ripple (dB)	Enter the desired pass band ripple in dB for Chebyshev I and Elliptic filters. Note: Pass Band Ripple vs. Pass Band Return Loss is an option selectable in the Options Panel.
Pass Band Width	Enter the pass band width for band pass and band stop filters. The text changes to "Upper Corner Frequency" when "Corner Freqs" is selected in the Pass Band Def box.
Ripple Def.	Displayed for Bessel filters when Equiripple Delay is checked and for Delay Filters. Select desired group delay equiripple definition in %Magnitude or ripple period.
Ripple Period	Displayed for Bessel filters when Equiripple Delay is checked and for Delay Filters. Enter the period of the desired prototype equiripple. A value of zero yields a maximally flat Bessel type group delay. The entered period will be the average length of all group delay ripple cycles.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Root Raised Cosine	Displayed only when analog or IIR Raised Cosine filters are selected. Check to compute Root Raised Cosine filters.
Set Order	Calculates the minimum filter order required to achieve desired pass band and stop band attenuation and frequency requirements, or in the case of Delay filter, group delay requirements.
Single Point Ripple	Confines Chebyshev and Elliptic pass band and/or stop band ripple to a single point. Useful for improving element value spread and group delay. One or three zeros are left and zero/infinity, as determined by the "Nonfin" (nonfinite) selection box.
Standard Pass Band Atten	Sets the pass band attenuation to the standard value for the selected filter type.
Stop Band	Displayed for Bessel Filters only. Check to create a stop band for the Bessel filter.
Stop Band Atten	Enter the desired stop band attenuation in dB for Chebyshev II, Hourglass, and Elliptic filters when "Specify Attenuation" is checked in the Stop Band Def box. Not displayed for Bessel, Butterworth, or Chebyshev I filters.
Stop Band Def	Select the desired stop band definition for Chebyshev II, Hourglass, and Elliptic

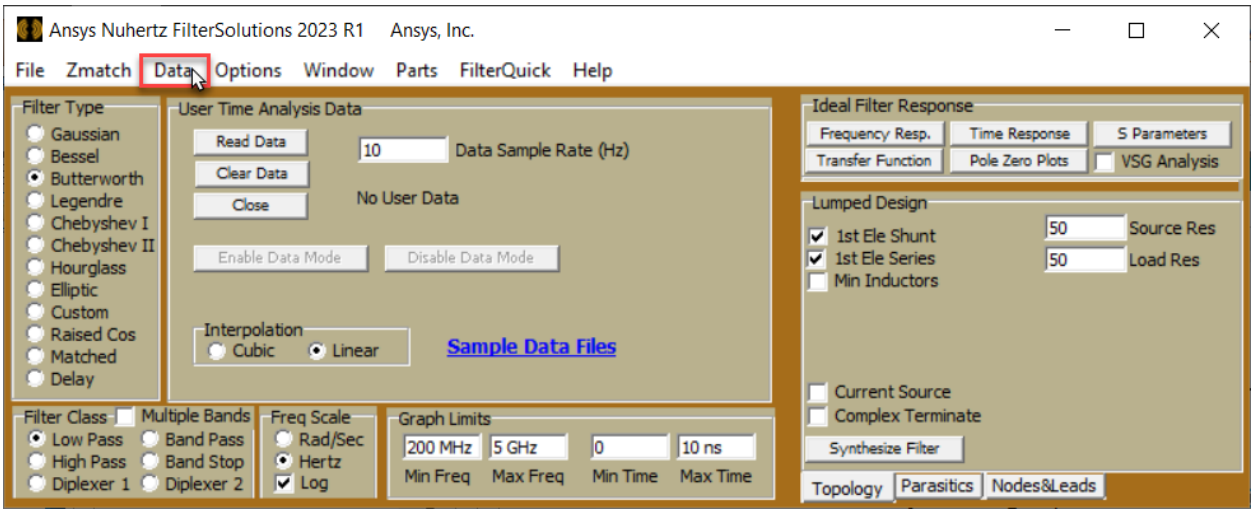
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	filters. Not displayed for Bessel, Butterworth, and Chebyshev I filters
Stop Band Freq	Enter the desired stop band frequency for low pass and high pass Chebyshev II, Hourglass, and Elliptic filters when "Specify Freq" is checked in the Stop Band Def box. Not displayed for Bessel, Butterworth, or Chebyshev I filters.
Stop Band Ratio	Referred to as Ws in some texts. Enter the desired stop band ratio for Chebyshev II, Hourglass, and Elliptic filters when "Specify" is checked in the Stop Band box. Not displayed for Bessel, Butterworth, or Chebyshev I filters. for Chebyshev II, Hourglass, and Elliptic filters when "Specify Ratio" is checked in the Stop Band Def box. Not displayed for Bessel, Butterworth or Chebyshev I filters.
Stop Band Width	Enter the desired stop band width for low pass and high pass Chebyshev II, Hourglass, and Elliptic filters when "Specify Freq" is checked in the Stop Band Def box. . Not displayed for Bessel, Butterworth, or Chebyshev I filters.
Stop Width 1/2	Displayed for triplexers only. Enter the stop band width of the lower and upper stop band gaps respectively. (Arithmetic difference between the two stop band corner frequencies.)
Taps	"Taps" replaces "Order" for FIR filters.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	The number of Taps rather than the order defines FIR filters. The number of taps is one greater than the number of states used to create the filter.
Transition	Displayed for Gaussian Filters only. Select a dB attenuation point to transition from Gaussian to a steeper roll off characteristic.
Upper Corner Frequency	Enter the upper corner frequency of a band pass or band stop filter when "Corner Freqs" is selected in the Pass Band Def box.

To access the **User Time Analysis Data** group box select **Data** in the ribbon menu. To return to the **Filter Attributes** select **Data** again.



The User Time Analysis Data group box options are defined as follows:

Read Data	Select to read in a data file.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Clear Data	Select to clear the user data.
Close	Close the User Data box
Enable Data Mode	After data is read in, data mode may be enabled or disabled at user discretion.
Disable Data Mode	After data is read in, data mode may be enabled or disabled at user discretion.
Interpolation	Select the desired interpolation, cubic or linear, for calculations between data points.
Data Sample Rate (Hz)	Enter the rate in which the user data was sampled.

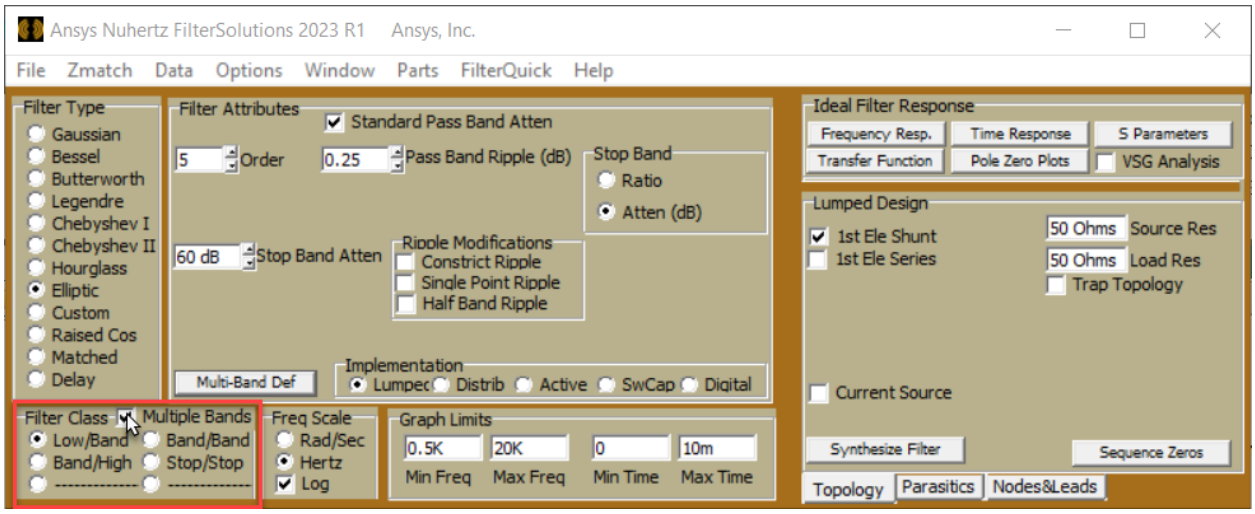
Multi-Band Definitions Spread Sheet

Available for lumped, active, switched capacitor, and digital filters.

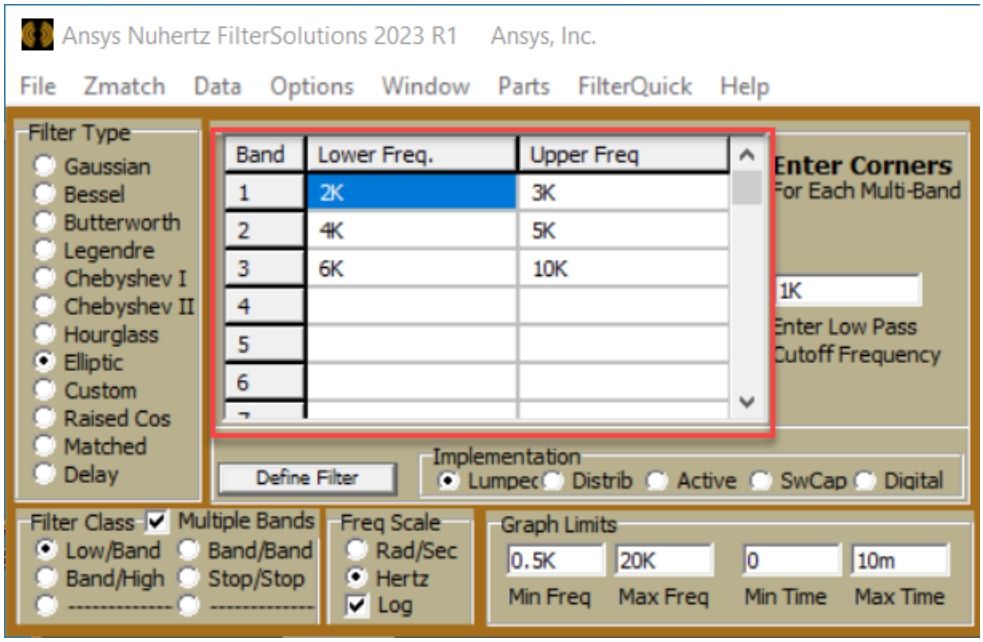
To open the Multi-Band filter input definitions spreadsheet:

1. Select **Multiple Bands** in the **Filter Class** group box. A two column spread sheet appears that allows the entry of the corner frequencies of each band pass section (or stop band section in the case of a Stop Band Multi-Band filter).

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

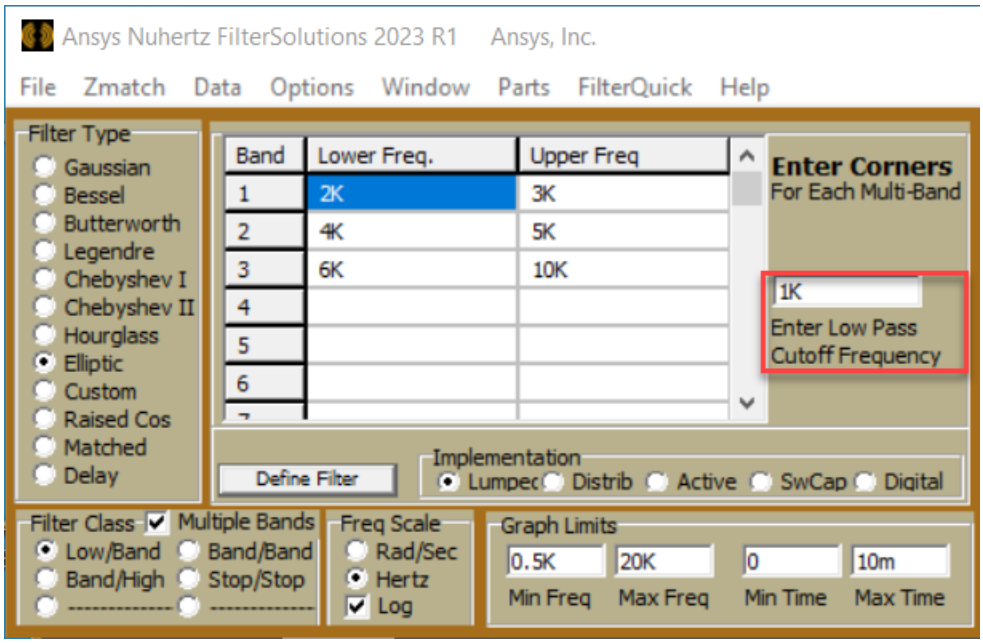


2. Enter Corners for each Multi-Band.



3. Enter the low pass or high pass cutoff frequency, if Low pass or High Pass Multi-Band filter is selected.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Select **Define Filter** to return to the **Filter Attributes** group box for more filter design parameter options.

Example:

To set up a 5th order Elliptic with 60dB stop band, enter all needed information into the main panel as shown in Figure 1.

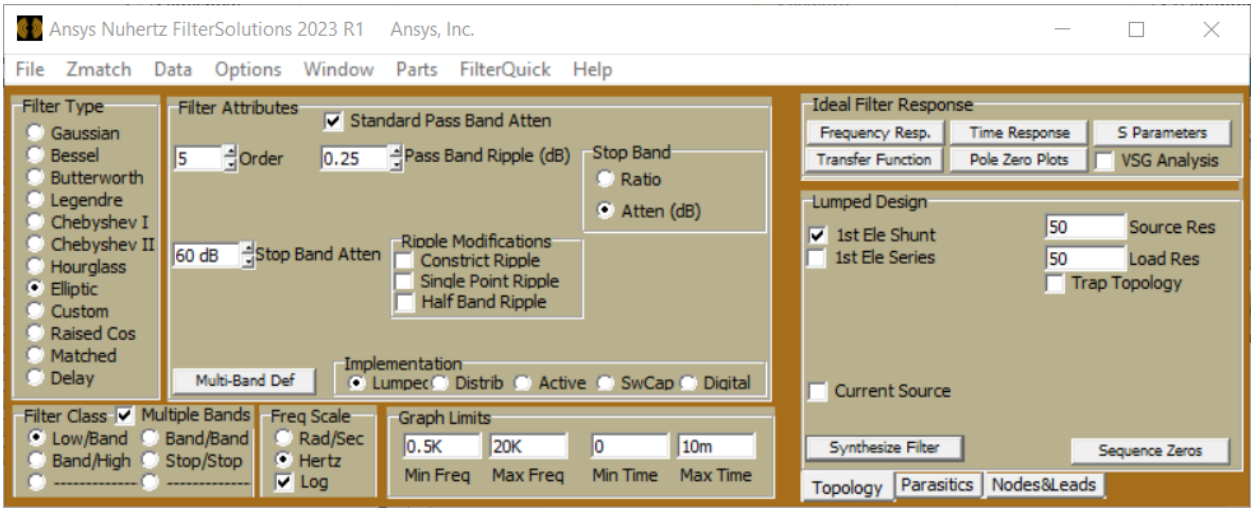


Figure 1: 5th Order Elliptic with 60dB stop band setup

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To set up the Multi-Band band pass sections and low pass cut-off frequency, select the **Multi-Band Def** button shown in Figure 1 above. The Multi-Band spread sheet will appear where the Filter Attributes used to be, as shown in Figure 2. Fill out the spread sheet with the desired corner frequencies of each band bass section. Enter the desired low pass cut-off frequency in the text box marked "Enter Low Pass Cutoff Frequency."

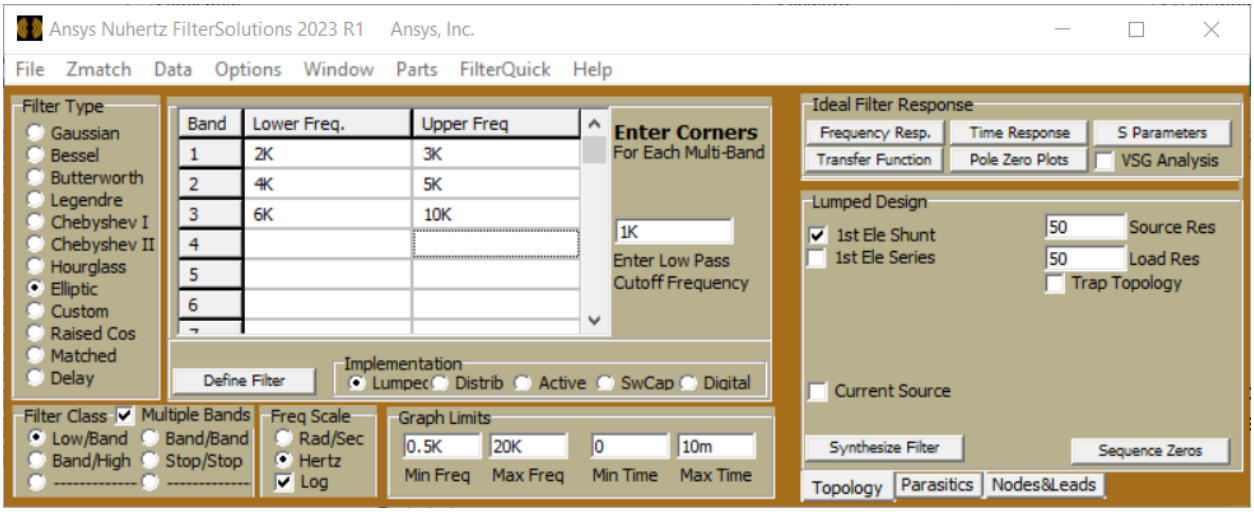


Figure 2: Low Pass Multi-Band Spread Sheet set up for three band passes and a low pass

When filled out correctly, the filter frequency response will look like Figure 3. Note that each corner frequency of the response matches exactly the values entered into the spread sheet in Figure 2 above.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

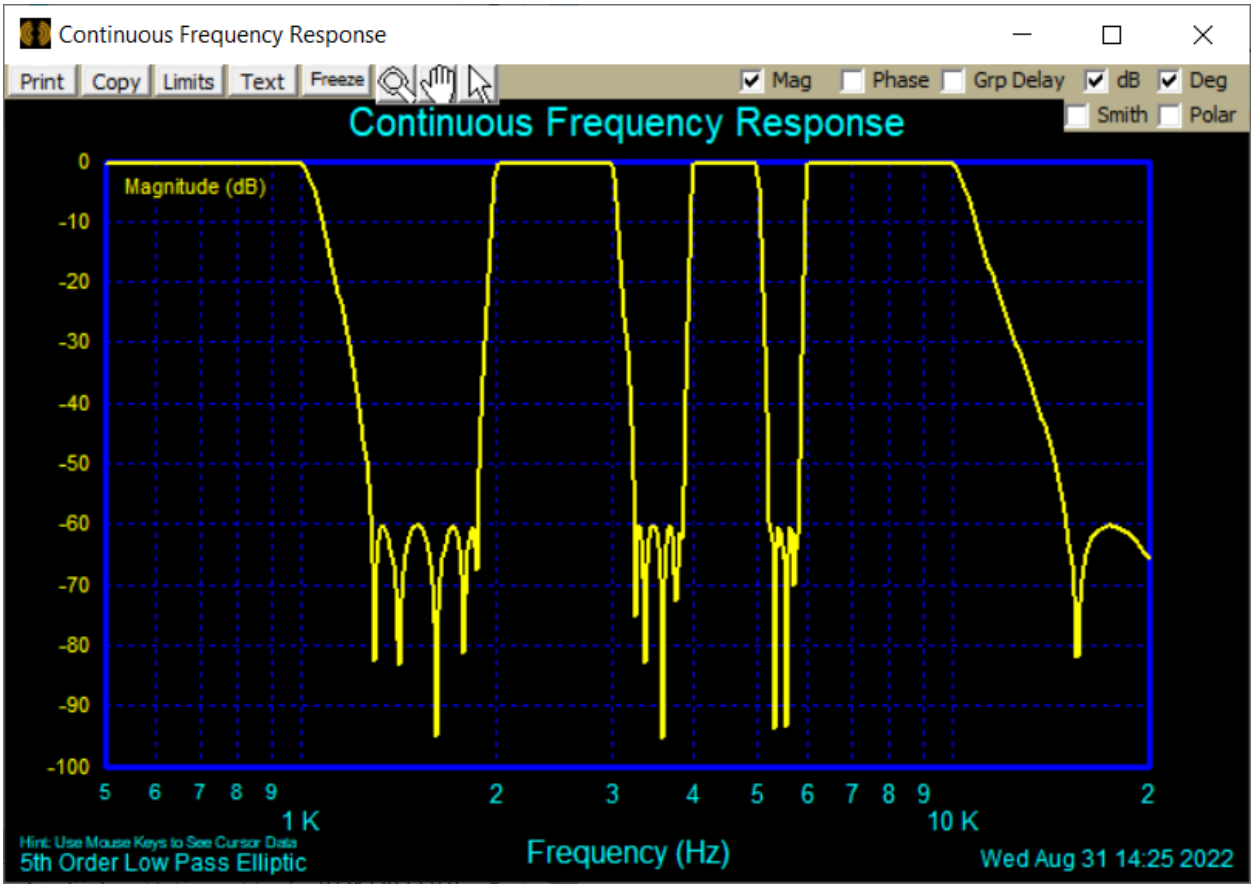


Figure 3: Frequency response of the Multi-Band Setup From Figure 2

Custom Filters

FilterSolutions provides the capability to create the filter based upon an existing Laplace Transform by selecting **Custom** in the Filter Type group box. Before selecting "Custom", have the implementation method already selected. If the Laplace Transform is a prototype, have the desired cutoff frequency and bandwidth already selected as well. Staged filters may be created by placing desired poles and zeros into the Custom Filter form entries.

Prototypes

A prototype transform is a low pass filter with a cutoff frequency of 1 radian per second. If **Proto S** is selected in the Custom Box, FilterSolutions will transform to the desired high pass, band pass or band stop filter, and provide frequency scaling prior to generating the passive or active filter. The user must enter pass band frequency, band width, and filter class.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Actual, S Transform

If **Act S** is selected along with **S Trans** in the lower left, FilterSolutions will set the magnitude of the filter, and generate a digital or analog circuit filter based upon exactly what was entered. It is important to note that not all transfer functions are realizable in passive circuit form, especially when the load and source resistors are equal. FilterSolutions will not generate band pass or band stop passive filters from actual Laplace transforms.

Actual, Mag Freq

If **Act S is** selected along with **Mag Freq** in the lower left, FilterSolutions will use an RMS error optimization algorithm to generate a digital or analog filter. Minimum and maximum transfer function stage Q may be entered as well as the number of transmission zeros and frequency limits for transmission zeros. Only low pass and high pass filters are permitted. FilterSolutions automatically detects low pass and high pass entries.

G Values

G values are the low pass, all pole prototype element values. These are used frequency as the starting point in microwave filter design. Select **Gvals** to enter a string of G values. G value strings must be terminated, that is, they must include the source resistor (Go) and the load resistance (G(n+1)). For equally terminated filters, Go and (G(n+1)) are both one (1.000).

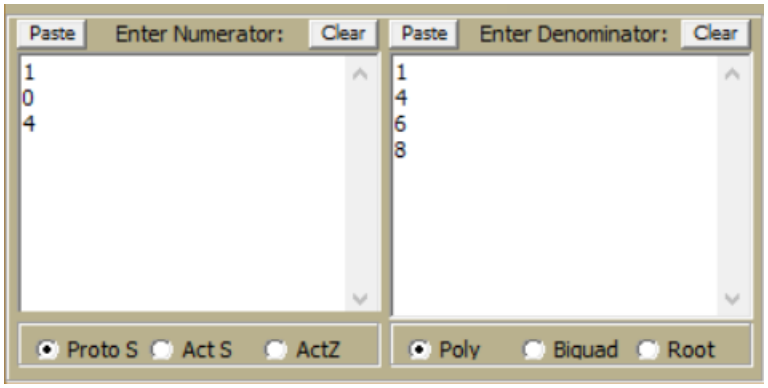
Digital

If **Act Z** is selected, FilterSolutions will assume that a Z transform has been entered and will output frequency and time traces and pole zero plots for the entered Z transform.

Formats

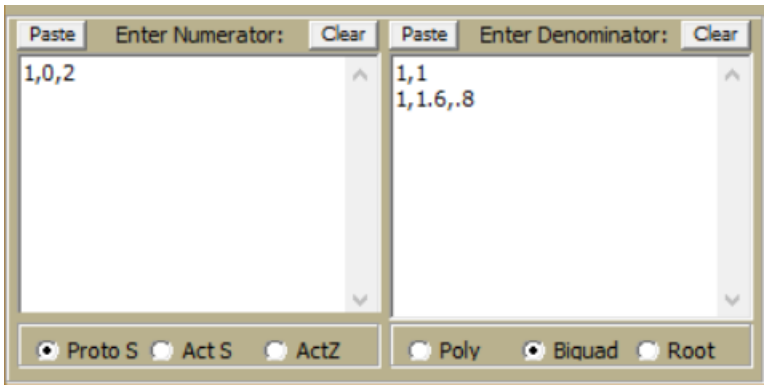
When entering transforms in polynomial form, enter the polynomial in descending powers.:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



$$\frac{S^2+4}{S^3+4S^2+6S+8}$$

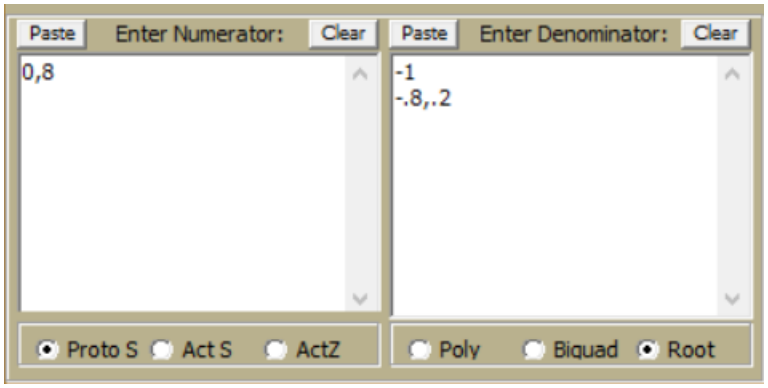
When entering biquads, use commas to separate biquad terms. Enter one biquad per line. The term "Biquad" as applied to this custom panel refers to any polynomial or order 2 or less.



$$\frac{S^2+2}{(S+1)(S^2+1.6S+0.8)}$$

When entering roots, use commas to separate real and imaginary part. Enter only one root per conjugate pair.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

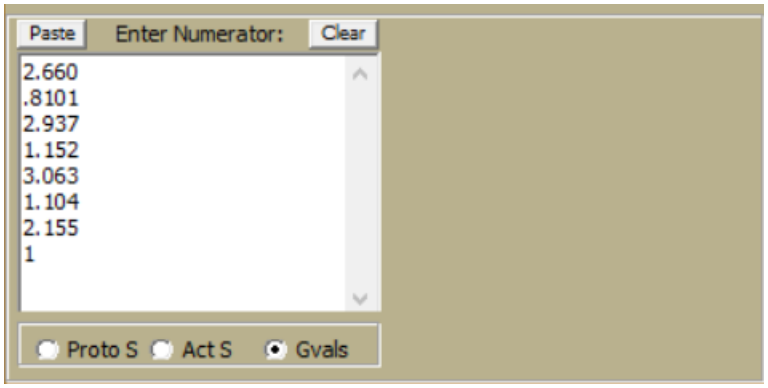


$$\frac{S^2 \pm J8}{(S+1)(S^2+0.8 \pm J0.2)}$$

Since roots do not contain any gain information, FilterSolutions will attempt to estimate the gain, but this estimated gain may not be what is expected by the user. If not satisfied with the gain from transfer functions generated from roots, it will be necessary for you to use the biquad or poly selection instead.

G Values

When entering G values, start with Go at the top, and G(n+1) At the bottom. Below are the entries for a 6th order Chebyshev with 1dB of passband ripple. Note that Go is the normalized source resistance of 2.660.



Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Adding Stop Band and Off-Axis Zeros

Butterworth and Chebyshev I filters have the option of adding stop band zeros, dual real zeros, and quadruplet zeros to the filters. Strictly speaking these filters are no longer true Butterworth and Chebyshev filters, but may be referred to as "Maximally Flat" and "Equirripple Passband" filters.

- Stop band zeros are entered with single numbers, and are placed on the imaginary axis.
- Dual real zeros use the suffix, "RE" or "re" to define them as real, and are placed equally spaced across the real axis.
- Quadruplet zeros and entered with two number separated with a comma, that is, "A, B" where A and B designate the real and imaginary parts. Quadruplet zeros are spaced equally across the real and imaginary axis.

Imaginary, or stop band, zeros are useful to increase the cutoff attenuation of a filter. Dual real and quadruplet zeros are useful for altering the phase and group delay, such as in a delay equalization process.

Select **Add Tx Zeros** in the main control panel to add the additional zeros to these filters. If the user owns a Passive or Transmission line license, the zeros may be placed at user specified positions by unchecking the **Auto Sequence** option. The Auto Sequence option is not displayed if a passive or transmission line license is not owned.

Select **Add Stop Band Zeros** to open the Add Zeros menu, shown in Figure 1. Select the **Auto Sequence** option for the zeros to be place sequenced automatically in passive and transmission line filters.

The form below is also in **Ratio** format. This mean that the stop band zero frequency is calculated as a ratio of the pass band frequency or pass band width. The advantage to using Ratio format is that you may switch between low pass, high pass, band pass, and band stop filters without changing the ratio. Band pass and band stop filters calculate the width between stop band zeros when in ratio format.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

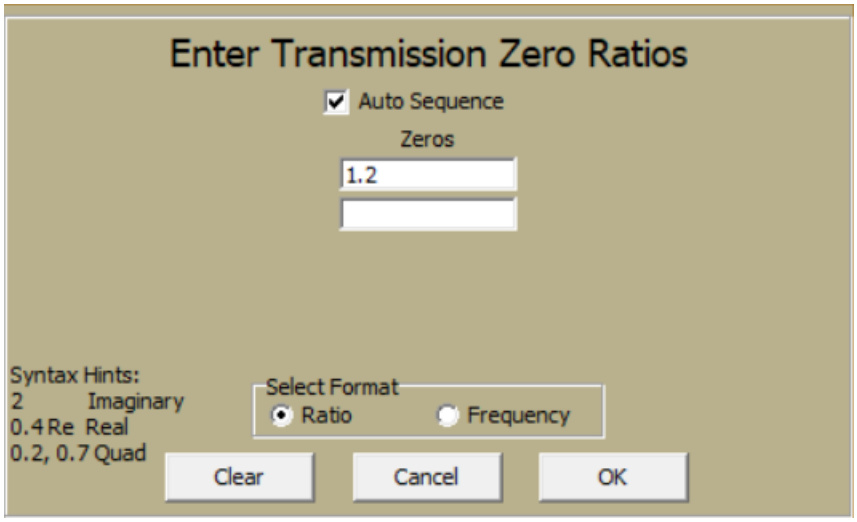


Figure 1: Auto Sequence Zeros in Ratio Form

Figure 2 below is in Manual Sequence mode. The **Position** column contains an integer specifying the element location in the filter to place the LC tank in that will generate the respective zero. Click **Default Sequence** to compute the default set of sequence integers.

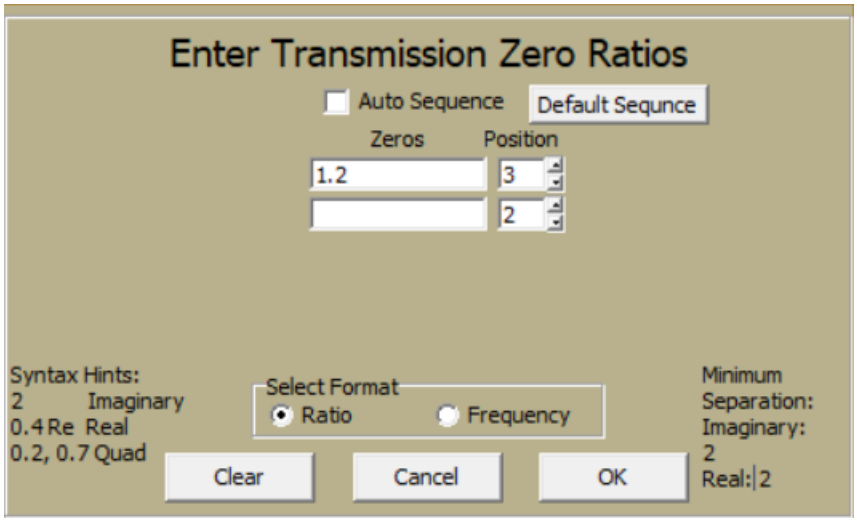


Figure 2: Manual Sequencing more

Figure 3 below is in **Frequency** mode. In Frequency mode, the exact frequency is specified of the desired stop band zero. Band pass and band stop filters specify the width between zeros.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

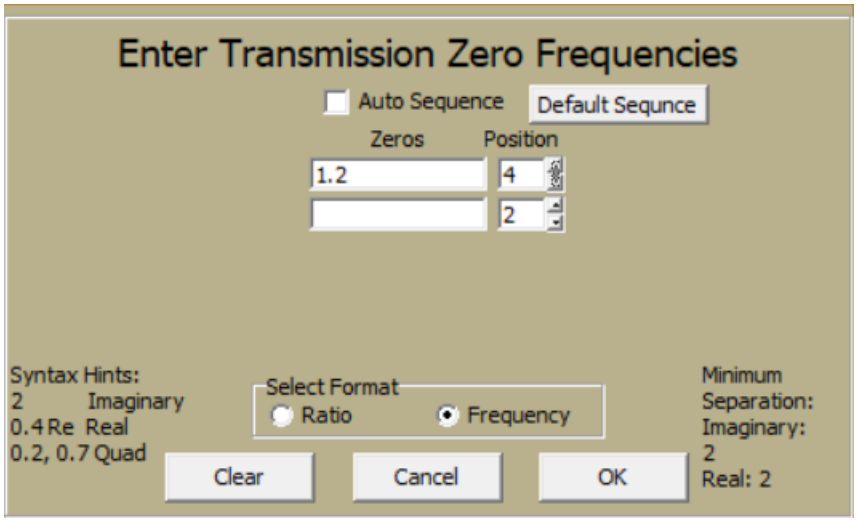


Figure 3: Frequency Mode

An example of entries for imaginary, real, and quadruplet zeros are shown below in Figure 4 in frequency mode:

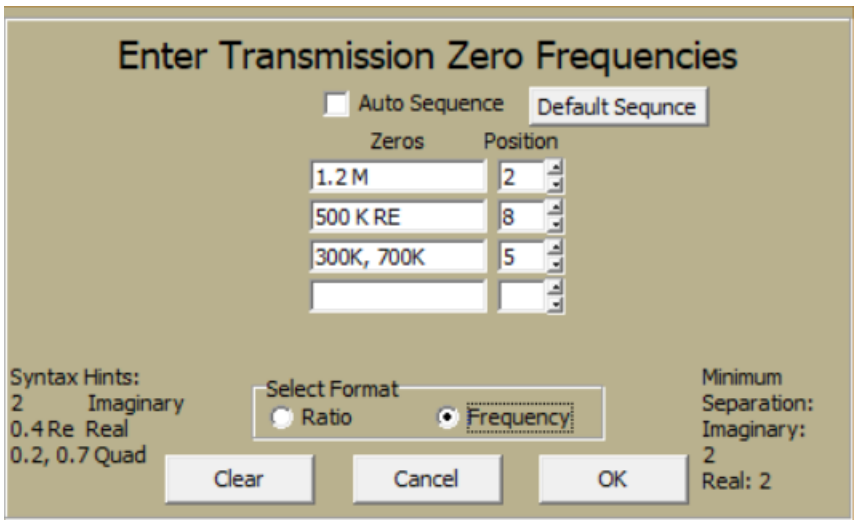


Figure 4: Imaginary, Real, and Quadruplet Zero Entries

Lumped Selections Definitions

Topology Tab

Ideal Parameters

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Frequency Response	Displays the ideal continuous frequency response of the filter that has been defined. For most equally terminated filters, the magnitude and phase will be the same as S12/S21 when VSG is left unchecked.
Pole Zero Plots	Displays ideal S21 (Forward Transfer) and S11 (Reflection Coefficient). S21*S21 + S11*S11 = 1 for ideal lossless conditions. Not displayed for zero ohm source or infinite ohm load resistance, or other cases when a reflection coefficient is known not to exist.
S Parameters	Displays the ideal reflection coefficient frequency response (S11) of the filter you have defined. Not displayed for zero ohm source resistance, or other cases when a reflection coefficient is known not to exist.
Time Response	Displays the ideal continuous time response of the filter that has been defined.
Transfer Function	Displays the Laplace Transform of the ideal filter that has been defined in standard, cascade, or parallel forms.
VSG Analysis	Displayed for doubly terminated filters only. Check to include the bias due to the source resistance to appear in the transfer function, frequency response, impedance response, and time response of the filter. For example, if the source resistance is equal to the load resistance, checking this box will cause a -6dB offset will appear in

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	the frequency response, and a factor of .5 will appear in the transfer function and time response. Uncheck the box to remove the gain offset due to the source resistance. Note: an equally terminated filter with VSG unchecked will display will produce S12/S21 in the frequency response magnitude.

Lumped Design

1sr Ele Shunt	Check to display filters whose first element after the generator is a shunt element.
1st Ele Series	Check to display filters whose first element after the generator is a series element.
Bridged T	Displayed for low pass and band pass stop band filters. Check to synthesize the filter stop band zeros in Bridged T form instead of LC tank dorm.
Cap Grounds	Displayed for band pass filters with stop bands. Check to force a shunt node capacitance at each node. Useful to absorb the physical parasitic node capacitance into a design without introducing parasitic frequency response degradation.

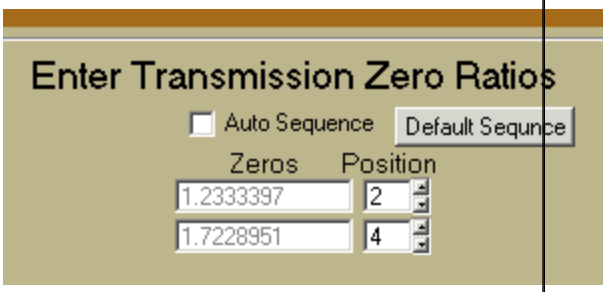
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Complex Terminations	Check to create filters with complex table driven terminations.
Define Load	Click to display the complex load definition table.
Define Source	Click to display the complex source definition table.
Equal Inductors	Displayed when it is possible to design a filter using only one inductor value. Check to design the filter with one inductor value. Note: If "Set Source Res" is selected, possibly two inductor values are required.
High / Low Pass	Displayed for band pass filters. Check synthesize band pass filters with cascaded low pass and high pass topologies.
Hi/Lo 1	Displayed in lieu of " High / Low Pass" with identical functionality when "HiLo 2" is displayed
Hi/Lo 2	Displayed for band pass filters with stop band filters of 10th orders or below. Computes a minimum inductor (or capacitor) high/low pass filter topology. Odd orders require N-1 inductors (or capacitor), even order require N, where N of the order of the filter.
LC on 2nd	Visible for singly terminated, even order stop

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Position	band filters with the stop band modified for even order (Even Order Mod and Mod Stop checked). Moves the first tx zero (LC tank) from the third element position. Useful in the high pass legs of diplexers.
Load Res	Enter the resistance of the load the filter is to drive. Enter "Inf" for open or very high resistive loads.
Match Impedance	Displayed for mismatched band pass termination impedances only. Check to automatically match the load and source impedance.
Minimum Cap/Ind	Displayed for Zigzags and Hi/Lo 2 Only. High / Low pass 2 filters are of a minimum inductor or capacitor topology. Select to choose element type count to minimize.
Min Inductors	Displayed for low pass and high pass filters only. When this is checked, FilterSolutions only displays the filters with the lowest inductor count of the all the filters that meet the other requirements.
Other Termination Impedance	Displayed in Zmatch mode when "Autofil Other Term" is checked. Enter the desired real impedance used to fill in the other impedance table when a single variable Touchstone file is used to fill in a Zmatch impedance table entry.
Q comp	Visible when load and source resistance is significantly different and load is finite.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Check to create circuits that automatically compensate for degraded performance due to finite component Q. The option is only available for unequal load and source resistances. Compensation works best for zero ohm source resistance, and is marginal for source to load ratios between 0.2 and 5.
Sequence Zeros	Displayed when more than one non-manual (Elliptic, etc.) stop band zeros (reflection poles)are present only. Selection produces a pop-up window to manually sequence zeros:  Enter the desired position for each zero. Zeros must be separated by at least two. The Default Sequence setting produces the arrangement that is most likely to minimize the element value spreads.
Set Source Res	Displayed for filters that generate a source resistance other than that specified, such as a Zigzag, but may have the source resistance set by adding capacitors or introducing other possibly undesirable effects, such as requiring two inductor values

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	instead of one. Check to force the source resistor to the entered value at the expense of one additional capacitor.
Source Res	Enter the resistance of the source that is driving the filter. Enter "0" for zero or very small resistive sources. If the source is a current source, a parallel zero source resistance is taken to be infinite.
Synthesize	Synthesizes and displays the RCL circuits containing the characteristics of the filter that has been defined.
Type I	Displayed for Zmatch only. Select alternating L's and C's, and alternating orientations of reactive elements to design matchi9ng networks.
Type II	Displayed for Zmatch only. Select LC series and parallel pairs to design lumped matching networks.
Zigzag	Displayed for Chebyshev II, Hourglass and Elliptic band pass filters only. Check to create a minimum inductor (or minimum capacitor) Zigzag filter. Odd orders require N-1 inductors (or capacitor), even order require N, where N of the order of the filter.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parasitics Tab

Def Cap Rp	Enter default capacitor parallel resistance parasitic
Def Cap Rs	Enter default capacitor series resistance parasitic. (ESR value)
Def Cap Q	Enter default capacitor Q parasitic
Def Cap Ls	Enter the default series inductance parasitic.
Def Ind Rp	Enter default inductor parallel resistance parasitic
Def Ind Rs	Enter default inductor series resistance parasitic. (DCR value)
Def Ind Q	Enter default capacitor Q parasitic
Def Ind Cp	Enter the default parallel capacitance parasitic.

Nodes & Leads Tab

Node Cap	Enter the expected shunt capacitance of the element type node pads. Each element will have one on each side.
Lead Ind	Enter the expected series inductance of the element leads. Each element will have one of these on each side.
Compensate	Check to make automatic adjustments to element values, when possible, to offset the effects of node capacitance and lead inductance.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Distributed Selections Definitions

Topology Tab

Ideal Parameters

Frequency Response	Displays the ideal continuous frequency response of the filter that has been defined. For most equally terminated filters, the magnitude and phase will be the same as S12/S21 when VSG is left unchecked.
Pole Zero Plots	Displays the ideal filter poles and zeros on a complex plane.
S Parameters	Displays ideal S21 (Forward Transfer) and S11 (Reflection Coefficient). S21*S21 + S11*S11 = 1 for ideal lossless conditions. Not displayed for zero ohm source or infinite ohm load resistance, or other cases when a reflection coefficient is known not to exist.
Time Response	Displays the ideal continuous time response of the filter that has been defined.
Transfer Function	Displays the Laplace Transform of the ideal filter that has been defined in standard, cascade, or parallel forms.
VSG Analysis	Displayed for doubly terminated filters only. Check to include the bias due to the source resistance to appear in the transfer function, frequency response, impedance response, and time response of the filter.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	For example, if the source resistance is equal to the load resistance, checking this box will cause a -6dB offset will appear in the frequency response, and a factor of .5 will appear in the transfer function and time response. Uncheck the box to remove the gain offset due to the source resistance. Note: an equally terminated filter with VSG unchecked will display will produce S12/S21 in the frequency response magnitude.

Distributed Design

1/4 Len Freq	Displayed whenever Set 1/4 Len Freq is checked. Enter the desired quarter length frequency. , That is, the frequency at which open becomes grounds and grounds becomes opens. Also the frequency which low pass filters achieve their first notch.
1/2 Len Freq	Displayed whenever Set 1/2 Len Freq is checked. Enter the desired half length frequency. That is, the frequency at which open becomes grounds and grounds becomes opens. Also the frequency which low pass filters achieve their first notch.

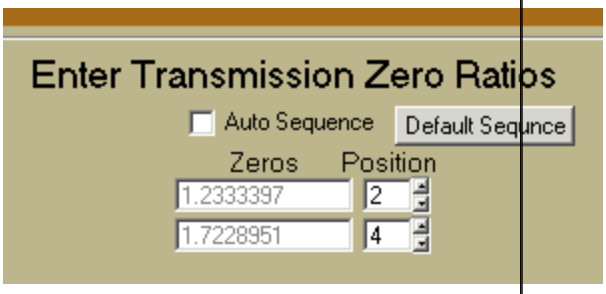
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1st Ele Shunt	Check to display filters whose first stub after the generator is a shunt stub.
1st Ele Series	Check to display filters whose first stub (or segment) after the generator is a stub element.
Combine Stubs	Check to combine resonators into a single stub. Uncheck to use two separate stubs for resonators. Single stub resonators have significant aliasing.
Enable Extensions	Displayed for tapped and untapped Interdigital (no wideband) only. Enables the extension of all the internal open ends to potentially improve optimization results.
Equal Width Approximation	Displayed for interdigital, combline, and band stop notch resonators. Interdigital and Combline: Forces all resonator widths to the same value. (Pinned interdigitals do not set the outer hairpin widths.) Band stop: Forces the center resonator widths to the same value. Improves the accuracy of EM port tuning, and is useful as a convenience, but may also introduce small errors.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Equal Width Conductors	Displayed for Parallel Edge Coupled with substrates only. Check to synezesis all conductors to be the same width.
Extension Length	Displayed for Hairpins only. Useful in tuning for S11. Enter the desired length, if any, of the outer hairpin extensions. This will be only the outer legs for Tuning Type 1, and both legs of the outer hairpins for Tuning Type 2.
Load Res	Enter the resistance of the load the filter is to drive. Enter "Inf" for open or very high resistive loads.
Mitered Corners	Displayed for resonator filter selections. Check to apply 50% miter to the corners. Useful to minimize narrow gaps in wide band filters.
Open Stub Ground	Displayed for band stop notch resonators only. Replaces the resonator grounds with open en 1/4 length stubs to simulate a ground at the notch frequency.
Overlay Capacitance	Displayed for suspended substrates only. Check to use broadside coupled lines to simulate series capacitance in high pass and band pass filters.. This removes the need for a lumped capacitor, and is more accurate than using coupled resonators.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Quick Optimize	Check to apply quick optimization to the filter design. Takes much more computer time, but attempts to provide a more accurate solution than raw synthesis alone.
Rotate Resonators	Displayed for Parallel Edge Coupled with substrates only. Check to rotate the resonators as needed to obtain a net horizontal, or other specified angle, layout. The default angular entry of "0"specifies a net horizontal layout. Resonator rotation is automatically applied as needed to meet net angle user input.
Select ... Topology	Select the desired filter topology. The selection will change for low pass, high pass, band pass, and band stop selections.
Sequence Zeros	Displayed when more than one non-manual (Elliptic, etc.) stop band zeros (reflection poles)are present only. Selection produces a pop-up window to manually sequence zeros:  Enter the desired position for each zero.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Zeros must be separated by at least two. The Default Sequence setting produces the arrangement that is most likely to minimize the element value spreads.
Set 1/4 Len Freq	Displayed for low pass and high pass filter selection when the option exists. Check to manually set the quarter wave length frequency, that is, the frequency at which open becomes grounds and grounds becomes opens. Also the frequency which low pass filters achieve their first notch. Uncheck to automatically use twice the cutoff or center frequency, or the value entered in the options table. Not displayed for Spaced Stubs or other filter types that fix the quarter length frequency by design.
Set 1/2 Len Freq	Displayed for band pass and band stop filter selection when the option exists. Check to manually set the half wave length frequency, that is, the frequency at which open becomes again, and grounds becomes grounds again. Also the frequency which band pass filters achieve their first notch. Uncheck to automatically use twice the cutoff or center frequency, or the value entered in the options table. Not displayed for resonator filters or other filter types that fix the quarter length frequency by design.
Set Line Width	Displayed for Hairpins only. Check to force a desired width of the hairpin lines or the

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Set Gap Width	width between the hairpin legs. Enter the desired value in the appropriate text box that appears when these items are checked.
Stub Taps	Displayed for Equal Width Parallel Edge Coupled only. Check to display the layout end segments as stubs. sometimes useful to obtain more desirable geometry.
Synthesize	synthesizes and displays the transmission line filters containing the characteristics of the filter you have defined.
Tuning Type 1	Displayed for Hairpins only. Sets both legs of the outer hairpins to adjustable extensions. This is useful in tuning for S11. This is useful in tuning for S11. Set the length of the extensions in the "Extension Length" text box.
Tuning Type 2	Displayed for Hairpins only. Sets only the outer legs of the outer hairpins to adjustable extensions. This is useful in tuning for S11. This is useful in tuning for S11. Set the length of the extensions in the "Extension Length" text box.
Use Coupled Lines	Check to use coupled resonators to simulate series capacitance in high pass and band pass filters.. This removes the need for a lumped capacitor, but tends to corrupt the frequency response, and may require significant optimization.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Use Series Caps	Check to use series lumped capacitors in high pass and band pass filters. The alternative is to use transmission line segments, which cause low frequencies to pass through. Displayed for band pass only. High pass selection is available in the drop down menu.
Via Ends	Displayed for Parallel Edge Coupled and Hairpins only. Not recommended. However, design flexibility may sometimes be increased by using the Via end solution as opposed to the more standard open end solution.

Substrate Tab

Substrate Type	Select between RLGC, stripline, microstrip, and suspended substrate. Note that RLGC is not necessarily a substrate, and the detailed substrate definitions go away.
Cover	Displayed for microstrip with and without suspended substrate. Check to place a cover above the conductor.
Unbalanced	Displayed for stripline only. Check to synthesize unbalanced striplins.
Substrate Parameters	Enter the appropriate design values. Loss Tangent is tied to the dielectric selection whenever "Er Selection Losses" are checked.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Geometry Tab

Adjust Length on Limit	Check to adjust the length of all limited stubs and segments in such a manner so as to maintain the desired lumped element impedance. In general, this increases the accuracy of the filter. Segments and stubs that do not simulate single lumped elements are not affected.
Alternate Stub Orientations	Check to alternate the vertical orientations of the stubs between up and down. This useful in minimizing EM interference between adjacent stubs, but also takes up more physical space.
Apply Limits	Check to apply the geometry limits entered in the text above the check box.
Fixed Width to Substrate Height Ratios	Check the appropriate box to synthesize all stubs and segments used to simulate the respective lumped element to the same width. Enter the desired width of the lines in units of substrate height ratios. The advantage is that the distributed planar filter is more robust with respect to geometry, and the frequency response about the cutoff frequencies is more accurate. The disadvantage is that the quarter length frequency is dispersed, which in the low pass case causes a spread of the notch following the cutoff frequency.
Max Gap	Enter the desired maximum gap width between conductors.
Max Width	Enter the desired maximum conductor width
Min Gap	Enter the desired minimum gap width between conductors.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Min Width	Enter the desired minimum conductor width
Split Height Radio	Displayed when Split Wide Stubs is selected, and radial and delta stabs are not selected. Enter the ratio between stub width and substrate height that triggers the splitting of the stubs. Al stubs wider than this ratio will be split.
Split Wide Stubs	Check to split wide stubs into an upper and lower stub. This has the effect of reducing stub thickness, but also takes more board space.

Radial Tab

Alternate Stub Orientations	Check to alternate the vertical orientations of the radial and delta stubs between up and down. This useful in minimizing EM interference between adjacent stubs, but also takes up more physical space.
Apply Limits	Check to apply the angular geometry limits entered in the text above the check box.
Deg	Displayed when Fixed Angle is selected Enter the desired stub angle fix all the stub angles to.
Delta Stubs	Use a delta (triangle) geometry to synthesize stubs.
Fixed Angle	Displayed when radial or delta stubs are selected. Check to synthesize all stubs to the same angle. Causes a dispersion of the quarter length frequency.
Max Angle	Enter the desired maximum gap width between conductors.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Max Angle	Enter the desired maximum conductor width
Offset From Feed line	Check to offset the radial or delta stub from the feed line. Useful for ease of manufacturing.
Offset Height Ratio	Displayed when Offset From Feed line is displayed. Enter the offset value in units of substrate heights.
Radial Stubs	Use a radial geometry to synthesize stubs
Split Angle	Displayed when Split Wide Stubs is selected, and radial or delta stabs are selected. Enter the angle in degrees that triggers the splitting of the stubs. Al stubs wider than this angle will be split.
Split Wide Angle Stubs	Check to split wide stubs into an upper and lower stub. This has the effect of reducing stub thickness, but also takes more board space.

Parasitics Tab

Def Cap Rp	Enter default capacitor parallel resistance parasitic
Def Cap Rs	Enter default capacitor series resistance parasitic. (ESR value)
Def Cap Q	Enter default capacitor Q parasitic
Def Ind Rp	Enter default inductor parallel resistance parasitic
Def Ind Rs	Enter default inductor series resistance parasitic. (DCR value)
Def Ind Q	Enter default capacitor Q parasitic

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Default Minimum Segment Lengths	Enter default minimum segment lengths to be inserted between adjacent stubs
Line Go	Enter the default dielectric conductance per unit length in RGCL tx lines
Line Ro	Enter the default conductor resistance per unit length in RGCL tx lines

Active Selections Definitions

Ideal Parameters

Frequency Response	Displays the ideal continuous frequency response of the filter that has been defined.
Pole Zero Plots	Displays the ideal filter poles and zeros on a complex plane.
Time Response	Displays the ideal continuous time response of the filter that has been defined.
Transfer Function	Displays the Laplace Transform of the ideal filter that has been defined in standard, cascade, or parallel forms.

Active Design

Absorb 1st Orders	Check to absorb isolated single poles into adjacent second order biquads. Only displayed when the filter is capable of absorbing isolated poles.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Absorb 2nd Orders	Check to absorb all pole 2nd order biquads into adjacent all pole 2nd order biquads to create 4th order stages. Only displayed under circumstances when the filter is capable of creating 4th order all pole stages, namely Positive and Negative SA's.
Active Implementation	Select one of the eight implementations for the active filter circuit.
Alternate Notch	Displayed for negative SAB's with notch stages. A simpler design with a lower quality notch is substituted.
Alternate Topology	Displayed for Leap Frog filters only. Check to synthesize a topology that flips the sign on each stage of the Leap Frog. Has the effect of changing the opamp count of any transmission zero stages. Note that odd order filters have a net sign gain reversal.
C in	Enter the estimated input capacitance of the opamps.
Circuits	Displays the active filter circuit of the type selected in the Active Implementation selection box.
Enhancement Ratio	Enter the desired default Positive Enhancement Ratio for negative SAB's. Only displayed when Negative SAB or Parallel implementations are selected and when Pos Enhance is checked.

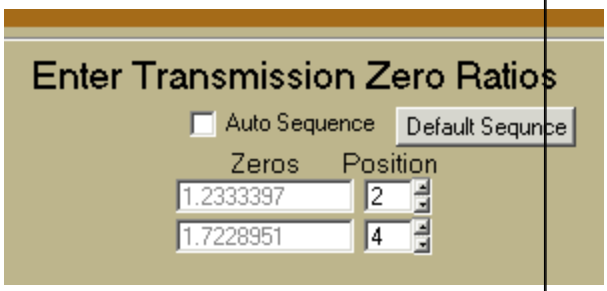
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Equal Caps Gain	Displayed for positive SAB's low pass selections for non-stop band filters. The gain is set such that the two capacitors are of equal value.
Equal Resis Gain	Displayed for positive SAB's high pass selections for non-stop band filters. The gain is set such that the two input resistors are of equal value.
Final Gain Stage	Check if you desire an additional gain stage to set the filter to the gain value you entered. FilterSolutions will only add this gain stage if it cannot set the gain any other way.
Gain	Enter the desired gain of the filter. FilterSolutions will attempt to set the filter with this gain. If it cannot do so, you may check Final Gain Stage for FilterSolutions to add an additional gain stage needed to set the final gain of the filter.
Gain*BW	Enter the estimated Gain Bandwidth product of the opamps.
High/Low Stages	Displayed for band bass filters only. Create band pass filters with high pass and low pass sections whenever possible. This switch is hidden when Alternate Notch and Positive Feedback Enhancement is

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	selected, but it is still functional.
Ideal Opamps	Check if you wish the active filter analysis to be performed with ideal opamps. This greatly speeds up the simulations.
Ideal Resistors	Check if the resistors are ideal, or you do not wish to consider shunt parasitic capacitance of integrated resistors in the simulations.
Parameters	Select which parameters, circuits or opamps, you would like to display and modify.
Pos Enhance	Check to add positive feedback enhancement to negative SAB's. Only displayed when Negative SAB's or Parallel implantations are selected.
R Constant	Enter the default resistive constant to be used in the active filters.
R in	Enter the estimated input resistance of the opamps.
R Out	Enter the estimated output resistance of the opamps.
Resistor Default Shunt C	Displayed when "Ideal Resistors is unchecked. Enter the default node parasitic capacitance of integrated resistors to be used in active simulations.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sequence Zeros	Displayed when more than one non-manual (Elliptic, etc.) stop band zeros (reflection poles)are present only. Selection produces a pop-up window to manually sequence zeros:  Enter the desired position for each zero. Zeros must be separated by at least two. The Default Sequence setting produces the arrangement that is most likely to minimize the element value spreads.
Synthesize	Synthesizes and displays the active filter circuit of the type selected in the Active Implementation selection box.
Slew Rate	Enter the estimated slew rate of the opamps in V/S.
T Caps	Displayed when negative SAB all pass, even notch, or band pass stages may be created. These stages will use capacitors instead of resistors in the T section of the stage.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TwinT on Even Notch	Displayed for positive SAB's with notch stages. Check to create TwinT Notch stages for even gain stages. Otherwise an injector resistor will be used.

Switched Capacitor Selections Definitions

Ideal Parameters

Frequency Response	Displays the ideal discrete frequency response of the filter that has been defined.
Pole Zero Plots	Displays the discrete filter poles and zeros on a complex plane.
Time Response	Displays the ideal discrete time response of the filter that has been defined.
Transfer Function	Displays the Z Transform of the filter that has been defined in standard, cascade, or parallel forms.

Switched Capacitor Design

Digital Implementation	Select the desired digital implementation for the switched capacitor filter.
Final Gain Stage	Check if you desire an additional gain stage to set the filter to the gain value you entered. FilterSolutions will only add this gain stage if it cannot set the gain any

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	other way.
FIR	Check to create an FIR (Finite Impulse Response) switched capacitor filter.
FIR Type	Displayed for FIR filters only. Select the desired FIR type.
Gain	Enter the desired overall gain of the filter.
High/Low Stages	Displayed for band bass filters only. Create band pass filters with high pass and low pass sections whenever possible. This switch is hidden when Alternate Notch and Positive Feedback Enhancement is selected, but it is still functional.
Min Cap Spread	Check to automatically select the digital IIR implementation that produces the minimum capacitance spread.
Min Cap Val	Enter the desired minimum capacitance value for filter capacitors (switched and unswitched)
Prewarp	Displayed for Bilinear implementations only. Check to apply prewarping to the

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	bilinear transformation.
Synthesize	Synthesizes and displays the switched capacitor filter circuit of the type selected in the Active Implementation selection box.
Switch Rate	Selected the clock frequency for the switched capacitors in the filter.
Switched Capacitor Implementation	Select an implementation for the active filter circuit.

Digital Selections Definitions

Ideal Parameters

Frequency Response	Displays the ideal discrete frequency response of the filter that has been defined.
Pole Zero Plots	Displays the discrete filter poles and zeros on a complex plane.
Time Response	Displays the ideal discrete time response of the filter that has been defined.
Transfer Function	Displays the Z Transform of the filter that has been defined in standard, cascade, or parallel forms.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Digital Design

Correct for Sinx/x	Check to automatically correct Sin(X)/X sampling error.
Digital Transformation	Select between the Bilinear, Matched Z, Impulse Invariant, or Step Invariant method to digitize the continuous filter. Displayed for IIR filters only.
FIR Type	Select the desired type of FIR filters. Displayed for FIR filter selections only.
IIR/FIR	Select IIR or FIR filter generation
Modified Invariant	Produces a variation of the impulse and step invariant transformations. Some time based errors are introduced, but the Z Transform result may be useful to some applications, such as switched capacitor filters.
Optimization Option	Displayed for FIR high pass, band pass, and band stop filters. Select the desired optimization option to synthesis the filter.
Prewarp	Check this to correct for error due to warping of the frequency response when a Bilinear transformation is performed. Only displayed when "Bilinear" is selected in the Digital Transformation box.
Sample Rate	Select the rate you wish to sample the filter at.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sinx/x Order	Visible only when "Correct for Sinx/x" is selected. Select the desired order of the Sinx/x compensation.

If FIR Design is selected, the Filter Attributes contain the following entry selections for the indicated FIR filter:

Alpha	Roll off Constant for Raised Cosine and Root Raised Cosine filters
Kaiser	Kaiser Constant Changes the shape of the Kaiser Window. The Kaiser Constant is usually set between 1 and 12.
Dolph-Cheb	Stop Band Attenuation The Dolph-Cheb FIR filter allows limited control of stop band attenuation.

Main Menu Selections Definitions

File	Open and save designed filters. The default suffix for saved filter files is FTR.
Zmatch	Displays the Zmatch Control Panel.
Data	Read in a textual data file containing user data

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	to be filtered and analyzed in the time domain.
License	Trial licenses display "Order" on this button. Enter the name and password license information. Displays the license agreement
Order	Permanent licenses display "License" on this button. Enter the name and password license information to activate the license. Displays the license agreement.
Options	Opens and closes the options panel Opens and closes the Initialization panel used to set custom initial setting upon program launch. Opens and closes the Color selection panel used to select program colors.
Window	Rearrange non-minimized windows into tile and cascade layouts. Minimizes and normalizes all windows. Selects between Floating Windows (windows floating over the desktop) and Embedded Windows (windows embedded in an MDI backplane)

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parts	Select which standard parts list to edit for use in updating filter element values to the nearest standard part.
Help	Displays the Help dialog Displays the About program version information.

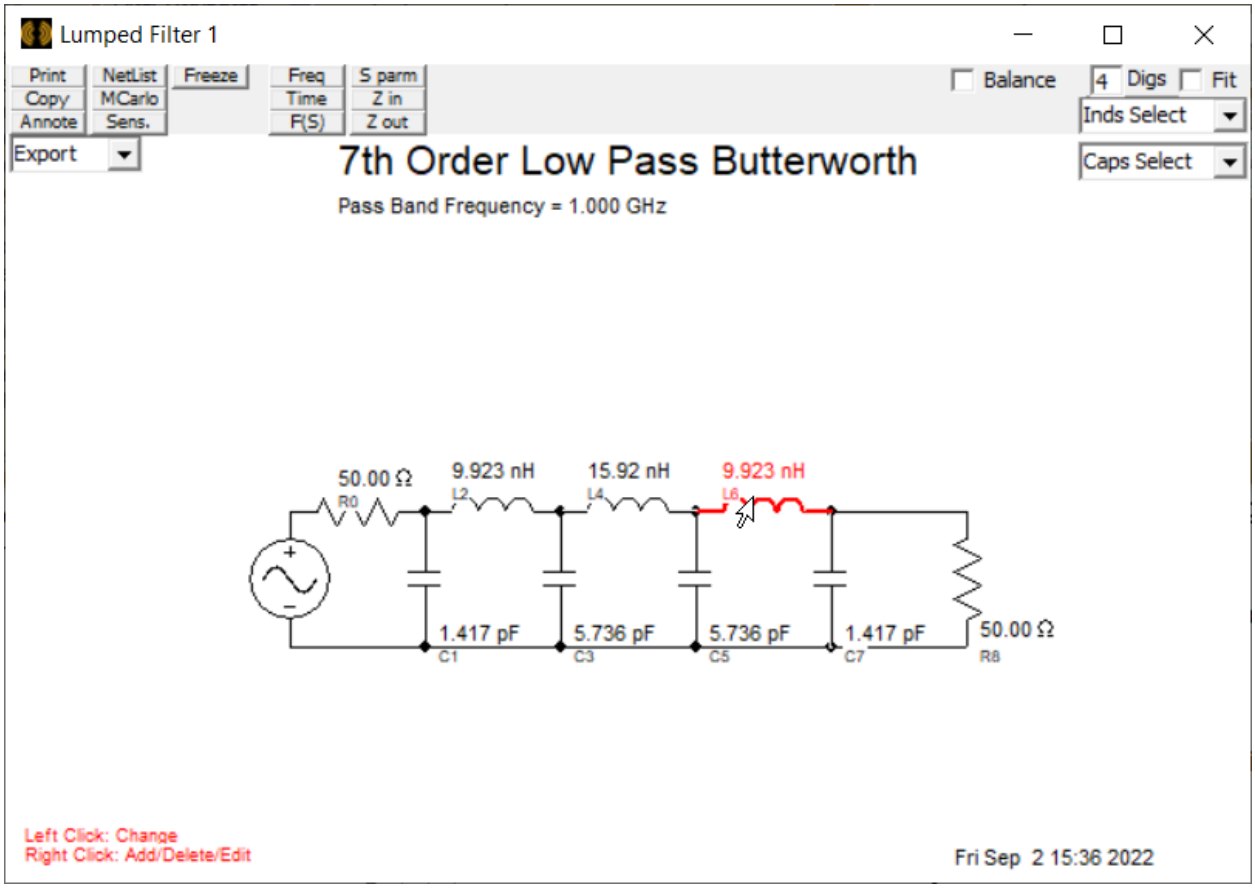
Standard Parts

FilterSolutions provides a flexible means to update the lumped or active filter elements values with the nearest value from the available standard parts, or the nearest 1%, 5%, and 10% standard part value. Up to three standard parts databases are maintained by **FilterSolutions**. If you need more than three, they may easily be maintained in another text or word processing document and copied or pasted in and out of FilterSolutions.

To update filter parts to the nearest value in the parts list:

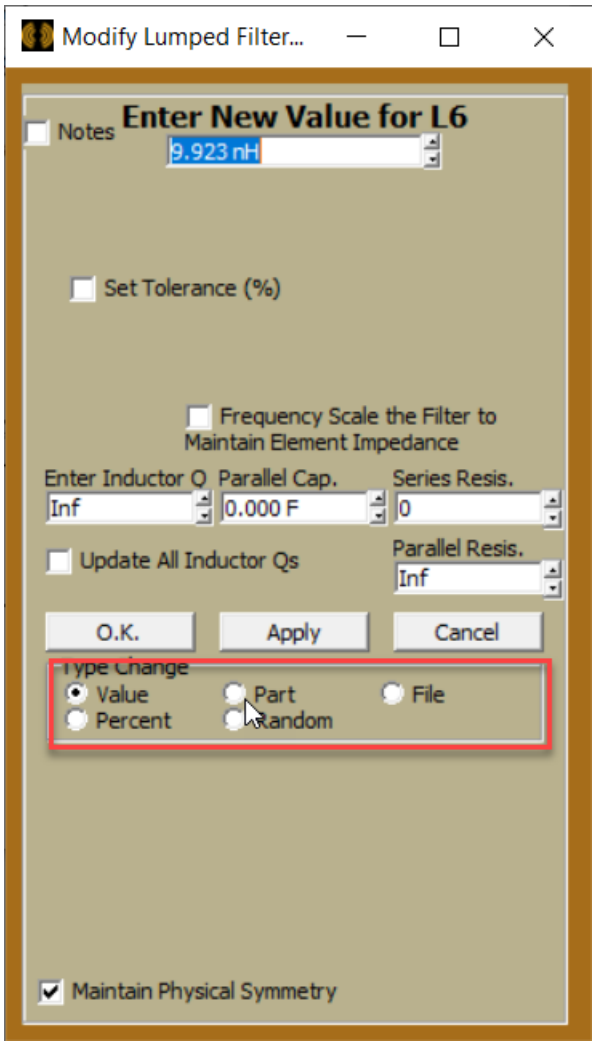
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select an element in the filter display.



2. Select **Parts** in the **Type Change** group box.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Select **OK**

Standard part updates are also available in the upper right corner of passive and active schematics.

Passive and active filters may have individual element or all like elements set to the nearest standard value from the selected database. The checkbox at the bottom of the Change control panel determines if one or all elements are updated.

You may select the default element value or the currently assigned element value as the base for the nearest part. A selection box in the change control panel determines the choice.

The upper part of the FilterSolutions standard parts window is shown below with some sample standard parts.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting the Q (Lumped filters only)

To associate a specific part value with a specific Q, add "Q=50", where 50 is an example Q value to the part line. For example:

Setting Series and Parallel Parasitic Resistance (Lumped filters only)

To associate a specific part value with a specific resistance, add "Rs=5" and "Rp=525", where 5 is an example series resistance value, and 525 is an example parallel resistance. The standard parts form is shown in Figure 1 below.

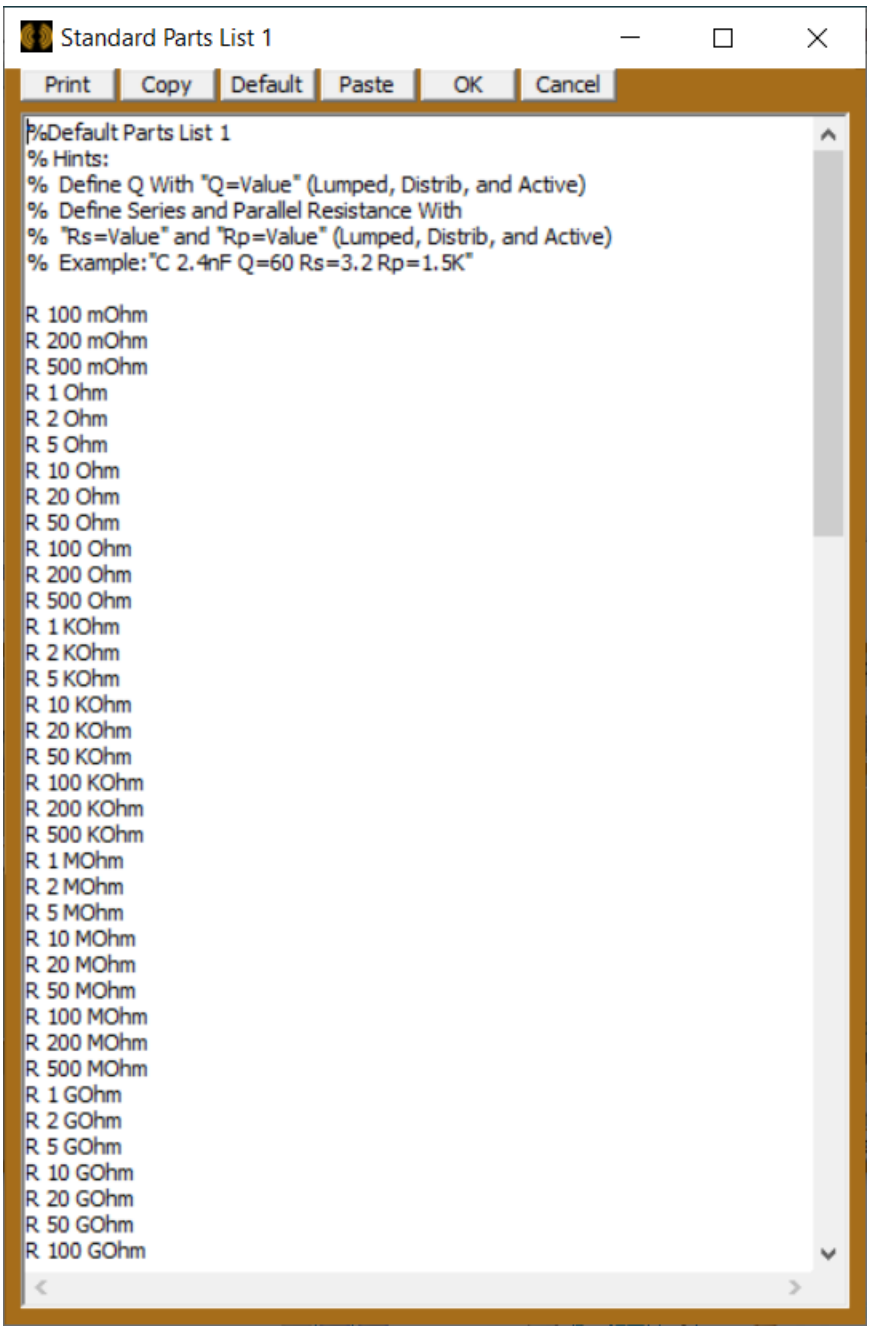
For Example:

"L 32uH Q=40 Rs = 3.5 Rp=7.25K" will have the effect of setting all inductors whose nearest part value is 32 uH to a Q of 40, a series resistance of 3.5 Ohms, and a parallel resistance of 7.25K Ohms.

Saving in an External File

FilterSolutions saves the user defined parts list in its own file. However, if it is desired to save the parts list in an external file, press Ctrl+C to copy the parts list and Ctrl+P to paste in the external file.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

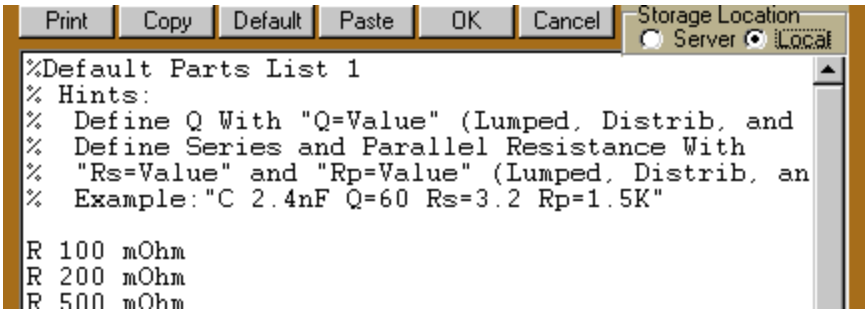


Saving on a common network server (Floating licenses only)

Network users of FilterSolutions may save standard parts on the network manager with the use of an Administrator's password. Network users may retrieve this common standard parts lists for use in their projects. This helps insure that all users on a project are using the same standard parts list. Users and administrators may select in the upper right where their part lists come

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

from, as shown in Figure 2 below. Server parts are read-only for users, and read/write for administrators. The license key issued by Nuhertz determines user/administrator status.



Parts Files

If administrators do not wish to use the administrative passwords to upload standard parts to Nuhertz Server, parts files may be manually copied into their respective locations instead. Client FilterSolutions parts files are located in the FilterSolutions_ini directory Default: (C:\Program Files (x86)\Nuhertz Technologies\FilterSolutions_ini\) under the Nuhertz Technologies program directory. They are names, parts0.dat, parts1.dat and parts2.dat for standard parts lists 1, 2, and 3 respectively. These files may be manually copied into the server Nuhertz Server program directory (Default: C:\Program Files (x86)\Nuhertz Technologies\Nuhertz Server\) as well, removing the necessity of providing an administrators password to the individual(s) charged with maintaining the parts list.

The buttons across the top perform the following functions:

Print	Print the displayed standard parts
Copy	Copy the displayed parts to the windows clipboard. Select the portion to be copied, or select all or nothing to copy all text in the text box.
Default	Sets the standard parts to the default parts list.
Paste	Paste text from the windows clipboard into the standard parts text box.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

OK	Save you standard parts list in the FilterSolutions database and exit.
Cancel	Exit without saving the parts list.
Storage Location	Determines where the standard parts lists is read from. Administrators may write to the server parts lists. Users may only read from it.

Parts List Formats:

Comments	Precede the comment line with a "%" character. Up to 10 comments are allowed.
Numerical Entries	Integer, floating, and engineering notation are all acceptable. Six significant digits are stored in the database. Example: 12, 0.00047, 470n, 50pF are all acceptable numerical entries. All numerical entries are sorted in ascending order in the database. Up to 500 numerical entries are allowed per element type.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Capacitors	Precede the capacitance numerical entry with a "C" or "c" character.
Inductors	Precede the inductance numerical entry with a "L" or "l" character.
Resistor	Precede the capacitance numerical entry with a "R" or "r" character.

1%, 5%, 10% and 20% Standard Parts

The Change Control Panel has selections for 1%, 5%, 10%, and 20%. When these options are selected, the nearest part value is selected from a list of standard industrial values. These values are listed below:

20% Parts	1.00, 1.5, 2.2, 3.3, 4.7, 6.8, 10.0, 15.0...
10% Parts	1.00, 1.20, 1.50, 1.80, 2.20, 2.70, 3.30, 3.90, 4.70, 5.60, 6.80, 8.20, 10.00, 12.0...
5% Parts	1.00, 1.10, 1.20, 1.30, 1.50, 1.60, 1.80, 2.00, 2.20, 2.40, 2.70, 3.00, 3.30, 3.60, 3.90, 4.30, 4.70, 5.10, 5.60, 6.20, 6.80, 7.50, 8.20, 9.10, 10.00, 11.00...
1% Parts	1.00, 1.02, 1.05, 1.07, 1.10, 1.13, 1.15, 1.18, 1.21, 1.24, 1.27, 1.30, 1.33, 1.37, 1.40, 1.43, 1.47, 1.50, 1.54, 1.58, 1.62, 1.65, 1.69, 1.74, 1.78, 1.82, 1.87, 1.91, 1.96, 2.00, 2.05, 2.10, 2.15, 2.21, 2.26, 2.32, 2.37, 2.43, 2.49, 2.55, 2.61, 2.67, 2.74, 2.80, 2.87, 2.94, 3.01, 3.09, 3.16, 3.24, 3.32, 3.40, 3.48, 3.57, 3.65, 3.74, 3.83, 3.92, 4.02, 4.12, 4.22, 4.32, 4.42, 4.53, 4.64, 4.75, 4.87, 4.99, 5.11, 5.23, 5.36, 5.49, 5.62, 5.76, 5.90, 6.04, 6.19, 6.34, 6.49, 6.65, 6.81, 6.98, 7.15, 7.32, 7.50, 7.68, 7.87, 8.06, 8.25, 8.45,

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	8.66, 8.87, 9.09, 9.31, 9.53, 9.76, 10.0, 10.2...

Options Menu

Several options are available to select from by selecting "Options" in the Main Menu

Enable Real Time Updates	Check to force an automatic update of all visible schematics and graphs each time a change is made to the control panel. This has the effect of increasing efficiency in design and analysis.
Enable Hints	Check to enable Hint text to pop up on all controls when the mouse cursor is passed over the control
CSV Data Format	Check to display and save data in Comma Separated Values (CSV) format. Also saves sensitivity tables in CSV format. CSV format is useful when importing data into Excel, or other tools that support CSV format.
Auto Freq Scale	Check to automatically scale the frequency and time analysis axes. This has the effect of increasing design and analysis efficiency, but may not be the desired scale. If auto frequency scale is desired, enter the number of decades to view in the "Decades of Frequency" text entry.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

User Defined % Autoscroll Updates	Autoscroll updates are set by default to change the associated text entry by 5% pet click. If a different percentage is desired, check the box and enter the desired percentage to update the text entry value by.
Enable Singly Terminated Approximations	Doubly terminated filters are sometimes more accurately synthesized as singly terminated filters with a parasitic resistance in the non-terminated end. Check this option if it is desired to include such filters in widely terminated design requirements.
Enable Exports of Filters to Non-installed Third Parties	Check to enable the export panels of third parties, even when the third party is not installed. This permits the file save functions, if any, to be used to save filters in the third party script or project text file format for use later on a different PC. Only third parties that support direct COM translatable scripting or project text exports are supported with this feature.
Use Dash Pattern in Graphs	Check to display graphs with unique dash-dot patterns for each line. This is beneficial to distinguish between the lines when colors are inadequate.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Always Launch With FilterQuick/Advanced	If it is desired to always launch with one of the two user design panels, select it here. Otherwise the last user panel closed will be the first user panel launched.

Data Points to Graph Options:

Data Points to Graph	Enter the number of desired data points to plot in the analysis graphs. Fewer points mean faster analysis. Multiconductor analysis requires more computer time, and therefore defaults to fewer points.

Windows Dimensions Options:

Window Dimensions	Enter the number of pixels to set the floating and embedded schematic and simulation graph windows to. FilterQuick user interface is not affected.

Synthesis and Measurement Options:

Pass Band Ripple Definition	Pass band ripple may be defined in terms of S11 or S12.
Phase Linearity Measurement	Select between **group delay and phase deviation** for phase linearity measurements. Group delay is the negative of the phase with respect to frequency, is measured in units of seconds, and is the default. Phase deviation is the difference in the unwrapped phase angle and the least

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	squares linear regression of the unwrapped phase angle. Phase deviation unwrapping is at the 360 degree transitions.
Phase Deviation Regression	Select between the full window or just the pass band to compute linear regression needed for the phase deviation measurement.

Layouts Options:

Length Units	Only displayed if tx lines are activated. Select the desired units of length. Meters (m) is the default.

S-Parameter Options:

S-Parameter Interp Order	Select the interpolation order of simulated S-Parameter files. 3rd order is the usually the most accurate, but may cause undesired loopy behavior if too much frequency separation exists between points in the file. 2nd order tends to be oscillatory and is not normally recommended. 1st order interpolation is the most stable, but may result in accuracy loss, especially with group delay. Group delay may show undesired sawtooth patterns when 1st order interpolation is selected.
S-Parameter Optimization	Since S-Parameter files are normally used to approximate inductor or capacitor responses, internal optimization assumed an inductor or capacitor, and interpolates along a path expected by an inductor or capacitor to

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	increase interpolation accuracy. If an S-Parameter file is imported for any reason that is not a simple inductor or capacitor approximation, it is generally better to select the "General" option for best results.
Touchstone Exports	FilterSolutions provides frequency response exports in the form of Touchstone format S, Y, or Z parameter format, in both mag/ang or real/imaginary format. Select whichever format is desired.

Library Location Options:

Edit Nuhertz Library Path	Used for user-customized libraries. If it is desired to use a customized library, select the path and drive of the upper level library directory. It may be necessary to close and restart FilterSolutions before the new path works.

Engineering Notation

For the designer's convenience, FilterSolutions allows numbers with large or small exponents to be entered in standard engineering terms. Units may be omitted is desired. For example, a 120 pF capacitance may be entered as 1.2E-07, 120p, or 120pF. FilterSolutions will interpret any of these as 1.2E-07.

The table below provides standard engineering notation for the designer's reference.

Notation	Description	Value
f	femto	E-15

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

p	pico	E-12
n	nano	E-09
u	micro	E-06
m	milli	E-03
K	Kilo	E+03
M	Mega	E+06
G	Giga	E+09
T	Tera	E+12
P	Pecta	E+15

Transfer Functions

This chapter contains the following topics:

- Continuous Transfer Function Display
- Discrete Transfer Function Display
- Transfer Function Vectors

Continuous Transfer Function Display

Print	Print the transfer function
Copy	Copy the display to the windows clipboard. If only a portion of the display is to be copied, depress the left mouse key, and drag the mouse across the desired portion. A box will outline the portion selected for copy. Then select the copy option.
Vec	Bring up the Vector display box that will copy the transfer function numerator and denominator as vectors to the windows clipboard for use by other applications.

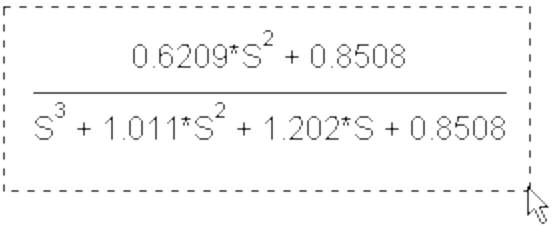
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Freeze	Create a duplicate display that freezes the data at the time of selection.
Exit	Terminates the transfer function display
Prototype	Displays the prototype transfer function
Stan	Displays the transfer function in Standard form.
Casc	Displays the transfer function in Cascade form. This is useful for active filter design.
Para	Displays the transfer function in Parallel form. This is useful for active filter design.
Sig Dig	Enter the number of digits to display in transfer function numerals.
fit	Fit the transfer function display to the window.

To copy only a portion of the display to the clipboard:

Depress the left mouse key and drag cursor across area to be copied. Select the Copy button while the outline is shown.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Discrete Transfer Function Display

Print	Print the transfer function
Copy	Copy the transfer function image to the windows clipboard. If a rectangle is selected by dragging the mouse, only the inside of the rectangle will be copied.
Vec	Bring up the Vector display box that will copy the transfer function numerator and denominator as vectors to the windows clipboard for use by other applications.
Ccode	Generates automatic C code that creates the displayed digital filter. The C code maintains the exact form (Stan, Casc, Para) and precision (Sig Dig entry) of the displayed Z Transform. Be certain to enter in SigDig the desired precision of the filter prior to generating C code.
Test	Test the filter performance and effect of fixed precision errors in the displayed form (Stan, Casc, or Para) and with the entered significant digits (SigDig) with the time response trace, frequency response trace , or pole/zero plot.
Freeze	Create a duplicate display that freezes the data

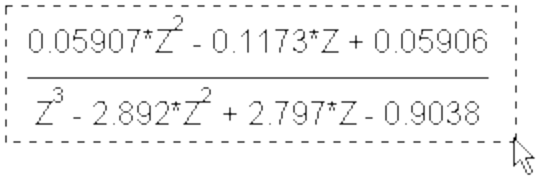
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	at the time of selection.
Exit	Terminates the transfer function display
Stan	Displays the transfer function in Standard form.
Casc	Displays the transfer function in Cascade form. This is useful for minimize errors due to finite precision.
Para	Displays the transfer function in Parallel form. This is useful to minimize errors due to finite precision.
Sig Dig	Enter the number of digits to display in transfer function numerals.
fit	Fit the transfer function display to the window.
Digital Format	Select the digital output format for Z transform constants.
Num Scale Den Scale	Displayed for fixed point only. Enter the maximum range for fixed point numbers in the displayed Z transform. Separate range selections are available for numerator and denominator, except in Cascade mode. In Cascade mode, separate ranges are available for the gain and the coefficients.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

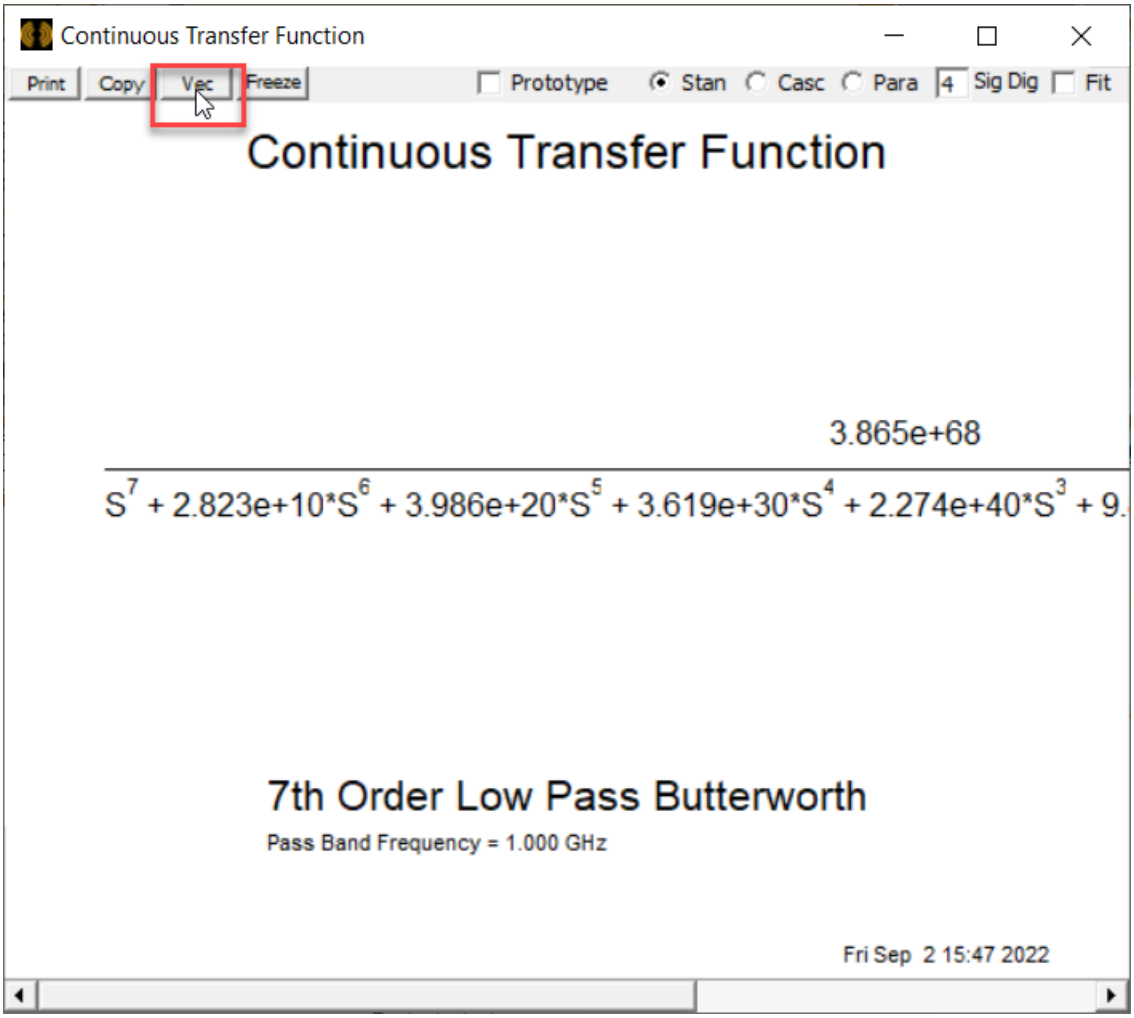
To copy only a portion of the display to the clipboard:

Depress the left mouse key and drag cursor across area to be copied. Select the Copy button while the outline is shown.

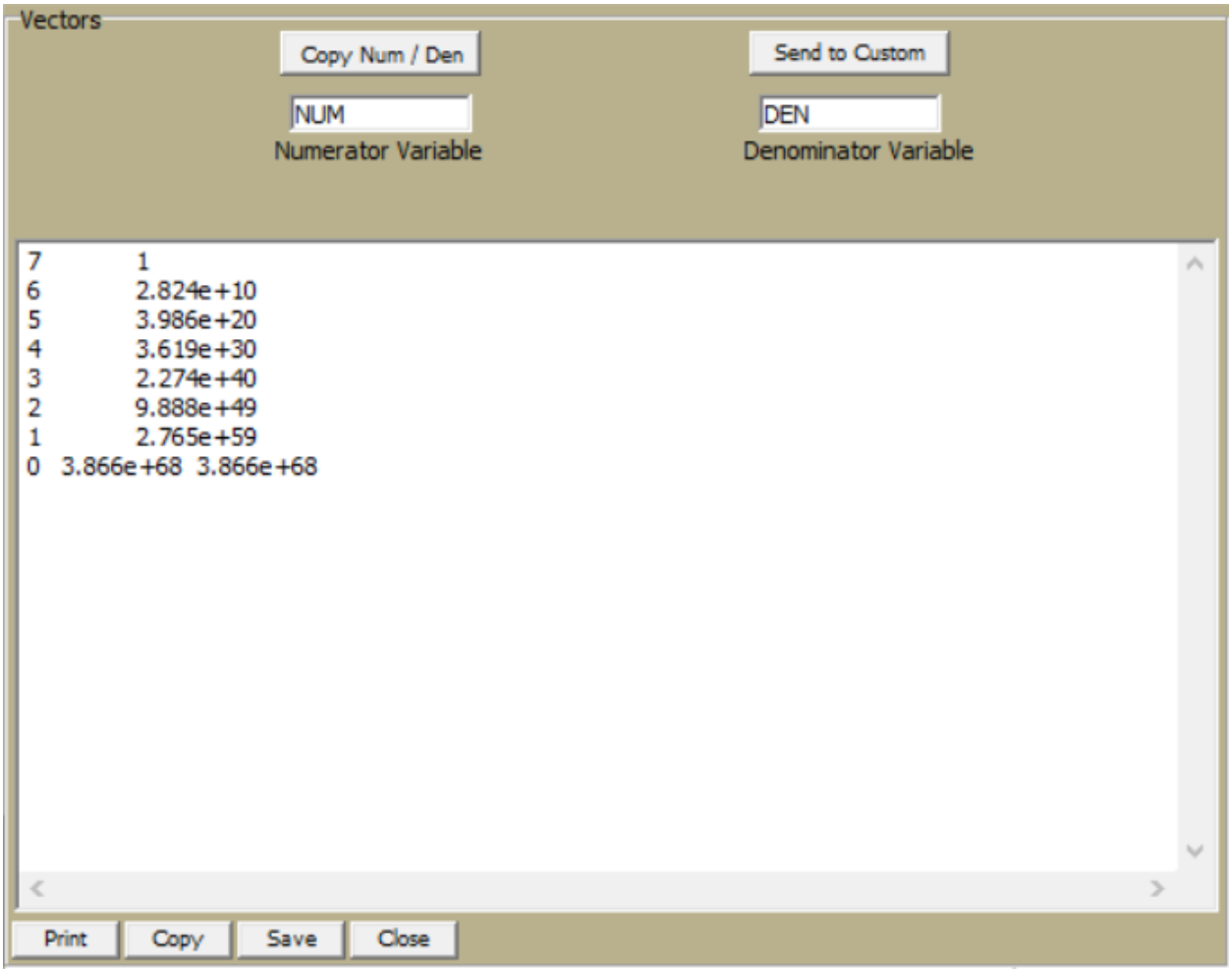


Transfer Function Vectors

Select the **Vec** button on the continuous or digital Transfer Functions displays, or the F(S) button on any circuit display to display the Vector control panel .



Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

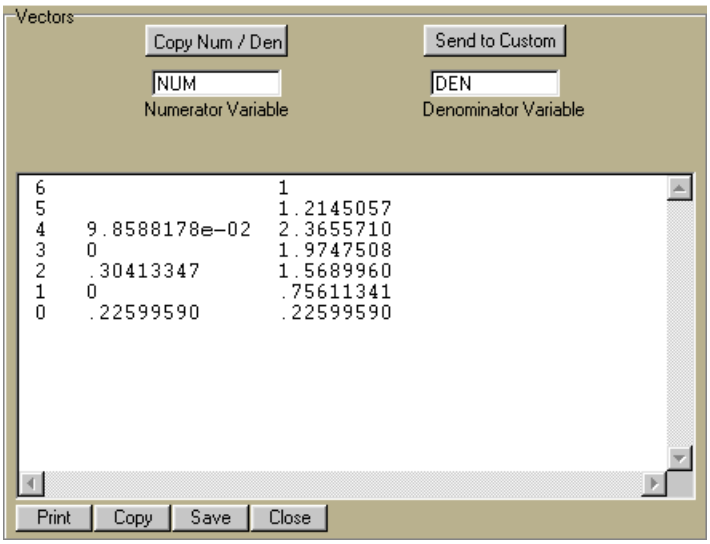


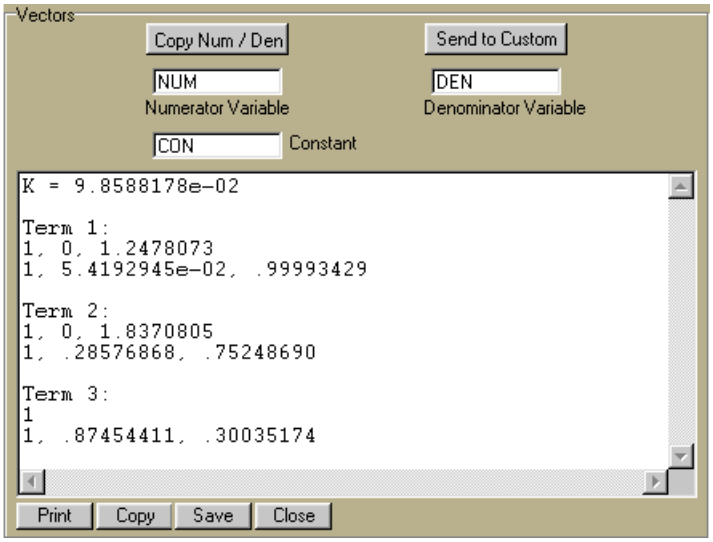
The vector control panel is used to copy the displayed transfer function to the windows clipboard in vector format that is compatible with Matrix-x and Matlab or to the FilterSolutions Custom Filters windows. For compatibility purposes, the continuous transfer function vector is copied in terms of S and the discrete transfer function vector is copied in terms of Z**-1, starting with Z**0 on the left and descending powers to the right. Note that Z transforms with denominators of higher order than the numerator will pad the Z**-1 vectors with 0's to the left until the left most term is Z**0.

Circuit Transfer function vectors with finite Q use the displayed resistance at each finite Q element to model a finite Q transfer function. This model is only valid for frequencies near the frequency used to calculate the displayed resistance.

The Form of the transfer function (Standard, Cascade, or Parallel) is preserved in the copied vector.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





Copy Num/Den From Standard Form	Copies the numerator and denominator vector of the transfer function along with the associated variables into the windows clipboard. The discrete transfer function shown in the fist illustration above will be copied as:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	NUM = [2.3383724e-04, -4.6048332e-04, -2.3378396e-04, 9.2107321e-04, -2.3378396e-04, -4.6048332e-04, 2.3383724e-04] DEN = [1, -5.8623845, 14.337149, -18.722583, 13.768864, -5.4066461, .88560049] %Hit Enter Key: The variable names and number of significant digits is selectable by the user. This feature may be used to copy and paste directly into Matlab, paste into a Matlab M-file or Matrix-x Mathscript file, or pasted into any other ASCI file.
Copy Num/Den **From Cascade or Parallel Form**	Copies the numerator and denominator vector of the transfer function along with the associated variables into the windows clipboard. The discrete transfer function shown in the second illustration above will be copied as: NUM1 = [1, 0, 1.2478073] DEN1 = [1, 5.4192945e-02, .99993429] NUM2 = [1, 0, 1.8370805] DEN2 = [1, .28576868, .75248690] NUM3 = [1] DEN3 = [1, .87454411, .30035174] CON = 9.8588178e-02 %Hit Enter Key:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	The variable names and number of significant digits is selectable by the user. The variable names and number of significant digits is selectable by the user. This feature may be used to copy and paste directly into Matlab, paste into a Matlab M-file or Matrix-x Mathscript file, or pasted into any other ASCI file.
Numerator Variable	Enter the desired name of the variable that will represent the numerator when Copy Num/Den is selected.
Denominator Variable	Enter the desired name of the variable that will represent the denominator when Copy Num/Den is selected.
Send to Custom	Sends the Z transform to the Custom windows. The format will match the Custom format selection. This feature is useful when designing staged filters.
Constant Variable	The Cascade and Parallel forms both contain an additional constant. For Cascade form, this constant is a multiplier. For Parallel form, this constant is an offset. In both cases, the Constant Variable is used to define the Matlab variable that will contain this value. Constant Variable is not displayed for Standard form.
Print	Print the text window contents

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Copy	Copy some or all the text window contents to the clipboard. No highlighted text results in all text being copied.
Save	Save the text to a file
Close	Close the vector window.

Graphical Displays

This chapter contains the following topics:

- Pole Zero Plot Display and Real Time Updates
- Pole Zero Modifications
- Time Display
- Frequency Display
- Reflection and Impedance Display
- Smith Charts and Polar Plots
- Change Graph Limits

Pole Zero Plot Display

Print	Print the display
Copy	Copy the display to the windows clipboard.
Limits	Activate the Limits Control Panel to change the plot limits.
Zoom	Select Zoom mode. Select "Zoom" again to deselect Zoom mode. When in Zoom mode, left mouse click to zoom in and the right mouse click to zoom out. Use "Limits" to restore to original scale.

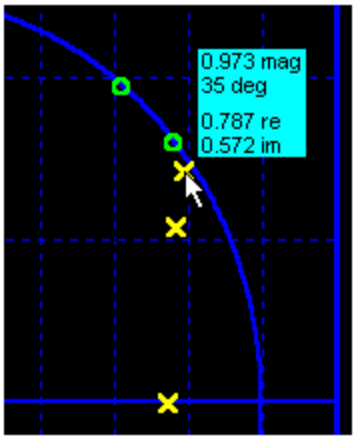
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Drag	Select Drag Mode. Select "Drag" again to deselect drag mode. When in Drag mode, use the cursor to drag the entire complex plane. Use "Limits" to restore to original position.
Freeze	Create a duplicate display that freezes the poles and zeros at the time of selection.
Exit	Terminates display
Base	Check to display the baseline poles and zeros on the display.
RTU	Real Time Update. Displayed for continuous and digital transfer function plots only. Check this box to cause other FilterSolutions displays to update in real time and poles and zeros are manually manipulated.
Restore Passband	Displayed for Butterworth and Chebyshev I filters with transmission zero in RTU mode. Check to recompute the pass band in real time.
Update Zeros	Displayed when "Restore Passband" is checked and zeros have been manually updated. Click to set current zeros positions permanent in the pole/zero plane and in the zeros table.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Prototype	Displayed for continuous transfer function plots only. Displays the prototype poles and zeros. Only prototype poles and zeros are shown for diplexers.
Polar	Displayed for continuous transfer function and reflection coefficients only. Check to display the plot in polar form using Wo and Q as coordinates.
Zeros	Check to display the zeros.
Poles	Check to display the poles

To locate a pole or zero location, insure Drag mode and Zoom mode are deselected, and place the cursor on the pose or zero and select as shown:



Poles and zeros may be shifted with the cursor and left mouse key, or **added and deleted with the right mouse key**. **A Right click while depressing the left key will add all pass poles and zeros.**

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Pole Zero Plot Modifications

FilterSolutions contains the powerful ability to modify the pole zero plots with the mouse and cursor. Any S or Z transfer function pole or zero may be repositioned to anywhere on the plot. Poles and zeros may be added anywhere on the plot prior to creating a filter. Conjugate poles and zeros and all pass poles and zeros are automatically move in unison. Laplace Transform poles and zeros may be manipulated by their prototype poles and zeros or their actual poles and zeros.

When a pole or zero is relocated, a dark blue pole or zero remains at the original location. Select **Rest** in the graph to restore poles and zeros to their original location.

Available actions are:

Action	Effect
Position Cursor Over Pole or Zero Depress Left Mouse Key Move Pole or Zero with Cursor	Move pole or zero
Position Cursor Over Pole or Zero Right Mouse Click	Add/Delete Pole or Zero
Position Cursor Over Desired Pole Location Depress and Hold Left Mouse Key Right Mouse Click	Add All Pass Poles and Zeros

When manipulating a pole zero plot, it is important to note that **not all pole zero combinations are realizable with passive circuit filters.** Actual band pass and band stop Laplace transforms that have been manipulated will not generate passive circuit filters. Use Prototypes to modify pass band and stop band filters.

> **Important: Displayed filters that result from pole/zero combinations that are not realizable are labeled "ERROR"**

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To see the affects of the new pole/zero locations, activate any FilterSolutions function (Circuits, Frequency, etc.) after moving the poles and zeros.
- To see the affects of the poles and zeros in real time, check the **RTU** box in the pole/zero plot control panel.
- To recompute the pass band characteristics of Butterworth and Chebyshev filters with transmission zeros, select the **Restore Pass Band** option.
- To permanently update Butterworth and Chebyshev transmission zeros, select **Update Zeros**.

Time Display

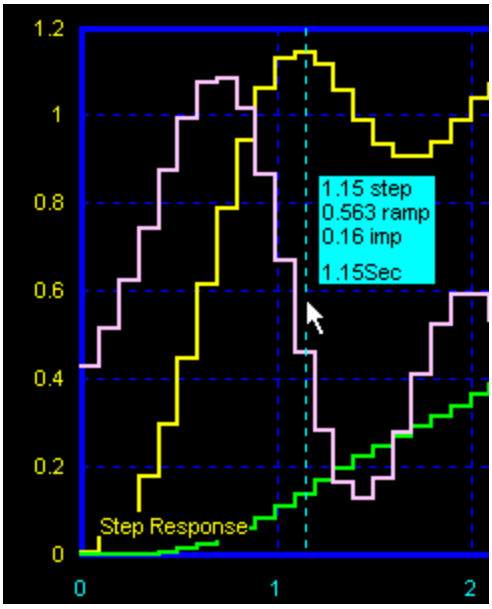
Print	Print the display
Copy	Copy the display to the windows clipboard.
Limits	Activate the Limits Control Panel to change the plot limits.
Text	Displays time data in textual form.
Freeze	Create a duplicate display that freezes the data at the time of selection.
Exit	Terminates display
Base	Displayed when circuits have been modified and for digital filter tests. Check to display the baseline time trace on the display.
Relative dB	Check to display time traces in units of dB, with the maximum value always set to 0dB. The dB value at any point in the trace is then always

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	relative to maximum dB.
Step	Check to plot the step response. Not displayed for user data analysis
Ramp	Check to plot the step response. Not displayed for user data analysis
Impulse	Check to plot the step response. Not displayed for user data analysis
Output	Displayed for user data analysis Check to display the filtered user input data.
Input	Displayed for User Data Analysis Check to display the raw user input data.

To determine the position of the traces at any point on the plot, place the cursor on the point and **left click** as shown below. The light blue box contains the value of the traces where they intersect the dotted line. **Right click** to permanently embed the cursor pointer into the graph. Right click again to remove the pointer.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The textual data display contains four control buttons:

Print	Print the text window contents
Copy	Copy some or all the text window contents to the clipboard. No highlighted text results in all text being copied.
Save	Save the text to a file
Close	Close the text window.

Frequency Display

Print	Print the display
Copy	Copy the display to the windows clipboard.

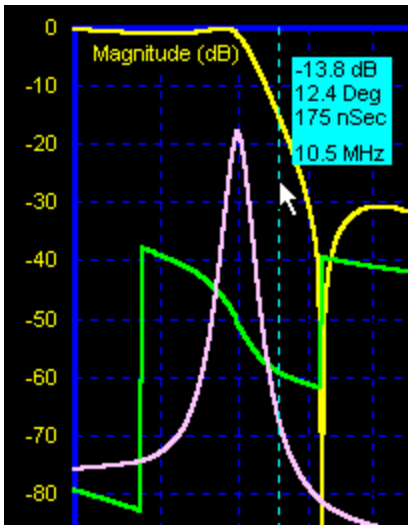
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Limits	Activate the Limits Control Panel to change the plot limits.
Text	Display frequency data in textual format.
Freeze	Create a duplicate display that freezes the data at the time of selection.
Exit	Terminates display
Base	Displayed when circuits have been modified and for digital filter tests. Check to display the baseline frequency trace on the display.
Magnitude	Check to plot the frequency magnitude response
Phase	Check to plot the frequency phase response
Group Delay	Check to plot the frequency group delay response
dB	Check to plot the magnitude in dB as opposed to arithmetic
Deg	Check to plot the phase curve in degrees as opposed to radians.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Smith	Displays a Smith chart. (passive, tx line, and ideal frequency responses only)
Polar	Displays a Polar plot.
Zgrid	Places an impedance overlay on Smith charts.
Ygrid	Places an admittance overlay on Smith charts, turning it into a Jones chart.

To determine the position of the traces at any point on the plot, place the cursor on the point and **left click** as shown below. The light blue box contains the value of the traces where they intersect the dotted line. **Right click** to permanently embed the cursor pointer into the graph. Right click again to remove the pointer.



The textual data display contains four control buttons:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Print	Print the text window contents
Copy	Copy some or all the text window contents to the clipboard. No highlighted text results in all text being copied.
Save	Save the text to a file
Close	Close the text window.

Reflection and Impedance Displays

The reflection coefficient and input impedance are displayed as a frequency display. The table below is a duplicate of the table shown in the frequency display section.

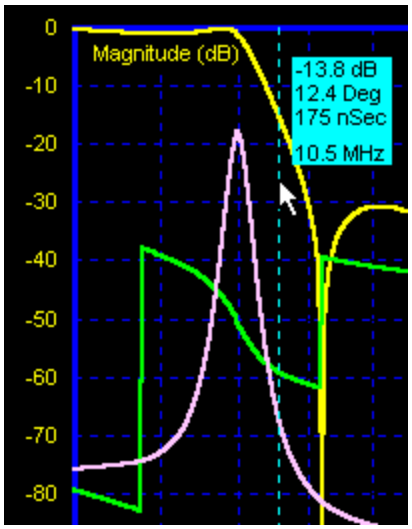
Print	Print the display
Copy	Copy the display to the windows clipboard.
Limits	Activate the Limits Control Panel to change the plot limits.
Text	Displays the reflection data in textual form.
Freeze	Create a duplicate display that freezes the data at the time of selection.
Exit	Terminates display

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Base	Displayed when circuits have been modified and for digital filter tests. Check to display the baseline reflection trace on the display.
Magnitude	Check to plot the reflection or impedance magnitude response
Phase	Check to plot the reflection or impedance phase response
Group Delay	Check to plot the reflection or impedance group delay response
dB	Check to plot the magnitude in dB as opposed to arithmetic
Deg	Check to plot the phase curve in degrees as opposed to radians.
Smith	Displays a Smith chart.
Polar	Displays a Polar plot.
Zgrid	Places an impedance overlay on Smith charts.
Ygrid	Places an admittance overlay on Smith charts, turning it into a Jones chart.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To determine the position of the traces at any point on the plot, place the cursor on the point and **left click** as shown below. The light blue box contains the value of the traces where they intersect the dotted line. **Right click** to permanently embed the cursor pointer into the graph. Right click again to remove the pointer.



The textual data display contains four control buttons:

Print	Print the text window contents
Copy	Copy some or all the text window contents to the clipboard. No highlighted text results in all text being copied.
Save	Save the text to a file
Close	Close the text window.

Smith Charts and Polar Plots

Polar plots consist of an array of plotted complex points with the magnitude indicated by distance from the origin and the phase angle indicated by angle from the right horizontal (other

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

systems may choose the vertical or another baseline angle). Since frequency information is not included in such a plot, FilterSolutions indicates a few scattered frequency points, and the left and right mouse keys indicate frequency, magnitude, and angle information for the nearest polar point.

Smith Charts

Smith charts are polar plots with a Cartesian impedance translation overlay. This is useful when plotting polar reflection data so that the grid indicates normalized Cartesian impedance. If a "Ygrid" is selected (Also known as a **Jones Chart**), then normalized Cartesian admittance is used as the overlay, and admittance data can be directly read from the reflection polar plot.

For polar plots, the maximum range of the magnitude is displayed on the chart. For Smith and Jones charts, the maximum range is always one by definition, so no maximum range is displayed.

Supported Displays

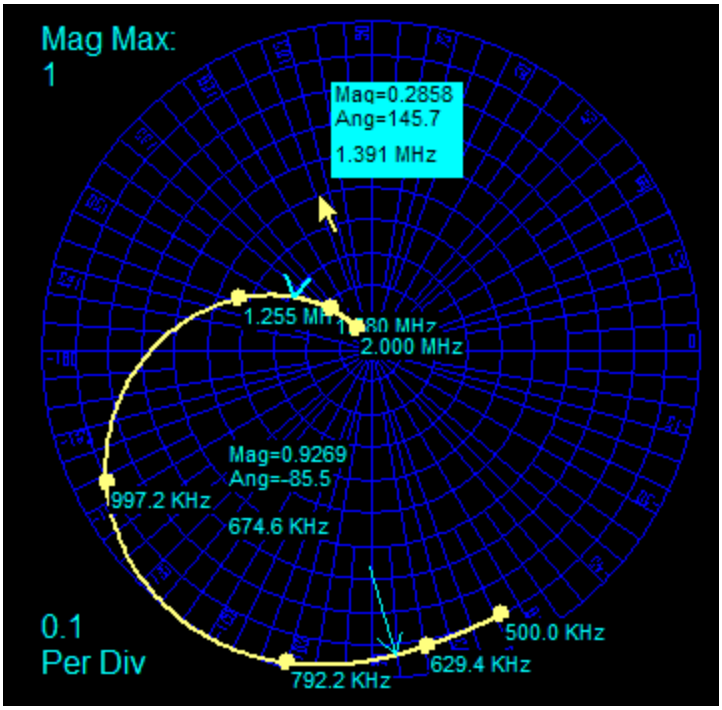
All frequency responses displays support a selection for the display of Polar plots. All reflection coefficient displays support a selection for the display of Smith charts, Jones charts, and polar plots. All passive LC and transmission line frequency response displays and reflection response displays support a selection for Smith charts, Jones charts, and polar plots. Active, switched capacitor, and digital filter frequency responses do not support Smith charts or Jones charts due to the lack of reflection features in those filters.

Cursor Data

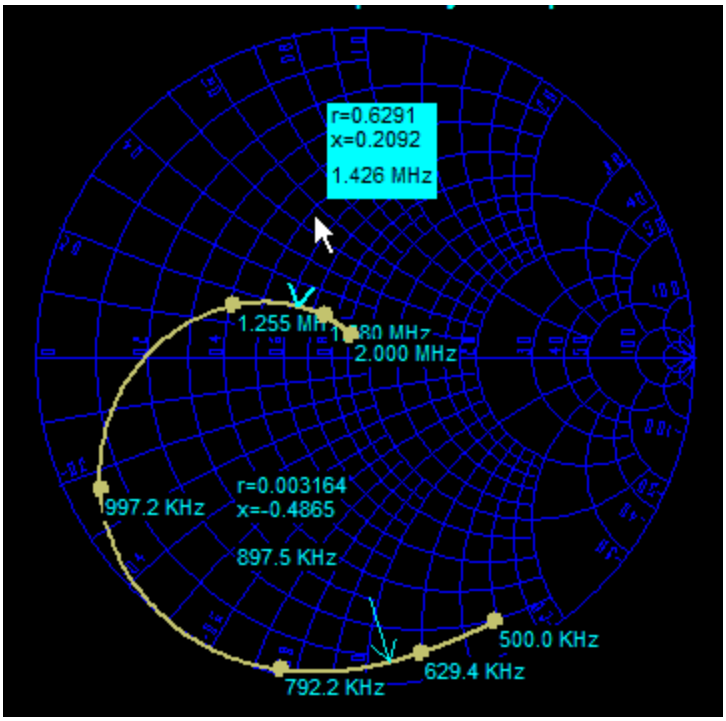
The left mouse key places the information on the screen for the duration of the key press, and the right mouse key places the information onto the screen permanently, or until the right mouse key is depressed again at the frequency point. Polar plots display the polar data, Smith and Jones charts display the Cartesian impedance data (r and x).

The example below shows a depressed left mouse key in the upper center displaying the current cursor data for the thick arrow below the data box, and a previously depressed right mouse key below with cursor data pointing to the selected point with the thin arrow.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Polar plot cursor data from left and right mouse keys.

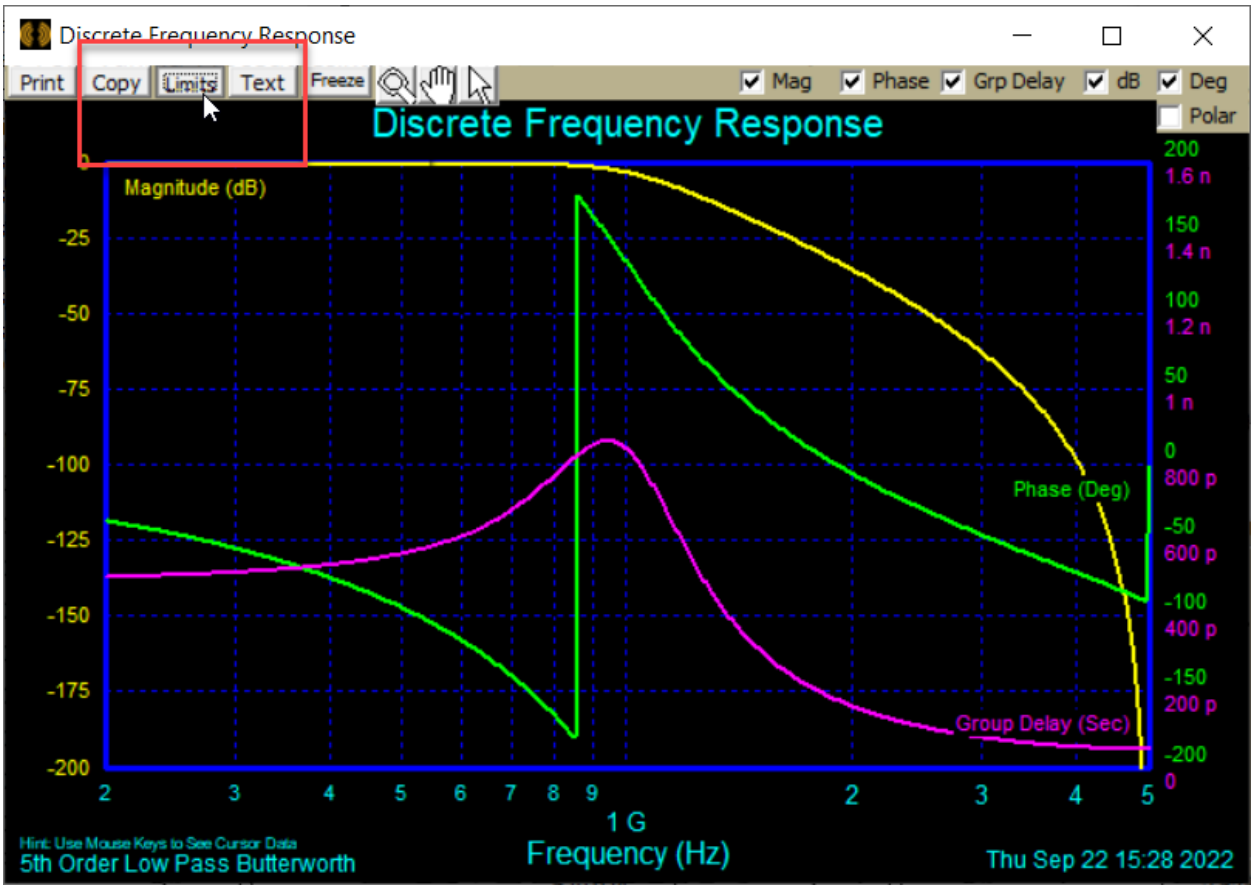


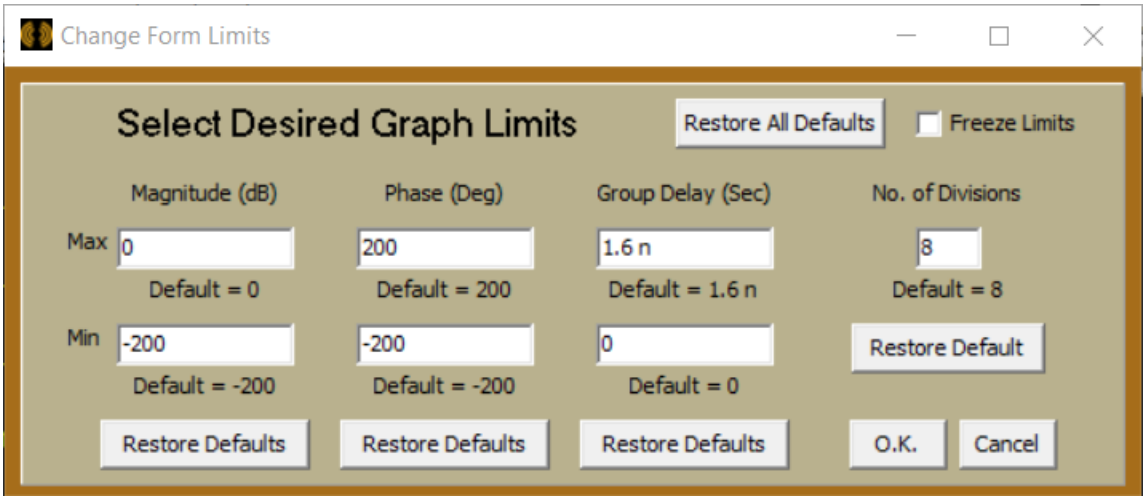
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Smith chart cursor data from left and right mouse keys.

Change Graph Limits

To display the Change Form Limits panel, select **Limits**.





Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Limits control panel is used to change the graphs limits if the auto-scale limits are not desired. Limits may also be changed in the Pole/Zero plots with the zoom and drag keys.

Max	Print the display
Min	Enter the new desired minimum value
No of Divs	Enter the new desired number of divisions
Restore Defaults	Restore the original default values.
Freeze Limits	Check to keep current graph limits for all future graph of the same type. The "Types" of graphs are frequency, time, reflection, and pole/zero graphs.

Circuit Displays

This chapter contains the following topics:

- Lumped Circuits Display
- Distributed Display
- Active Circuits Display
- Editing Schematics Video

Lumped Schematic Display

Print	Print the display
Copy	Copy the display to the windows clipboard. If only a portion of the display is to be copied, depress the left mouse key, and drag the mouse across the desired

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	portion. A box will outline the portion selected for copy. Then select the copy option.
Annote	Allows you to add personal annotations to the circuit display. Annotations are passed down to circuit children functions.
Netlist	Displays the circuit netlist in the netlist Display Box.
M Carlo	Performs a Monte Carlo analysis, generating unique analysis curves for each statistical condition.
Sense.	Creates a sensitivity analysis table containing sensitivity data for each element in the filter at a specified frequency.
Freeze	Create a duplicate display that freezes the circuit at the time of selection.
Restore	Restored changed element values to their default values.
C tune	Displayed when only inductor values have been manually changed. Click to tune the capacitors to minimize the pass band error resulting from the manual inductor changes.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

L tune	Displayed when only capacitor values have been manually changed. Click to tune the inductors to minimize the pass band error resulting from the manual capacitor changes.
Tune	Displayed when any inductor or capacitor values have been manually changed. Click to tune either the capacitors or inductors to minimize the pass band error resulting from the manual changes to the opposite type elements.
Freq	Display the frequency trace of the displayed circuit. Baseline traces of the unmodified circuit show up in the background.
Time	Display the time trace of the displayed circuit. Baseline traces of the unmodified circuit show up in the background.
F(S)	Displays and exports the transfer function of the filter with or without modifications.
Sparam	Displayed when finite source and load resistors are defined. Displays the S parameters (Scatter Matrix) S21, S12, S11, and S22. S21/S12, and S11/S22 are combined into one trace when they are the same. S11 and S22 are identical to the input and output reflection coefficients respectively and are always the same in physically

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	symmetrical filters. S21 and S12 are identical to the forward and reverse frequency response from the generator, adjusted by 2*sqrt(Rg/Rl) and 2*sqrt(Rl/Rg) respectively, which makes them both the same.
Z in	Display the input impedance trace of the displayed circuit as seen by the source resistor. Baseline traces of the unmodified circuit show up in the background.
Z out	Display the output impedance trace of the displayed circuit as seen by the load resistor. Voltage sources are assigned zero impedance. Current sources are assigned infinite impedance. Baseline traces of the unmodified circuit show up in the background.
Coupled Resonators	Displayed for band pass filters without stop bands (Gaussian, Bessel, Butterworth, Legendre, and Chebyshev I). Check to display the coupled resonator band pass filters.
Tubular	Displayed for band pass filters without stop bands (Gaussian, Bessel, Butterworth, Legendre, and Chebyshev I). Check to display the tubular band pass filters.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

X Coupled	Displayed for Chebyshev I and Elliptic filters. Check to display the cross coupled resonator filter.
Balance	Displayed only when no all pass stages exist in the circuit. Check to display the lattice balanced filter topology.
Balance 1	Displayed only when all pass stages exist in the circuit. Check to display the balanced filter topology with Bridged T all pass stages.
Balance 2	Displayed only when all pass stages exist in the circuit. Check to display the balanced filter topology with lattice all pass stages.
Amplitude Equalization	Displayed for finite Q filters with zero source resistance. Select between shunt, series, or equal impedance equalization schemes, or "none" for no equalization.
Inductor Parts Select	Select the desired standard parts list or standard percent part to update all inductors in the schematic to.
Capacitor Parts Select	Select the desired standard parts list or standard percent part to update all capacitors in the schematic to.
Q and Res	Displayed for filters with finite Q elements. Select "Res" to display resistance values

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

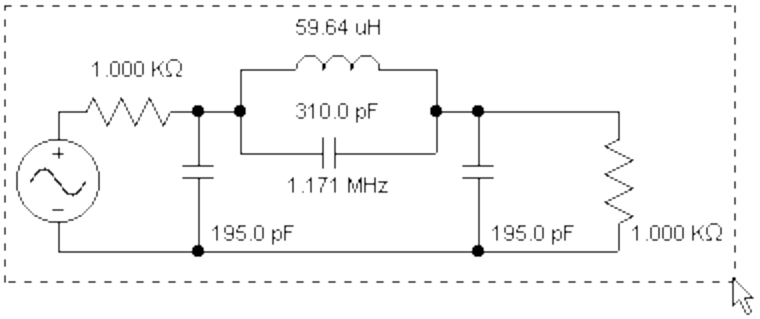
	calculated from the dominant frequency of the element. This is the resonant frequency for LC pairs, and the filter cutoff or center frequency otherwise.
Coupled Resonator Control Box Parameters	Control box displayed when coupled resonators are selected. Used to modify the resonator properties.
Tubular Control Box Parameters	Control box displayed when tubular filters are selected. Used to modify the tubular properties.
Digs	Enter the number of digits to display in element value numerals.
Fit	Fit the circuit to the window.

- Circuit element values and Q may be changed by left clicking the mouse while the cursor is over the element.
- Circuit elements may be added or deleted by right clicking the mouse while the cursor is over the element.
- Circuit elements are highlighted when the cursor is over them and changes to the element are enabled.

To copy only a portion of the display to the clipboard:

Click and drag the cursor across area to be copied. Select the Copy button while the outline is shown.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Distributed Schematic Display

Print	Print the display
Copy	Copy the display to the windows clipboard. If only a portion of the display is to be copied, depress the left mouse key, and drag the mouse across the desired portion. A box will outline the portion selected for copy. Then select the copy option.
Annote	Allows you to add personal annotations to the circuit display. Annotations are passed down to circuit children functions.
Netlist	Displays the circuit netlist in the netlist Display Box.
M Carlo	Performs a Monte Carlo analysis, generating unique analysis curves for each statistical condition.
Freeze	Create a duplicate display that freezes the

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	circuit at the time of selection.
Restore	Restored changed element values to their default values. Only displayed when one or more elements has been modified from its original value.
Freq	Display the frequency trace of the displayed circuit. Traces include magnitude, angle, and group delay. Baseline traces of the unmodified circuit show up in the background.
Time	Display the time trace of the displayed circuit. Baseline traces of the unmodified circuit show up in the background.
Sparam	Displayed when finite source and load resistors are defined. Displays the S parameters (Scatter Matrix) S21, S12, S11, and S22. S21/S12, and S11/S22 are combined into one trace when they are the same. S11 and S22 are identical to the input and output reflection coefficients respectively and are always the same in physically symmetrical filters. S21 and S12 are identical to the forward and reverse frequency response from the generator, adjusted by 2*sqrt(Rg/Rl) and 2*sqrt(Rl/Rg) respectively, which makes them both the same.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Z in	Display the input impedance trace of the displayed circuit as seen by the source resistor. Baseline traces of the unmodified circuit show up in the background.
Z out	Display the output impedance trace of the displayed circuit as seen by the load resistor. Voltage sources are assigned zero impedance. Current sources are assigned infinite impedance. Baseline traces of the unmodified circuit show up in the background.
Export	Export filter to other software. Select software format to export to.
Coupled Resonators	Displayed for band pass filters without stop bands (Gaussian, Bessel, Butterworth, and Chebyshev I). Check to display the coupled resonator band pass circuit to be displayed.
Layout View	Check to view the physical layout of the filter. Detailed design information is not displayed., but a distance scale bar is shown.
Other Info	Check to displays the alternate filter parameters of Lo, Co, Ro, Go. Not functional in Layout View mode.
Q and Res	Displayed for filters with finite Q elements. Select "Res" to display resistance values calculated from the dominant frequency of

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

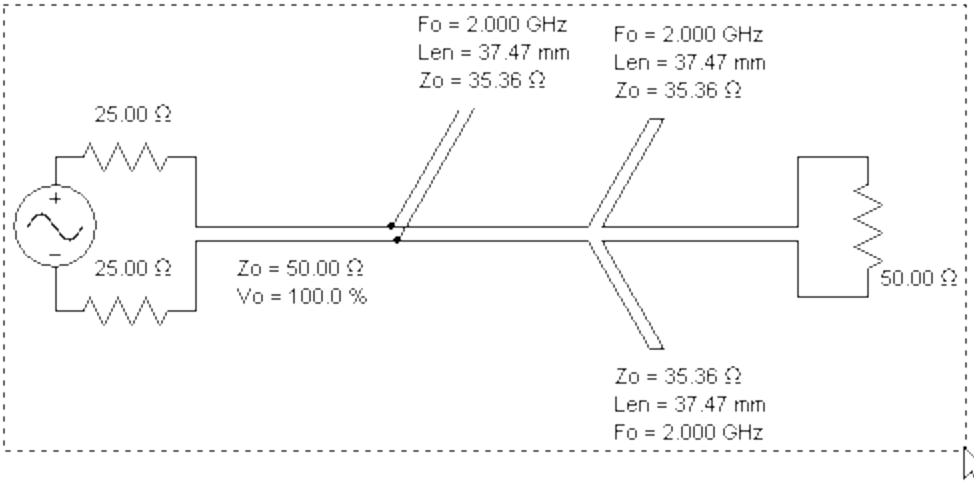
	the element. This is the resonant frequency for LC pairs, and the filter cutoff or center frequency otherwise.
Resonator Control Box Parameters	Control box displayed when coupled resonators are selected. Used to modify the resonator properties.
Digs	Enter the number of digits to display in element value numerals.
Fit	Fit the circuit to the window.

- Stub parameters, segment parameters, circuit element values, and element Q may be changed by left clicking the mouse while the cursor is over the element.
- Stubs may be added, deleted, lumped or unlumped by right clicking the mouse while the cursor is over the element.
- Resonators may be combined to one or split to two by right clicking the mouse while the cursor is over the element.
- Stubs, segments, and elements are highlighted when the cursor is over them and changes to the element are enabled.

To copy only a portion of the display to the clipboard:

Click and drag the cursor across area to be copied. Select the Copy button while the outline is shown.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Active and Switched Capacitor Circuit Display

Print	Print the display
Copy	Copy the display to the windows clipboard. If only a portion of the display is to be copied, depress the left mouse key, and drag the mouse across the desired portion. A box will outline the portion selected for copy. Then select the copy option.
Annote	Allows you to add personal annotations to the circuit display. Annotations are passed down to circuit children functions.
Netlist	Displays the circuit netlist in the netlist Display Box.
M Carlo	Performs a Monte Carlo analysis, generating unique analysis curves for each statistical condition.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sense.	Creates a sensitivity analysis table containing sensitivity data for each element in the filter at a specified frequency.
Freeze	Create a duplicate display that freezes the circuit at the time of selection.
Restore	Restored changed element values to their default values
Freq	Display the frequency trace of the displayed circuit. Baseline traces of the unmodified circuit show up in the background.
Time	Display the time trace of the displayed circuit. Baseline traces of the unmodified circuit show up in the background.
F(S)	Displays and exports the transfer function of the filter with or without modifications.
Z in	Display the input impedance trace of the displayed circuit. Baseline traces of the unmodified circuit show up in the background.
Cap Compute	Recalculate resistor values around selected capacitor values for all stages when possible. Some capacitors may be recalculated. If resistors cannot be calculated around chosen capacitors, an error message is displayed.

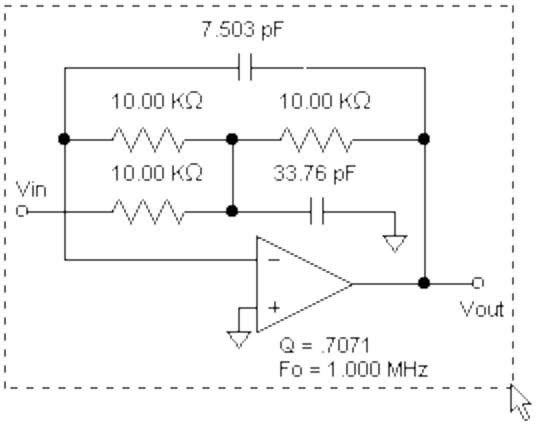
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Caps Select	Select capacitors to be used for analysis for all stages.
Resis Select	Select resistors to be used for analysis for all stages.
Digs	Enter the number of digits to display in element value numerals.
Fit	Fit the circuit to the window.

- Circuit element values and Q may be changed by left clicking the mouse while the cursor is over the element.
- Circuit elements are highlighted when the cursor is over them and changes to the element are enabled.

To copy only a portion of the display to the clipboard:

Click and drag the cursor across area to be copied. Select the Copy button while the outline is shown.



Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Editing Schematics Video

See the Online Help to view the Editing Schematics Video.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6 - Delay Equalizers

This chapter contains the following topics:

- About Delay Equalizers
- All Pass Pole Zero Placement
- Automatic Pole Zero Placement
- Off Axis Real and Quadruplet Zeros
- Group Delay and Phase Deviation
- Bridged Element Delay Equalization Video

About Delay Equalizers

It is possible to flatten the pass band group delay of any filter by designing and installing a delay equalizer. A delay equalizer is created by adding all pass sections to the filter, or by designing around off-axis zeros. The all pass, by definition, has no effect on the filter magnitude frequency response along the JW axis. It does, however, affect the phase and group delay. Off axis zeros drastically change the frequency response, however, internal design techniques are used to restore the equiripple pass band response in the case of a Chebyshev I, or the maximally flat pass band response in the case of a Butterworth, with the effect of maintaining the required magnitude response along the JW axis, but altering the phase and group delay.

Delay equalizers are created by placing all pass poles and zeros or off axis zeros in the location necessary to achieve the desired effects on the group delay. FilterSolutions supports **manual delay equalization** for all filters, and **automatic all pass delay equalization** for low pass and band pass filters. Manual off axis zero equalization is supported for low pass, high pass, and band pass passive, active, sw cap, and digital filters.

In the continuous plane, all pass functions consist of a pole and a zero spaced equal and opposite the imaginary axis. In the digital domain, the pole and zero exist on the same radius of the unit circle and are inversely proportional in distance from the origin.

In the continuous plane, off axis zeros consist of a pair of zeros spaced equal and opposite the imaginary axis. In the digital domain, the zeros exist on the same radius of the unit circle and are inversely proportional in distance from the origin.

All pass stages are more direct and easier to use, but are less efficient in that they involve canceling poles and zeros. Off axis zeros are more sophisticated, but also more efficient in that poles and zeros do not cancel.

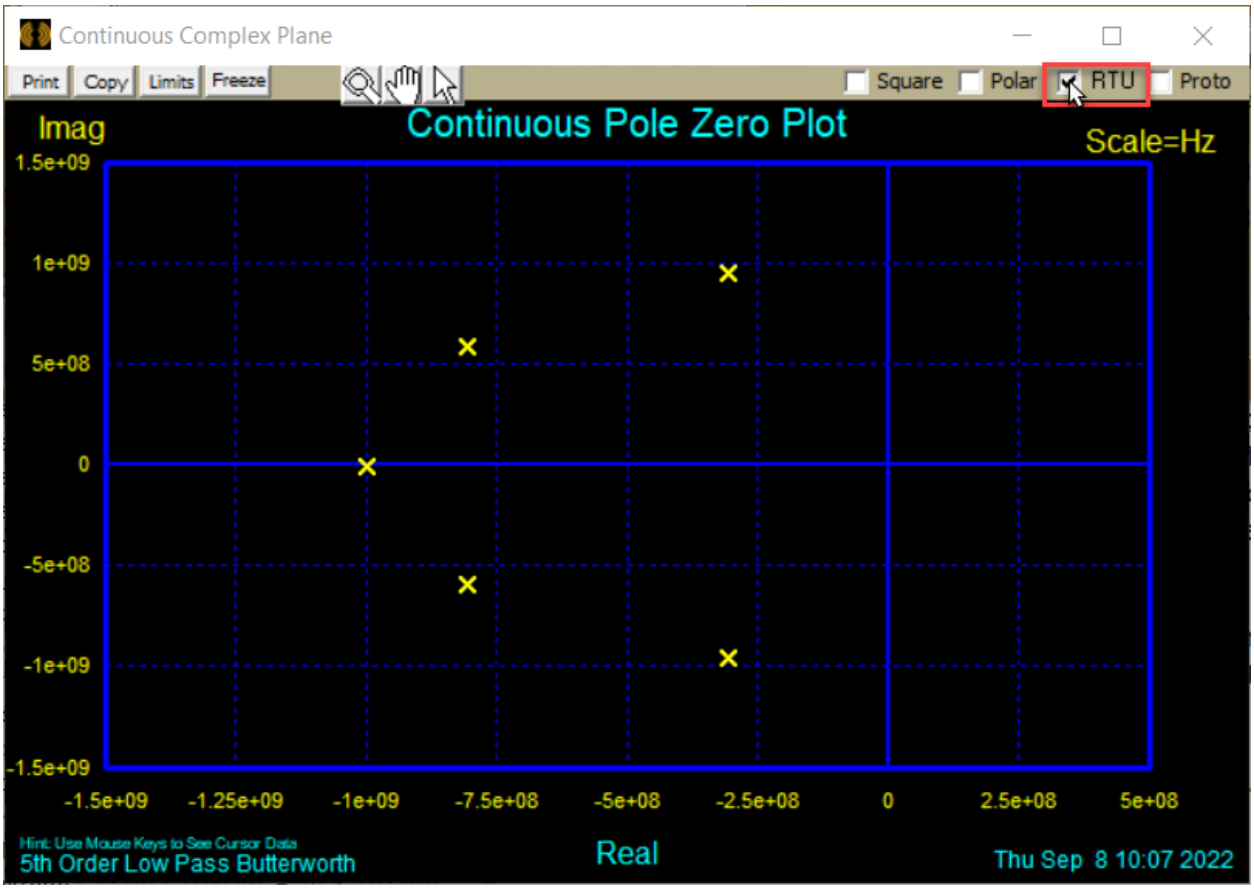
To create an all pass addition in FilterSolutions using the pole/zero plots:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Place the cursor at the point on the pole/zero plot where the pole is to be located.
2. While holding down the left mouse key, click the right mouse key. The all pass poles and zeros will be created at their appropriate positions.

You may slide any all pass pole or zero around with the cursor, and all associated poles and zeros will move in unison. FilterSolutions will automatically install the zeros all pass sections into the zeros filter.

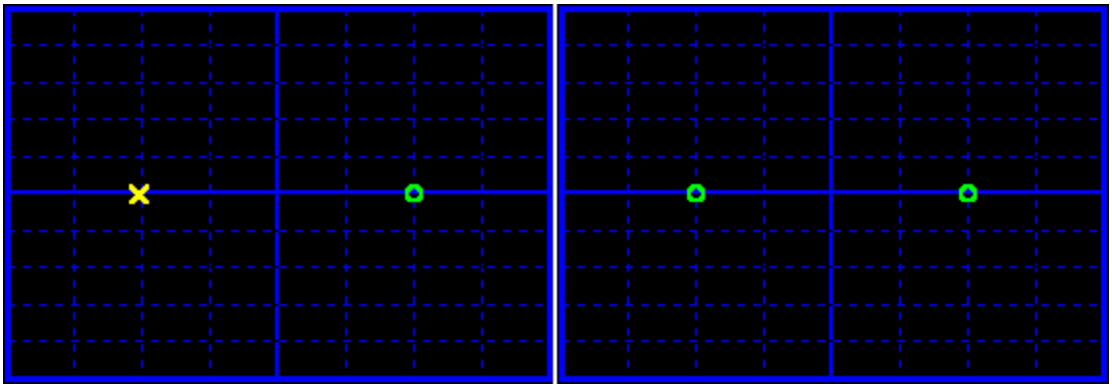
To watch the group delay of the zeros filter update in real time as you move the zeros all pass poles and zeros around, check the "RTU" box on the pole/zero plot control bar while the zeros frequency graph is displayed.

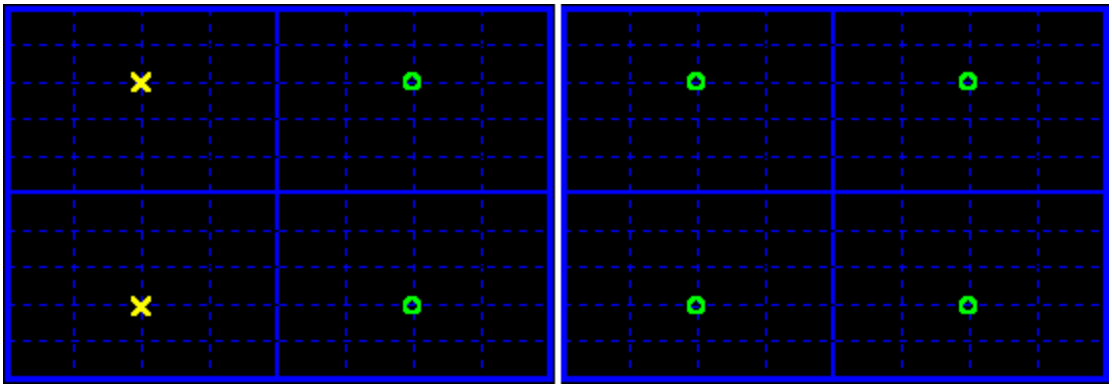


Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important: Off Axis zeros must be created using the **Add Zeros** table. Checking the **RTU** and the **Restore Passband** checkboxes in the pole/zero plots permits the off - axis zeros to be manually slid around while the poles are recalculated to restore the required pass band response.

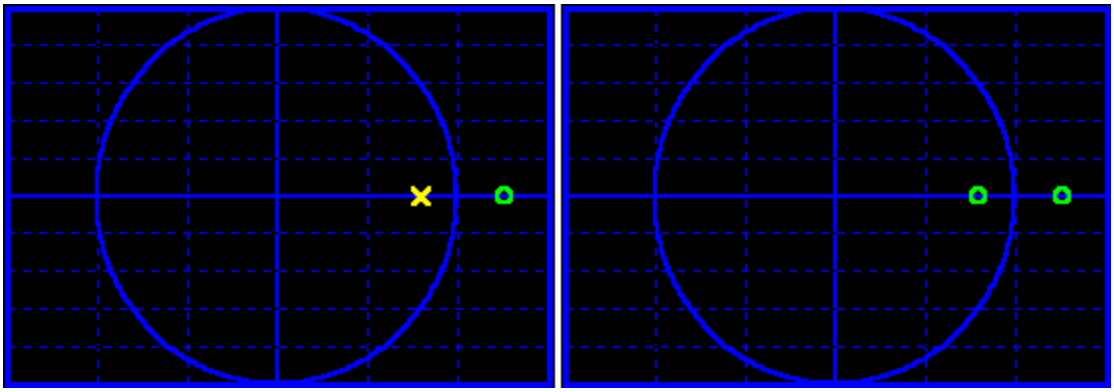
For analog filters, the pole zero plot of real and complex all pass functions and off-axis zeros are as shown below:

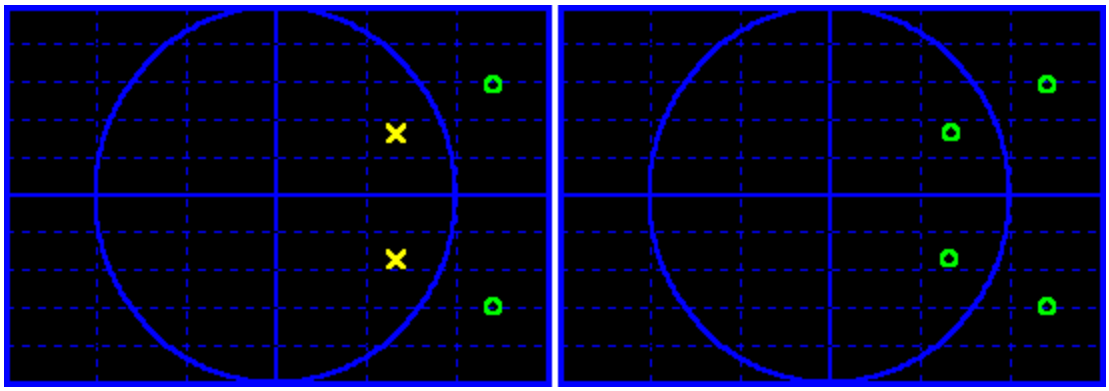




Digital real and complex all pass functions and off-axis zeros are shown below:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





All Pass Pole Zero Placement

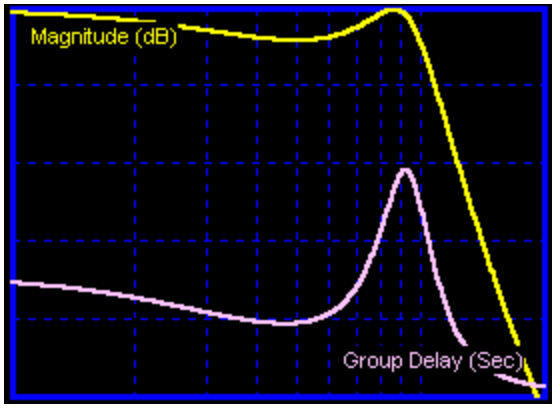
A stable all pass always has the pole to the left of the JW axis in analog filters, and inside the unit circle in discrete filters. To manually create a delay equalizer, experimentally slide the pole around until the desired effect is achieved on the pass band group delay. Check the RTU box on top of the pole/zero to watch the group delay update in real time.

Example:

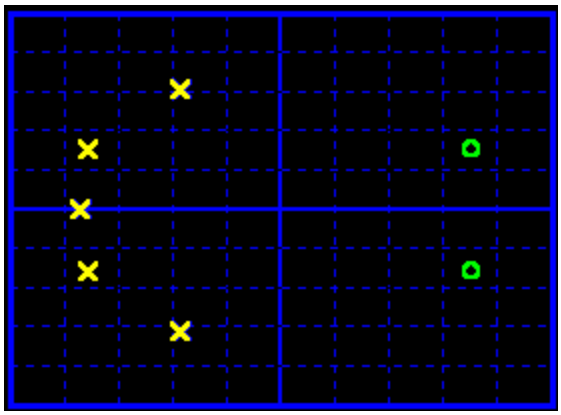
A third order Chebyshev I filter with 2 dB of ripple in the pass band requires some delay equalization.

The basic frequency response :

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

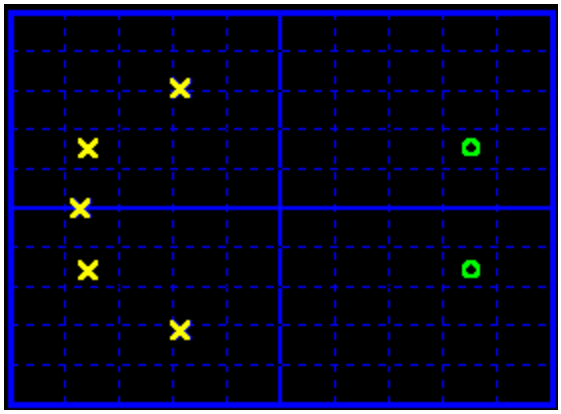


A single second order all pass function is added to the pole zero plot:

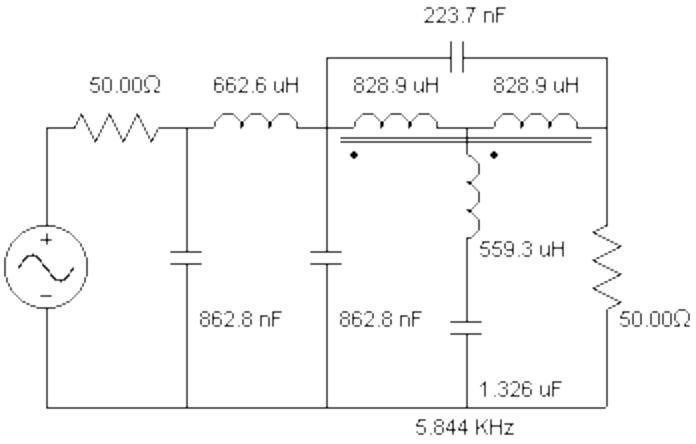


The new frequency response has an identical frequency magnitude response, and a more equalized group delay response as shown:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



A new passive circuit with a cutoff frequency of 10KHz and 50 Ohms source and load includes an all pass section realized with coupled coils:

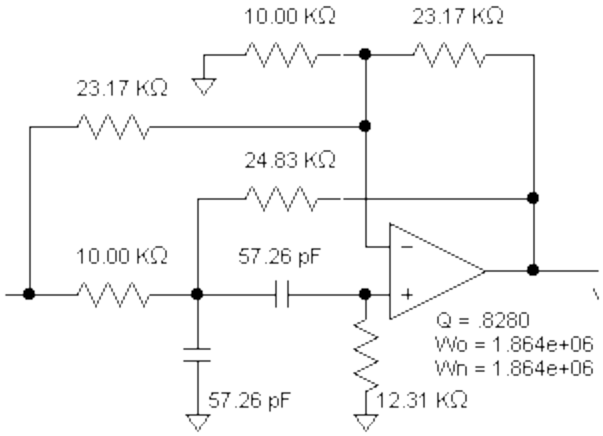


The coupled coils are not necessary if a balanced circuit is constructed. See the Passive Filter section of this Help file for translations from unbalanced to balanced circuits.

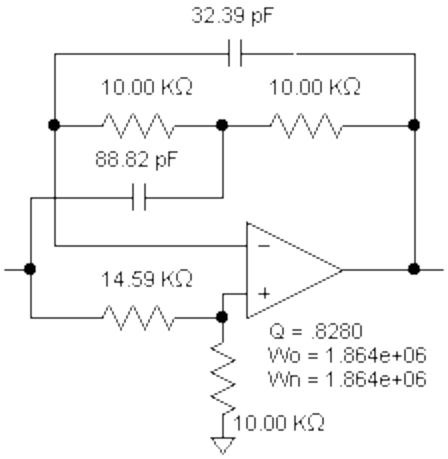
Active circuits may have the all pass section cascaded as a positive SAB, negative SAB, Thomas biquad, Akerberg-Mossberg biquad, or QIC biquad. The positive and negative SABs and QIC biquads are shown below.

Postive ASAB All Pass:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

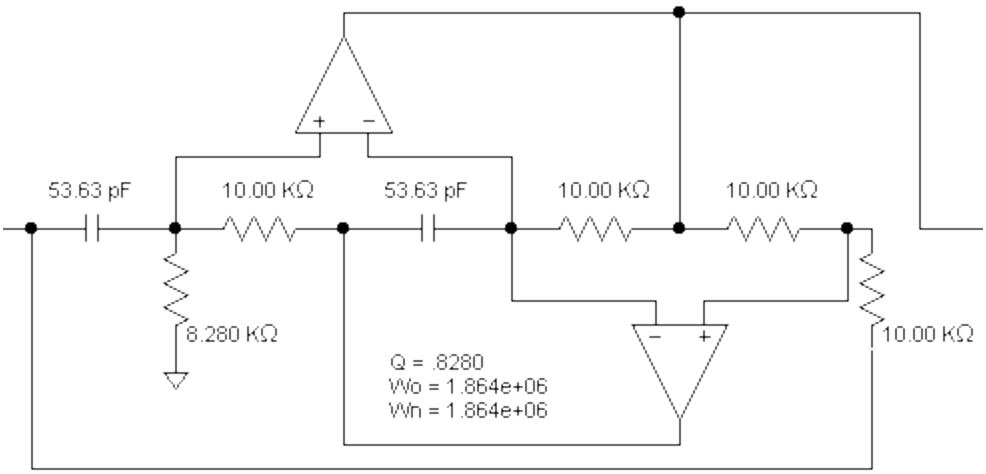


Negative SAB All Pass:



QIC All Pass:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Automatic Pole Zero Placement

If the filter is a low pass or band pass, and is a Butterworth, Chebyshev I, Chebyshev II, Hourglass, or Elliptic, then all pass poles and zeros may be automatically positioned by checking the **Delay Equalize** box on the FilterSolutions Main control panel. Enter the desired order of the delay equalizer in the adjacent text box, and create the filter. If **Passband** is checked, FilterSolutions will attempt to create a group delay with a minimum possible standard deviation within the pass band. To delay equalize out to a specified attenuation, uncheck **Passband** and enter the desired attenuation to equalize out to.

The ideal frequency graph of a filter created with an automatic delay equalizer will display the standard deviation of the group delay within the pass band in the upper left corner. This displayed value is updated in real time when poles and zeros are manipulated and the **RTU** box is checked.

For large filters, it is often possible to reduce the group delay standard deviation more by manually adjusting the all pass poles and zeros after performing an automatic equalization.

Band Pass Filter Delay Equalizers

Band pass delay equalization creates a special problem because the delay equalizer all pass sections do not necessarily have a prototype. Medium width band pass filters require equalizers that do not have a prototype, and narrow band filters require an equalizer that is most efficiently constructed using a prototype. For bandwidth to pass band ratios above 0.20, FilterSolutions uses a nonprototype equalization solution, and for ratios of 0.20 or less, a prototype solution is used.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Digital and Transmission Line Equalizers

Due to the inherent warping of the frequency axis, filters that have been successfully equalized for the ideal case may still exhibit undesired group delays when implemented as digital or transmission line filters. When this happens, it is necessary to manually equalize the digital or transmission line filter using real time pole/zero manipulation.

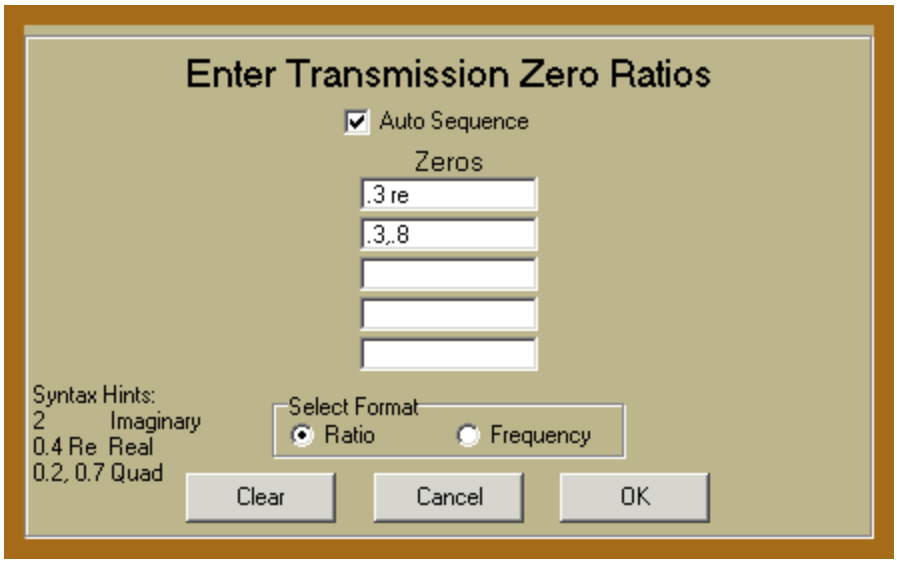
Off Axis Quadruplet and Dual Zeros

Off axis real and quadruplet zeros are supported for **Passive, Active, Sw Cap, and Digital Filters.**

Phase angle and group delay may be altered by the presence of dual and quadruplet off axis zeros. Unlike all pass stages, the mere addition of dual and quadruplet off-axis zeros also effects the pass band magnitude response, so additional calculations are needed to adjust the pole locations as needed to restore the pass band. However, it is generally more efficient to use quadruplet zeros in LC passive filters in that there are no canceling poles and zeros that are inherent in LC all pass stages, and may be more efficient in active and digital filters, depending upon the exact filter design requirements.

Seeding The Off-Axis Zeros

Before off-axis zeros may be manually moved around to adjust the group delay, they must be installed. Use the "Add Tx Zeros" button to enter seed zeros:



Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Band Pass Filters

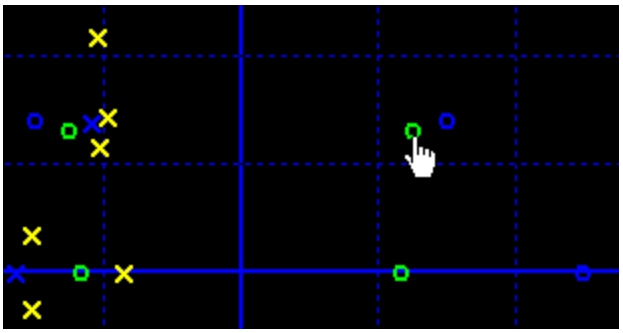
Band pass filters require either asymmetric or prototype equalization. For medium and wide band filters, asymmetric usually works best. Check **Asymmetric** in the main control panel prior to synthesizing the filter. Narrow band equalization usually works best with prototypes and even order cross coupled resonators. For narrow band filters, after synthesizing an **EVEN ORDER** band pass filter without the use of the **Asymmetric** selection, check the **Proto** switch in the upper right of the pole/zero plot. For lumped filters, select **X Coupled**, **Tubular**, or **Coupled Resonator** in the circuit schematic, to synthesize a cross coupled resonator topology. For active filters, select the **GIC Ladder** topology to synthesize delay equalizing bridged resistors.

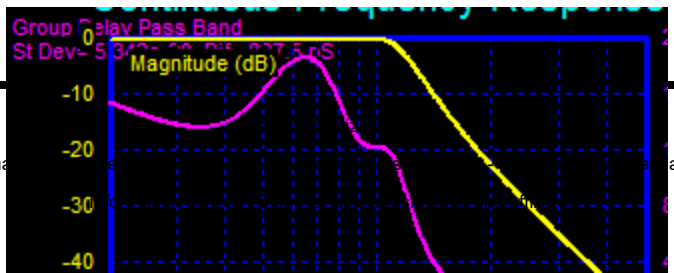
> **Note:** Cross coupled resonator band pass filters require an even order. Therefore, narrow band off-axis zeros equalization should always be performed with an even order selection.

Adjusting the Zero Locations

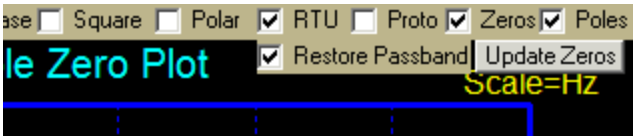
After checking the **RTU** box on top of the pole/zero plot, check the **Restore Passband** checkbox to watch the poles update to restore the pass band and the group delay update in real time. Check **Proto** for narrow band pass filters. (Medium and wide band pass filters should be equalized in Asymmetric synthesis mode.

Manually slide the around while a frequency trace updates automatically:

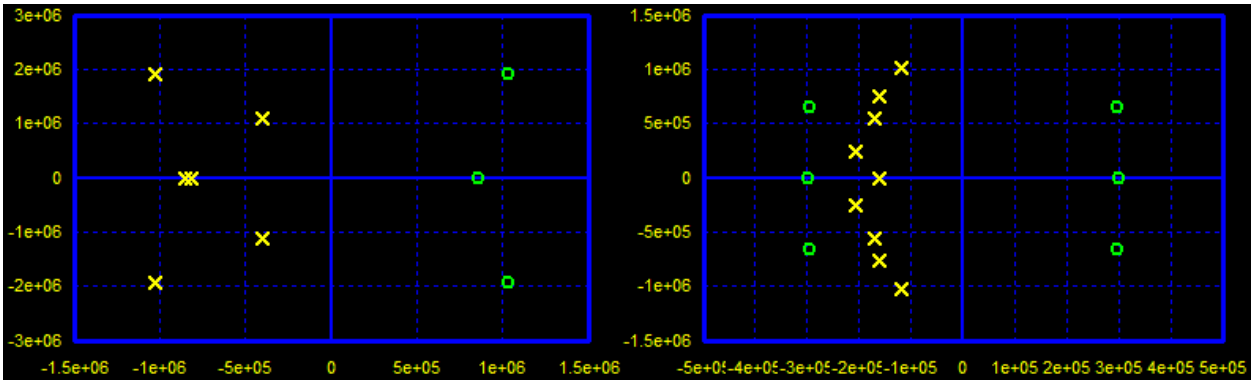




Once a desired frequency response is obtained, click the **Update Zeros** button to the right of **Restore Passband** to permanently update the pole locations, including zero values in the zero entry table:



Below, Figure 1 shows the traditional all pass pole/zero placement for equalizing a 3rd order, .2dB ripple Chebyshev on the left, and an equivalent 60 dB/decades of attenuation Chebyshev equalized with a dual zeros pair and a Quadruplet zero set. Note the additional zeros and poles on the Quadruplet equalized filter.

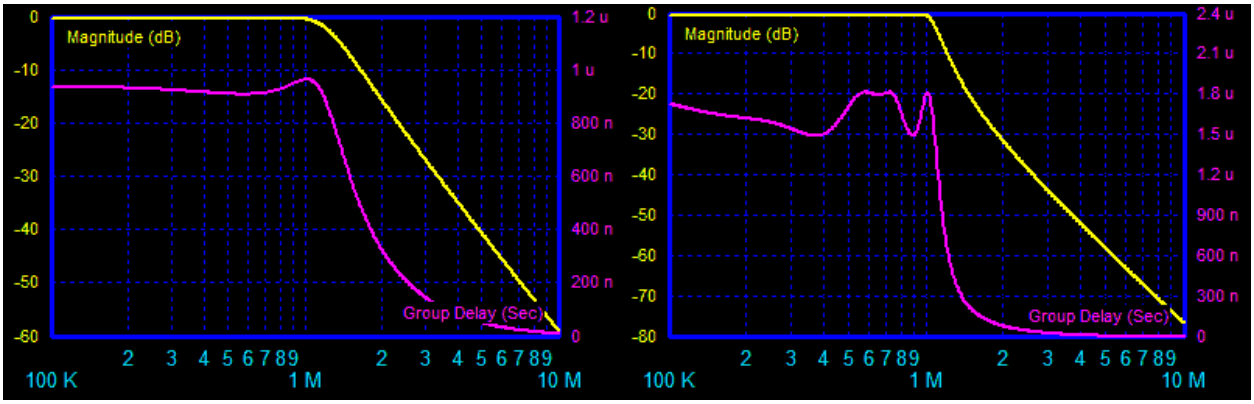


All Pass Quadruplet and Dual Zeros

Figure 1: Equalization Pole/Zero Placement

Below, Figure 2 shows the respective frequency responses. Both filters attenuate ate 60 dB per decade, due to the presence of three more poles than zeros. However, the Quadruplet equalized filter exhibits a significantly improved pass band response, a faster attenuation near the cutoff frequency of 1 MHz, and only a slight decay in equalization quality. This faster attenuation may result in a smaller order filter meeting a specified filter stop band attenuation requirement.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



All Pass Quadruplet and Dual Zeros

Figure 2: Equalization Frequency Response

Synthesizing The Filter

Figure 3A and 3B below shows the respective all pass and Quadruplet zero equalized passive LC filter. Note that both filters require the same number of inductors, and the Quadruplet zeros equalized filter requires only two more capacitors, and one less coupled coil. Generally speaking, the two LC implementations are somewhat comparable in complexity, but the Quadruplet zero equalized filter exhibits superior performance.

Real and quadruplet zeros are also supported in passive cross coupled resonator filters.

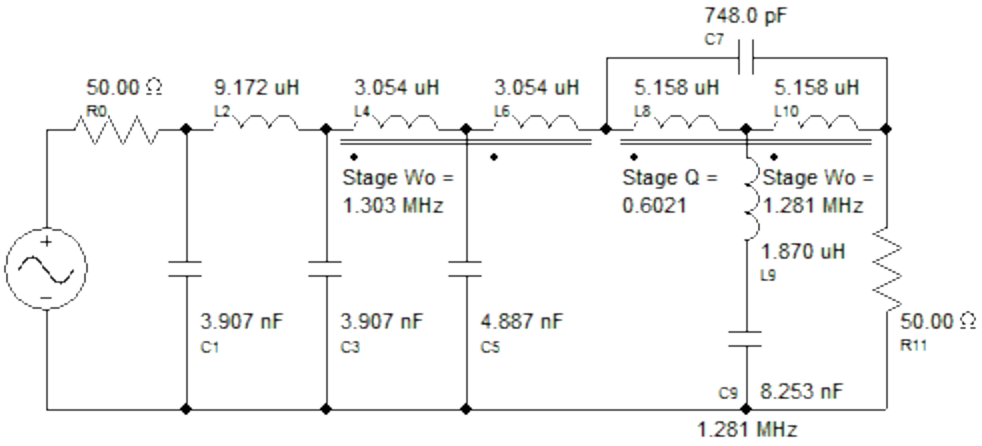


Figure 3A: All Pass Equalized LC filter

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

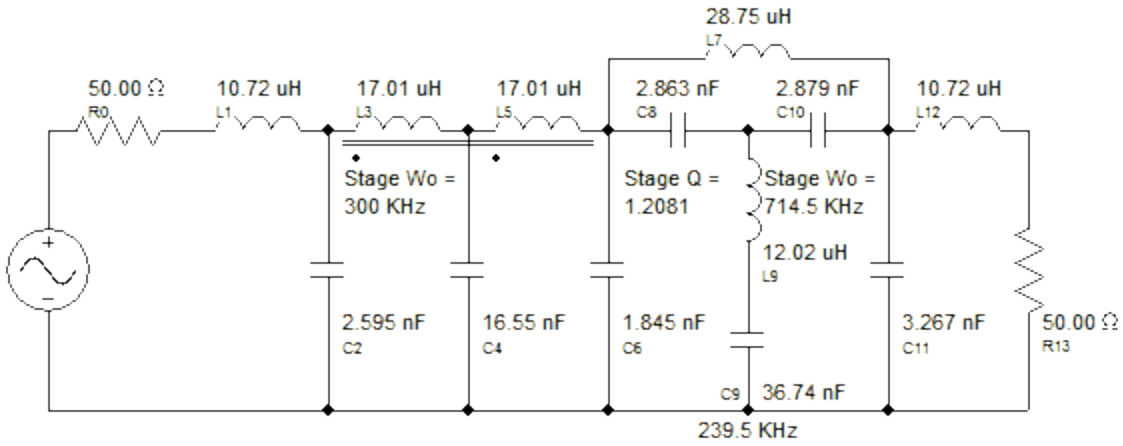


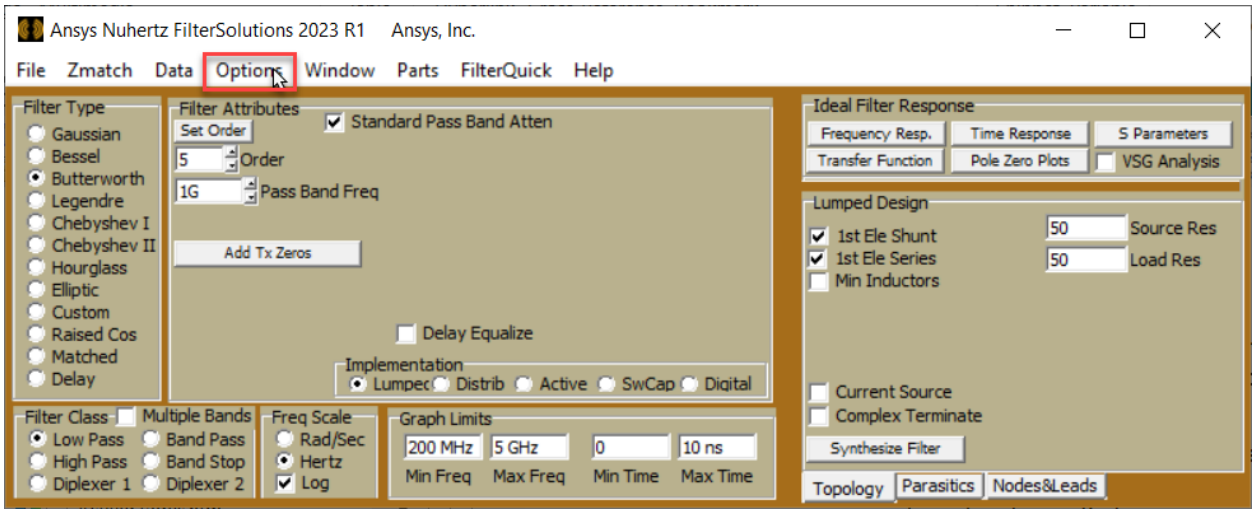
Figure 3B: Quadruplet and Dual Zeros Equalized LC filter

Group Delay and Phase Deviation

The quality of linear phase needed to minimize the distortion may be measured by Group Delay or Phase Deviation.

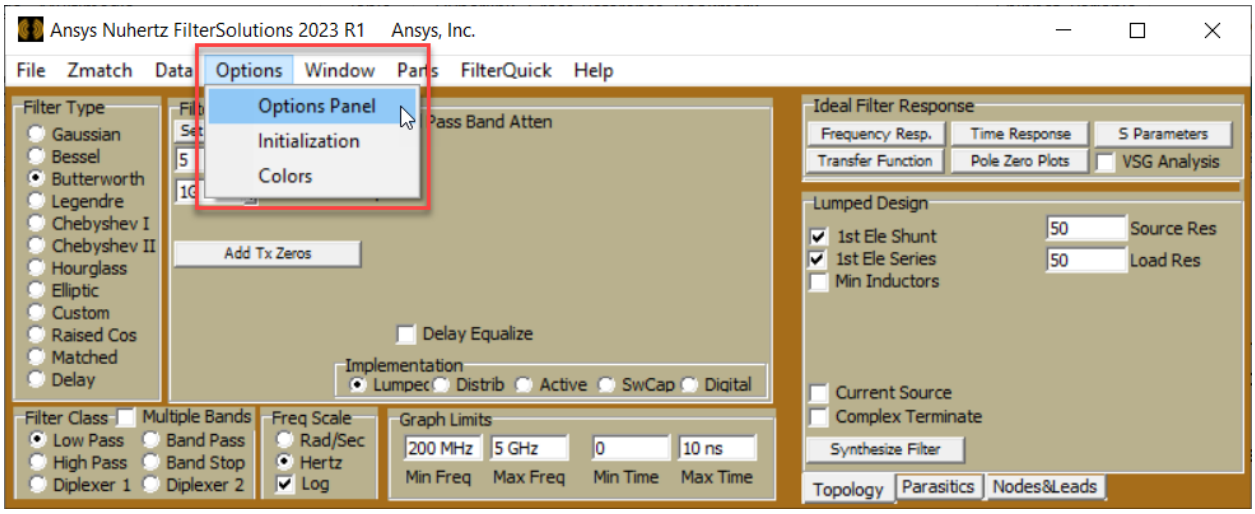
To select between the two options, for the frequency response graphic:

1. On the ribbon, select Options.



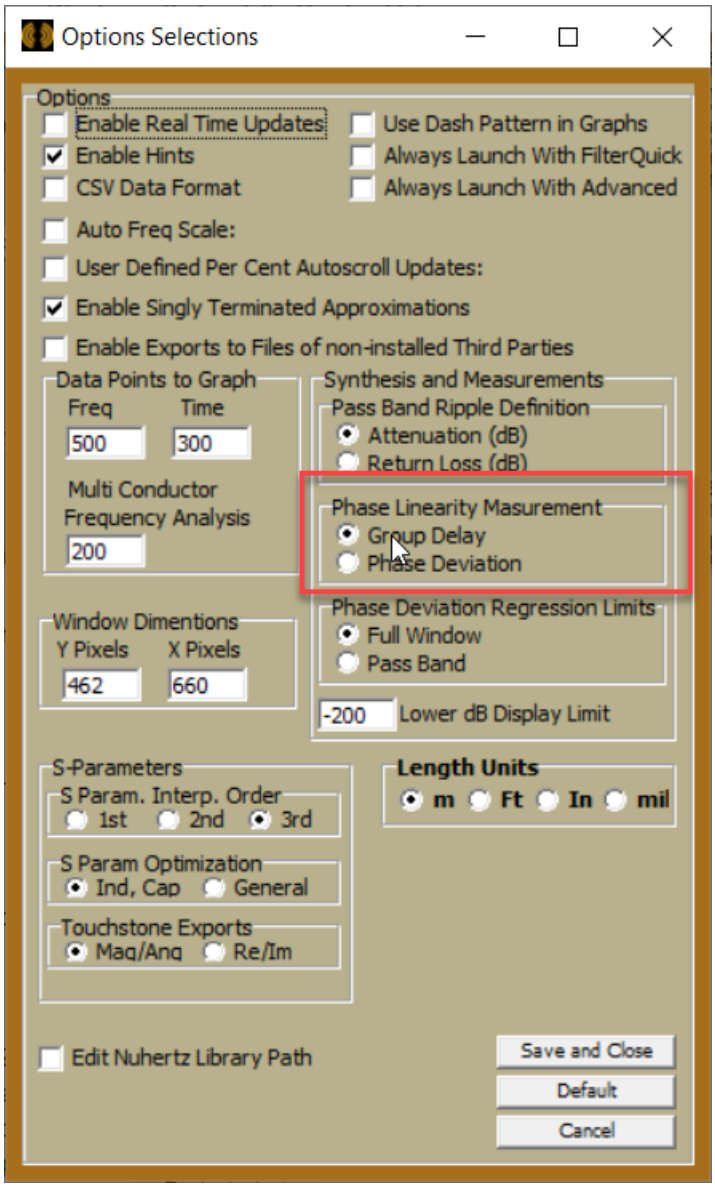
2. Select Options Panel

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Select between Group Delay and Phase Deviation in the Phase Linearity Measurement group box.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Group Delay

Group delay is simply the negative of the derivative of the phase angle in radian with respect to frequency in radians per second, and is a very simple and straightforward method to measure linear phase, with a purely horizontally flat group delay being ideal. Tuning and optimizing functions may be used to force a linear phase by forcing the group delay to be flat. The value of the group delay may vary greatly, so optimizers must be set to a reasonable value and force the desired frequency range to that value or within a range of values.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Phase Deviation

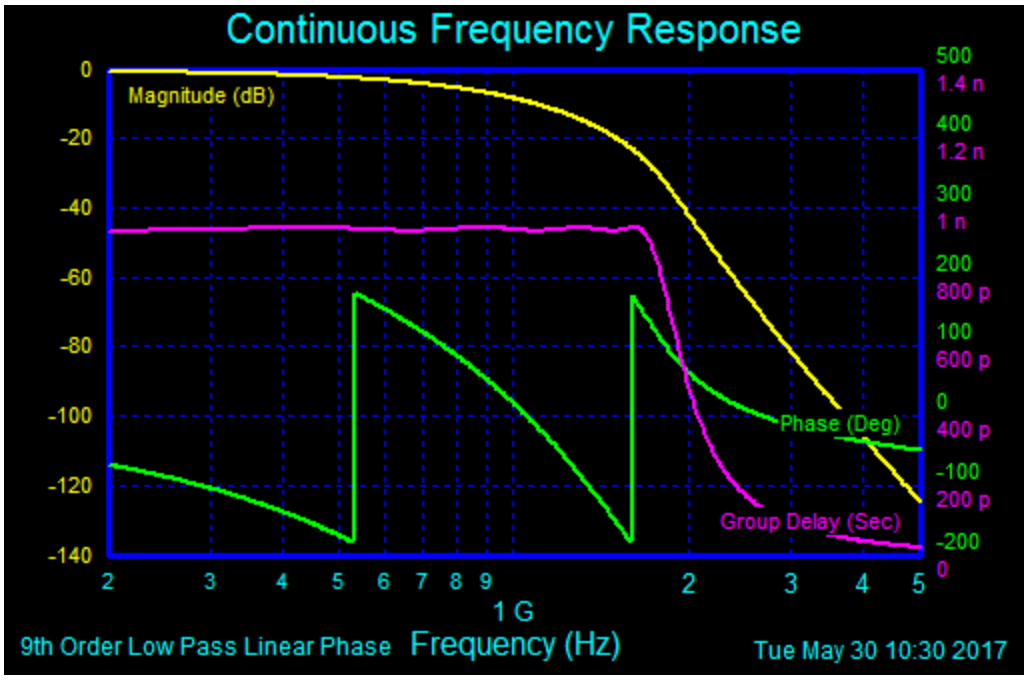
Phase deviation, or deviation from linear phase, is the difference between the linear regression of the unwrapped phase angle and the actual value of the unwrapped phase angle, providing a direct means to view the quality of the linear phase. The ideal value for linear phase is absolute zero across the desired frequency range. This has an advantage over group delay in that tuners and optimizers have a fixed target of zero to force the phase deviation, instead of an unknown or estimated flat value. The disadvantage is the linear regression must be limited to the frequency range where phase is desired to be linear, usually the passband, which somewhat complicates analysis and optimization.

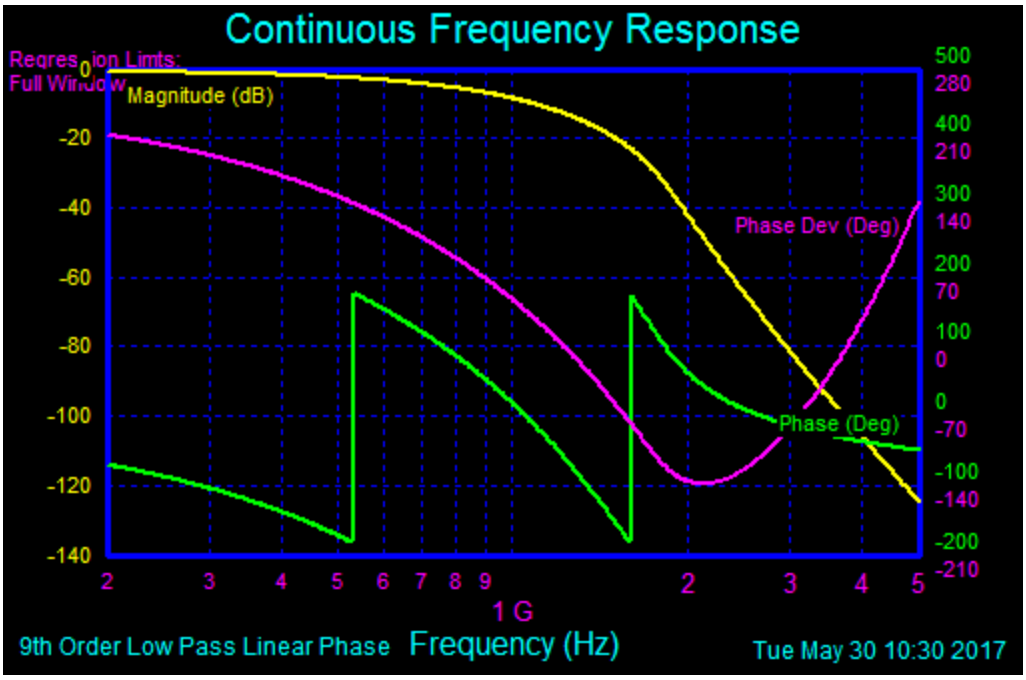
Phase Deviation Regression

In the Options Selection menu (Step 3), the Phase deviation option limits are selectable, full graphical window being the default. If it is desired to limit the frequency range to the frequencies desired to be linear, this default selection works. If a broader range of frequencies is desired to be viewed, then it may be necessary to limit the linear regression to just the passband. When setting limits to just the passband it is usually desirable to also set the phase deviation measurement limits using the "Limits" selection in the frequency response graphical window to zero in on the phase deviation for the desired frequencies and exclude phase deviation elsewhere.

The frequency response of a 9 pole Linear Phase filter with 1% group delay ripple is shown below in Figures 1A, 1B, 1C, and 1D with the various described methods of viewing the quality of linear phase.

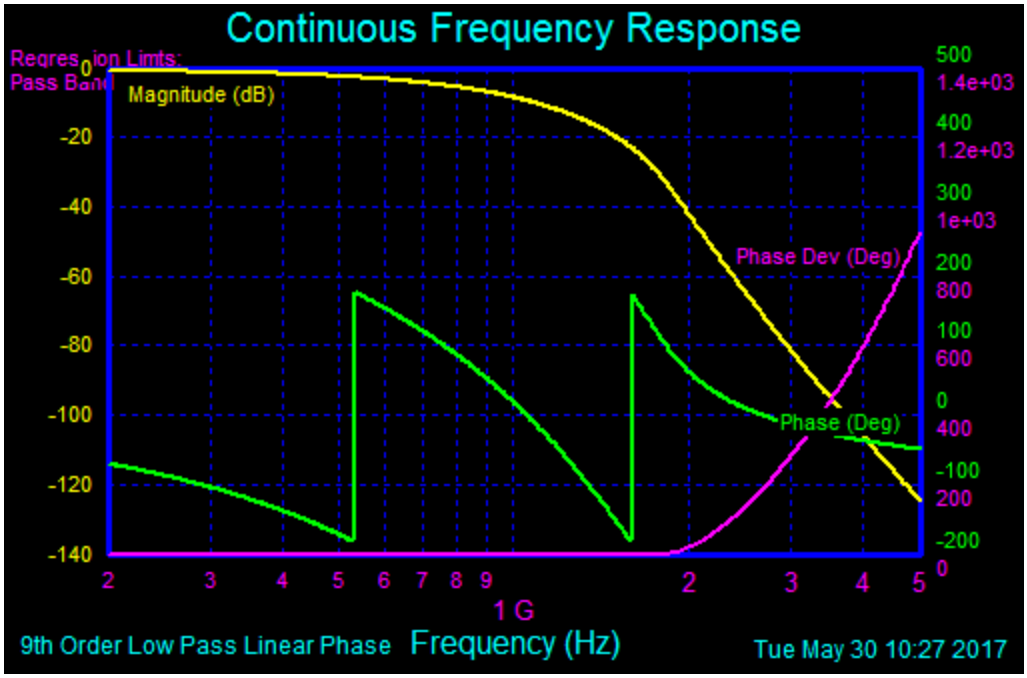
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

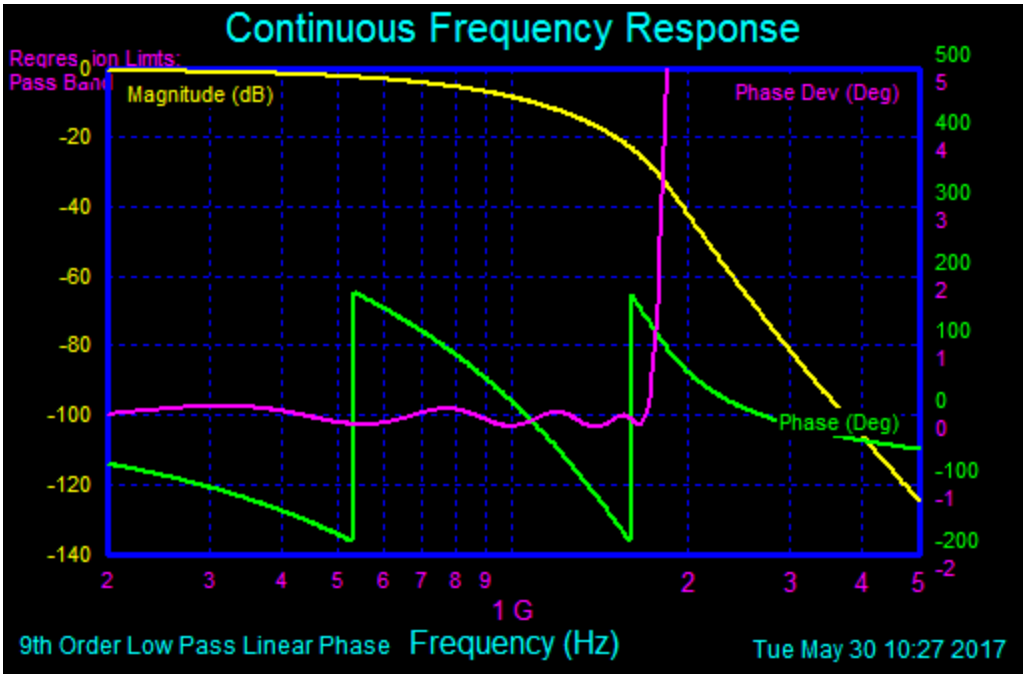




Figures 1A and 1B: Group Delay and Phase Deviation, Full Window

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





Figures 1C and 1D: Phase Deviation, Pass Band Regression, Default and Phase Deviation, Pass Band Regression With Adjusted Vertical Limits

Figures 1A - 1D: Various Forms of Linear Phase Measurements

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Custom Regression Limits

If it is desired to limit the phase regression to something other than the pass band or graphical window, Advanced Panel frequency response windows permit the selection of customized regression limits by selecting the "Set Limits" button the upper right, as shown in Figure 2.

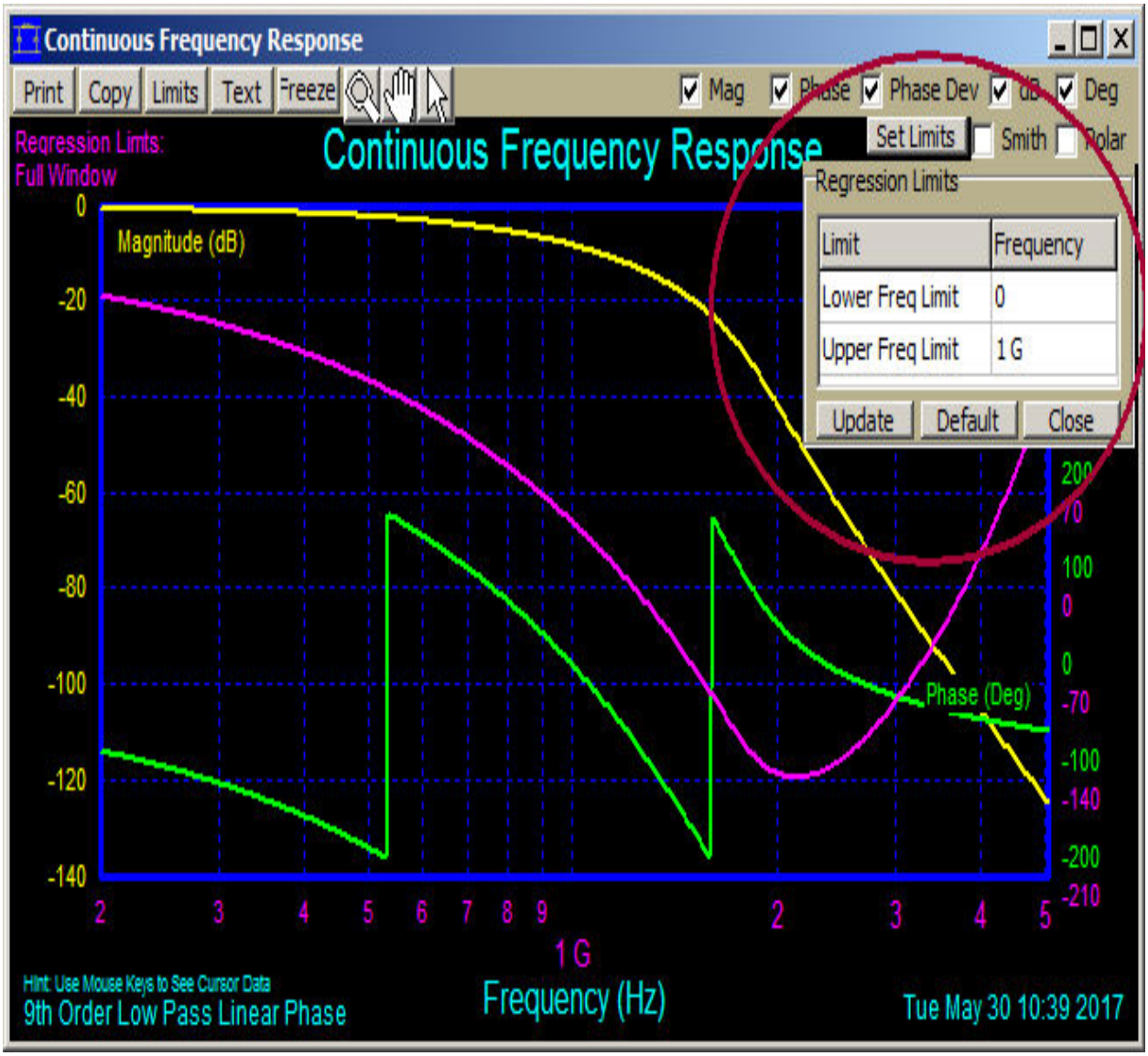


Figure 2: Setting Custom Regression Limits for Phase Deviation Measurements.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Bridged Element Delay Equalization Video

See the Online Help to view the Bridged Element Delay Equalization video.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7 - Arithmetic Frequency Symmetry

This chapter contains the following topics:

- Arithmetic Symmetry Definition and Purpose
- Arithmetic Group Delay Equalization
- Arithmetic Bessel and Gaussian Wide Band Equalization
- Asymmetric Improvements

Arithmetic Symmetry Definition and Purpose

Arithmetic symmetry design may be found in the Advanced Design panel.

The primary purpose of arithmetic frequency symmetry is to provide improved group delay response in wide band pass filters. Classical filter design provides geometric frequency symmetry, which in turn causes undesirable deviations in the group delay across the pass band. Given the group delay computational definition to be the derivative of the phase angle with respect to frequency, it is inherently intuitive that an arithmetically symmetrical response would provide a more desirable group delay response than a classical geometrically symmetrical response would.

Fortunately, an arithmetically symmetrical model exists with transmission line filters. Transmission line elements inherently warp the frequency response into repeating cycles of arithmetically symmetrical sections, each with an improved localized group delay response. Therefore, if the frequency response can be warped in a similar manner to what is seen in a transmission line response, an improved group delay response would be expected. But, since it is not practical or possible to implement the response with lumped elements, the arithmetically symmetrical response must be limited and approximated over the desired frequency range, namely zero to twice the center frequency. Direct manipulation of the poles and zeros of the filter design transfer function or characteristic functions with transmission line equations accomplish said objective.

Gaussian Example

Although Gaussian filters are normally used to shape square pulses, they also make convenient band pass filters if the rounding of the top meets design requirements. The classical geometric response of a 60% bandwidth filter is shown below in Figure 1, where it may be seen that the group delay is nowhere near flat. Figure 2 shows the same filter after arithmetic symmetry warping, and with a substantially improved group delay response. Figure 3 is the same filter again, but plotted in a linear frequency scale where the arithmetic symmetry of the round top is easily seen.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

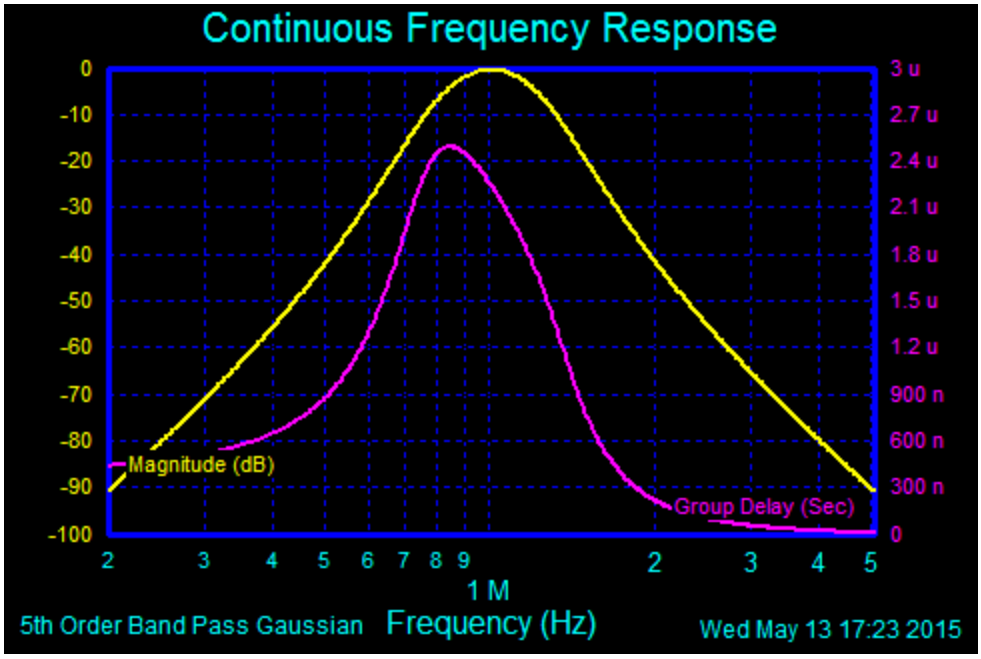


Figure 1: Classical Geometric Gaussian Band Pass response

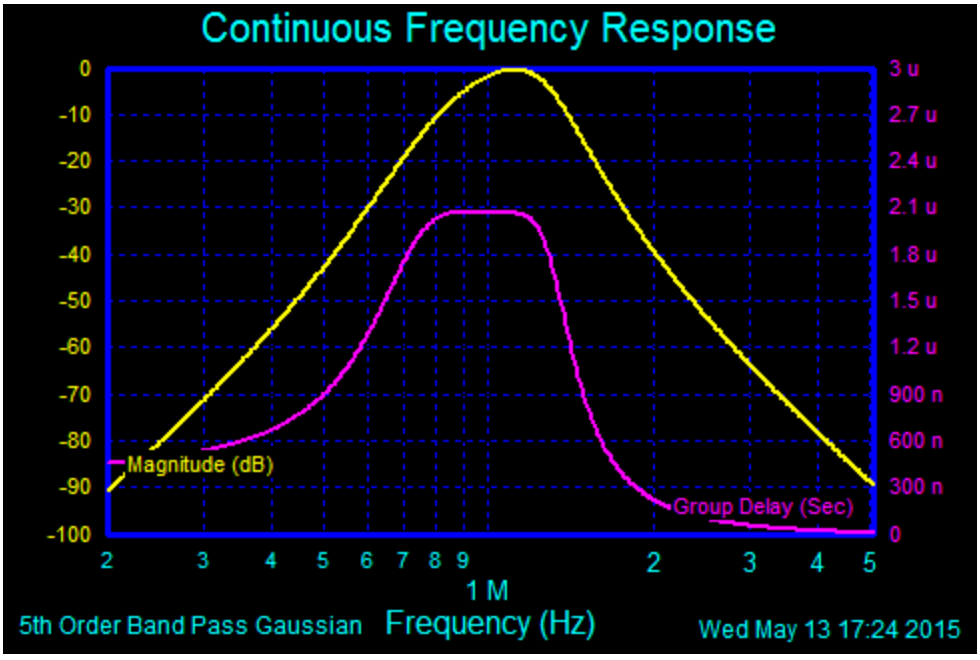


Figure 2: Arithmetic Approximate Gaussian Band Pass response

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

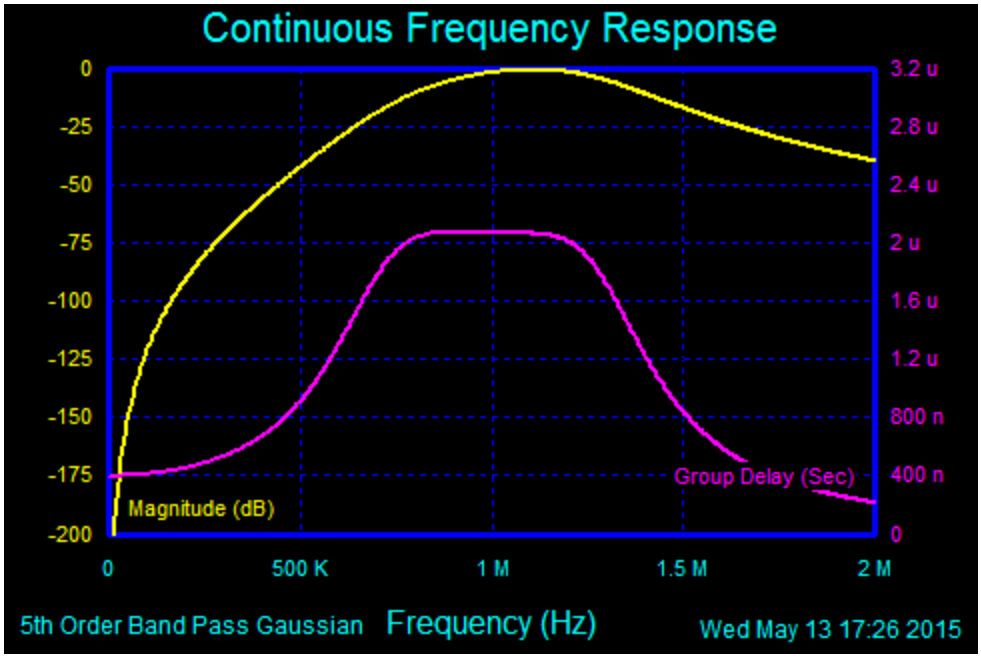


Figure 3: Arithmetic Gaussian Band Pass Response on a Linear Scale

Arithmetic Group Delay Equalization

Although Gaussian and Bessel filters produce nice group delay responses when synthesized with arithmetic symmetry, most applications require a flat pass band. This, in turn, has the effect of creating undesirable "rabbit ears" in the group delay across the pass band. Classical geometrically symmetric design further compound the problem by creating asymmetric rabbit ears. An arithmetically symmetric design keeps the group delay more symmetric across the pass band, and subsequently easier to flatten with delay equalizers.

Figure 1 shows a classical geometrically symmetric Hourglass repose with undesirable asymmetric group delay. Figure 2 shows the same filter designed with arithmetic symmetry, and a more desirable symmetric group delay. Figure 3 shows the arithmetically symmetric filter delay equalized without asymmetric rabbit ear effect.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

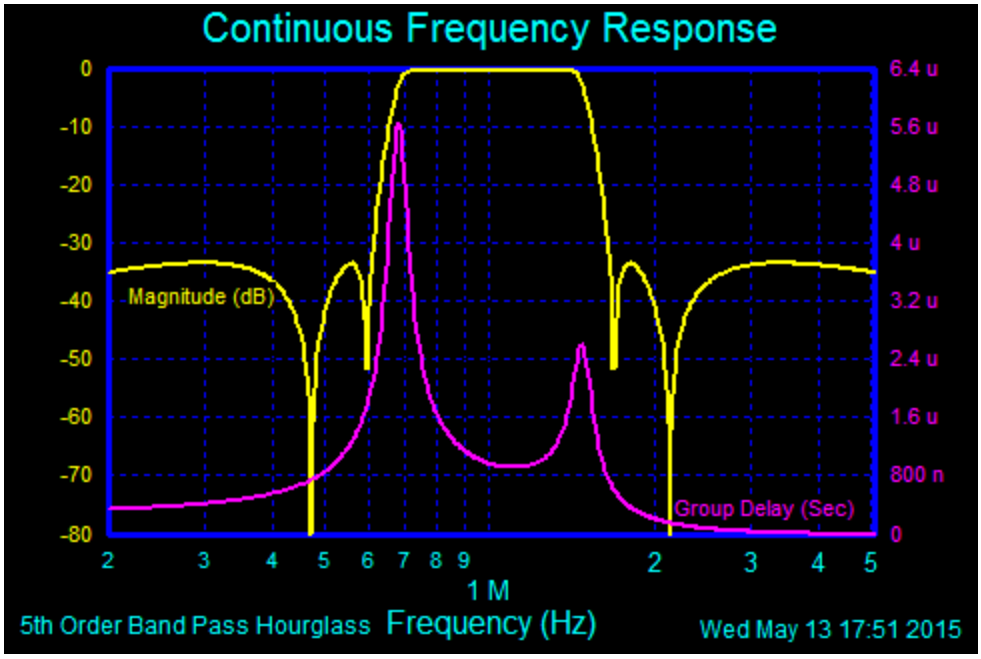


Figure 1: Asymmetric Group Delay Distortion in a Geometric Hourglass Filter

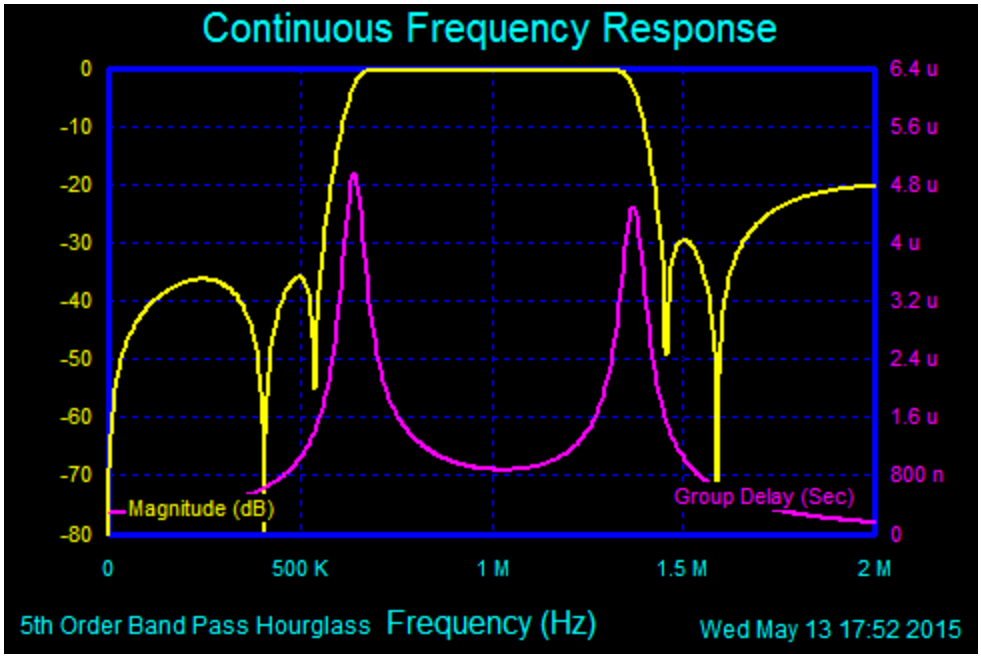


Figure 2: Approximate Arithmetically Symmetric Group Delay in a Arithmetically Symmetric Hourglass

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

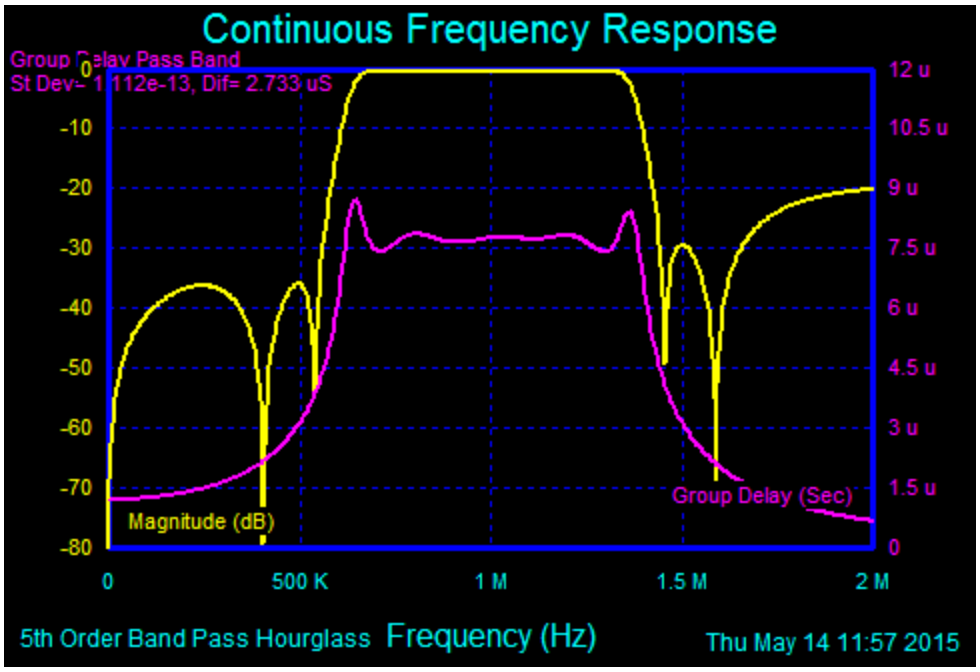
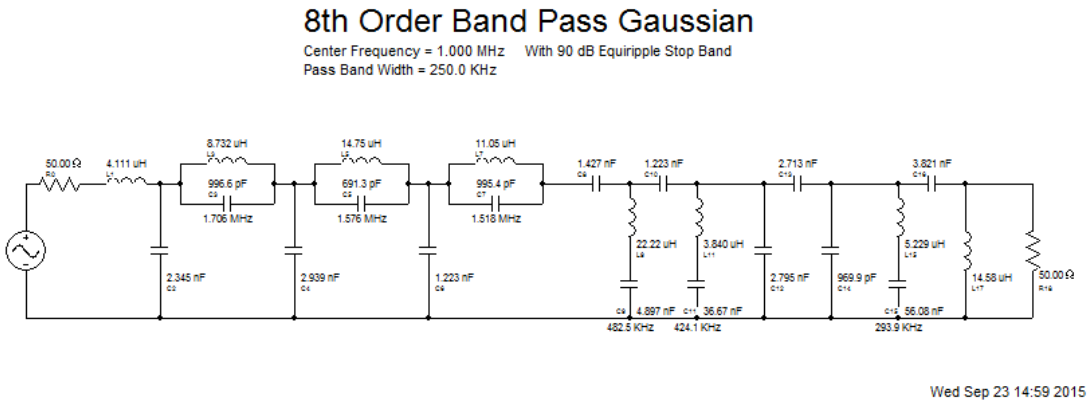


Figure 3: Delay Equalized Hourglass With Arithmetically Symmetrical Design

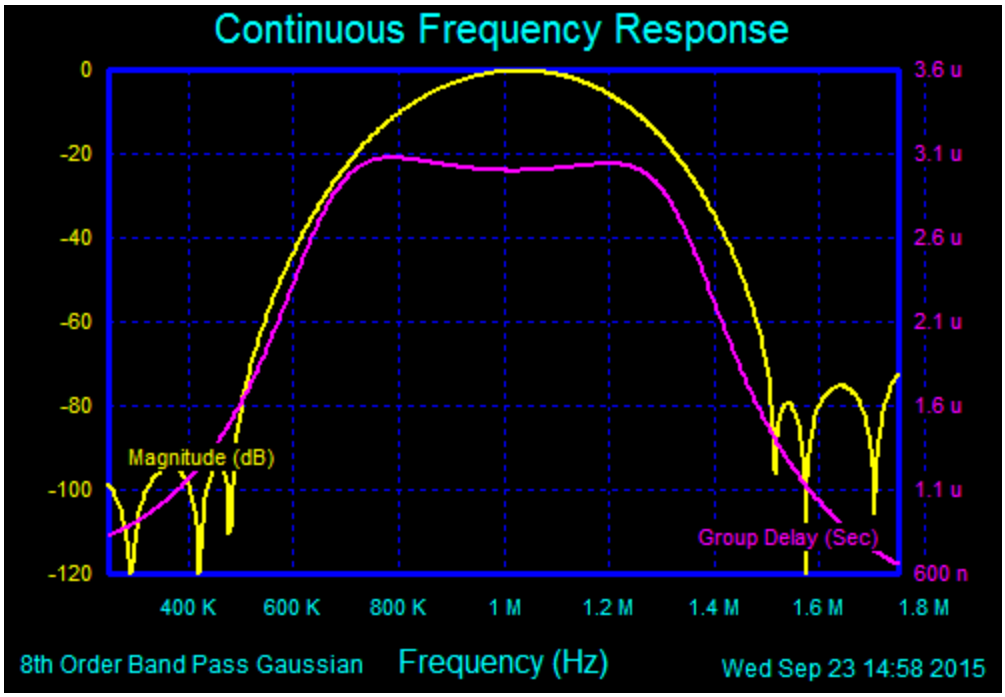
Arithmetic Bessel and Gaussian Wide Band Equalization

Bessel and Gaussian filters may be synthesized with a stop band and arithmetic symmetry to produce a wide pass band with equalized group delay characteristics, as shown below.



Gaussian Wide Band with High/Low Topology

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Frequency Response With Flat Group Delay of Above Gaussian Wide Band

Asymmetric Improvements

One of the drawback of arithmetic symmetry is the undesired warping of the frequency response toward the upper end of the arithmetic section, potentially requiring a larger filter to effectively suppress the higher frequencies. Hipower Filter Designer has the capability of mitigating this potential problem with asymmetric design capability. If more filtering is applied to the higher frequencies, then the undesired warping effects may be minimized.

Figure 1 shows an 80% bandwidth, five pole Elliptic filters design to attenuate 60dB in the stop bands. It may be seen that the lower stop band is fine, but the upper stop band is warped and fails to attenuate properly. Figure 2 applies asymmetric design requirements, five pole and 60 dB attenuation for the lower stop band, seven pole, 90 dB attenuation on the upper. This unwarps the upper stop band to a significant extent. Figure 3 shows the asymmetric design setup selection in the Advanced Design Panel.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

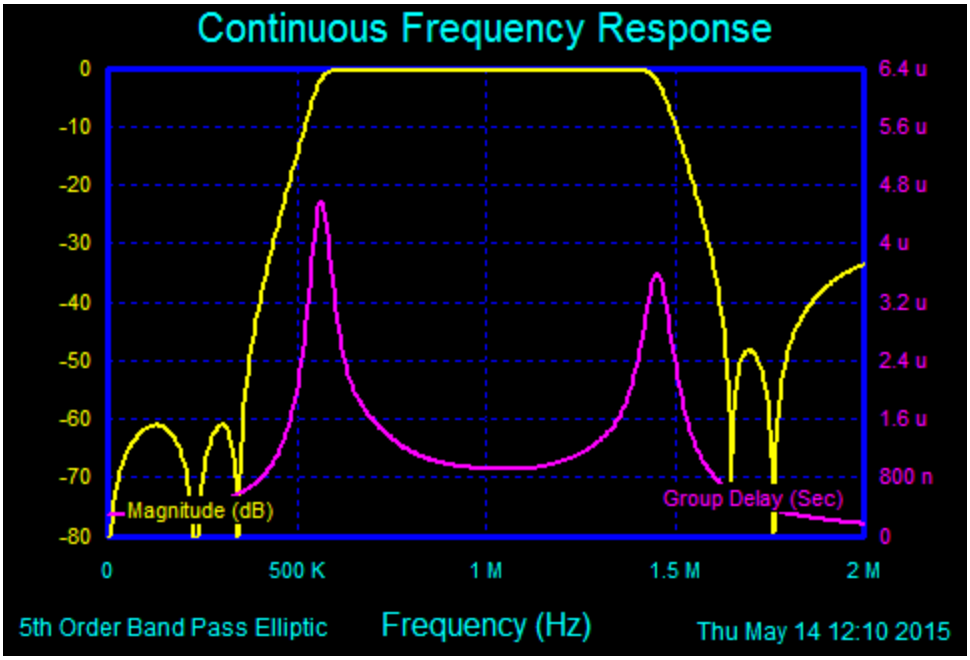


Figure 1: Arithmetic Elliptic Response with Warped Upper Stop Band.

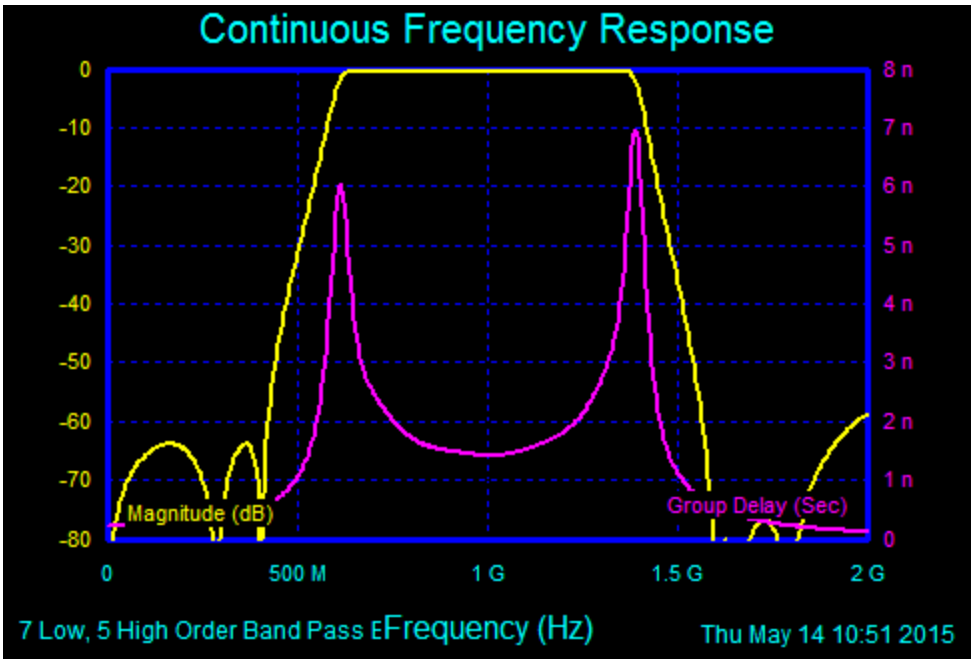


Figure 2: Arithmetic Response of Asymmetric Elliptic Function With Mitigating Effect of Upper Stop Band

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

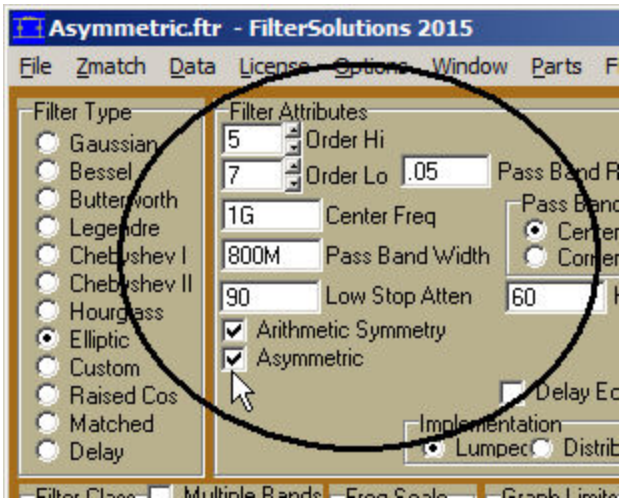


Figure 3: Asymmetric Design Setup in the Advanced Design Panel

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8 - User Data Analysis

This chapter contains the following topics:

- User Data Analysis
- User Data Files

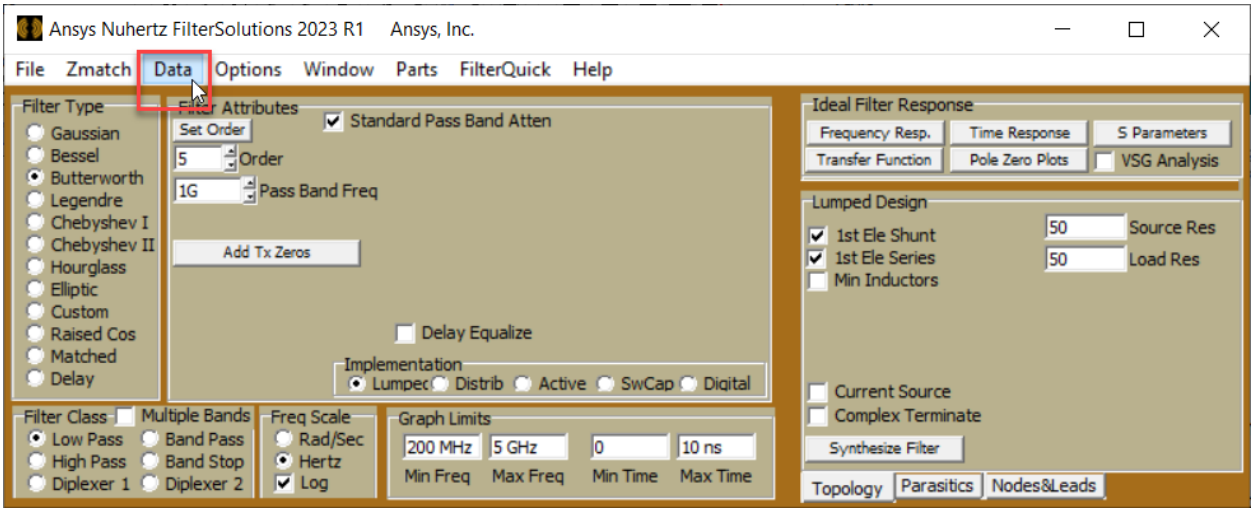
User Data Analysis

General

FilterSolutions allows data to be imported and filtered. The filters and raw data are displayed in the time analysis windows. See the section on *User Data Files* for information on formatting the data file.

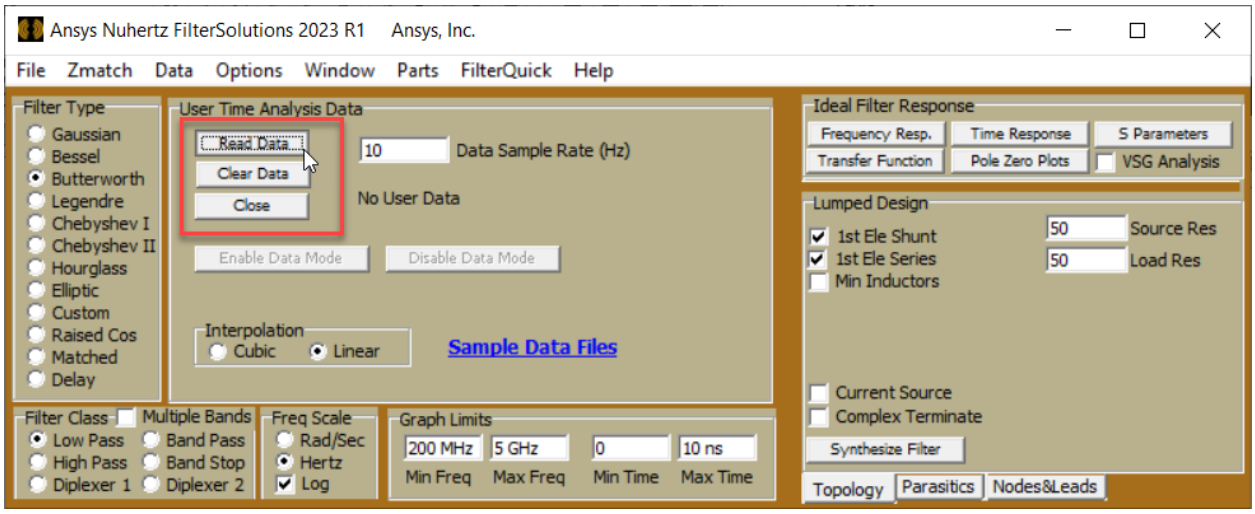
To import the file:

1. Select **Data** from the ribbon menu.



2. Select **Read Data**from the **User Time Analysis Data** group box.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Enter the rate in which the data was sampled in the text window.

User data mode may be enabled or disabled at the user's discretion once data is read in. All time analysis operations performed while user data is entered and into the system and enabled will be on that user data. To clear out the user data, select **Clear Data**. Up to 5,000 user data points may be entered.

Sampling

There are no restrictions on the sampling rate. The data points are splined together internally, so even if a few points are provided, adequate interpolation is provided. The spline may be cubic or liner, depending on the state of the "Interpolation" selection box in the User Data control panel.

Digital Filters

If a point by point filtering process is desired, you must match the filter sample rate to the data sample rate. Otherwise, the user data will be interpolated to obtain values at the digital filter sample points.

Output Data

Your filtered data, as well as the raw input data may be viewed both graphically and textually. Time analysis graphs will depict the input and output data as dependent variables instead of the usual step, ramp, and impulse time responses. Select the **Text** on the time analysis graphs to view and/or save filtered text data

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

User Data Files

The user data file only needs a string of numbers in a vertical column Each number in the file represents sampled data. After reading the file,

The following is an example a valid data file that is capable of being read:

- - - - - - - - - - - - - - - (Top of File)

4.232

-2.2

-1.32

-.02

4.1122

3.0922

-1.11

4.231

5.231

-3.2

- - - - - - - - - - - - - - - (Bottom of File)

The above set of numbers represent 9 sampled data points. The number of sampled points is echoed out just below the sample rate text entry box. The maximum number of points that may read is 5,000.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9 - Parts Libraries

This chapter contains the following topics:

- Parts Libraries
- Customizing Libraries
- User Library Directories
- Designing With Manufacturer's Parts Video

Parts Libraries

A limited set of inductor and capacitor S-Parameter and Spice model libraries, and opamp Spice file libraries and inductor and capacitor spice models ship with the product. The libraries are accessible by the patent pending process of clicking the "Menu" selection in schematic inductor and capacitor part selection drop-down boxes, "Menu" in the "Part" and "File" selection of the Standard Part Selection in the Change Control Panel (accessible by left-clicking an element), or the "Opamp Menu" button in an active filter netlist.

Schematic Drop-Down Selections

Lumped and active filter schematics have inductor and capacitor drop-down selection boxes in the upper right, as seen in Figure 1. Select, "Menu" to get the parts selection menu, as seen in Figure 2. Select the desired family of parts, and all of the like parts in the filter will be replaced by an S-Parameter or Spice model file that most closely matches the ideal part.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

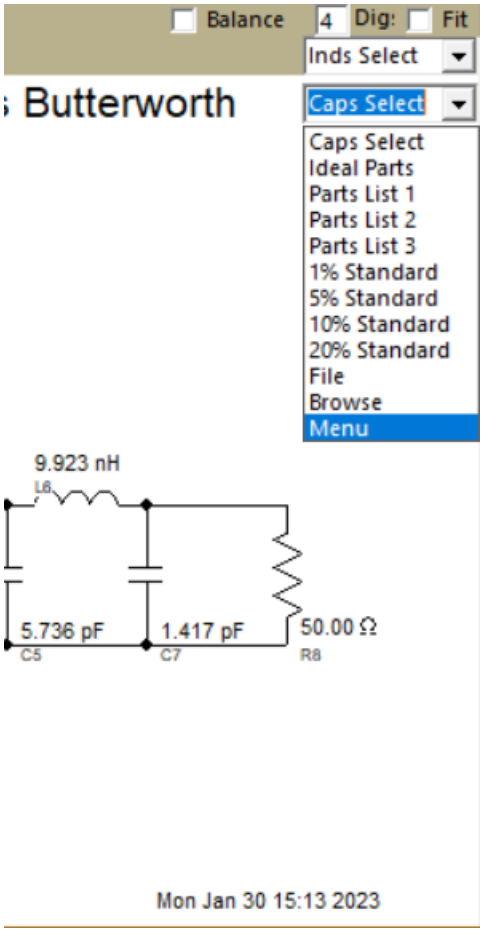


Figure 1: Schematic Parts Selection Drop-Down Boxes.

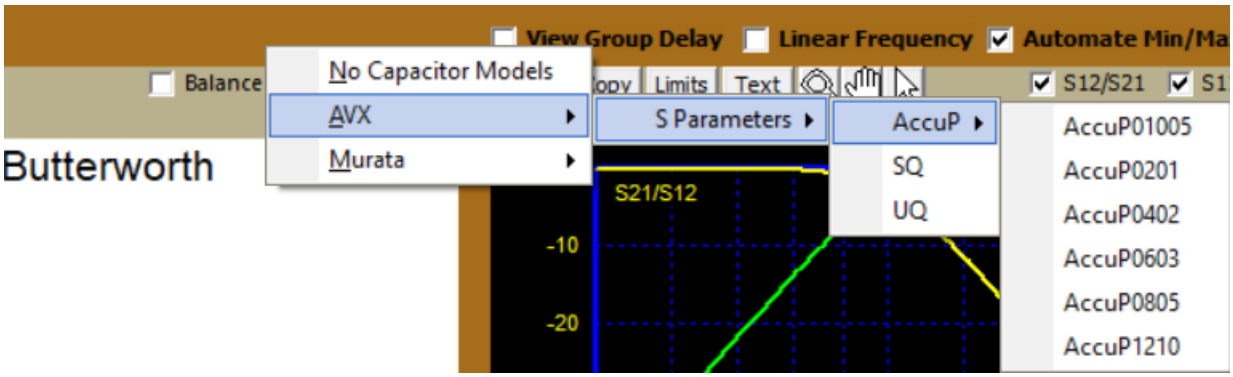


Figure 2: Part Selection Menus

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Change Control Panel Part Selection

If it is desired to set an individual part to the nearest S-Parameter or Spice model of a specific family, left click on that part to display the Change Control Panel. In the change Control Panel, select, "Part", "File", and "Menu", as seen in Figure 3.

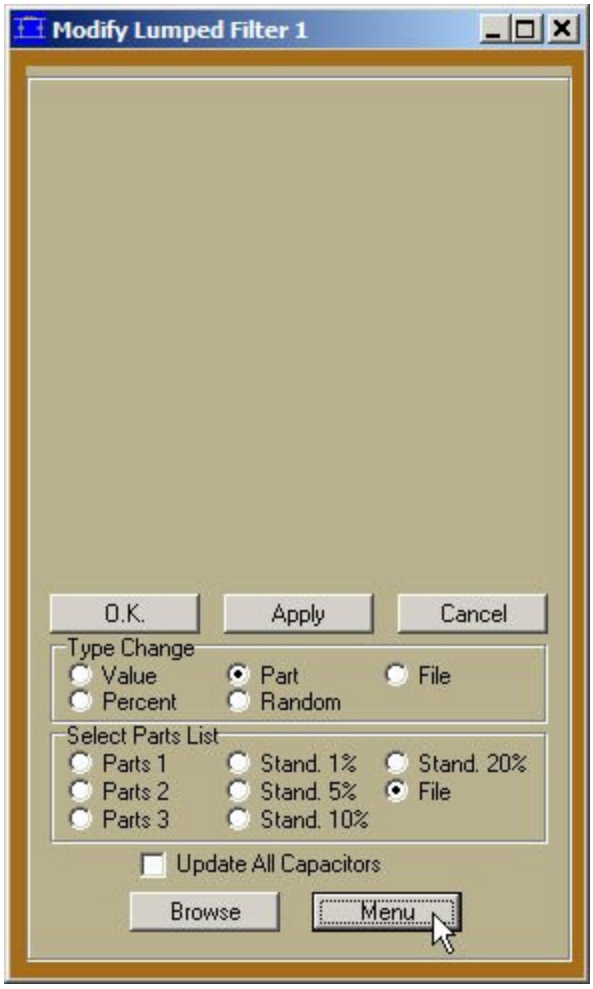


Figure 3: Change Control Panel

Individual Part Selection

Automated S-parameter and Spice model selection optimized by the part only, not the final design. It is possible that a different combination of parts may result in more desirable design performance, especially when the Q and reactive parasitics of the candidate parts varies. It is possible to tune the design by selecting specific S-parameter files and Spice models for individual parts. To do this, left click the desired element, and select, "File". Browse for the desired S-parameter or Spice model file, as shown in Figure 4.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

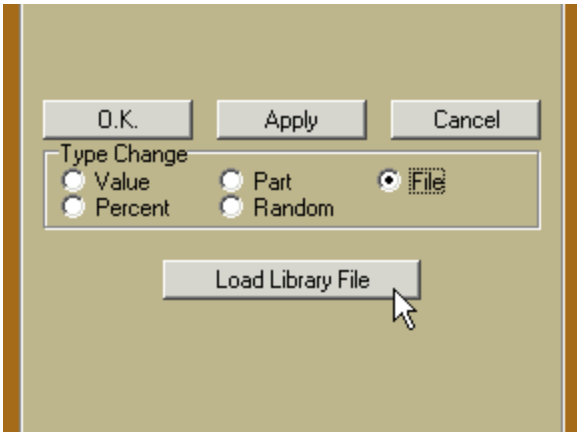


Figure 4: Selecting Individual S-Parameter or Spice Models

Active Filter Opamp Selection

Active filters are most accurately simulated with manufacturer physical opamp models. The option exists in the active filter netlist section to insert manufacturer opamp Spice models into the netlist, as shown in Figure 5, and to launch an application with the active filter netlist for simulation. Opamps are selectable from a patent pending pop-up menu that loads the default or user0customizable opamps selections. If an application to read .CKT documents exists, such as Microcap, a "Launch" button will appear to launch the application and import the netlist.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

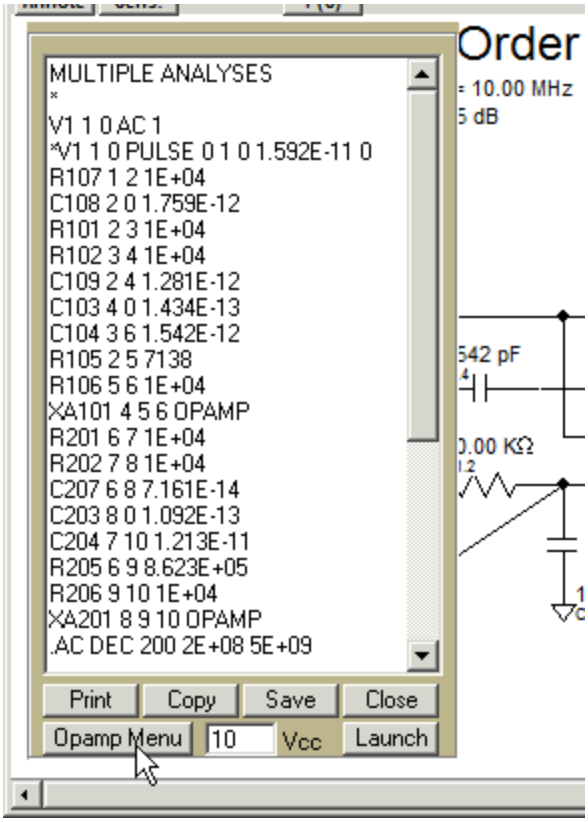


Figure 5: Manufacturer Opamp Model Insertion Into an Active Filter netlist

Customizing Libraries

The libraries that ship with the product are limited and cannot be kept current by the product. Therefore, it is necessary for the user to customize the library as needed to keep it current and to insert needed parts that may not be included with the shipped library.

To Customize Your Libraries:

The patent pending process of customizing user libraries is as follows:

1) In the options panel ("Options Panel" under "Options" in the main menu), check, "Edit Nuhertz Library Path", as shown in Figure 1. Select the desired drive and the desired directory to house the customized library, and select, "Select Nuhertz Library Path". **The selected directory must include, "Parts Libraries" directory**, which in turn must include the directories of Capacitors, Inductors, and Opamps which in turn must contain the families and part files that ere desired to be accessible.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

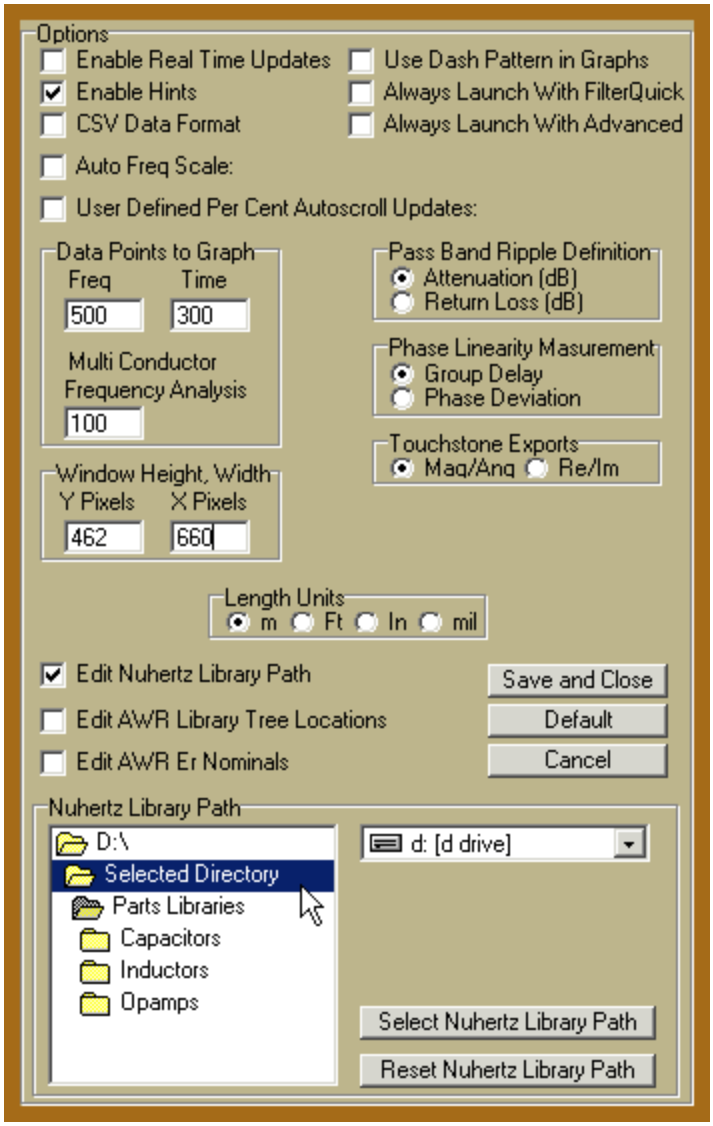


Figure 1: Editing the Library path

Restoring the Libraries

If it is necessary to restore the libraries to the shipped default of no parts libraries, simply select, "Reset Nuhertz Library Path" in the options panel, as seen in Figure 1, above.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

User Library Directories

Individual directories may be loaded with candidate s-parameter or spice model files for use in designs. The best fit s-parameter or spice model file will be selected for use for each part, collectively or individually.

Select the "Browse" selection in schematic inductor and capacitor part selection drop-down boxes, and browse for the directory containing the files. The directory will be remembered and subsequently accessed by selecting, "File" in the respective drop down box, as seen in Figure 1.

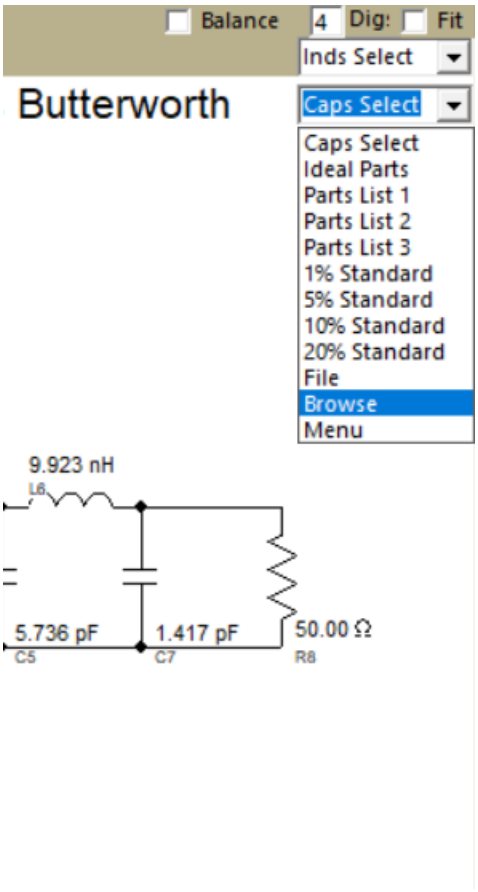


Figure 1: Selecting a user specified directory of candidate S-Parameter or Spice Model Files.

Designing With Manufacturer's Parts

See the Online Help to view the Designing with Manufacturer's Parts video.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10 - Zmatch Impedance Matching

This chapter contains the following topics:

- General Information
- Using Zmatch
- Lumped Impedance Matching
- Distributed Impedance Matching

General Information

This chapter contains the following topics:

- What is Zmatch?

Zmatch Impedance Matching

The Zmatch feature is a general purpose, easy-to-use, single point and broad band, lumped and distributed impedance matching program. Zmatch allows one to define the frequency dependent complex lumped source impedance or transmission line impedance, frequency dependent complex load impedances and the desired order of matching network. Zmatch will calculate one to four matching networks consisting of lumped elements, RLGC transmission lines, or planar layouts, and provide analysis results of circuit imperfections due to real world parasitics.

Using Zmatch

This chapter contains the following topics:

- The Control Panel
- Editing and Tuning Matching Circuits

The Control Panel

No. of Matching Elements/Stubs	Select the number of desired elements (for lumped) or stubs (for distributed) in the matching network
Number of Elements	Displayed when 4+ is selected in the No. of Matching Elements. Enter the number of desired reactive elements in the matching network

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Frequencies	Select optimization for individual frequency points or for all frequencies distributed in the input table.
JX Definition	Select the desired format to read and display imaginary impedance: Reactance: Pure Reactance Inductance: Equivalent Inductance in Henrys Capacitance: Equivalent Capacitance in Farads
Z Definition	Select an impedance defined by Polar, Cartesian, or Parallel coordinates. The parallel definition for Zmatch is defined as the real entry in parallel with the imaginary entry.

Touchstone Data Importation.

Impedance rows may be filled in with data from a single dimension Touchstone format file using controls to the right of the impedance tables. All data is transformed to meet the settings selections in the above table. For example, if "Polar" data is selected, Touchstone data is transformed into polar format and inserted into the impedance table.

Lumped Filter Alternate Impedance Entries.

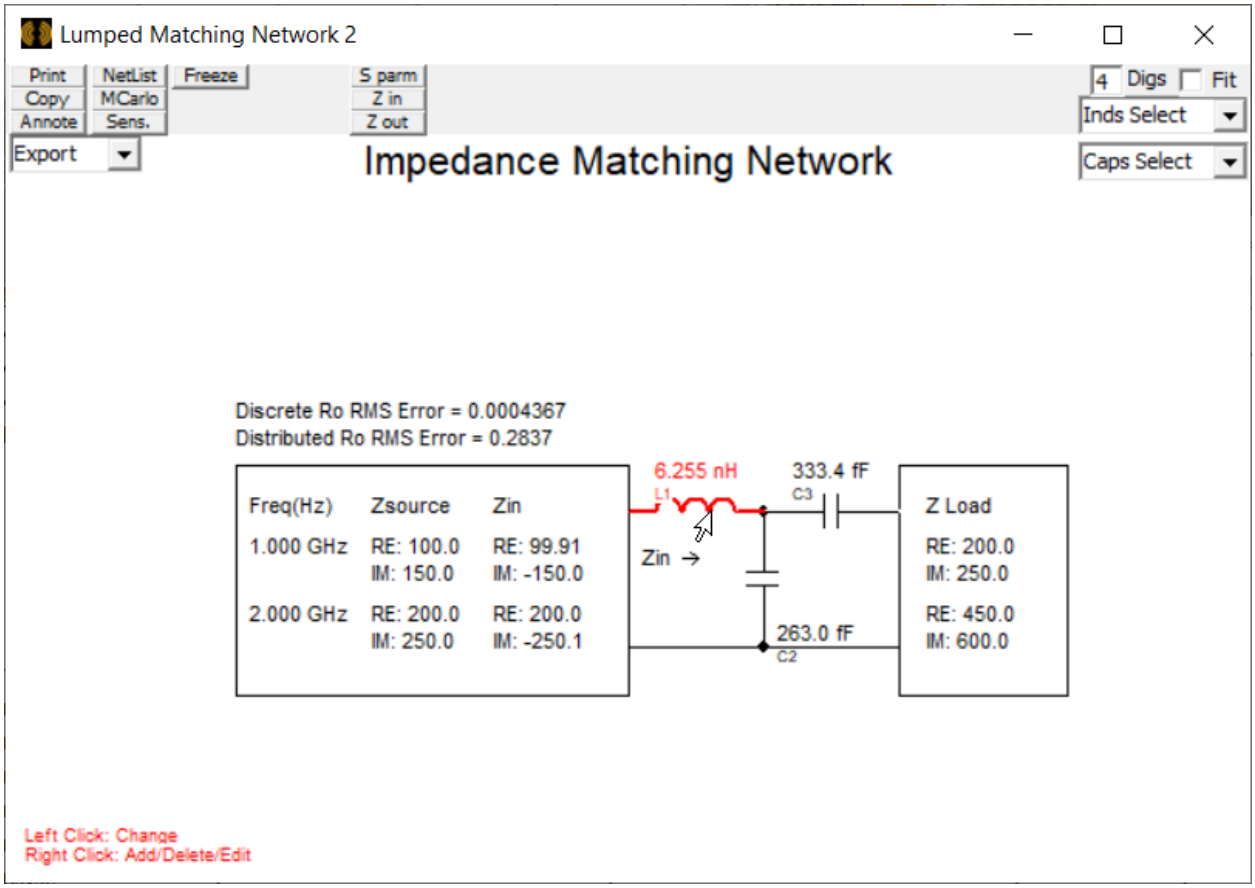
Zmatch for lumped elements has separate impedance columns for the source and the load. If one of the impedance columns is to be a fixed resistance, check **Autofil Other Term** to automatically fill in with the value entered into the **Other Termination Impedance** text box. Leave unchecked to enter just one impedance column at a time with a Touchstone file.

Tuning Matching Circuits

To tune the matching network by freezing the value of an element and then recompute the elements to achieve a new match:

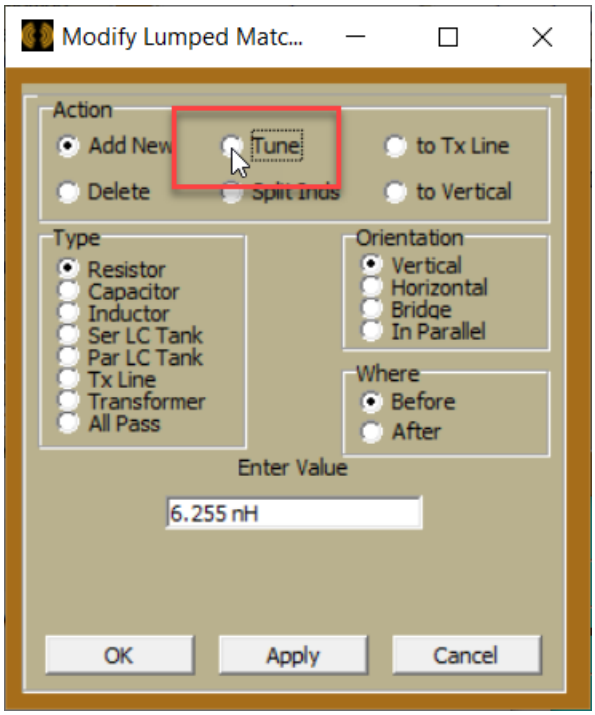
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Right-click an element of the circuit.

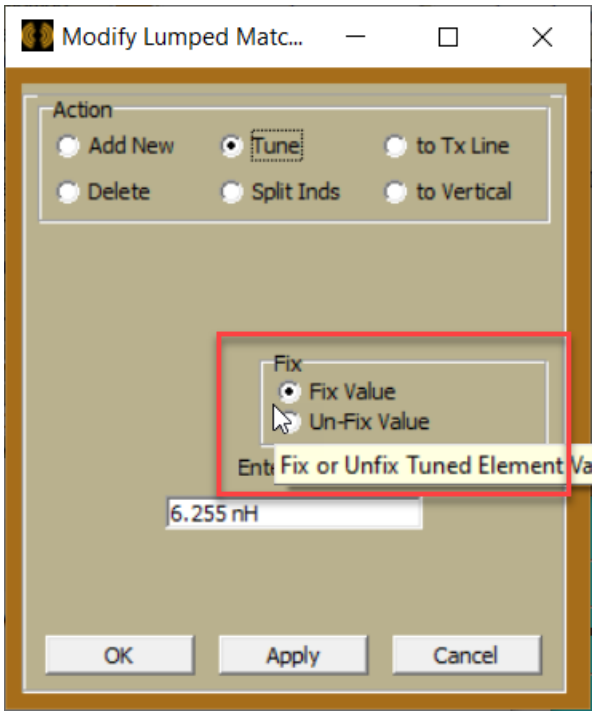


2. Select Tune in the action group box.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

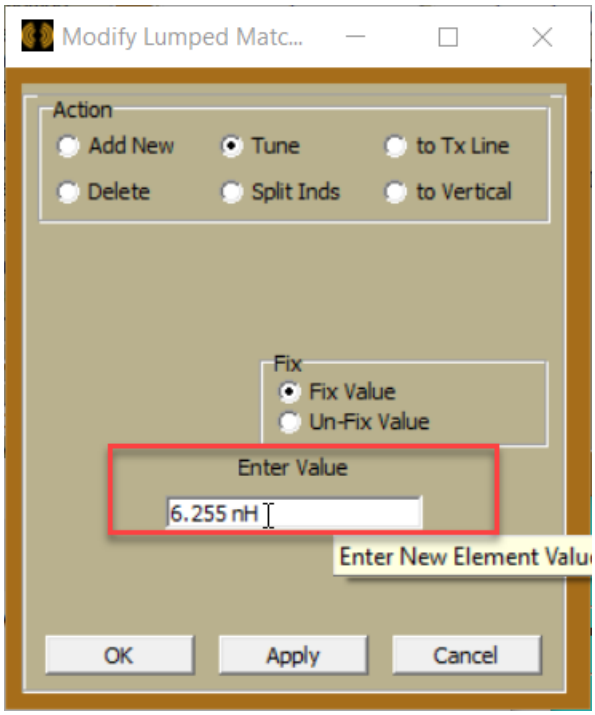


3. Select Fix Value.



4. Enter the desired value of the element.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5. Select Apply.

Selecting the "Fix Value" option above has the effect of freezing or "fixing" the element or stub value to that specified for all future tuning action of the network.

Lumped Impedance Matching

This chapter contains the following topics:

- Concepts and Theories
- Reactive Elements
- Single Point
- Multiple Point
- Broadband Matching

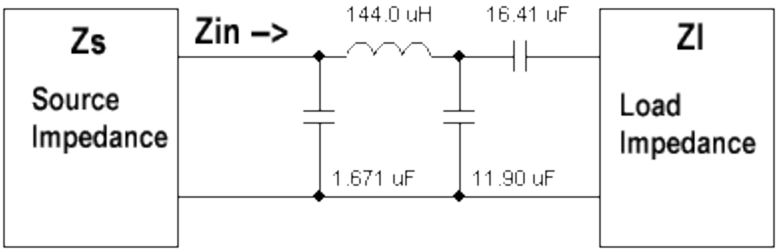
Concepts and Theories

An impedance matching circuit is a circuit positioned between a source impedance and a load impedance that serves to produce zero reflections when a frequency voltage or current source is applied. Ideally, the resulting network will produce zero reflections at all frequencies. In reality, it is only possible to match a handful of frequencies, or to minimize the RMS reflection magnitude over a range of frequencies.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Frequency Matching Networks

RC networks may be used to match a source and a load impedance, as shown below:



If the network is designed properly, then the reflections to the source are zero. Zero reflection occurs when the conjugate of Zin is equal to Zs. This may be easily achieved for one frequency point, and sometimes several frequency points. If it is desired to match impedances over a wide band of frequencies, then it is usually necessary to minimize the RMS magnitude of the reflections.

Reflection Coefficient

The reflection coefficient is defined mathematically as:

$$\rho = \frac{Zin - Zs^{*}}{Zin + Zs}$$

Where :
ρ is Reflection Coefficient
"*" Designates conjugate

Note that when Zin is the conjugate of Zs, no reflections occur. The matching network must be designed to either produce zero reflections at one or more frequencies, or to minimize reflections over a broad range of frequencies.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reactive Elements

Zmatch uses reactive elements to match impedances. The lossless nature of these elements insures that all of the available power is transferred to the load. However, real reactive elements are not purely lossless, but have frequency dependent loss functions defined by the Q of the element. True lossless reactive elements have a Q of infinity. Real elements have finite Q values. Zmatch will consider the finite Q of the reactive elements when computing a matching network.

Inductors

The impedance of the inductors may be expressed as follows:

$$Z = J\omega L + \frac{\omega L}{Q}$$

In addition, series resistance (DCR) and parallel resistance of the inductor are supported an analysis parasitics and are compensated for in the matching computations.

Capacitors

The impedance of the capacitors may be expressed as follows:

$$Z = \frac{1}{J\omega C} \| \frac{Q}{\omega C}$$

Where $\|$ designated a parallel operator

In addition, series resistance (ESR) and parallel resistance of the inductor are supported an analysis parasitics and are compensated for in the matching computations.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

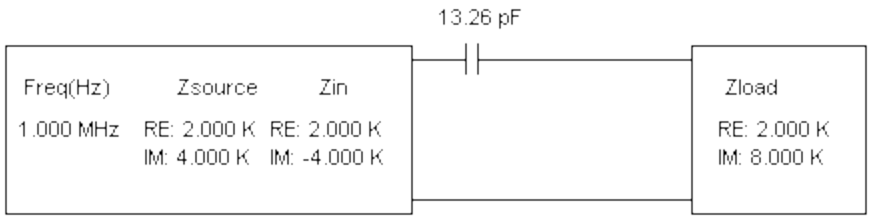
Single Point Impedance Matching

Single point frequency matches usually require two reactive elements. One reactive element is required if the load and source impedance only differ in phase. Occasionally, three reactive elements are required to produce a match.

Generally, only one matching solutions is possible for single element matches, four solutions are possible for two element matches, and an infinite number are possible for three element matches. This means that a three element matching network may be designed that is tunable even if only one or two elements are required to match the load.

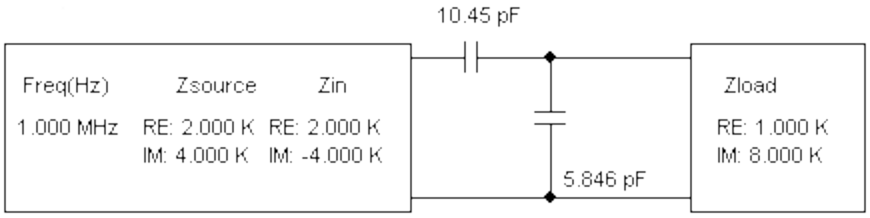
Single Element Match

The load and source both have the same resistance. Therefore, only a series reactive element is needed to translate the load reactance to the source reactance. In this case, a series capacitor reduces the input reactance.



Double Element Match

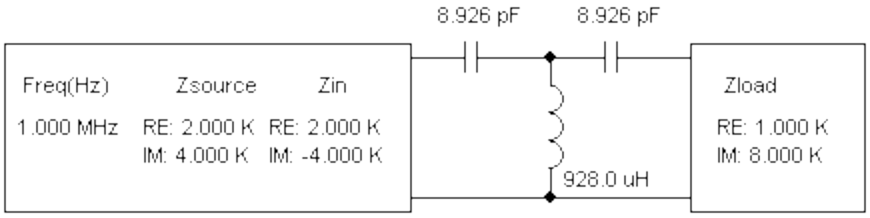
When the load an source resistance are unequal, at least two reactive elements are required. Generally, four solutions are possible. Occasionally, less than four solutions are possible, or even no solutions are possible. When this happens, more reactive elements are needed.



Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

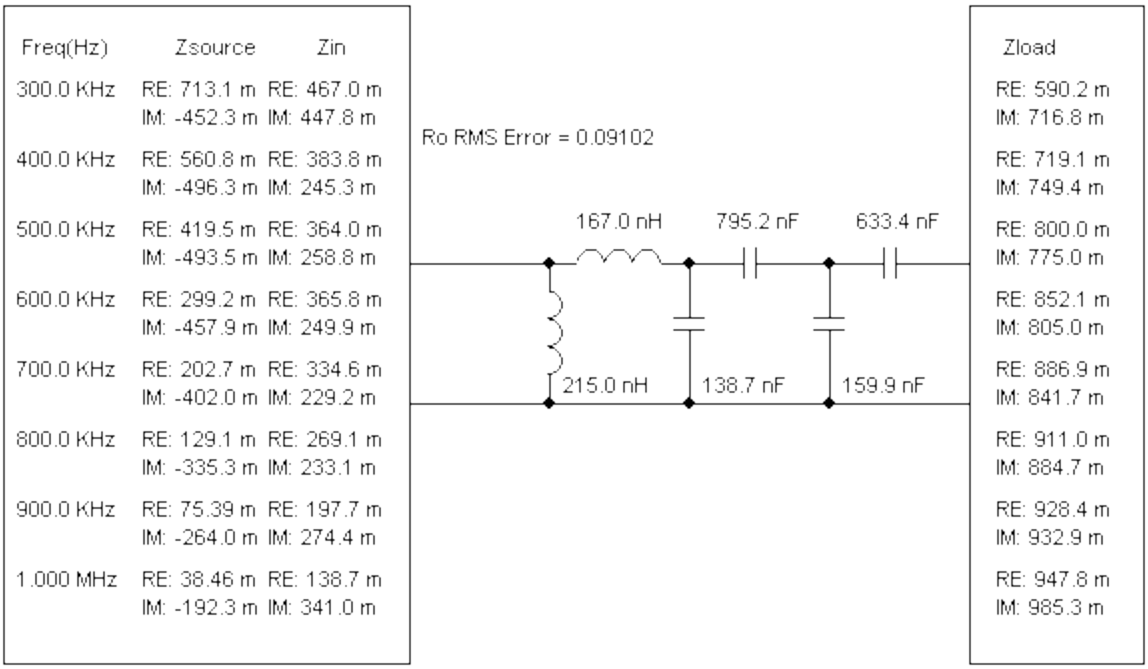
Triple Element Match

Generally, an infinite number of three element solutions are possible. In the example below, equal capacitors are used. It is usually possible to set one of the three element values to any desired value, then recalculate the other two to obtain a match.



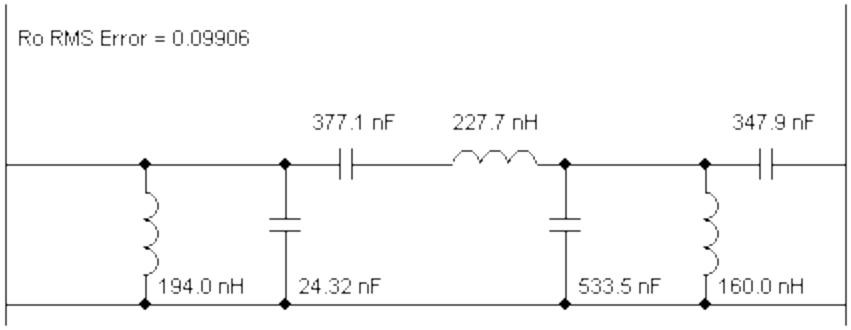
Multiple Point Impedance Matching

If it is necessary to match multiple frequency points, such as in a broad band application, it is possible to minimize the RMS of the sum of the reflections from each frequency point. Any number of elements may be used, however better matches may be obtained with more elements.



Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Alternatively, a band pass LC topology may sometimes produce better results. This topology is referred as "Type II" in Zmatch.



Broadband Impedance Matching

Matching individual frequency points doesn't necessarily produce the most efficient match for a given frequency spectrum. Better matches may be produced by increasing the number of matching frequency points, or by selecting "**Broadband"** in the "**Frequencies"** selection. Zmatch will then attempt to minimize the RMS reflection for 25 frequency points equally spaced across the frequency spectrum entered in the impedance table. This usually has the effect of minimizing the RMS reflections across the spectrum at the expense of more reflections at the individual frequency points.

Graphical displays show both the RMS error of the individual frequency points and the distributed frequency.

Distributed Impedance Matching

This chapter contains the following topics:

- Concepts and Theories
- Tx Line Stubs
- Single Point
- Multiple Point
- Broadband Matching

Concepts and Theories

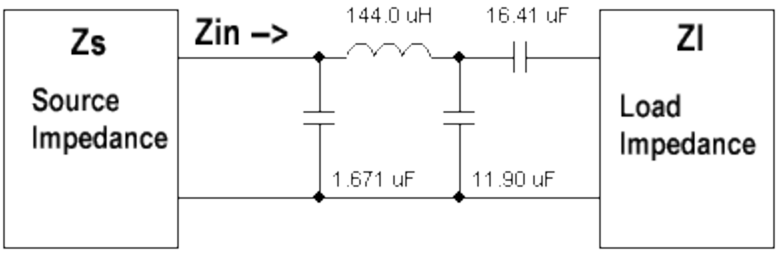
An impedance matching circuit is a circuit positioned between a source impedance and a load impedance that serves to produce zero reflections when a frequency voltage or current source is applied. Ideally, the resulting network will produce zero reflections at all frequencies. In reality, it

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

is only possible to match a handful of frequencies, or to minimize the RMS reflection magnitude over a range of frequencies.

Frequency Matching Networks

RC networks may be used to match a source and a load impedance, as shown below:



If the network is designed properly, then the reflections to the source are zero. Zero reflection occurs when the conjugate of Zin is equal to Zs. This may be easily achieved for one frequency point, and sometimes several frequency points. If it is desired to match impedances over a wide band of frequencies, then it is usually necessary to minimize the RMS magnitude of the reflections.

Reflection Coefficient

The reflection coefficient is defined mathematically as:

$$\rho = \frac{Zin - Zs^{*}}{Zin + Zs}$$

Where :
ρ is Reflection Coefficient
"*" Designates conjugate

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that when Zin is the conjugate of Zs, no reflections occur. The matching network must be designed to either produce zero reflections at one or more frequencies, or to minimize reflections over a broad range of frequencies.

Transmission Line Stubs

Zmatch uses shorted transmission line stubs of specific length and positioned at specific locations to match impedances. Stubs are placed in parallel with the main line. The object is to arrange stubs and segments to create a net load impedance such that RMS reflections are minimized.

Transmission Line Stub Impedance

The impedance of any ideal shorted stub of given length may be expressed as follows:

$$Zin = J * Zo * \tan(\varpi * len / V)$$

Where:

$\text{len} = \text{Transmission Line Length}$

$$\text{Zo} = \sqrt{\frac{\text{L}}{\text{C}}}$$

$$V = \frac{1}{\sqrt{LC}} = \text{Speed of Propagation}$$

$\text{L} = \text{Inductance per unit length}$

$\text{C} = \text{Capacitance per unit length}$

$$\text{J} = \sqrt{-1}$$

Transmission Line segment impedance

Matching stubs are connected by transmission line segments. The impedance of a transmission line segment of any given length and load may be expressed as follows:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$Zin = Zo\frac{Z1 + J*Zo*\tan(\varpi*len/V)}{Zo + J*Z1*\tan(\varpi*len/V)}$$

Where:

len = Transmission Line Length

$$Zo = \sqrt{\frac{L}{C}}$$

$$V = \frac{1}{\sqrt{LC}} = \text{Speed of Propagation}$$

Z1 = Load Impedance

L = Inductance per unit length

C = Capacitance per unit length

$$J = \sqrt{-1}$$

Single Point Impedance Matching

Single point impedance matching requires at least one stub and one segment between the stub and the load. Two unique solutions are possible. Zmatch provides both solutions. If the stub and segment lengths of both solutions are not desirable, then multiple stubs may be used. Multiple stub single point matching has no unique solution, so tuning stub lengths and segment lengths are possible.

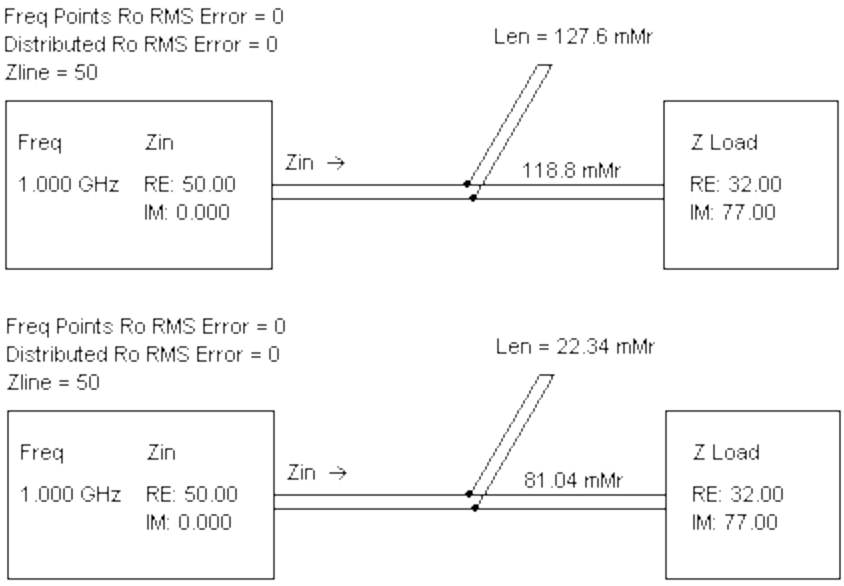
Single Stub Match

Two unique solutions are possible. Stub and segment lengths may be edited and tuned, but in all cases a matching error will result.

The following are two solutions to match a 32+J77 Ohm load to a 50 Ohm line at 1GHz. Both cases provide a perfect match with an RMS reflection error of zero.

Single Frequency Point One Stub Matches

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

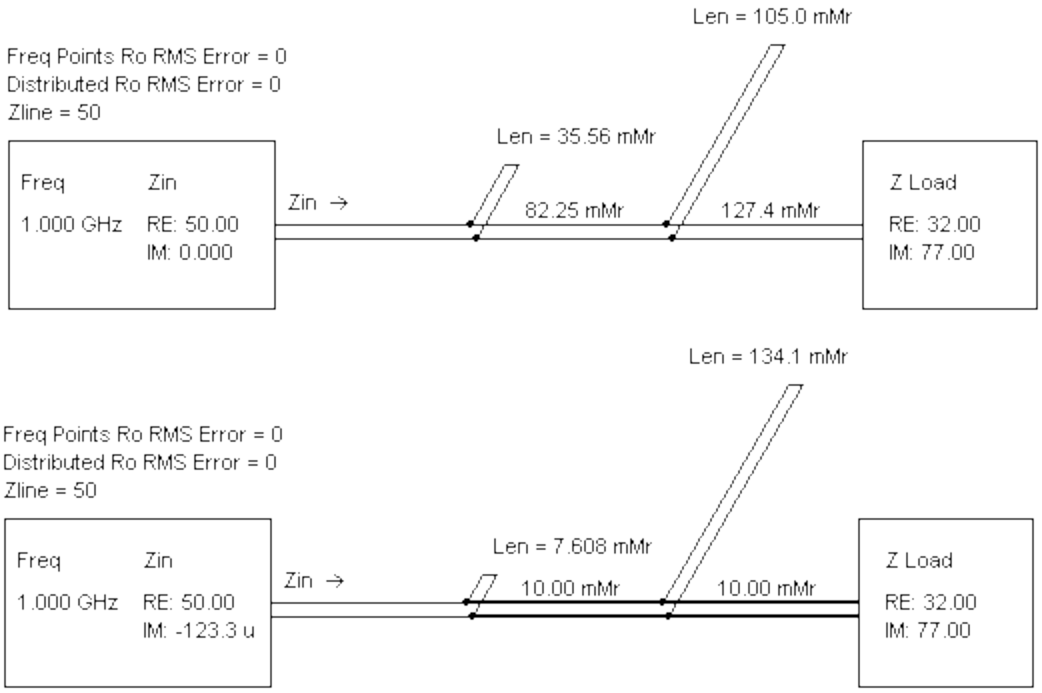


Double Stub Match

Sometimes, neither of the two single stub solutions are acceptable. An infinite number of solutions are obtainable when matching one frequency with two stubs. It is almost always possible to tune one stub or one element and frequency possible to tune two components and still obtain a perfect match, a "component" being defined as a stub or segment. Tuning may be accomplished by clicking a stub or segment, selecting "Tune", and entering the desired segment or stub length. The following shows the same frequency match as shown above, but with two stubs. The second match is accomplished with equally spaced stubs only 10mM apart.

Single Frequency Point Double Stub Matches

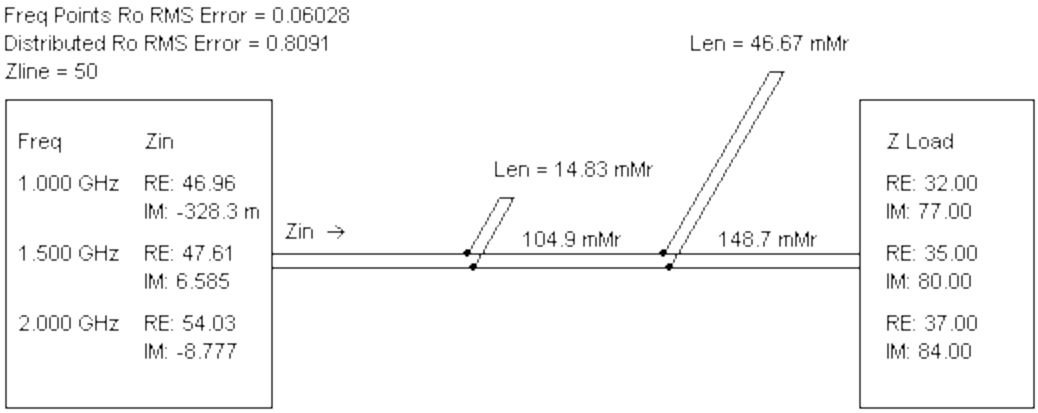
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Multiple Point Impedance Matching

If it is necessary to match multiple frequency points, such as in a broad band application, it is possible to minimize the RMS of the sum of the reflections from each frequency point. Any number of elements may be used, however better matches are obtained with more elements. To perfectly match each selected frequency point, the number of stubs much be equal to or greater than the number of defined frequency points. The following example uses two stubs to match three frequencies with reasonably good results.

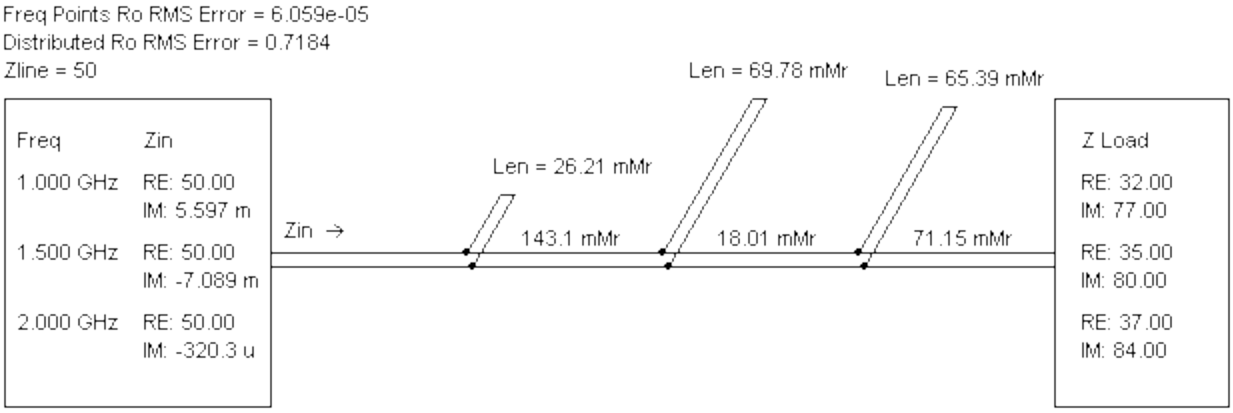
Triple Frequency Point Double Stub Match



Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If more accuracy is desired, three stubs may effectively provide a perfect match at all frequencies, as shown below

Triple Frequency Triple Stub Matches



Broadband Impedance Matching

Matching individual frequency points doesn't necessarily produce the most efficient match for a given frequency spectrum. Better matches may be produced by increasing the number of matching frequency points, or by selecting "**Broadband"** in the "**Frequencies"** selection. Zmatch will then attempt to minimize the RMS reflection for 25 frequency points equally spaced across the frequency spectrum entered in the impedance table. This usually has the effect of minimizing the RMS reflections across the spectrum at the expense of more reflections at the individual frequency points.

Graphical displays show both the RMS error of the individual frequency points and the distributed frequency.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11 - Multi-Band Filters

This chapter contains the following topics:

- Low Pass Multi-Band
- High Pass Multi-Band
- Band Pass Multi-Band
- Band Stop Multi-Band
- Trap Topology

Low Pass Multi-Band

(Available for lumped, active, switched capacitor, and digital filters).

Low pass multiband filters provide finite attenuation at DC and infinite attenuation at infinity. A low pass cut-off occurs within the frequency band, and one or more band passes occur between the low pass cut-off frequency and infinity. Band pass frequency stages are defined by their corner frequencies.

FilterSolutions supports low pass multiband filters for lumped, active, switched capacitor, and digital filters. Digital filter bilinear transformations include a prewarping option to preserve the frequency positions of the band pass sections and the low pass cur-off frequency.

Figure 1 is an example of an Elliptic low pass multi-band filter with a low pass cut-off frequency of 1MHz, and band pass stages at 2-3 MHz, 4-5 MHz, and 6-10 MHz. Stop band attenuation is set to be 60 dB.

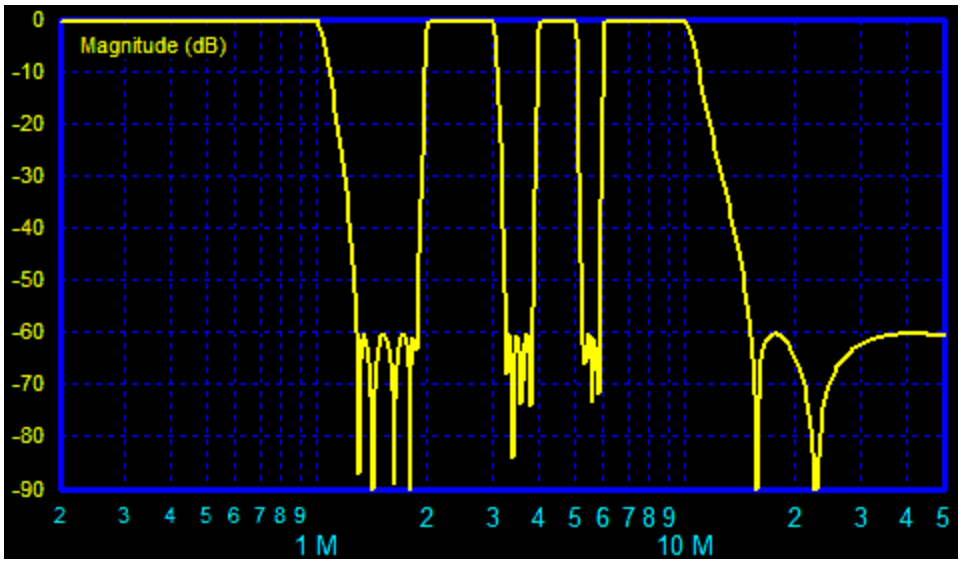


Figure 1: Low Pass Multi-Band Filter

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

High Pass Multi-Band

(Available for lumped, active, switched capacitor, and digital filters).

High pass multiband filters provide finite attenuation at infinity and infinite attenuation at DC. A high pass cut-off occurs within the frequency band, and one or more band passes occur between the high pass cut-off frequency and DC. Band pass frequency stages are defined by their corner frequencies.

FilterSolutions supports high pass multiband filters for lumped, active, switched capacitor, and digital filters. Digital filter bilinear transformations include a prewarping option to preserve the frequency positions of the band pass sections and the high pass cur-off frequency.

Figure 1 is an example of an Elliptic high pass multi-band filter with a High pass cut-off frequency of 10MHz, and band pass stages at 1-2 MHz, 3-4 MHz, and 5-6 MHz. Stop band attenuation is set to be 60 dB.

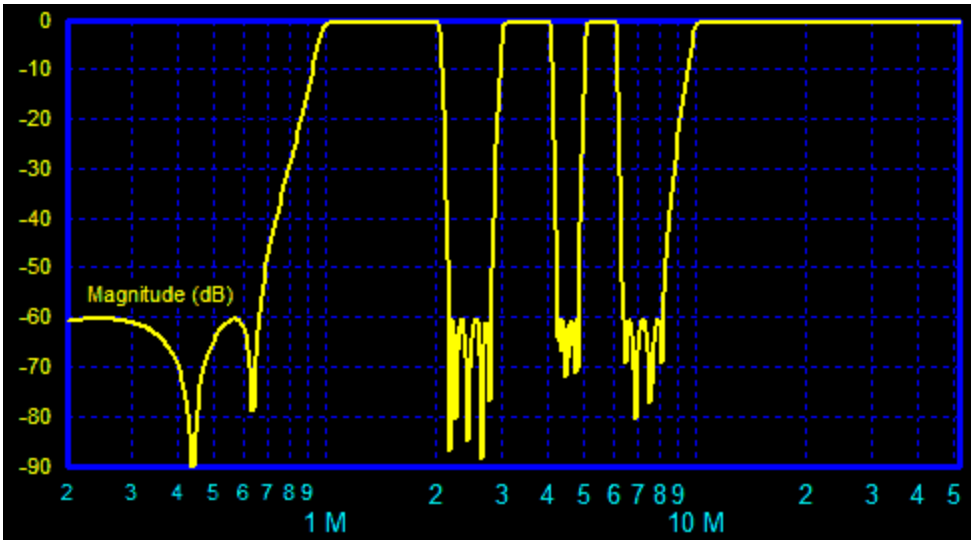


Figure 1: High Pass Multi-Band Filter

Band Pass Multi-Band

(Available for lumped, active, switched capacitor, and digital filters).

Band pass multiband filters provide infinite attenuation at DC and infinity, with one or more band passes occurring at finite frequencies. Band pass frequency stages are defined by their corner frequencies.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FilterSolutions supports band pass multiband filters for lumped, active, switched capacitor, and digital filters. Digital filter bilinear transformations include a prewarping option to preserve the frequency positions of the band pass sections.

Figure 1 is an example of an Elliptic band pass multi-band filter with band pass stages at 1-2 MHz, 4-5 MHz, and 6-10MHz. Stop band attenuation is set to be 60dB.

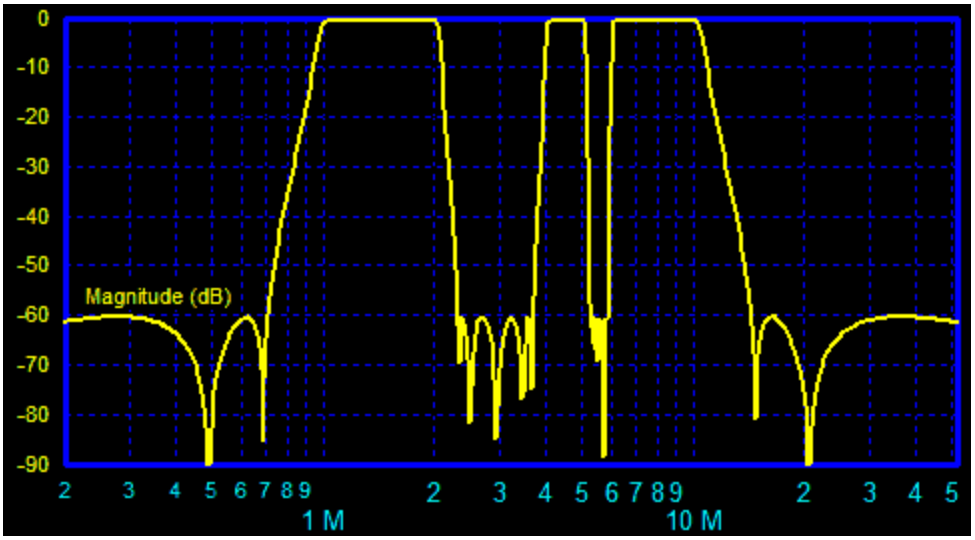


Figure 1: Band Pass Multi-Band Filter

Band Stop Multi-Band

(Available for lumped, active, switched capacitor, and digital filters).

Band stop multiband filters provide finite attenuation at DC and infinity, with one or more band stops occurring at finite frequencies. Band stop frequency stages are defined by their corner frequencies.

FilterSolutions supports band stop multiband filters for lumped, active, switched capacitor, and digital filters. Digital filter bilinear transformations include a prewarping option to preserve the frequency positions of the band stop sections.

Figure 1 is an example of an Elliptic stop pass multi-band filter with band stop stages at 1-2 MHz, 4-5 MHz, and 6-10MHz. Stop band attenuation is set to be 60dB.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

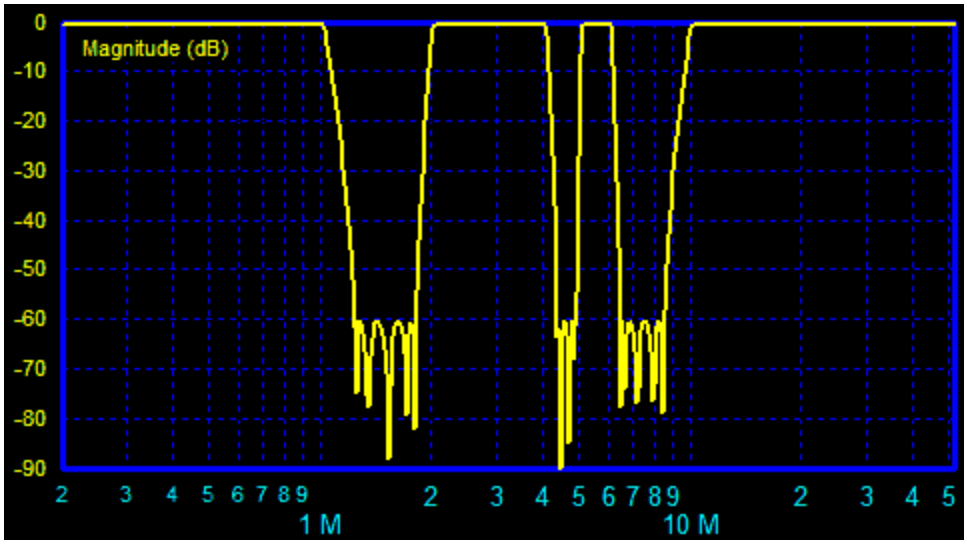


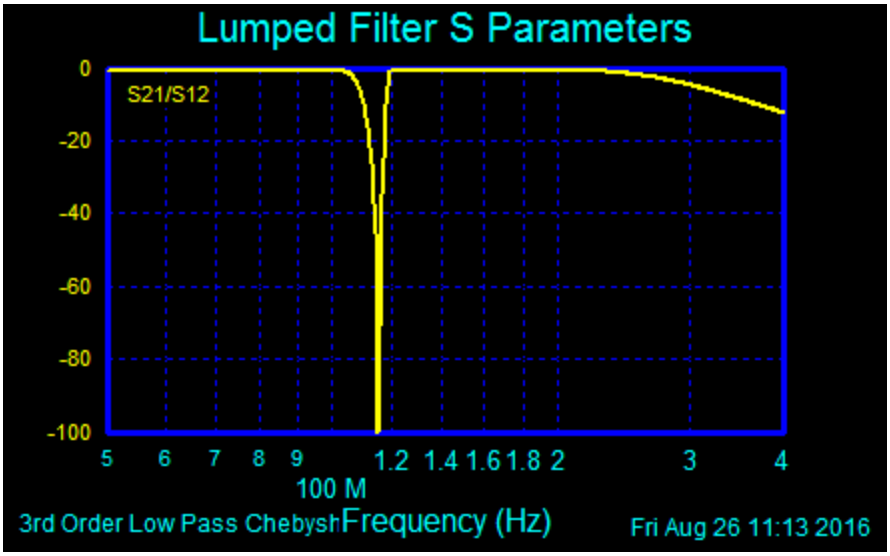
Figure 1: Band Stop Multi-Band Filter

Trap Topology

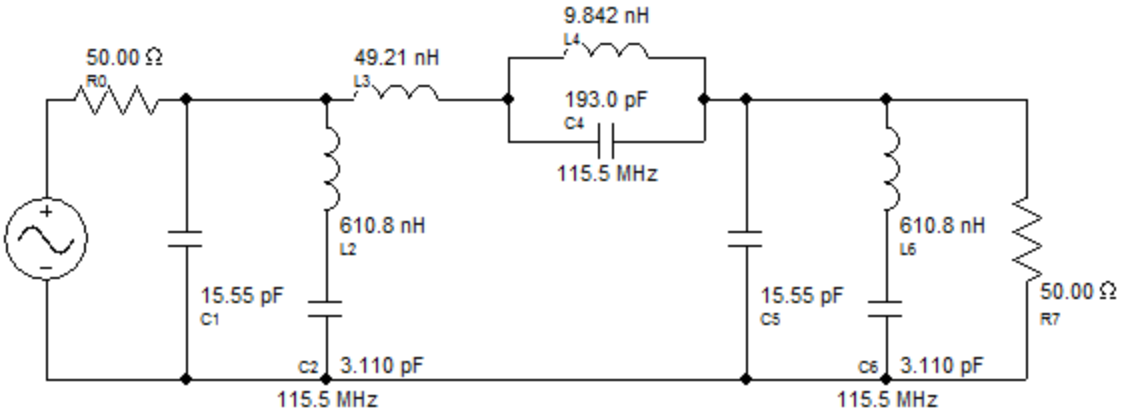
Low pass and high pass multiband filters are essentially low pass and high pass notch filters, containing one or more notch sections. When transformations are applied to place a series element on the shunt resonators and a parallel element on the series resonators, the resulting topology may be referred to as a Trap topology. The advantage is that some applications contain parasitics that may be absorbed fully or in part by these elements. Further user transformation with the use of the right mouse click are possible to better fit the filter to application needs.

Chebyshev 1 Low Pass Notch S12 Response:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

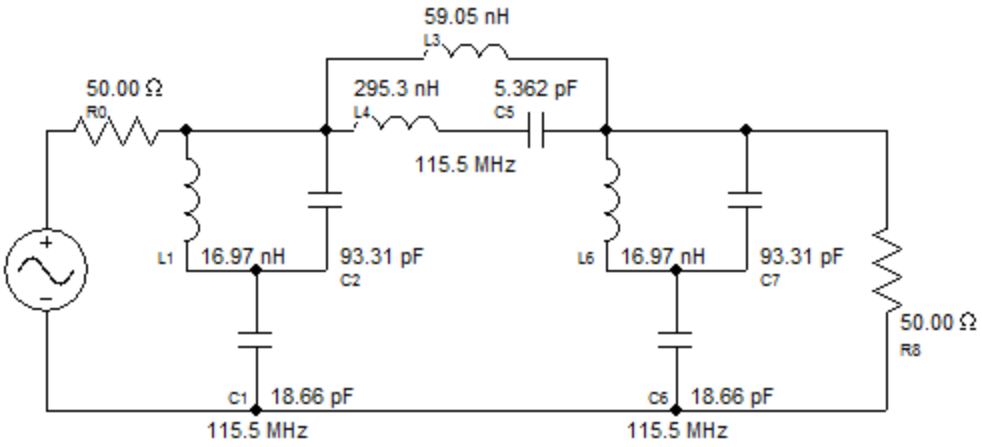


Standard Chebyshev I Low Pass Notch Topology:



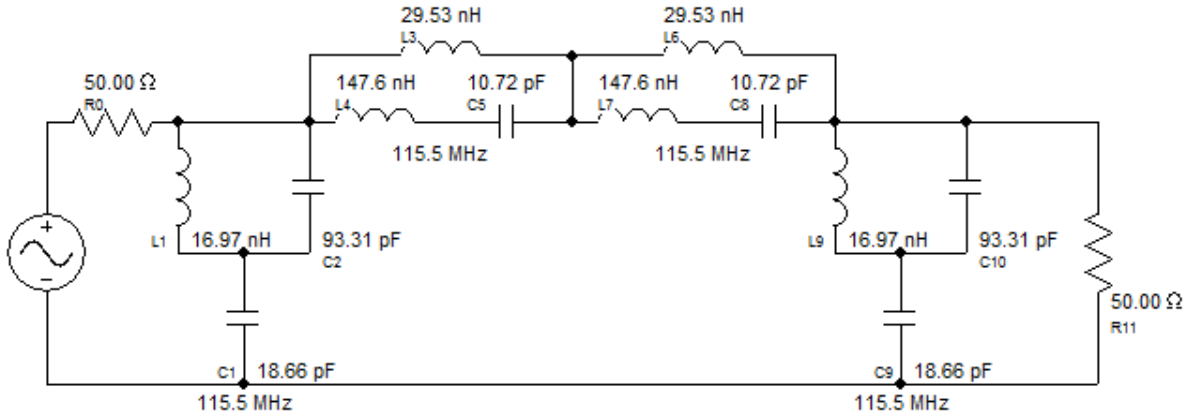
Trap Chebyshev I Low Pass Notch Topology:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



User-Modified Trap Chebyshev I Low Pass Notch Topology With Physical Symmetry:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12 - Lumped Filters

This chapter contains the following topics:

- Passive LC Features
- Working With Lumped Filters
- Sensitivity
- Modelithics Exports to Electronic Desktop
- Automated Discrete Optimization
- Exporting to Other Software

Passive LC Features

This chapter contains the following topics:

- Circuits Synthesis and Analysis
- Singly and Doubly Terminated Filters
- Finite Q
- Lumped Parasitic Models
- Node Capacitance and Lead Inductance
- Diplexers
- Coupled Resonator Band Pass Filters
- Tubular Band Pass Filters
- Improving Frequency Symmetry
- Coupled Resonator Band Pass Filters
- Quadruplet Coupled Resonators
- Bridged Tubular and Resonator Filters
- Minimum Inductor Zigzag Filters
- Matching Unequally Terminated Filters
- Bridged T Topologies
- Equal Inductor Band Pass Filters
- Cascaded High and Low Pass Topologies
- Amplitude Equalizers
- Shunt Node Capacitance Compensation
- Even Order Modification
- All Pass Sections
- Balanced Lumped Circuits

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Circuit Synthesis and Analysis

Terminations

FilterSolutions produces both single termination and double termination filters. The value of the source resistance determines the choice. When zero is entered as the source resistance, only single terminated filters are displayed. When the source resistance is equal to the load resistance, only double terminated filters are displayed. When a finite source resistance is unequal to the load resistance, both single and double terminated filters are created. Of these, only the filters that meet the frequency magnitude characteristics of the desired filter are displayed.

Transfer Functions

The Transfer Function button on the face of the Main Control Panel displays the transfer function of the desired filter.

Time Response

The Control Panel time response displays the design time response of the desired filter. It is computed by stimulating the transfer function of the desired filter.

The circuit time response uses the user-changed element values and is computed directly on the displayed filter.

Frequency Response

The Control Panel frequency response displays the design frequency response of the desired filter. It is computed by substituting Jw into the filter transfer function.

The circuit frequency response uses the user-changed element values and is computed directly on the displayed filter.

Reflections

The Control Panel reflection button displays the design reflection coefficient. This reflection is derived by:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\rho(S) = \sqrt{1 - 4(Rg/Rl)G(S)G(-S)}$$

Using Left Plane Roots Only

Where:

$\rho(S)$ = Design Reflection Coefficient

Rg = Source Resistance

Rl = Load Resistance

G(S) = Design Transfer Function of the Filter.

Note that the above equation is a square root function, so two reflection solutions exits that differ by 180 degrees.

The individual circuit reflection button displays the individual circuit reflection coefficients. The circuit reflections will change when circuit element values are changed. The circuit reflection coefficient is derived by:

$$R(S) = (Z(S) - Rg) / (Z(S) + Rg)$$

Where:

R(S) = Circuit Reflection Coefficient

Z(S) = Impedance of the Circuit Less the Source Resistor

Rg = Source Resistance.

If designed correctly, the above reflection equation will yield the same result as one of the solutions from the previous equation.

Singly and Doubly Terminated Filters

Doubly terminated filters are filters with a finite non zero termination at both ends. Single terminated filters are filters with a finite nonzero termination at one end, and an open or shorted termination at the other. Equally terminated filters are doubly terminated filters with identical finite nonzero terminations at each end.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Equally terminated filters usually, but no always optimized for the best reflection response. with a finite nonzero source resistance. In some cases, it is not physically possible to synthesize with equal terminations, and in others a step up/down transformer is required.

FilterSolutions supports singly, doubly, and equally terminated filters. Singly terminated filters are supported for shorted voltage sources, open current sources, and open loads.

Doubly and Equally Terminated Filters:

Enter the load and source resistance into the appropriate text entries on the Passive control panel. Check the reflections response to insure you are getting the desired reflections.

Singly Terminated Filters:

To synthesize a shorted voltage source filter, enter "0" in the Source Res text entry box and select Voltage source.

To synthesize an open current source filter, enter "Inf" in the Source Res text entry box and select Current source.

To synthesize an open load filter, enter "Inf" in the Load Res text entry box and select Current source.

Impedance Matching

Purely reactive loads and sources may sometimes be incorporated directly into a singly terminated filter. For example, if a filter load is exactly one 1mH and an open end filter design requires 2mH, the 2mH inductor may be composed of a 1mH inductor in series with the load. Likewise, if the filter design requires 500uH, the 500uH inductor may be composed of a 1mH indictor in parallel with the load.

> **Note:** "0" and "Inf" entries in the text entry boxes have the same effect in that they are interpreted to mean 0 or infinity as required., so synthesizing both voltage and current driven singly terminated source filters may be accomplished by simply entering a "0".

Finite Q

When operating in RF frequencies, it may be necessary to consider the finite Q of the passive components, usually just the inductors, before and after creating a filter. FilterSolutions allows

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the entry of a default inductor and capacitor Q for the filter and to change the Q of each individual component for use in frequency, reflection, and time analysis. When doing an analysis with finite Q, the dark blue base trace represents the filter with infinite Q for quick visual comparison purposes.

Select **Real Parameters** in the main Lumped control panel to enter default values. Use the left mouse key to edit values.

Finite Q effects are only included in the analysis of individual passive filters. The Main Control Panel analysis functions do not include these effects. The reason for this is that the effect of finite Q is unique to the individual passive filter.

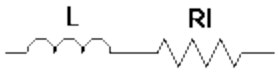
If the filter has zero source resistance or unequal load and source resistance, FilterSolutions has the capability of automatically altering the filter to partly compensate for the degradation effects of the finite Q. This is done by "Stretching" the prototype poles of the filter along the real axis until the frequency response at the cutoff frequencies has been mostly restored. This compensation feature works best for Zero ohm source resistance. If the source to load ratio is between 0.2 and 5.0, the automatic Q compensation feature has a marginal effect. Filters with equal or nearly equal load and source resistances may not be compensated in this manner, but may be amplitude equalized.

You may compensate unequally and singly terminated filters for finite Q effects yourself by opening a pole/zero plot, and dragging the prototype poles to the left and right to "stretch" the real pole placement until you like the circuit frequency response of the desire circuit. Try dragging the left most pole(s) farther to the left first. Frequently, this is all that is needed. **To watch the effect of pole/zero manipulation in real time**, check the "RTU" box in the pole/zero control panel.

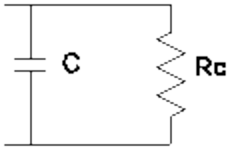
If the filter has a finite source resistance, you may compensate for finite Q with the amplitude equalizers. Check the "Amp Equalize" box above the circuit to be equalized to do this.

Inductors and capacitors may be modeled according to the following expressions and models:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



$$Rl = L\omega/Q$$



$$Rc = Q/(C\omega)$$

When FilterSolutions displays a resistance to represent finite Q, the frequency used to calculate the resistance is the frequency displayed for the component. If no frequency is displayed, the cutoff or center frequency is used.

Reactive Element Parasitic Models

In addition to finite Q, series and parallel resistance parasitics and reactive parasitics are also modeled. Select, **Parasitics** in the main Lumped control panel to enter default values. Series inductor resistance is sometimes referred to as DC series resistance (DCR) and series capacitor resistance is sometimes referred to as equivalent series resistance (ESR). Capacitors support series inductor parasitics, and inductors support parallel capacitance parasitics.

Like finite Q, the analysis is available in the schematic analysis and amplitude equalization is available to mitigate the lossy parasitic effects.

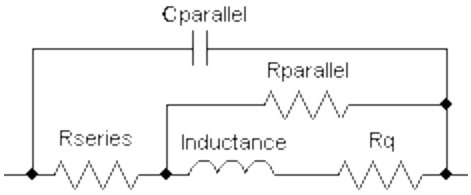
Standard parts lists may also include parasitic values by adding the terms, "Q= ", "Rs= ", "Rp= ", and "Ls=" for capacitors, and "Cp=" for inductors to an inductor or capacitor line.

The following models are used for analysis purposes when series, parallel, and/or finite Q effects are entered.

Inductors

The model elements of Q resistance, series resistance, parallel resistance, parallel capacitance and inductance are arranged according to the following inductor model:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



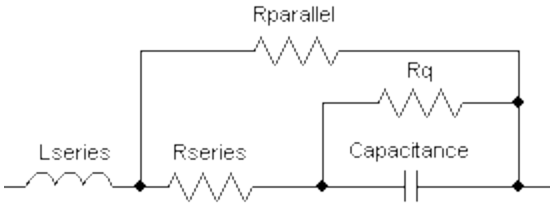
Inductor model with Parasitic Effects

Rq is computed as a function of frequency: Rq = (L*W)/Q

where L is inductance in Henries, and W is frequency in R/S

Capacitors

The model elements of Q resistance, series resistance, parallel resistance, and series inductance capacitance are arranged according to the following capacitor model.



Capacitor model with Parasitic Effects

Rq is computed as a function of frequency: Rq = Q/(C*W)

where C is capacitance in Farads, and W is frequency in R/S

Node Capacitance and Lead Inductance

Parasitic models and S-Parameter simulations of lumped elements don't necessarily account for node capacitance resulting from the pad attachments and/or lead inductance resulting from conductor leads of the lumped element. Yet, node capacitance and lead inductance do have a parasitic affect on the performance of the filter, and the parasitic affect may be significant.

FilterSolutions permits the simulation of both node capacitance and lead inductance on all lumped elements. A single node capacitance and a single lead inductance may be entered into the parasitic page of the lumped design control panel at design time, as shown in Figure 1 below. In addition to an optional compensation function is available that alters the filter design

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

capacitance and inductance values where possible to compensate for the node capacitance and lead inductance.

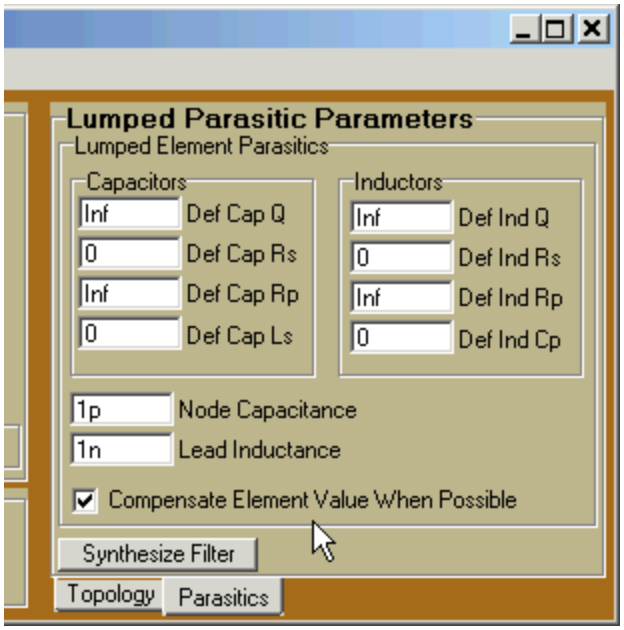


Figure 1: Lumped Parasitic Page with Node Capacitance, Lead Inductance, and Element Value Compensation

Node Capacitance and Lead Inductance Models

Each capacitor, resistor, and inductor in the filter is simulated with an internal model that adds node capacitance and lead inductance to the element. Parasitic element models and S-Parameter element models remain unchanged between the node capacitance and lead inductance, as seen in Figure 2 below.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

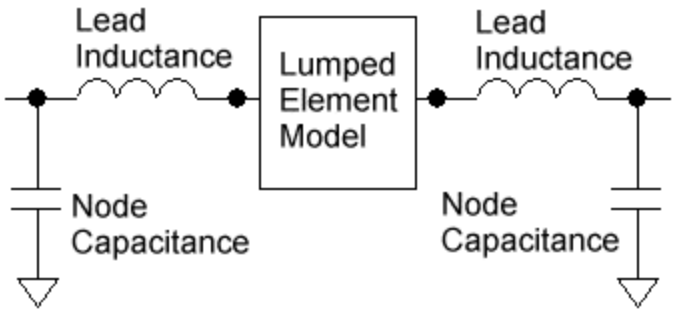


Figure 2: Lumped Element Model with Lead Inductance and Node Capacitance.

About Diplexers

In the FilterSolutions Main Control Panel, the **diplexer 1** option designs low pass/high pass diplexers, and the **diplexer 2** option designs band pass/band stop diplexers or low pass, band pass, high pass diplexers, otherwise known as triplexers.

The basic diplexer consists of a high pass and a low pass filter, or a band pass and band stop filter in parallel that are driven from the same source and usually contains RC series tank elements in parallel to reduce undesired reflections.

The low pass and high pass legs, or the band pass and band stop legs of the diplexer are synthesized using normal low pass and high pass design parameters except that the reactive elements are calculated assuming a zero ohm source resistance. This is done to achieve a desirable reflection response, but slightly degrades the pass band frequency response. When an even order diplexer is specified, the highest frequency finite transmission zero is automatically mapped to infinity. The reflection zeros are left alone. Otherwise stated, the Even Order Mod option is automatically performed on the transmission zeros, but never on the reflection zeros.

When designing a low pass/high pass diplexer with stop band filters (Chebyshev II, Hourglass, or Elliptic filters) it is advantageous to use even order filters so that the number of inductors is minimized. Selecting "Asymmetrical" and entering an odd order high pass leg may further reduce the inductor count.

Contiguous Diplexers

Contiguous diplexers exist when the half power point (-3.01dB) frequency is the same for both legs of the diplexer. If the pass band ripple is set properly, all reflections will be below -20dB and no reflection compensation elements are needed. **To create contiguous diplexers, uncheck**

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the "Standard Pass Band Atten" checkbox, enter -3.01dB as the pass band attenuation. For high/low pass diplexers, set the bandwidth to zero. For band pass/stop diplexers or triplexers, set the outer bandwidth equal to the inner bandwidth.

Noncontiguous Diplexers

If a separation in frequency exists between the diplexer legs, a noncontiguous diplexer is created. Undesirable reflection may occur near the cutoff frequencies of the legs. Reflection compensating elements may be needed to force the reflections into the frequency gap between the legs and away from the frequencies passed by the diplexer legs.

The values of the compensation elements may be calculated by:

$$Lo = \frac{1}{Y(\varpi_2 - \varpi_1)} \qquad Co = \frac{1}{Lo\,\varpi_1\varpi_2}$$

Where:
ϖ_2 is the upper corner frequency in R/S
ϖ_1 is the lower corner frequency in R/S
Y is the imaginary part of the average admittance of the diplexer at the upper and lower corner frequencies without the compensating elements

Reference:
"On the Design of Filters by Synthesis"
by Rudolf R. Saal and E. Ulbrich, page 55.

If a band pass/band stop diplexer or a triplexer is designed, two compensators are used, one for each frequency transition.

Triplexers

Triplexers are just like diplexers, except they have three legs instead of two, a high pass, low pass and band pass. Two reflection compensators are required, one for the low pass to band pass gap, and one between the band pass and high pass gap. The two gap center frequencies are the geometric mean of the arithmetic width of the gap.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Band Pass Diplexers

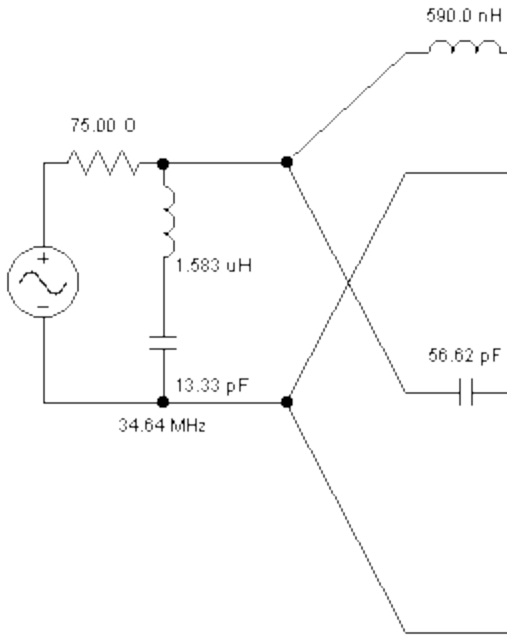
Diplexers composed of two band pass legs may be synthesized by selecting "Diplexer 1" and "Band Pass" in the Diplexer Type selection box. FilterSolutions will create three reflection compensator circuits, one for the center frequency, one above the upper band pass, and one below the lower band pass. The two outer frequency compensators are physically drawn on the diplexer legs, and are installed or removed as desired with the use of the "Compensate" selection on top of the diplexer schematic. The two center frequencies are the geometric mean of the arithmetic width of the respective band pass filters.

The center compensator conforms to the above defining equations. The outer compensators are approximations designed to minimize the reflections from the outer frequency portions of the band edges. Sometimes manual manipulations of the outer compensating elements may help improve band edge filter performance.

Diplexer Example, Low Pass/High Pass

An example of a diplexer is shown below. Only the compensator is shown, but it is composed of a 30MHz low pass filter in parallel with a 40Mhz high pass filter and has a parallel compensator. The rule of thumb is to keep the reflections outside the middle frequencies (30 to 40 MHz in this case) at least -20dB down. To achieve low reflections, it is generally necessary to set the pass band ripple to 100mdB or lower. If negative elements are generated in the diplexer, raise the dB of the pass band ripple until all elements are positive.

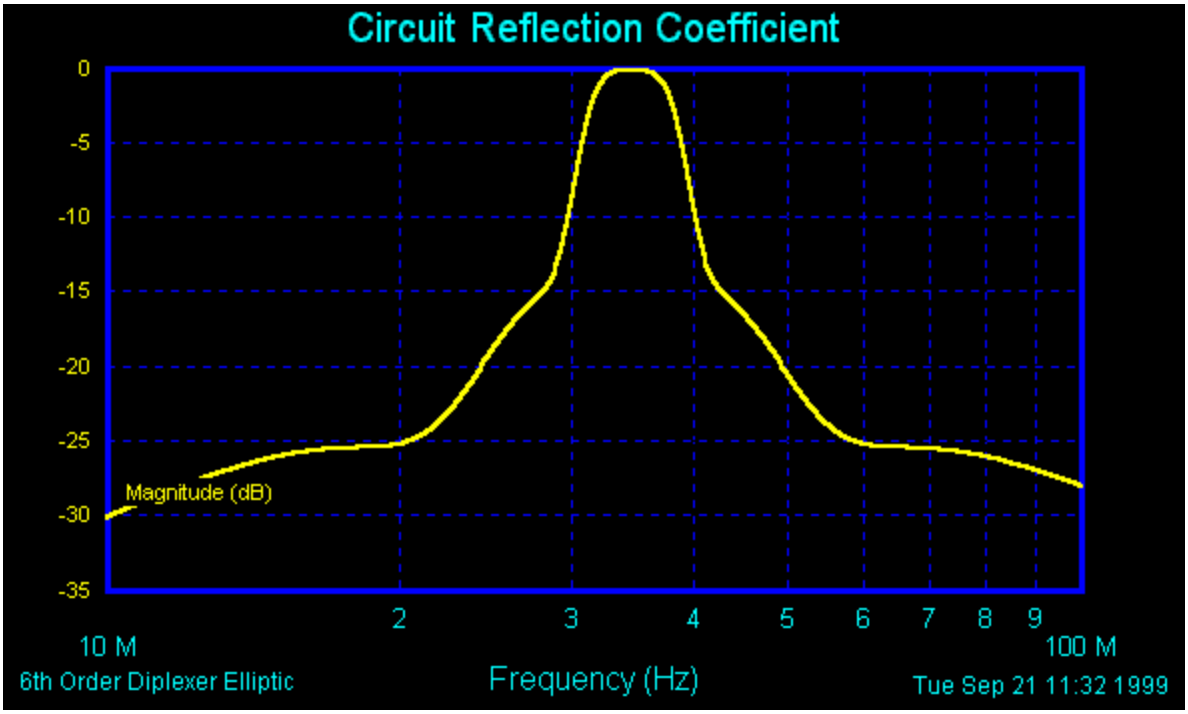
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Low Pass/High Pass Diplexer Reflection Compensation

Without the parallel compensator, this circuit would contain undesirable reflections that do not meet the -20dB rule as shown here.

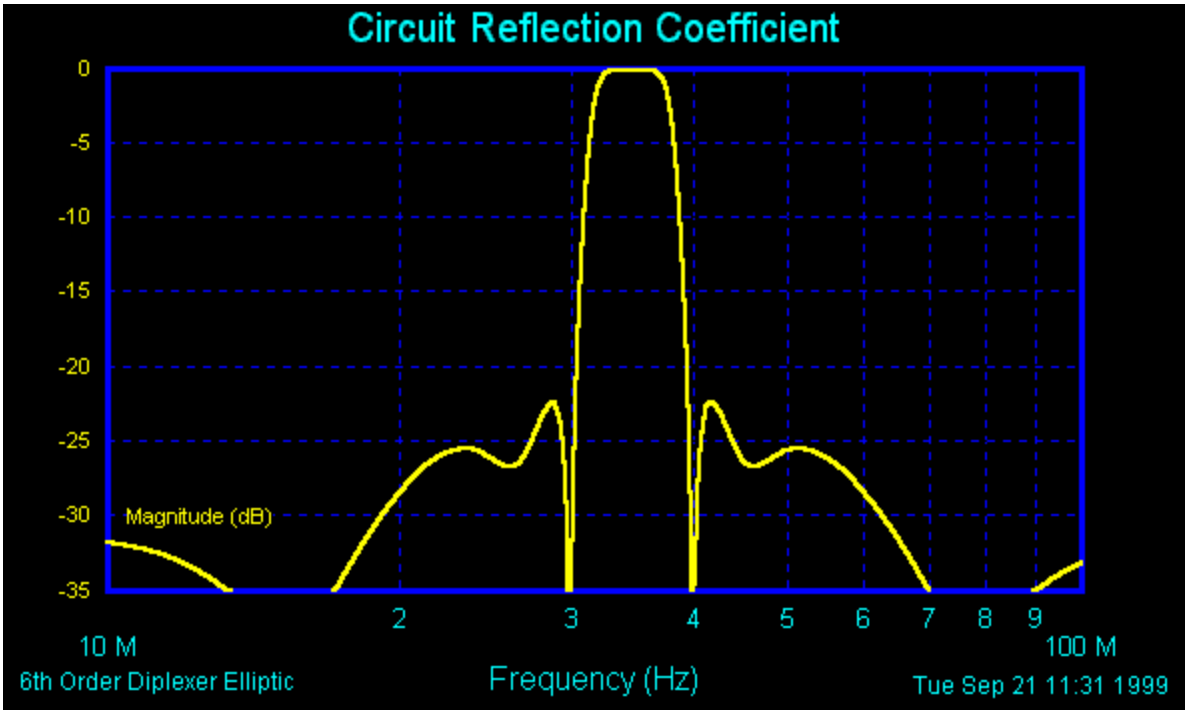
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Uncompensated Reflections

With the parallel compensator, this circuit contains much more desirable reflections that are well below -20dB outside the middle frequency area.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

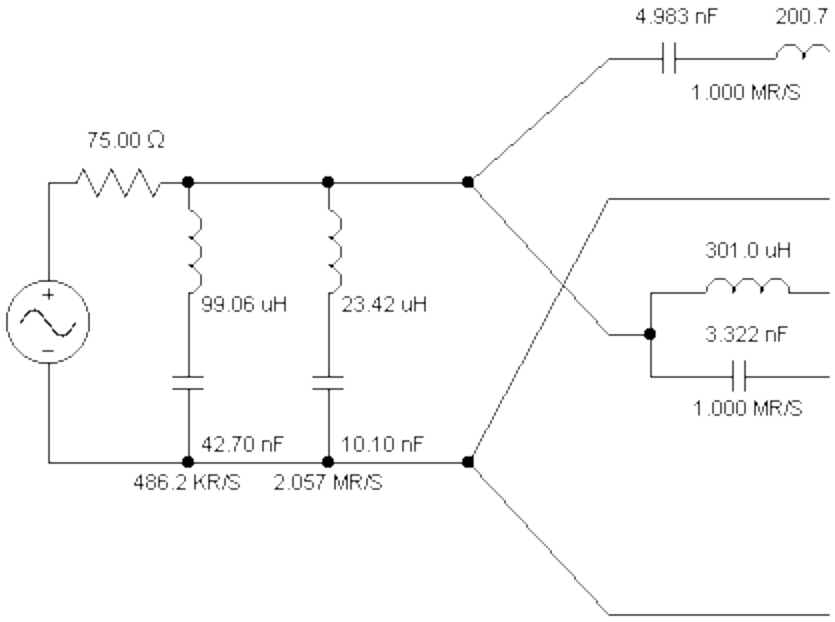


Compensated Reflections

Diplexer Example, Band Pass/Band Stop

An example of a band pass/band stop diplexer is shown below. The upper leg is a band pass filter, and the lower is a band stop filter. There are two reflection compensators, one for the low break frequency, and one for the high break frequency. The computation of the compensator element values is, however, the same as for low pass/high pass diplexers. In the case of triplexers, all computations are the same, except three legs exist to compute the compensation elements instead of two.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



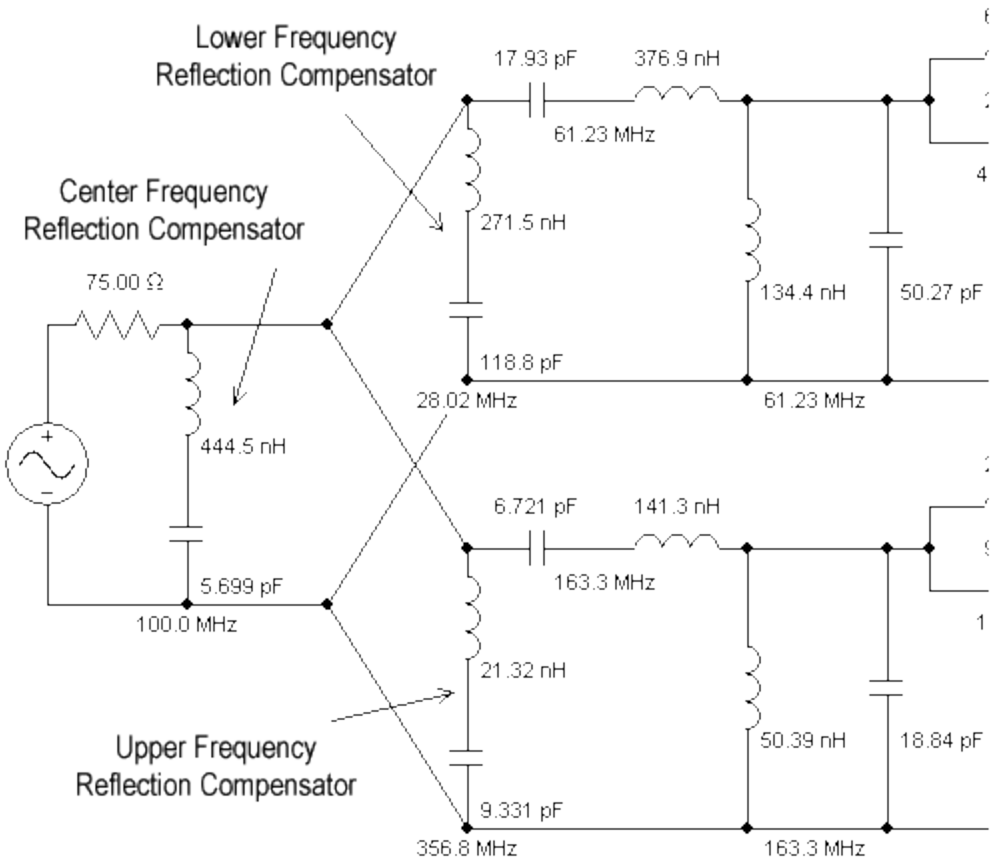
Band Pass/Band Stop Diplexer Reflection Compensation

Diplexer Example, Band Pass

Band pass diplexers may be defined by the band widths of the inner and outer corner frequencies respectively (Band Pass 1 Selection) or by the center frequency and band widths of each band pass leg (Band Pass 2 selection.)

An example of a band pass diplexer is shown below. Both legs of the diplexer are band pass filters. The leading reflection compensator is the center frequency. The leading parallel LC tank circuits on each leg are outer frequency reflection compensators.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



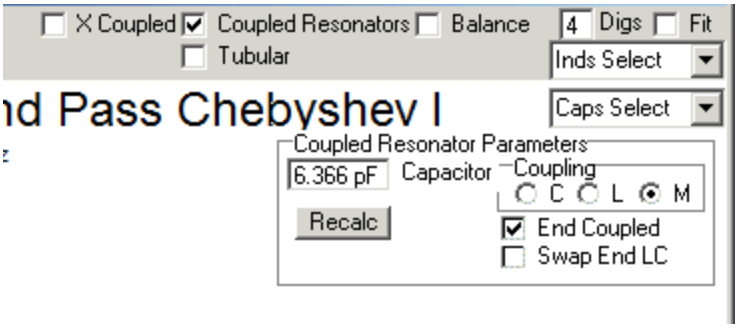
Band Pass Diplexer Example

Coupled Resonator Band Pass Filters

Coupled resonator band pass filters are narrow band approximations of band pass filters. The advantages to using coupled resonators over classical band pass filters are: more desirable element values at high frequencies, and flexible element value selections.

Coupled resonator filters are selectable by checking the "Coupled Resonators" checkbox in the band pass schematic tool bar in schematics that support coupled resonator filters, as shown below:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Coupled Resonator Filter Selection Example

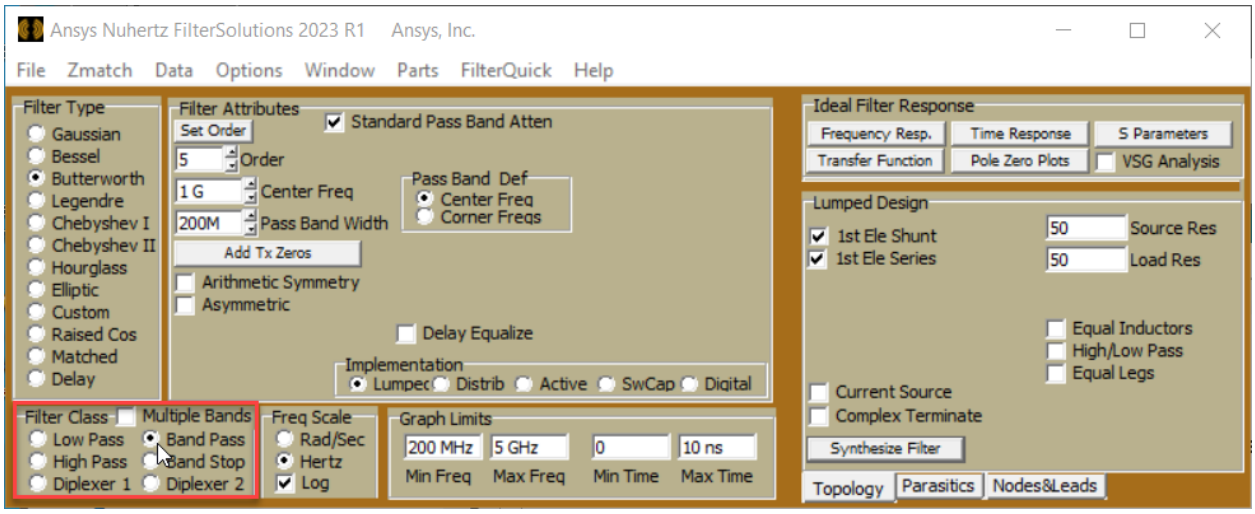
Coupled resonator filters consist of a series of LC resonant pairs or mutually coupled coils coupled together with capacitors or inductors. Usually capacitors are chosen due to the lower cost and higher performance. FilterSolutions allows one to select one of the LC resonant element values, and will calculate the other element values needed to create the desired filter.

> **Important:** To create band pass resonant filters, either Gaussian, Bessel, Butterworth, Legendre, or Chebyshev Type 1 filter type **must** be selected. Coupled resonator filters may not be created from filters with stop band zeros.

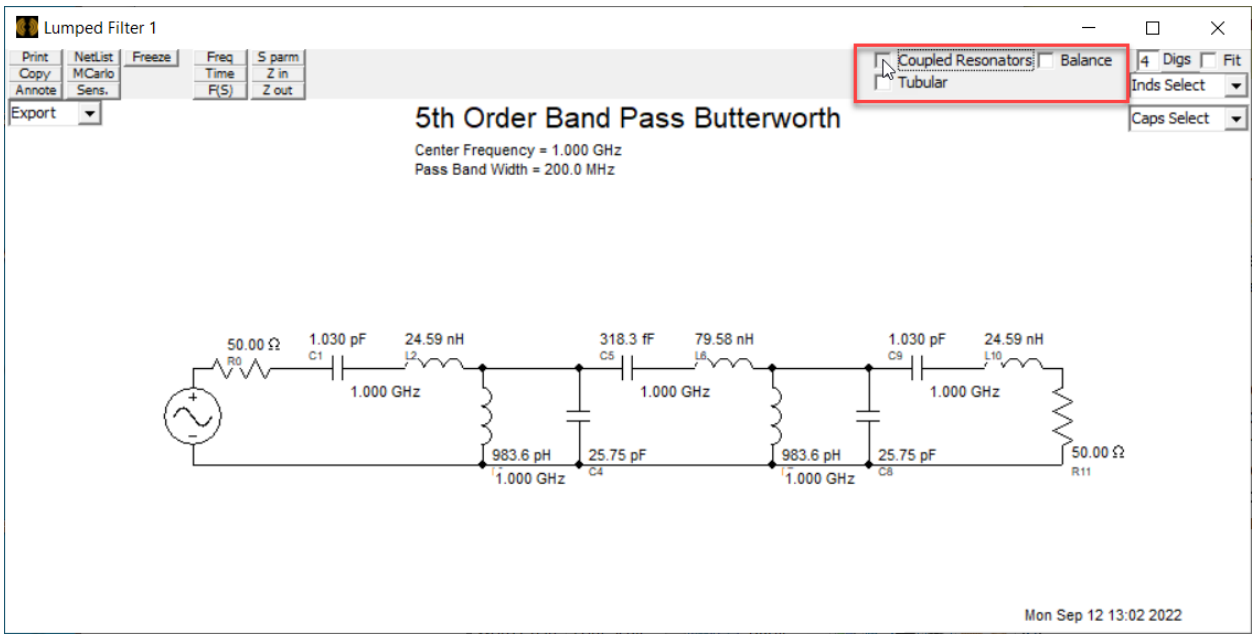
To create a band pass resonant filter:

1. Select the desired filter type from the Filter Type group box.
2. Select **Band Pass** in the Filter Class group and then create the passive circuits.

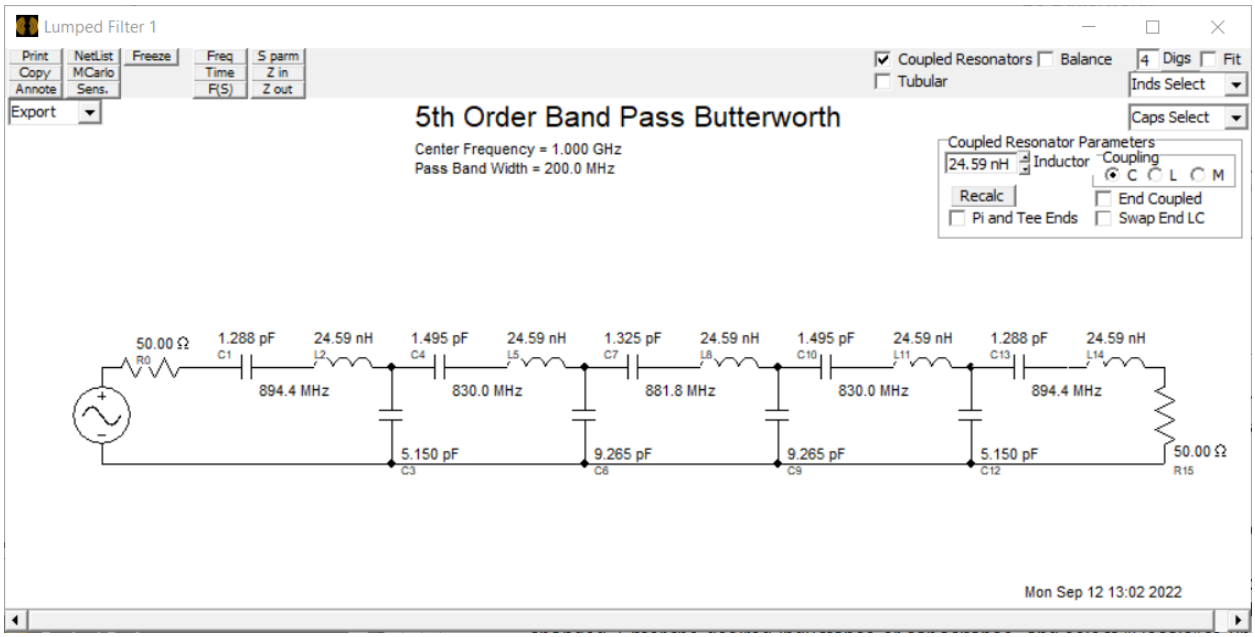
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. The passive circuit will have a check box in the menu for **Coupled Resonators** select this option.



Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



A coupled resonant band pass filter will be displayed. The first element of the filter will match the series/shunt selection in the Passive Control Panel. If the selected pass band is too wide, you will receive an error message informing you that you need to reduce the width of the pass band.

When the resonant filter is displayed, the Coupled Resonator control panel is displayed in the upper right of the circuits window. You are offered the choice of inductor, capacitor, or mutual coil coupled resonators, coupled resonators with or without end coupling elements, and you may select new resonator element values. Capacitor coupled resonators may have their inductors changed. Inductor coupled resonators may have their capacitors changed. Mutual coupled resonators may have their capacitors changed. Enter the desired inductance or capacitance, and select "Recalc" to update the filter. Coupled resonator filters without end coupling elements require third order are higher to redesign with different elements.

Swap End LC for More Symmetric Response

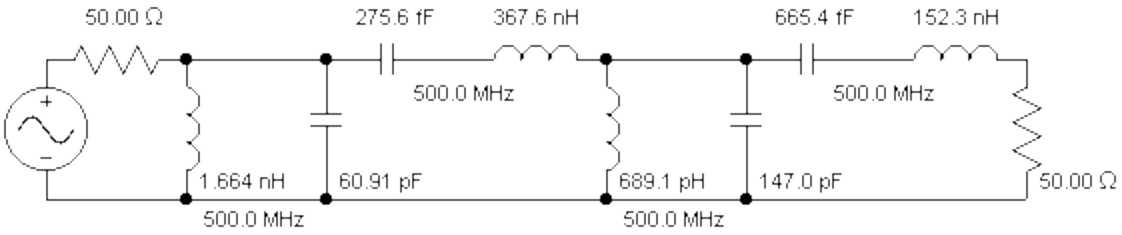
The "Swap End LC" option is available whenever "End Coupled" is checked. Coupled resonator filters have an asymmetrical frequency response characteristic that may be undesirable. If a more symmetric frequency response is desired, selecting the "Swap End LC" option will swap the positions of the outer inductors and capacitors, and apply a narrow band transformation that will restore a narrow band frequency response approximation that is more symmetric, but at the expense of slightly more error in the pass band. The additional error is generally negligible for narrow band filter, but increased for mid band filters. Visual inspection is recommended prior to using the "Swap End LC" option.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Alternatively, the right mouse key may be used to swap outer inductor and capacitors on any band pass filter with an outer inductor and capacitor of differing orientation.

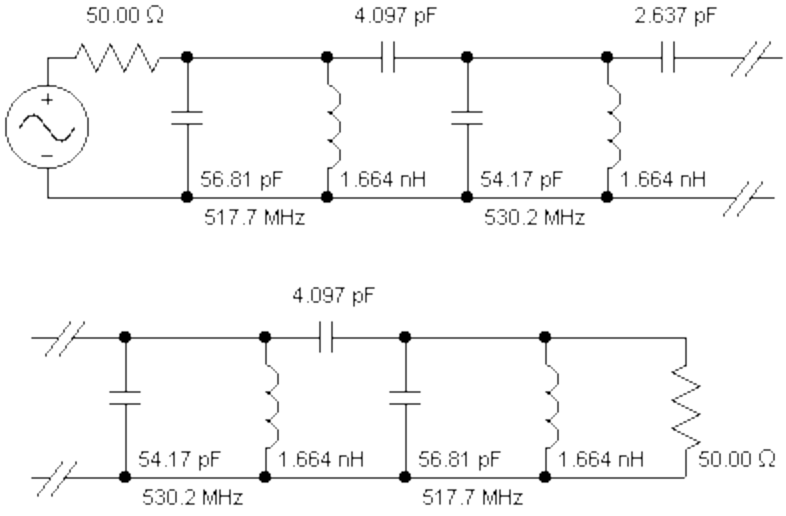
Coupled Resonator Example:

Below is a classical 4th order Butterworth band pass filter with center frequency 500MHz and a band width of 40MHz. Notice some capacitor values are less than 1 pF, and some inductor values are below 1 nH.



Classical Band Bass Filter

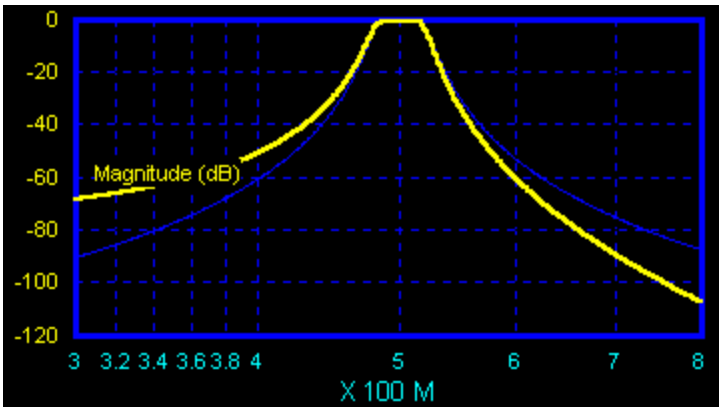
Below is the series resonance capacitor coupled band pass filter: Capacitor values are now more reasonably in the low pF range, and all inductor values are 10 nH. However, more capacitors are required to build this filter.



Capacitor Coupled Series Resonator Filter

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

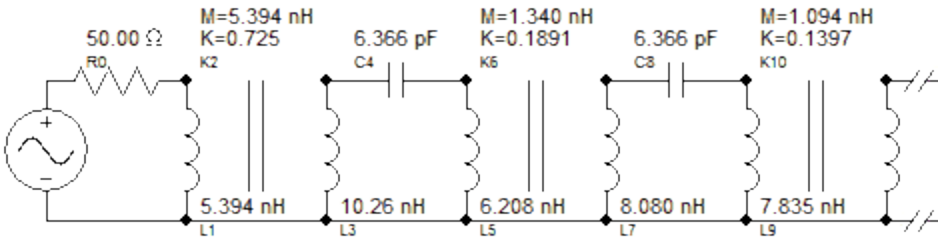
The magnitude frequency response error is shown below. Dark blue baseline traces depict the true Butterworth response. Yellow depicts the coupled resonator filter response. Note that the error is only significant at frequencies of very high attenuation.



Resonator Coupled Frequency Error

Inductor coupled and mutual coupled resonators skew in the opposite direction. End couplings reverse the skew direction from no end coupling solutions.

Mutual coupled resonators use vertical bars to denote coupled coil, which are in turn coupled with series or shunt capacitors.



Mutual Coupled Resonator Example

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tubular Band Pass Filters

> **Important:** To create band pass tubular filters, either Gaussian, Bessel, Butterworth, Legendre, or Chebyshev Type 1 filter type and Band Pass filter class **must** be selected. Tubular filters may not be created from filters with stop band zeros.

The passive circuit schematics will have a check box in the tool bar for "Tubular". Check this box, and a tubular band pass filter will be displayed. The first element of the filter will match the series/shunt selection in the Passive Control Panel. If the selected pass band is too wide, you will receive an error message informing you that you need to reduce the width of the pass band.

Tubular band pass filters are narrow band approximations of band pass filters. The advantages to using tubular over classical band pass filters are: more desirable element values at high frequencies, flexible element value selections, and all nodes are attached to a grounded capacitor. Any parasitic node capacitance that results in the physical construction of the filter may be subtracted from the design value of the grounded capacitor connected to that node. In most cases, the inductance values adjustable.

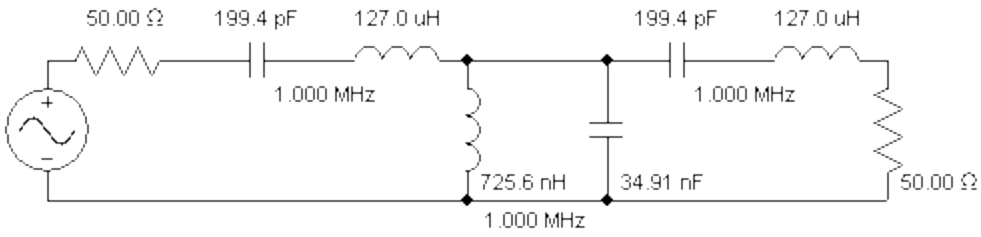
Tubular filters consist of an alternating series of inductors and capacitors, with each node containing a grounded capacitor. FilterSolutions permits tubular with or without a leading shunt capacitor, with an inductor or a capacitor first series element being, a "Minimum Elements" selection that minimizes the number of elements by skipping series capacitors, and an outer shunt inductors option to place the outer inductors shunt rather than series.

The "Minimum Elements" selection requires fewer parts, but has the disadvantage of requiring a wider range of required capacitor values.

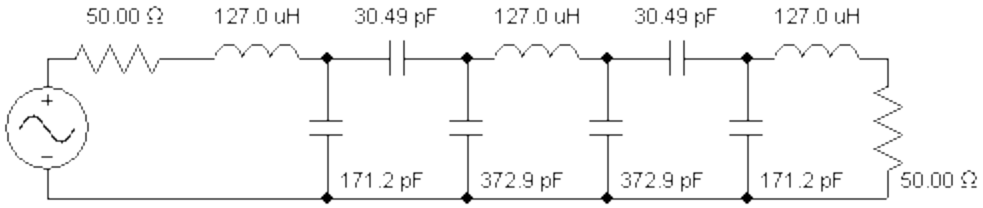
The outer shunt inductors option has the advantage of minimizing the outer frequency errors at the expense of increasing narrow band approximation errors near the center frequency.

When the tubular filter is displayed, the Tubular control panel is displayed in the upper right of the circuits window. You are offered the choice of leading inductor, leading capacitor, minimum parts, and outer shunt inductors; and you may recalculate with new tubular element values. To calculate the tubular filter with new parts, Enter the desired inductance, and select "Recalc" to update the filter. Tubular filters without leading shunt capacitors require third order or higher to redesign with different elements.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

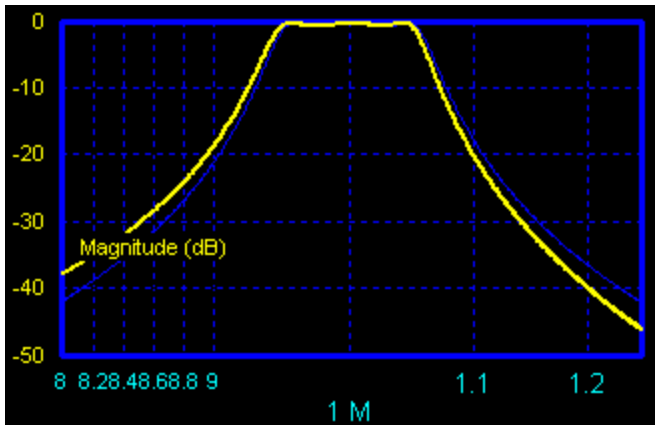


3rd Order Classicle Band Pass Filter



3rd OrderTubular Band Pass Filter
With Capacitor Ground to all Nodes

The magnitude frequency response error is shown below. Grey baseline traces depict the true Chebyshev response. Blue depicts the tubular filter response. Note that the error is only significant at frequencies of very high attenuation.



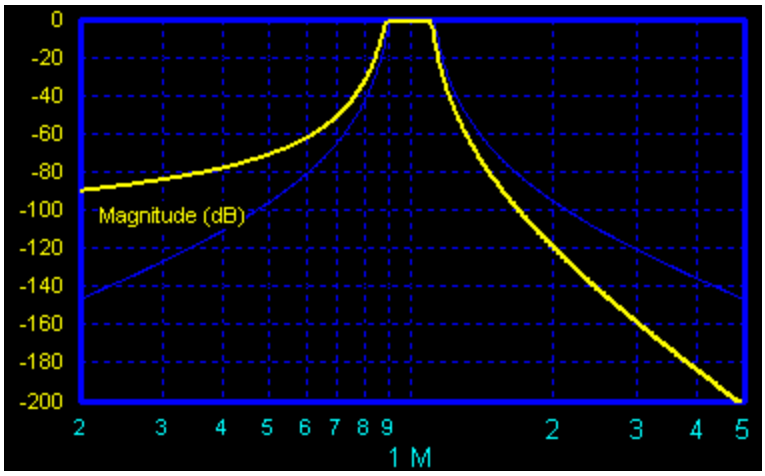
Tubular Frequency Error

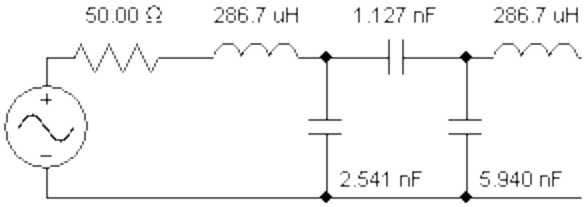
Outer Shunt Inductors

Tubular (and resonator) filters introduce an asymmetric frequency response error into band pass filters. The asymmetrical error may at times be undesirable. This asymmetrical frequency error

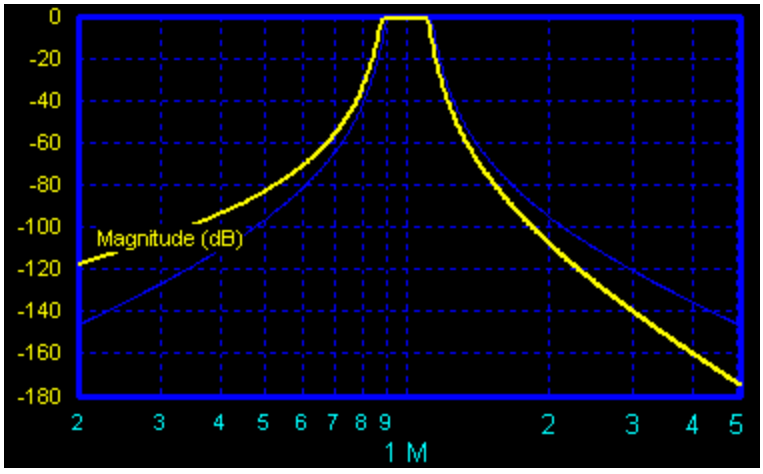
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

may be minimized by using the outer shunt inductor option. See examples of fourth order Chebyshev filters below.

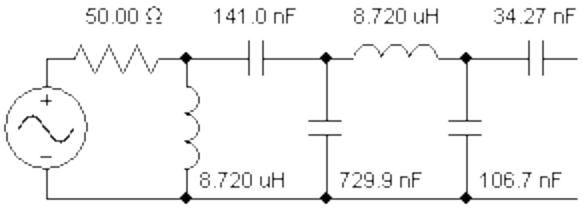




Standard Leading Series Inductor Tubular Design

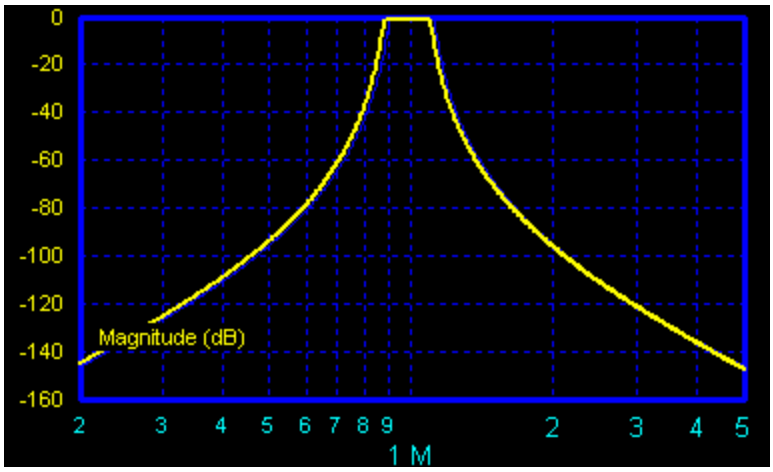


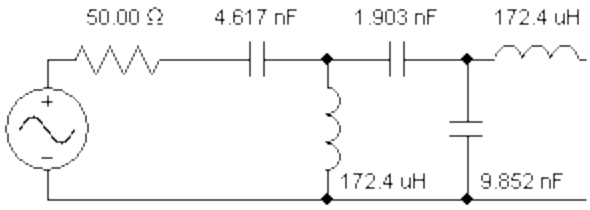
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Leading Shunt Inductor Tubular Design

With Reduced Asymmetrical Frequency Error





Leading Series Capacitor and Shunt Inductor Tubular Design

With Further Reduced Asymmetrical Frequency Error

Note that the leading series capacitor shunt inductor design is the most effective in reducing asymmetrical frequency error. However, this design produces the greatest narrow band

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

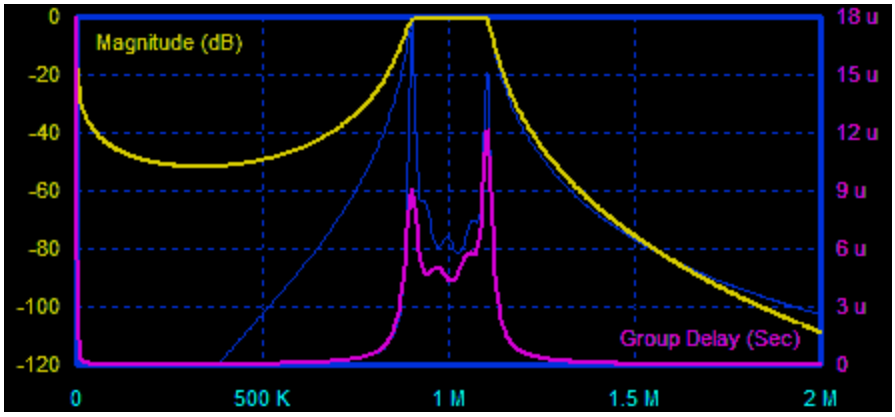
approximation error in the passband, and the default design does not have a shunt capacitor attached to the first node. A simple Norton transformation can place a shunt capacitor at the first node if desired.

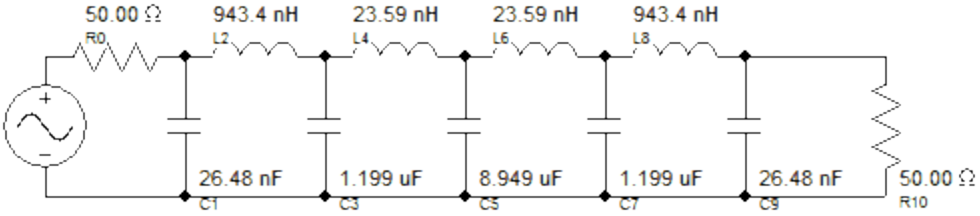
Low Pass Approximation Tubulars

This function is supported for Chebyshev I filters only. A low pass filter with a narrowly constricted passband behaves like a tubular filter in the frequency ranges about the pass band. The advantage is that a filter with fewer parts is required, and DC coupling is provided for applications that require band pass filters with DC coupling. The disadvantage is that the element sizes may differ dramatically within the filter, and many applications do not permit DC coupling.

Low pass approximation tubular filters may be useful when DC coupling and dramatically differing element values are not an issue. Such a filter may be designed manually by setting the cutoff frequency and percentage pass band constriction. The inclusion of this function into the Tubular menu is simply a matter of convenience.

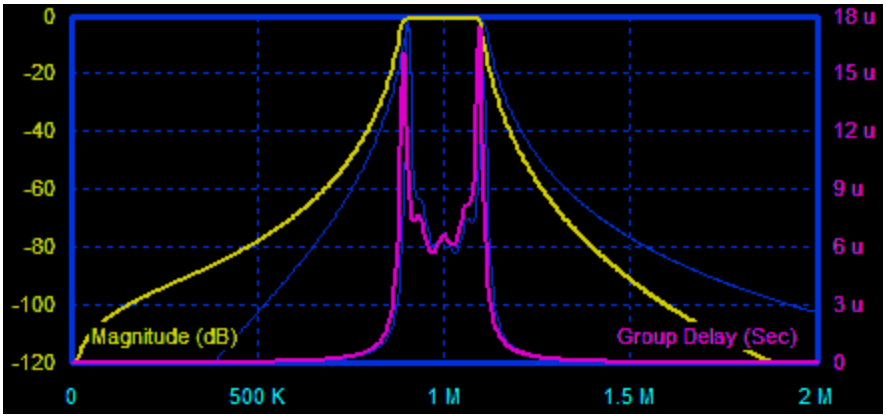
Below is an example of the low pass approximation tubular features. It is easily seen that the passband is essentially the same in both the true tubular and the low pass approximation; that the low pass approximation has far fewer elements, and that DC coupling and large part value spread exists in the low pass approximation tubular.

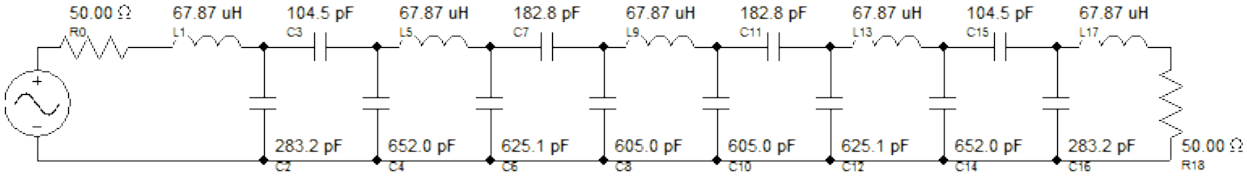




Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5th Order Low Pass Approximation Tubular





5th Order Actual Tubular

Improving Frequency Symmetry

Coupled Resonator and Tubular filters both produce asymmetrical frequency responses. In cases where this is undesirable, frequency symmetry may be improved by swapping the outer inductors and capacitors and transforming their values. This function may be performed directly by selecting the appropriate coupled resonator or tubular topology, as shown in Figure 1 below, or manually by selecting an outer element with the right mouse key.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

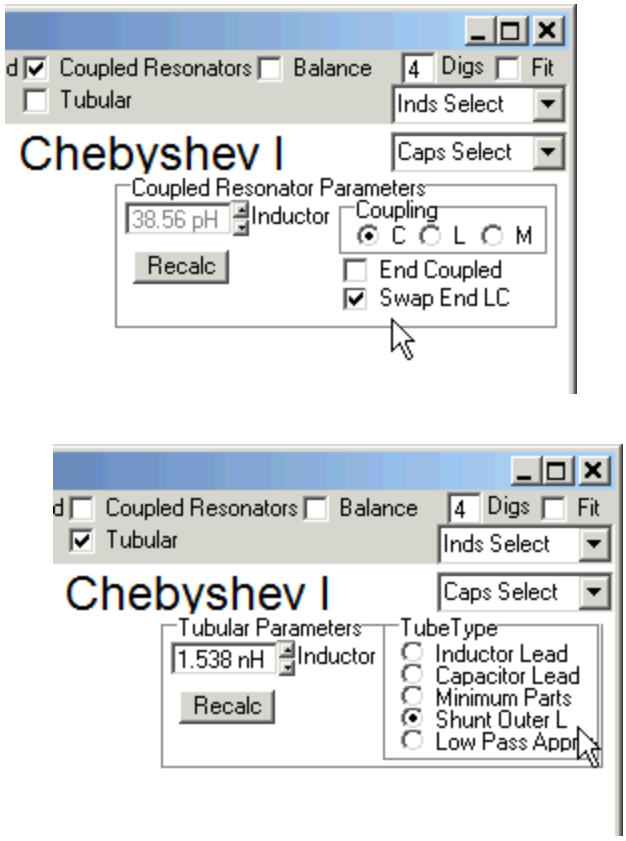


Figure 1: Coupled Resonator and Tubular LC Swapping Selection to Improve Frequency Symmetry

The effect of outer LC swapping on the frequency response is illustrated below with a seven pole top cap coupled resonator filter.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

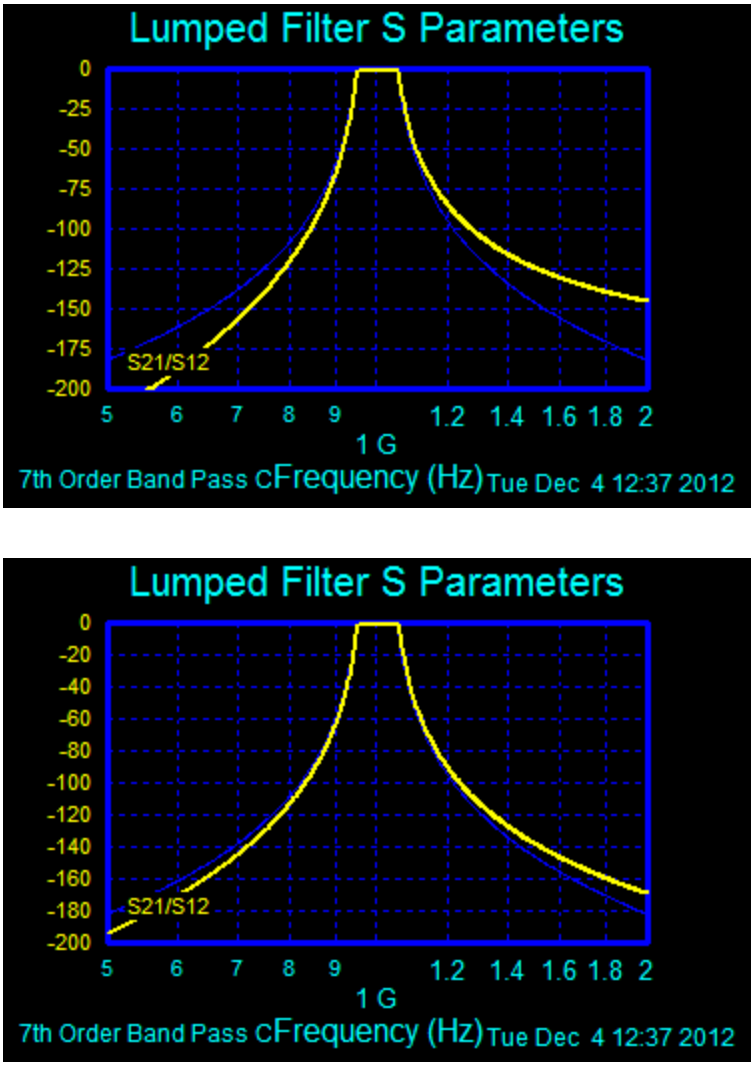


Figure 2: Effect of Outer LC Swapping on a Seven Pole Coupled Resonator Filter

Cross Coupled Resonator Filters

> **Important:** To create band pass cross coupled filters, Chebyshev Type I or Elliptic filter type, and Band Pass filter class **must** be selected.

Cross Coupled filters may be created from Chebyshev I filters with stop band zeros and off axis real and quadruplet zeros. **The passive circuit schematics will have a check box in the tool bar for "X Coupled"**. Check this box, and a cross coupled resonator band pass filter will be displayed if the filter design requirements meet all of the all the cross coupled design criteria If

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the selected pass band is too wide or other criteria is not met, you will receive an error message informing you that you need to reduce the width of the pass band or instructed to meet other criteria.

Design Flexibility

Right clicking the mouse key over a cross coupled resonator filter permits the selection of Norton Transformation of the cross couplings.

Unequally terminated filters may automatically match the load and source by checking the "Impedance Match" checkbox in the Lumped control panel on the right side of the Main panel. This has the effect of applying a Norton transformation as needed to convert an equally terminated Cross Coupled filter to an impedance matched unequally terminated filter.

Output

The full coupling matrix may be displayed by selection of the "Matrix" button in the schematic menu. The coupling matrix may then be printed or copied into other applications, as shown in Table 1 below.

0	0.08476	0	0	0	0.001248
0.08476	0	0.06018	0	-0.01325	0
0	0.06018	0	0.06758	0	0
0	0	0.06758	0	0.06018	0
0	-0.01325	0	0.06018	0	0.08476
0.001248	0	0	0	0.08476	0

Table 1, Coupling Matrix Output for 6 Pole Elliptic Bandpass

Required Criteria:

In addition to requiring Chebyshev I or Elliptic selection, cross coupled resonator filters must also meet the following:

I. Finite load and source resistance (singly terminated and unequally terminated not permitted).

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

II. Even order with even order modification. (The order must be even, 2, 4, etc., and "Even Order Mod" must be checked.)

III. Band width less than center frequency. To obtain accurate results, the band width should generally be much less than the center frequency.

IV. "Match Impedance" selected for unequal load and source resistance (Unmatched, unequally terminated not permitted).

Cross coupled resonator band pass filters are narrow band approximations of band pass filters. The advantages to using cross coupled over classical band pass filters are **more desirable and consistent element values**. The disadvantage is, of course, the need to couple inductors.

Cross coupled filters consist of cascaded sequence of vertical or circular LR resonators with at least the adjacent inductors coupled together. If stop band zeros are desired, such as in Elliptic filters, nonadjacent inductors are coupled as well, although the filter layout may be arranged to make all coupled inductors adjacent. Couplings may be displayed as coupling coefficients (-1<K<1) or mutual inductances (M = K*sqrt(L1*L2) at the users discretion.

All pass stages for delay equalization

Finite Q Analysis

FilterSolutions cross coupled resonator filters support finite Q analysis and compensation. However, as with all narrow band filters, finite Q compensation may have a negligible effect on performance. For analysis, FilterSolutions assumes that the mutual impedance resulting from finite Q is handled the same as mutual inductance, that is, Zm = K*SQRT(Z1*Z2).

Sensitivity and Monte Carlo Analysis

Cross coupling coefficients and mutual inductance is supported for both sensitivity and Monte Carlo analysis. The sensitivity chart shows entries for coupling coefficients (Kx_x) or mutual inductance (Mx_x) depending which mode the user has selected. Monte Carlos are selected by selecting an inductor, then selecting, "Random", "Update All Inductors" and "Update All Couplings" in the change control panel. Set the inductor tolerance to "0" to analyze just the cross couplings.

Optional Views

FilterSolutions supports three independent optional views of cross coupled filters:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Full Coupling Matrix View

This view displays the resonators with all non-zero coupling coefficients below with integer references to specific inductors that are coupled. The advantage of this view is that the full coupling matrix may be edited if desired. Passing the cursor over a vertical column of coefficients and selecting the left mouse key permits the editing of all coupling coefficients, even the zero ones, as shown in Figure 1 below.

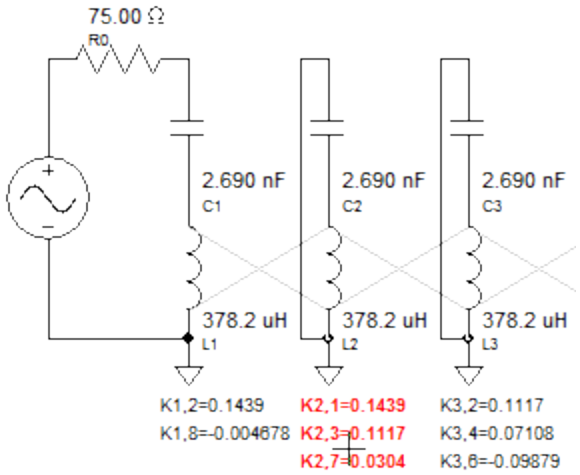


Figure 1: Editing the General Coupling Matrix

Specific Coupling Matrix View

This view displays only the couplings needed to create the filter, namely all adjacent couplings and necessary couplings of nonadjacent inductors needed to realize transmission zeros. Only the displayed couplings may be edited, as shown in Figure 2 below.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

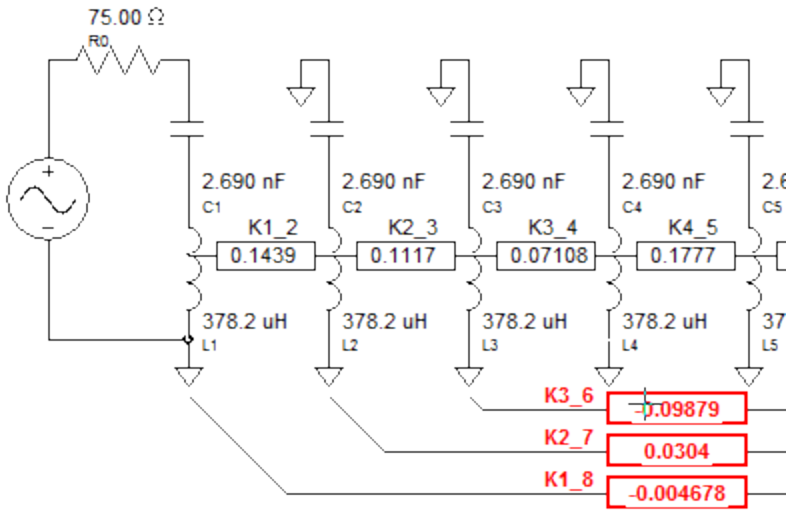


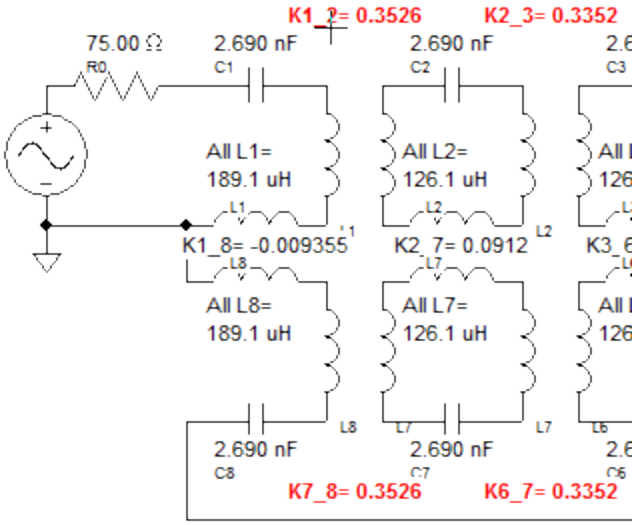
Figure 2: Editing the Specific Coupling Matrix

Adjacent Coupling Topology View

This view is like the Specific Coupling Matrix View, except that the resonators have been folded over to make couplings that were nonadjacent, in the Specific Coupling Matrix View, adjacent. Adjustments are made in the inductor and coupling coefficient values that are needed to maintain the frequency response. View 3 is highly desirable in that it may be used to design a physically realizable filter with coupled inductors all adjacent with each other.

The displayed couplings in this view may be edited just as in the Specific Coupling Matrix View, as shown in Figure 3.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Quadruplet Coupled Resonator Band Pass Filters

Full cross coupled filters have the disadvantage in that it may not always be realizable for all desired technologies. It is sometimes convenient to remove some resonator couplings so that only series quadruplet couplings remain. selecting, "Half Band Ripple" stop band in the main control panel. removes transmission zeros from the stop band, partly to enable the construction of he filter with series quadruplet couplings. Selecting "Single Point Ripple" in the stop band results in the symmetric construction of the filter. Deselecting, "Single Point Ripple" in the stop band results in a flat stop band, but requires an asymmetrical construction of the filter.

FilterQuick

In the FilterQuick interface, options exist for "Cross Coupled" and "Quadruplet Coupled" selections, with the option of a "Single Point Ripple" feature. Half Band Ripple is already selected for you when using the FilterQuick interface.

Coupling Matrix Differences

An 8 pole Elliptic is shown below in both standard form and in quadruple coupled form in the Tables 1 and 2 ad Figures 1 and 2 below:

Cross Coupled:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0	0.08216	0	0	0	0	0	-0.0009074
0.08216	0	0.05891	0	0	0	0.007371	0
0	0.05891	0	0.04518	0	-0.03506	0	0
0	0	0.04518	0	0.08148	0	0	0
0	0	0	0.08148	0	0.04518	0	0
0	0	-0.03506	0	0.04518	0	0.05891	0
0	0.007371	0	0.04986	0	0.05891	0	0.08216
-0.0009074	0	0	0	0	0	0.08216	0

Table 1: 8 Pole Elliptic Coupling Matrix in Standard Cross Coupled Form

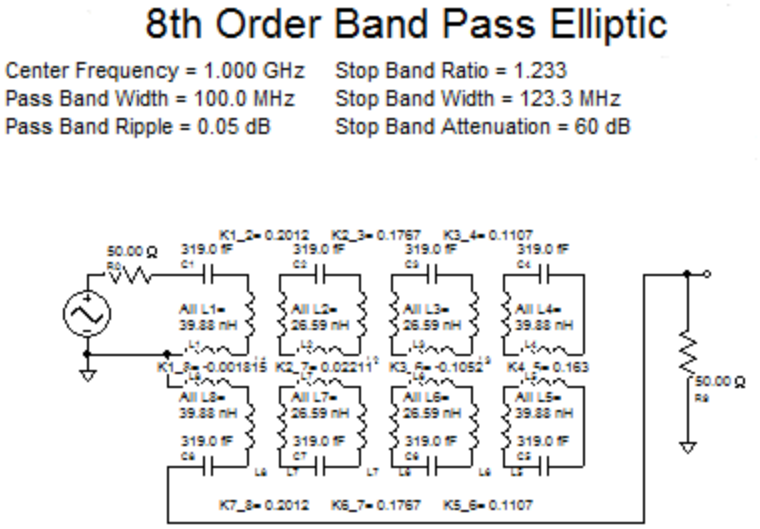


Figure 1: 8 Pole Elliptic Coupling Matrix Implementation in Standard Cross Coupled Form

Quadruplet Coupled with Single Point Ripple Stop Band:

0	0.08023	0	-0.01828	0	0	0	0
0.08023	0	0.07112	0	0	0	0	0
0	0.07112	0	0.05004	0	0	0	0
-0.01828	0	0.05004	0	0.05364	0	0	0
0	0	0	0.05364	0	0.05004	0	-0.01828
0	0	0	0	0.05004	0	0.07112	0
0	0	0	0.04986	0	0.07112	0	0.08023
0	0	0	0	-0.01828	0	0.08023	0

Table 2: 8 Pole Elliptic Coupling Matrix in Series Quadruplet Coupled Form

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

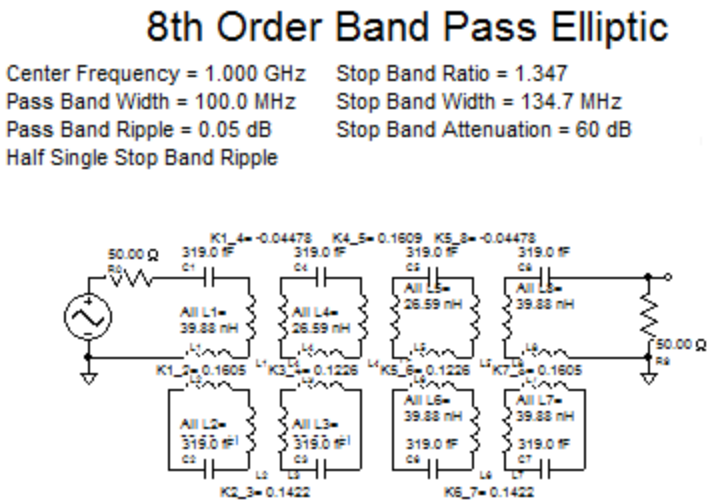


Figure 2: 8 Pole Elliptic Coupling Matrix Implementation in Series Quadruplet Coupled Form

Bridged Tubular and Resonator Filters

Bridged tubular filters and bridged coupled resonator filters may be used to synthesize delay equalized Chebyshev I filters without the need for all pass stages or coupled inductors. More information may be found under "Delay Equalizers - Off Axis Real and Quadruplet Zeros" section of the Help, or the short Help video, "Bridged Element Delay Equalization".

Bridged tubular filters consist of cascaded sequence PI capacitor sections separated by horizontal inductors. A series of one or more Bridge Capacitors connect the PI section on opposite sides of the filter, and are used to formulate off axis zeros, as shown below in Figure 1.

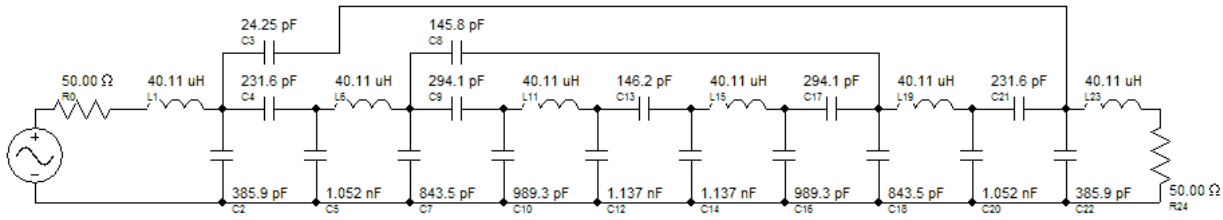


Figure 1. Bridged Tubular Topology Example

Coupled resonator filters are similar in that bridge capacitors or inductors are placed across the coupled resonator topology.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Important: To create band pass bridged tubular or bridged resonator filters, Chebyshev Type I Band Pass **must** be selected.

Bridged tubular filters may be created from Chebyshev I filters with stop band zeros and off axis real and quadruplet zeros.. **The passive circuit schematics will have a check box in the tool bar for "Tubular"**. Check this box, and a bridged tubular resonator band pass filter will be displayed if the filter design requirements meet all of the all the bridged tubular design criteria If the selected pass band is too wide or other criteria is not met, you will receive an error message informing you that you need to reduce the width of the pass band or instructed to meet other criteria.

Design Flexibility

Unequally terminated filters may automatically match the load and source by checking the "Impedance Match" checkbox in the Lumped control panel on the right side of the Main panel. This has the effect of applying a Norton transformation as needed to convert an equally terminated bridged tubular or resonator filter to an impedance matched unequally terminated filter.

Inductor or capacitor values may be manually selected after synthesis to resynthesize around all but the first and last inductor.

Outer inductors or capacitors may be shunted to improve or change frequency response asymmetry.

Required Criteria:

In addition to requiring Chebyshev I or Elliptic selection, bridged tubular resonator filters must also meet the following:

I. Finite load and source resistance (singly terminated and unequally terminated not permitted).

II. Even order with even order modification. (The order must be even, 2, 4, etc., and "Even Order Mod" must be checked.)

III. Band width less than center frequency. To obtain accurate results, the band width should generally be much less than the center frequency.

IV. "Match Impedance" selected for unequal load and source resistance (Unmatched, unequally terminated not permitted).

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Bridged tubular and resonator band pass filters are narrow band approximations of band pass filters. The advantages to using bridged tubular or resonator filters over classical band pass filters are **more desirable and consistent element values**.

Minimum Inductor Zigzag Filters

Band pass filters with stop bands (Chebyshev II, Hourglass and Elliptic) may have their inductor count reduced through the use of a zigzag filter implementation. A disadvantage to zigzag filters is the source resistance of the original design changes, and equal resistance terminations and single terminations are not possible in the design of the zigzag filter without introducing small errors.

Even Order Zigzags

Even order Zigzag filters modify the band pass function with the "Even Order Mod" function, which is automatically selected. The even order modified band pass function is then exactly reproduced. **Even order Zigzag filters require N inductors,** where N is the prototype order of the filter. Extremely narrow pass bands and extremely large order filters (>10) are more easily achievable when the design source resistor is equal to the design load resistor.

Odd Order Zigzags

Odd order Zigzag filters only achieve a close approximation to the ideal band pass function in that the transfer function must be modified to relocate two reflection zeros from the imaginary axis to the real axis, creating a parametric filter. **Odd order Zigzag filters require N-1 inductors**, where N is the prototype order of the filter. Note that odd order Zigzags require fewer inductors than even order. Generally, extremely narrow pass bands or extremely large order filters (>10) are not as achievable as even order equal design terminated Zigzag filters at this time.

Setting the Source Resistor

Pure Zigzag filters end up with a different source resistance than what is designed for, and equally terminated Zigzag filters are not realizable. It is, however, possible to set the source resistance to any desired value by adding one capacitor and performing a Norton transformation. This is step is performed automatically by checking the "Set Source Res" checkbox in the Passive Control Panel. If the desired terminations are unequal, it is sometimes possible to set the source without adding a capacitor by experimentally entering different design source resistances until the desired actual source is obtained.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Equal Inductors

Zigzag filter may be designed with only one inductor value for the entire filter. This comes at the expense of requiring more capacitors, and in the case of even order, setting the source resistance to a specific value. For odd order Zigzags, the single inductor value is selectable by the user within a specified range displayed in the upper right corner of the filter schematic. Setting a custom inductor value is generally at the expense of requiring the addition of one more capacitor. For even order Zigzags, the source resistance may be set at the expense of requiring two inductor values instead of one.

Equal inductor Zigzags generally **require large stop band attenuations or ratios** in order to achieve an all-positive element value design. Stop band ratios should be set generally within 2.5 to 4.0. If the Zigzag design requires small stop band ratios, then equal inductor designs are generally not realizable and cannot be used.

Negative Element Values

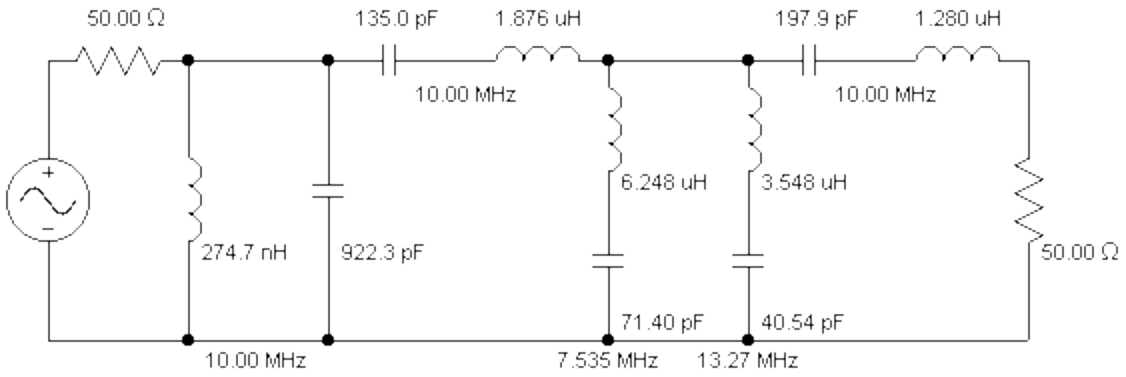
Zigzag filters occasionally produce negative element values, especially the odd order Zigzags. When this happens, the corrective action is to increase the pass band ripple or stop band ratio or attenuation until the negative values turn positive. If equal inductors is selected, it may be necessary to deselect equal inductors.

To create a Zigzag filter, select Chebyshev II, Hourglass, or Elliptic filter; nonzero source resistance, and check the "Zigzag" check box in the passive control panel.

Zigzag Filter Example

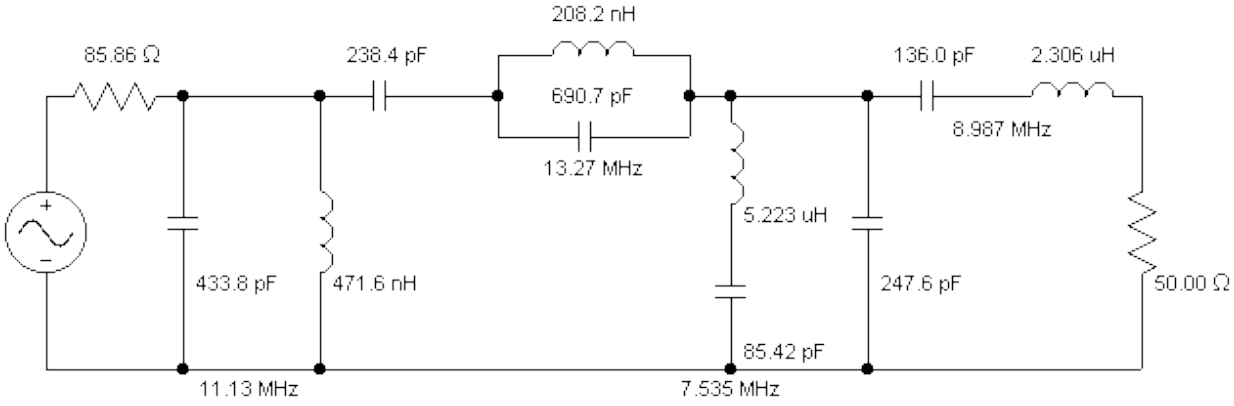
The following is a 4^{th} order classical Elliptic Filter. Note that there are five inductors.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Classical Band Pass Filter

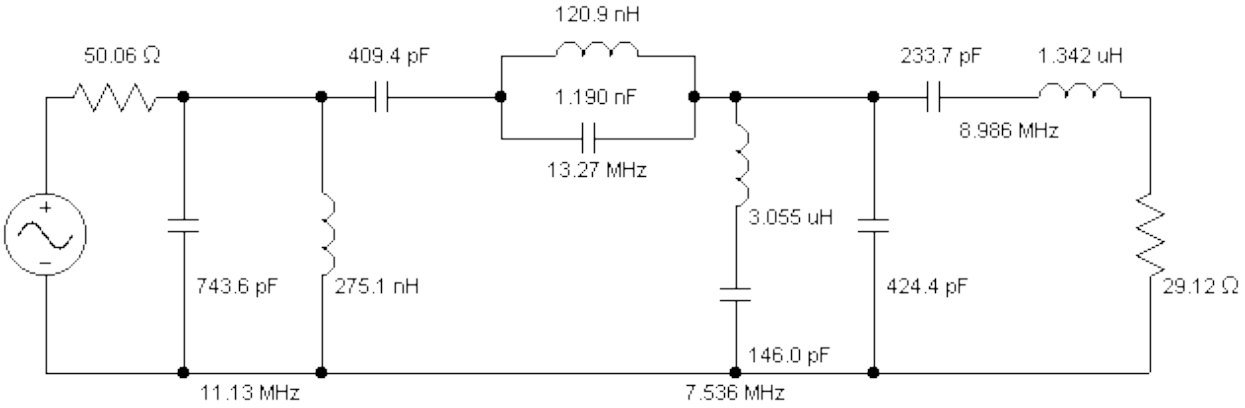
The following is an example of an equivalent 4th order zigzag filter. Note that there are only four inductors, and the source resistance is no longer 50 ohms.



Minimum Inductor Zigzag Filter

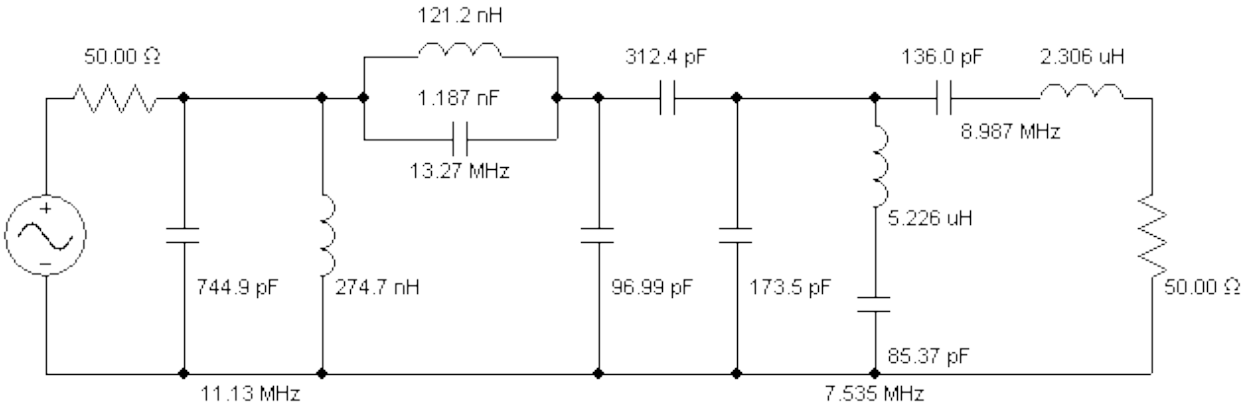
The following example is the same zigzag filter as above, but modified for equal source and load terminations. The price paid to set both the source and load resistors to arbitrary values is the additional 96.99 pF capacitor that appears in the middle of the filter. Note that the front half of the filter is impedance scaled with a Norton Transformation.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Equally Terminated Minimum Inductor Zigzag Filter

If only the source resistance requirement is fixed, it is possible to hand calculate the load resistance needed to create the required source resistance. In the above example, entering 29.12 ohms (50*50/85.86) as the load and source resistance will produce a zigzag filter with a 50 ohm source resistance, as seen below:

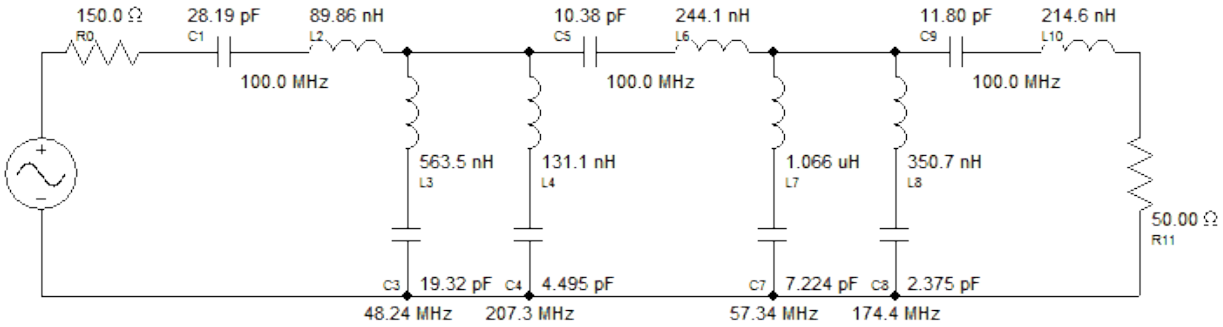


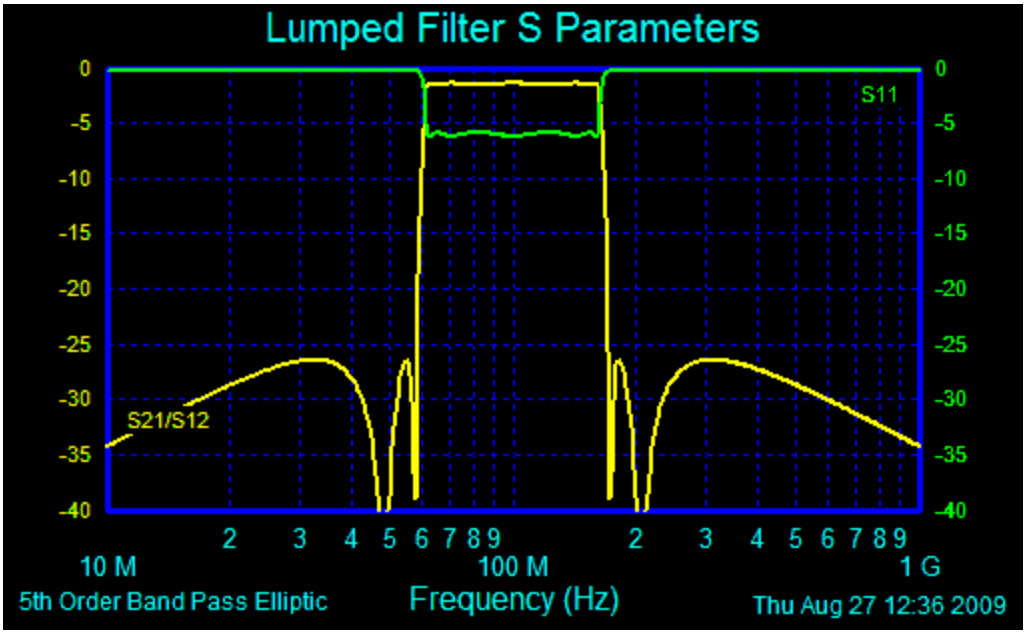
Zigzag Filter Adjusted to Achieve Only 50 ohm Source Resistance

Matching Unequally Terminated Filters

Filters designed with unequal terminations tend to have undesirable reflections (S11 and S22) that may not meet design specifications, as may be seen below in a classic unequally terminated **150 Ohm source and 50 Ohm load** Elliptic band pass filter. Note the poor S11 performance, as shown below in the termination mismatched 5th order classical Elliptic design.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





Impedance Mismatched filter and S12/S11

If the case of band pass filters, this potential problem is almost always correctable with the application of one or more Norton transformations to a comparably designed filter designed with equal terminations.

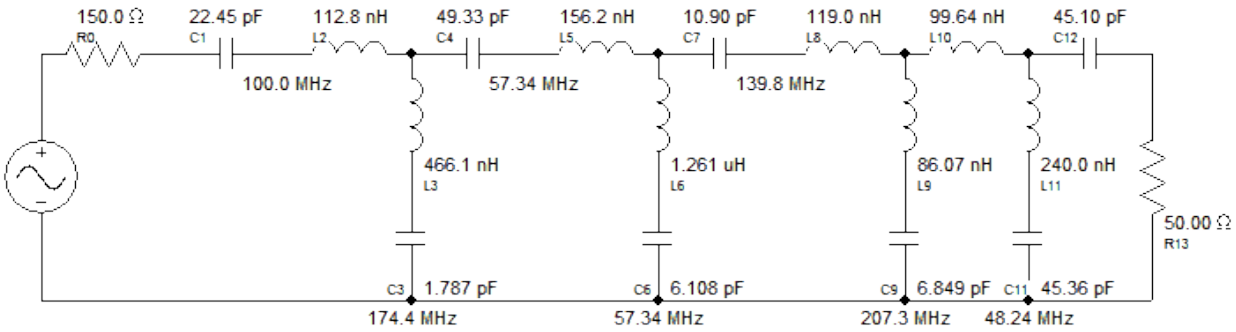
Norton transformations may be applied manually with the use of the right mouse key on any PI, TEE, or L set of like elements, PI arranged parallel LC resonators, and Tee arranged series LC resonators. Applying manual Norton transformations sometimes requires the rearranging of parts with the use of the right mouse key to get them into the needed PI, Tee, or L formation before a Norton may be applied.

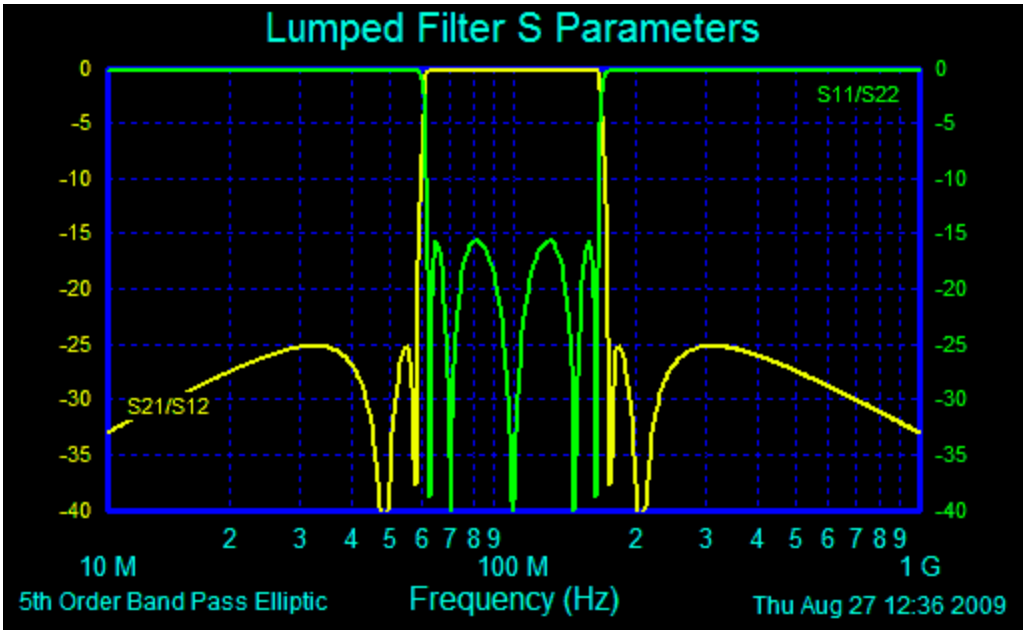
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Automatic Matching of Unequal Terminated Band Pass Filters

Checking the "Match Impedance" check box in the lumped control panel on the right side of the Main control panel has the effect of designing an equally terminated filter and applying the necessary Norton transformation to create the specified termination resistors. This is sometimes a significant time saver when multiple Norton transformations are needed and/or when element s require significant rearranging to get them into the required Norton configuration.

Below is the same classical band pass filter that has been automatically matched by internal rearrangement of parts and strategic application of Norton transforms. **Note the source is still 150 Ohms, and the load is still 50 Ohms.**





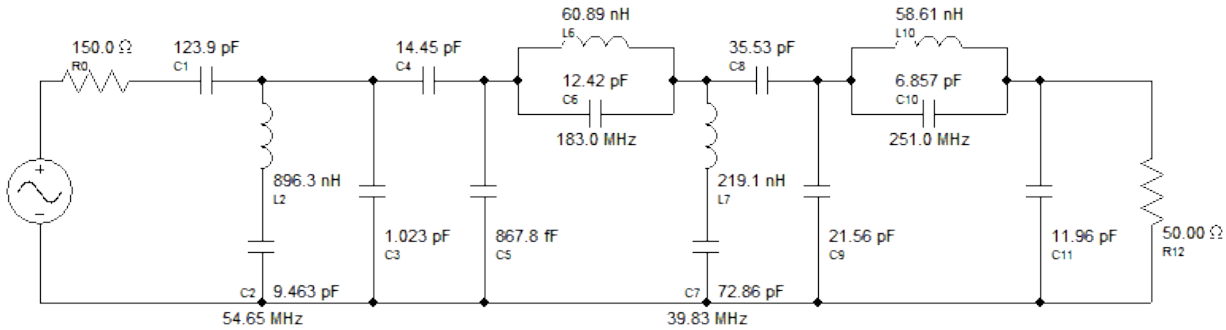
Impedance Matched filter and S12/S11

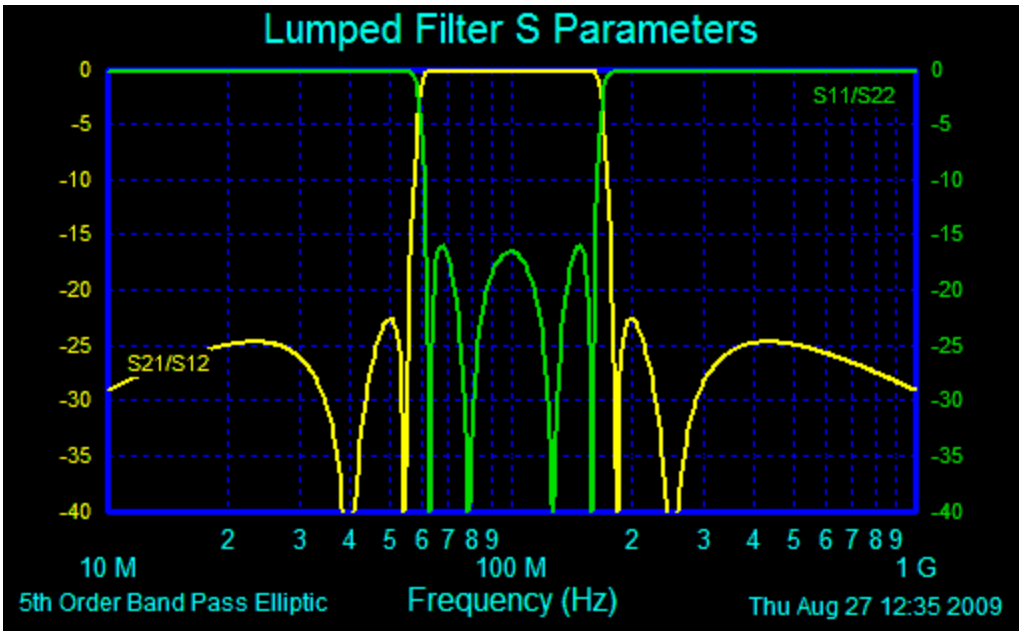
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note the greatly improved S11 performance through automatic impedance matching of mismatched terminations.

Matching Minimum Zigzag Filters

It should be noted that the above filter is more efficiently designed with a Zigzag topology as shown below, which would require only four inductors for both the matched and unmatched versions, instead of the seven unmatched and eight matched inductors needed for the classical design. **Source and loads are still 150 Ohm and 50 Ohm respectively.**





Impedance Matched Zigzag Filter and S12/S11

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

With only Four Inductors

Low Pass, High Pass, and Band Stop Filters

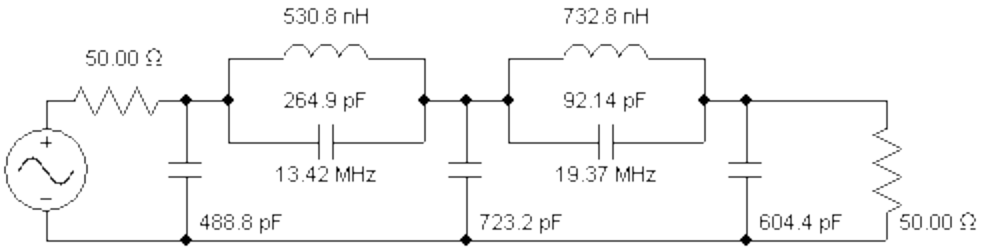
Any filter may have mismatched loads and sources matched up by adding a coupled coil transformer anywhere in the matched network, and applying a Norton transformation to the transformer. Adding a transformer to a network also blocks frequencies at DC and infinity, so if these frequencies must be passed, the transformer matching scheme will not work.

Transformers are added by right clicking any component in the network, and selecting **Add New** and **Transformers** from the pop-up window.

Bridged T Topologies

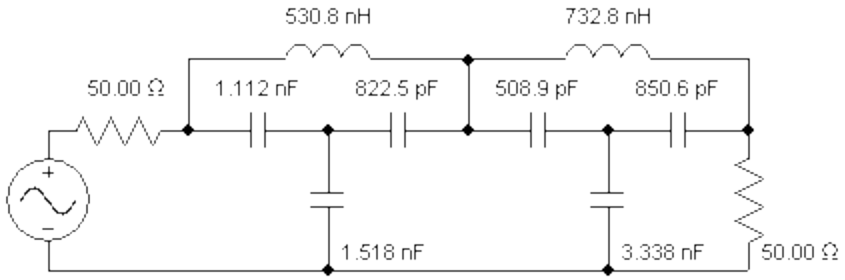
FilterSolutions offers the flexibility of synthesizing low pass and high pass stop band filters, that is, filters with transmission zeros in the stop band, with Bridged T topologies, named for the physical layout of what resembles a "T" with a bridge across the top. The classic LC tank topology is generally superior to the Bridged T topology in that fewer parts are required and low pass filters contain a shunt capacitor at all nodes. However, the Bridged T tends to require larger capacitor values. In situations where small capacitor values are impractical or not available, such as < 1pF, Bridged T topologies may be more desirable in that they can be synthesized with larger capacitors.

Below is an example of a 5th order Elliptic filter designed with both classic LC Tank and Bridged T topologies. Both are identical in frequency response and performance.



Classic LC Tank Topology

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Bridged T Topology

Note that the classic LC Tank topology requires one less capacitor than the Bridged T, and that the class LC Tank topology has a capacitor at each node that may be used to absorb parasitic node capacitance. In general, one would select the above classic LC design over the Bridged T.

However, if capacitor values below 500pF are not available for any particular reason, then the classic LC Tank design is not easily realizable. In this case, the Bridged T design may be more desirable.

Equal Inductor Band Pass Filters

All Pole Filters

All pole band pass and Zigzag filters nay be constructed using one inductor value only. See the Zigzag section for information regarding Zigzag equal inductor filters. For all pole filters, that is, filters without stop band zeros, the unequal inductor topology may be transformed into an equal inductor topology. Unlike odd order zigzag filter, the all-pole band pass filter requires a fixed single inductor value that is not user selectable. In the case of odd order equally terminated filters, this transformation has no effect on the source resistance. For even order and unequally terminated filters, the source resistance changes. If the source resistance is required to be a fixed value, the "Set Source Res" option will synthesize the filter with two inductor values instead of one.

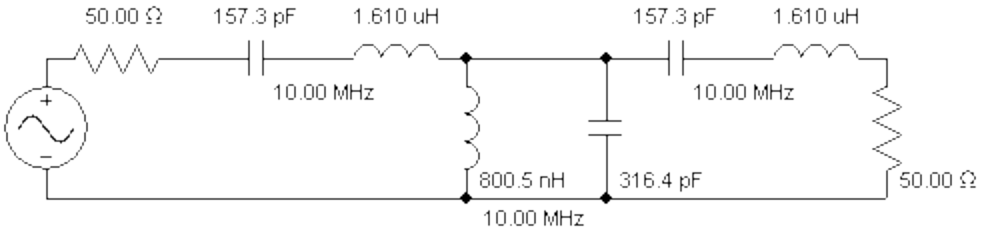
Synthesizing all pole filters with one inductor value has the obvious advantage of requiring fewer expensive inductor values. It also has the advantage that designs can be produced that place a shunt capacitor at each node. Shunt node capacitors may be used to absorb parasitic node capacitance and eliminate or minimize distortion due to parasitic node capacitance. For medium or wide band filters, this is especially desirable, because tubular filters require narrow bands and cannot be synthesized for medium and wide bands.

Medium and wide band equal inductor band pass filters sometime require negative element values. When this happens, it may be possible to remove them by narrowing the band width or

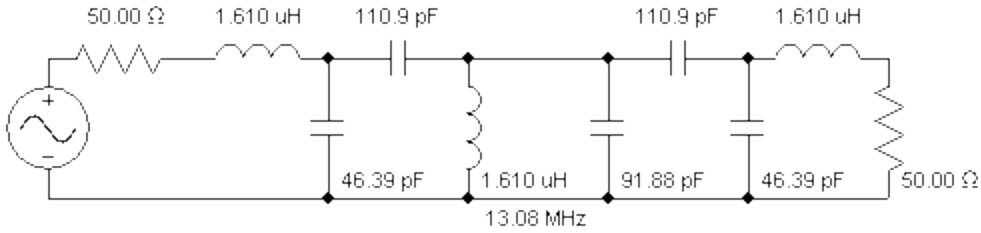
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

order. If it not feasible to narrow the band width or order, equal inductor will probably not work for the application and the option should be deselected.

The example below is a 3rd order Chebyshev with 10MHz center frequency and a 10MHz band width.



Classic Unequal Inductor Design



Equal Inductor Design with One Inductor Value

Note that the equal inductor design above contains only one inductor value and contains a shunt capacitor at every node. The expense for these nice features was the addition of two extra capacitors. Also note that physical symmetry is maintained in the equal inductor design.

Stop Band Filters ("Parametric" Filters)

Stop band filters of Chebyshev II, Hourglass, and Elliptic filters may have all shunt inductors or all series inductors set to the same value. Such filters are sometimes referred to as "Parametric". Equal inductor designs for stop band filters sometimes improves the inductor spread ratio for medium and narrow band filters, and in the case of shunt inductors, has the added benefit of providing a shunt capacitor to all nodes so that parasitic node capacitance may be easily absorbed. The price is that a few more inductors are required.

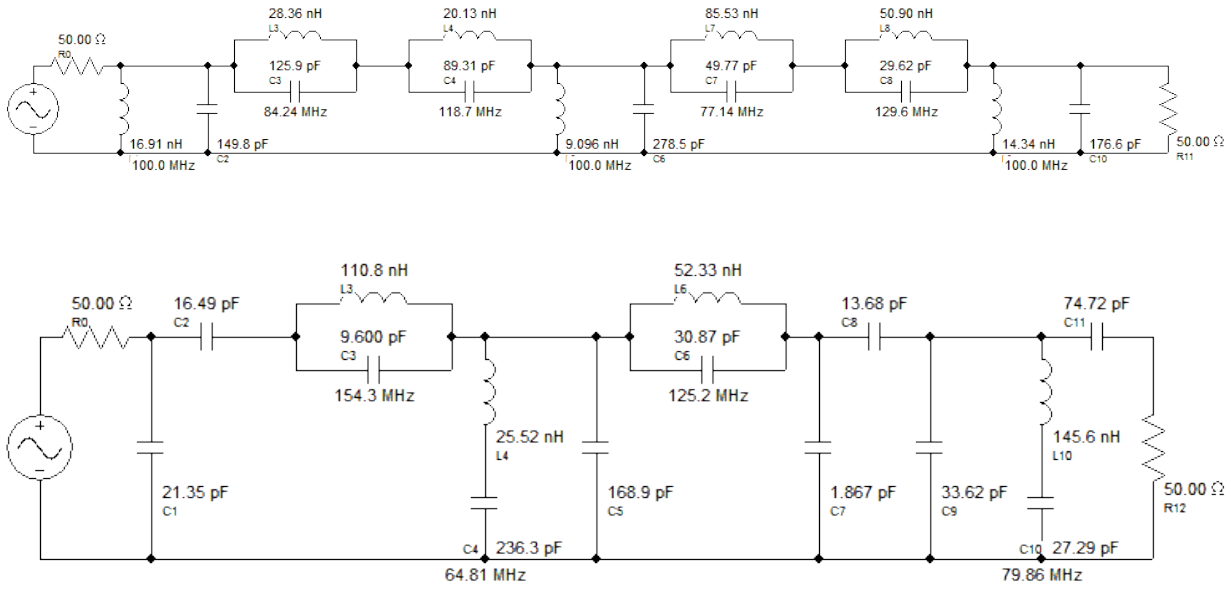
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

It should be noted that Equal Inductor Zigzag filters have the capability to reduce the inductor spread ratio to unity (perfect), but may result in negative inductor values. In these cases, an equal inductor classical band pass filter may offer improved inductor spread ratios over both the standard zigzag and the standard classical filters.

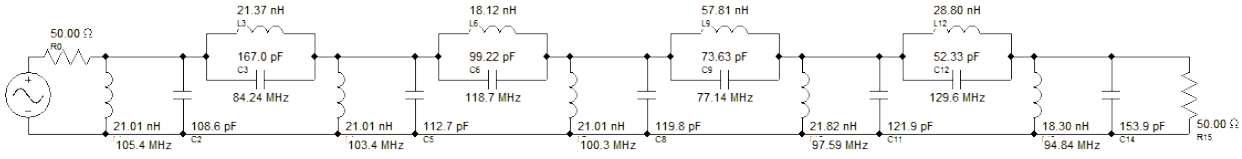
Equal inductor classical stop band filters alter the source resistance somewhat. If this is objectionable, the "Set Source" Res" selection will restore the source resistor at the expense of fewer equal inductors and possible inductor spread ratio.

Example:

Below is an example of a 20% band width, steep cutoff, 5th order Elliptic filter in classical and Zigzag topologies, followed by the equal inductor counterpart.



Classical and Zigzag Elliptic filters



Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Equal Inductor Classical Topology (Some shunt inductors altered slightly to restore 50 Ohm source resistance)

The equal inductor Zigzag filter would have negative elements in this case, and cannot be used for the vast majority of applications. The inductor spread ratios and inductor counts are as follows:

Topology	Inductor Ratio	Inductor Count
Classical	9.40	7
Zigzag	5.71	4
Equal Inductor, Zigzag	--- Negative Values	NA ---
Equal Inductor, Classical	3.19	9

It is easily seen that the Equal Inductor Classical topology produces smallest inductor spread ratio in this case, and is useful in cases where inductor count is not critical.

Cascaded High and Low Pass Topologies

Band pass filters may be implemented as integrated high and low pass elements, usually by applying a low pass to band pass conversion, or by cascading separate low pass and high pass stages. It is sometimes useful to try out both designs to evaluate element values for desirable ranges. Cascaded high/low pass topologies do not produce physically symmetrical filters and may not be realized with a single, or just two, inductor values.

Cascaded high/low pass filters are sometimes useful for implementing wide band filter, and tends to minimize the inductor spread separately in the low pass and high pass section of the filter. In the case of even order stop band filters, a minimum inductor count may be realized with the same inductor count as a Zigzag filter.

Cascaded high and low pass topologies are selected by checking the "High/Low Pass" checkbox in the Passive Design panel, or "Hi/Lo 1" or "Hi/Lo 2" checkboxes when they are displayed.

Source Resistor

Cascaded high and low pass synthesis frequently alters the value of the source resistance. If this is objectionable, selecting "Set Source Res" will usually fix the problem by adding an additional capacitor, or possibly an inductor, to force the source resistor back the original design value. Hi/Lo 2 selection cannot adjust the source resistor

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

All Pole Filters

The All Pole filters of Gaussian, Bessel, Linear Phase, Butterworth, Legendre, and Chebyshev I are the easiest to realize in cascaded high/low topology form. There are virtually no restriction for these topologies.

Stop Band Filters

The stop band filters of Chebyshev II, Hourglass, and Elliptic are realizable in cascaded high/low pass topologies. Even order filters of 10 orders or less have two option, **HiLo 1 and HiLo2.**

The Hi/Lo 1 option is produces a similar topology as the odd order.

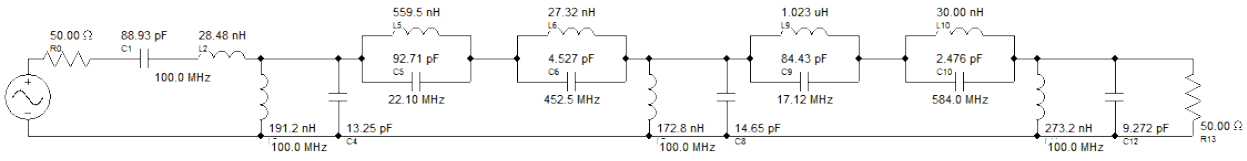
The Hi/Lo 2 option minimizes the inductor count, but results in a set source resistance value that cannot be restored.

Asymmetrical Band Pass Filters

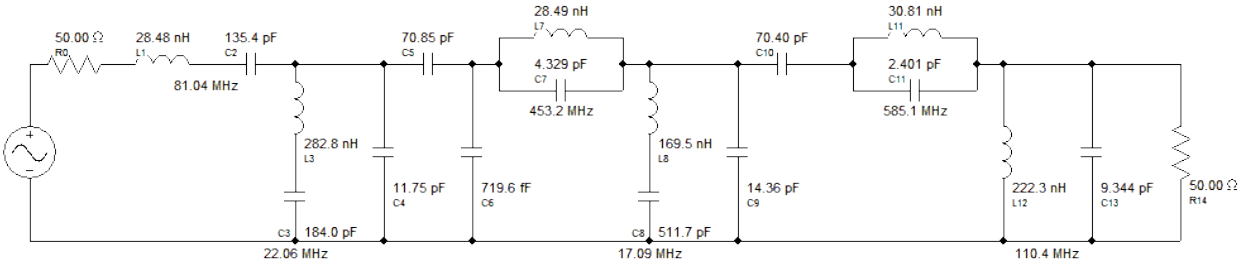
Frequency asymmetrical band pass filters may be realized with cascaded high/low pass stages, and are frequently more efficient to generate, especially when stop band zeros are required. When problems are encountered with stop band zero placement for Butterworth or Chebyshev I integrated asymmetrical band pass filters, the recommended corrective action is to switch to cascaded high/low pass asymmetric filters.

Example:

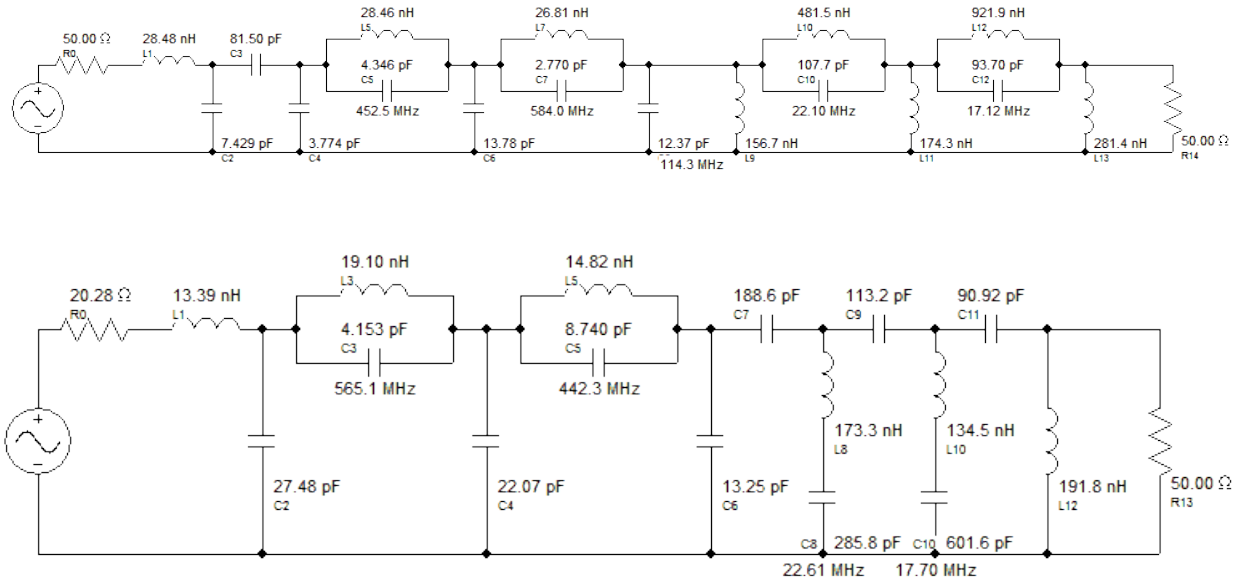
Below is an example of a 6th order, 300% band width, steep cutoff Elliptic design. Classical and Zigzag topologies are shown on top. Low/High pass topologies are shown below for comparison of inductor count and inductor spread. Classical and Zigzag equal inductor topologies produce negative values for this case and will not be used for comparison in that they have extremely few applications.



Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Classical and Zigzag Filter Topologies.



High/Low 1 and High/Low 2 Filter Topologies

The inductor spread ratios and inductor counts are as follows:

Topology	Inductor Ratio	Inductor Count
Classical	37.4	8
Zigzag	9.93	6
Minimum Inductor Topologies	--- Negative Values	NA ---
High/Low 1*	5.88 / 1.06	8
High/Low 2*	1.43 / 1.43	6

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

* High pass and low pass sections of the filter shown separately

It is easily seen that the High/Low pass 2 topology is the superior topology for wide band steep cutoff cases such as this when the source resistance is not important. If source resistance is important, then High/Low 2 or Zigzag work the best, depending on inductor count requirements.

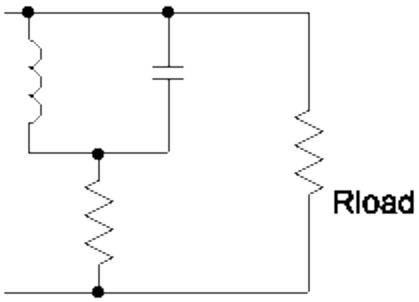
Amplitude Equalizers

Lumped filters with equal or near equal terminations cannot be compensated for finite Q by altering the pole/zero plot. Since most of the degradation due to finite Q is at the break frequency, it is possible to reduce the magnitude of the frequency response everywhere except the transition to produce an evenly attenuated frequency response. FilterSolutions supports series, shunt, and equal resistance amplitude equalization on filters with finite Q elements. **To amplitude equalize the filter, select "Shunt", "Series", or "Const" in the Amplitude Equalization selection box in the upper right of the circuit display.** This box are displayed whenever a passive filter is generated with finite Q.

The amplitude equalizer element values are changeable, just like all other element values in the filter. If not satisfied with a particular amplitude equalization effect, you may adjust the values in the equalizer to obtain a more desirable effect.

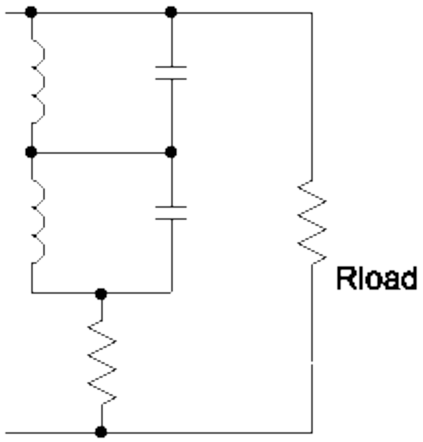
Shunt Amplitude Equalizers

Selecting "**Shunt Equalization"** in the Amplitude Equalization selection box on a passive circuit generated with finite Q elements generates a shunt amplitude equalizer. The addition of one the following RLC circuits at the end of the filter approximately achieves the amplitude equalization:



High Pass and Low Pass Shunt Equalizer

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

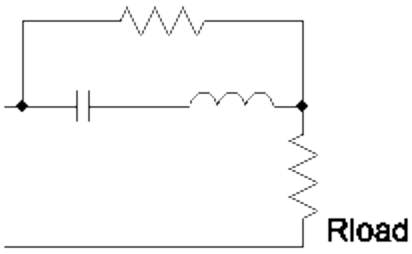


Band Pass and Band Stop Shunt Equalizer

If the resonant frequency of one LC tank is set to a filter break frequency, then attenuation is minimized at the break frequency, but is inserted everywhere else. If the Q of the LC tank(s) is set properly, an approximation of an attenuated infinite Q frequency trace will result. Since the shunt equalizer has the effect of decreasing the load impedance slightly, a slight distortion in the filter frequency response is introduced.

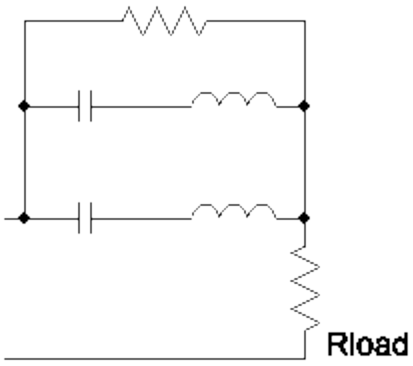
Series Amplitude Equalizers

Selecting "**Series Equalization"** in the Amplitude Equalization selection box on a passive circuit generated with finite Q elements generates a series amplitude equalizer. The addition of one the following RLC circuits at the end of the filter approximately achieves the amplitude equalization:



High Pass and Low Pass Series Equalizer

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

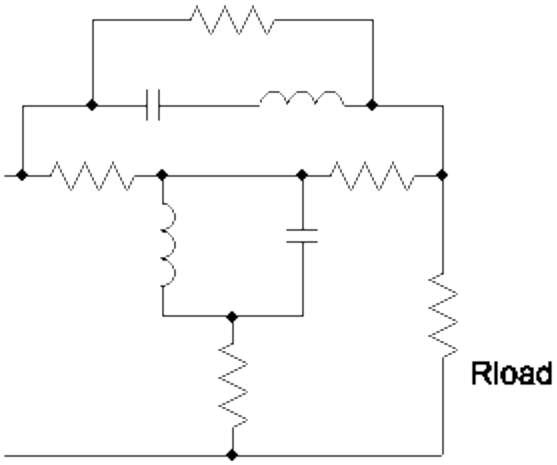


Band Pass and Band Stop Series Equalizer

If the resonant frequency of one LC tank is set to a filter break frequency, then attenuation is minimized at the break frequency, but is inserted everywhere else. If the Q of the LC tank(s) is set properly, an approximation of an attenuated infinite Q frequency trace will result. Since the series equalizer has the effect of increasing the load impedance slightly, a slight distortion in the filter frequency response is introduced.

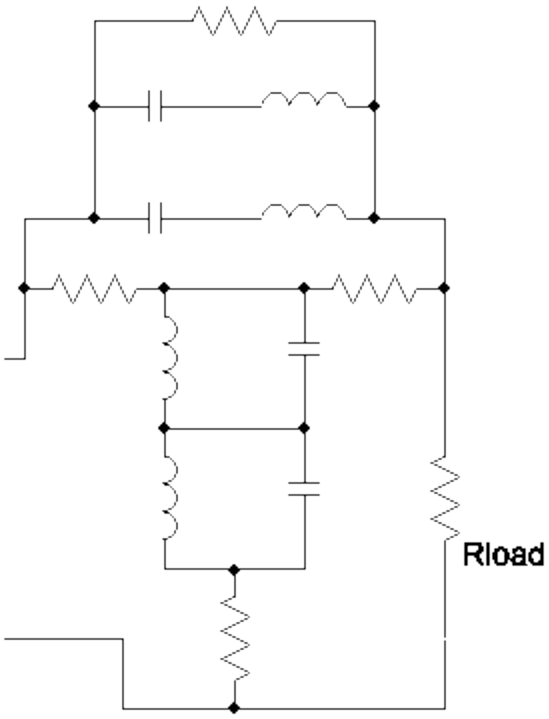
Constant Resistance Amplitude Equalizers

Selecting "**Constant Impedance"** in the Amplitude Equalization selection box on a passive circuit generated with finite Q elements generates an equal resistance amplitude equalizer. The addition of one the following RLC circuits at the end of the filter approximately achieves the amplitude equalization:



Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

High Pass and Low Pass Equal Resistance Equalizer



Band Pass and Band Stop Equal Resistance Equalizer

If the resonant frequency of one shunt and one series LC tank are set to a filter break frequency, then attenuation is minimized at the break frequency, but is increased everywhere else. If the Q of the LC tanks are set properly, an approximation of an attenuated infinite Q frequency trace will result. Since equal resistance equalizers do not increase or decrease the load impedance, **no distortion in the filter frequency response is introduced.**

When adjusting the equal amplitude equalizer, it is important to maintain the equal resistance property of the equalizer. This is accomplished by maintaining the following relations:

Rseries = Rsquare/Rshunt

Cseries = Lshunt/Rsquare

Lseries = Cshunt*Rsquare

Where:

Rsquare is the square of **Rload**.

The subscripts "series" and "shunt" refer to the series and shunt sections of the equalizer.

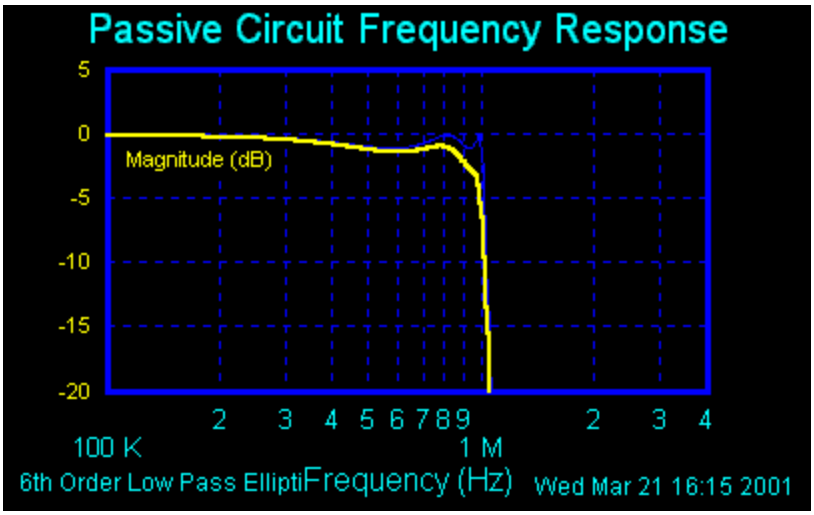
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Checking the **Maintain Equal Impedance** checkbox in the **Change Control Panel** will cause other elements in the amplitude equalizer to automatically update such that equal resistance is maintained.

Effect of Amplitude Equalization

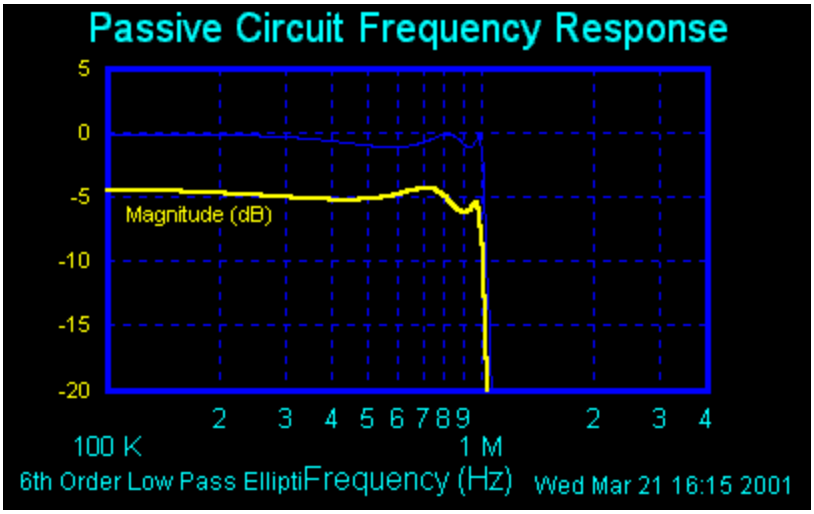
The effect on a low pass Elliptic filter with inductor q's of 40 and capacitor Q's of 150 is seen below. The dark blue trace is the ideal response. The yellow is the equalized finite Q response.

The following frequency graphs depict the equalization effect of finite Q on a sixth order Elliptic low pass filter with and without amplitude equalization. The dark blue trace is the ideal filter. The yellow trace is the actual filter with finite Q effects. The filter generating the curve has inductor Q's of 40 and capacitor Q's of 150.



Finite Q Effects Without Amplitude Equalization

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

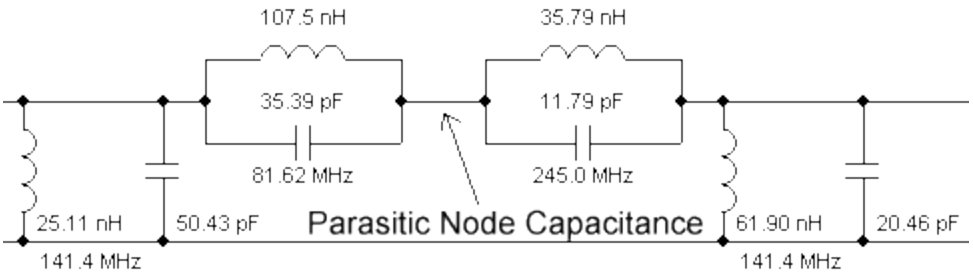


Finite Q Effects With Amplitude Equalization

Shunt Node Capacitance Compensation

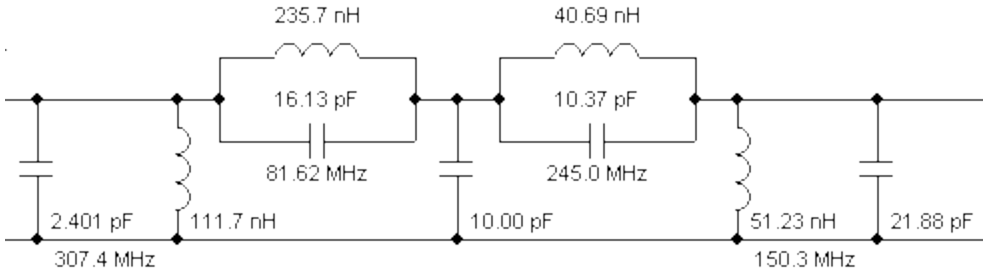
Nodes on printed circuit boards tend to have a shunt parasitic capacitance at each physical node. It is desirable to have a shunt capacitance in the filter design at each node so that the parasitic capacitance may be absorbed into the design capacitance. Tubular filters and some resonator filters do this for all pole band pass filters, but classical band stop filters present a problem in that the node between the LC zeros has no design shunt capacitor, leaving the filter vulnerable to frequency response degradation due to parasitic shunt node capacitance.

Fortunately, FilterSolutions contains an option to transform classical stop band filters into a design with user selectable shunt capacitance between the LC tank circuits.



Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FilterSolutions Shunt Node Capacitance compensation shown below transforms the design above into an equivalent circuit with a shunt capacitance between the two LC tanks. The new 10pF value is user-selectable. It should be set at least as high as the parasitic node capacitance in order to properly absorb the parasitic capacitance into the design.



The process of transforming the design also changes the source resistance value.

It is desirable to use even order filters so that the source resistance may be Nortoned back to where it should be. Check Set Source Res to force the source resistor back to its original value, or manually right click a Nortonable capacitor to set the source.

Even Order Modification

Description of the Problem

The even order filters of Chebyshev I, Chebyshev II, Hourglass, and Elliptic create a special problem in that their standard forms are not realizable with passive RCL circuits when their source and load resistances are comparable. **FilterSolutions** solves this problem by providing an option to modify these standard filters to a form of filter that is realizable with such termination.

If **Even Order Modification** is not selected for even order Chebyshev I filters, the source resistor is calculated that achieves the maximum power transfer for the filter, and is substituted for the user selected source resistance when necessary.

In general, for a filter to be realizable, a low pass filter frequency response must have a reflection zero at 0Hz and a transmission zero at infinity. For high pass filters the reflection zero must be at infinity, and the transmission zero must be at 0Hz. For band pass filters, the reflection zero must be at the center frequency, and the transmission zero at 0Hz and infinity. For band stop filters the reflection zero must be at 0Hz and infinity, and the transmission zero must be at the center frequency.

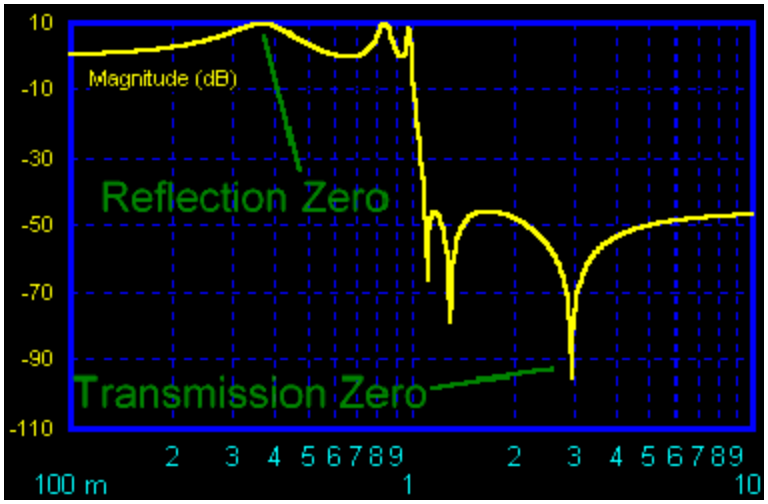
FilterSolutions displays the **Even Order Mod** option in the lower right part of the **Control Panel** which, when checked, modifies the filter frequency response to conform to both these requirements.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This even order modification is very important when designing diplexers because it minimizes the number of inductors needed to create the diplexer by allowing even order filters to be used.

Graphical Illustration

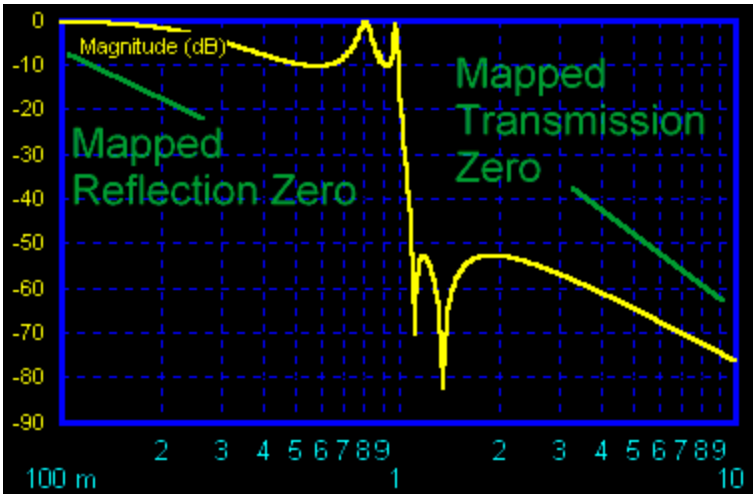
Below is a graphical description of the problem for low pass filters and the frequency modifying solution. This 6th order Chebyshev I filter illustrates the problem. The lowest frequency reflection zero for this 6th order filter is .4 Hz, and the highest frequency transmission zero is at 3 Hz. Because if this, this filter is not realizable with comparable load and source resistances.



To realize the 6th order Chebyshev I circuit, Filter Solution modifies the frequency response such that the lowest frequency reflection zero is mapped to 0Hz. The result is no longer a true Chebyshev I filter, but is rather a modified Chebyshev I filter that is realizable with RCL circuits of comparable source and load resistances.

As you can see, the lowest frequency reflection zero is now at 0Hz. It is still a 6th order filter with greater attenuation that the 5th order filter.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

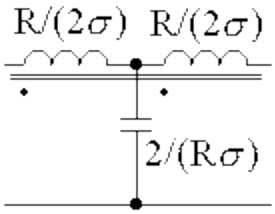


All Pass Passive Circuit Elements

FilterSolutions supports Bridged T topology balanced and unbalanced all pass sections, and lattice topology unbalanced states. (It should be noted that Bridged T and lattice structures are also used to support dual real and quadruplet zeros.) Unbalanced low Q all pass section require coupled coils. Balanced all pass sections do not. Equations and topologies are shown below.

Real All Pass Circuit Elements

Real all pass circuit sections are always of the following form:



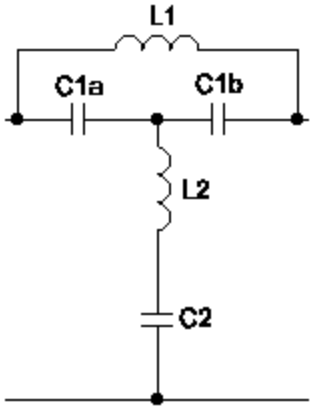
Where R is the load resistance, and Sigma is the distance from the origin.

Complex All Pass Circuit Elements

Complex all pass section may have three different forms. FilterSolutions selects the form that has the fewest inductors, has all positive element values, and all element values are realistic.

For all passes with a Q higher than one, the following elements are used:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Where:

$$L1 = 2R/(QWo)$$

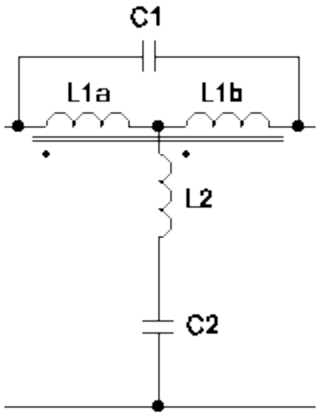
$$C1a = C1b = Q/(RWo)$$

$$L2 = QR/(2Wo)$$

$$C2 = 2Q/\left((Q^2 - 1)WoR\right)$$

For all pass section with Q less than one, the following is used:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Where:

$$L1a = L1b = L(1 + K)/2$$

$$C1 = Q/(2RWo)$$

$$L2 = L(1 - K)/2$$

$$C2 = 2/(RQWo)$$

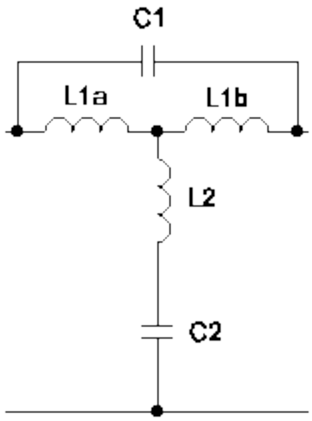
and

$$L = R(Q^2 + 1)/(2QWo)$$

$$K = (1 - Q^2)/(1 + Q^2)$$

On rare occasions, an all pass section with a Q of slightly greater than one results in an excessively large capacitance if the Hi Q all pass above is used. When this happens, the following is used:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Where:

$$L1 = L(1+K)$$

$$C1 = Q/(2RWo)$$

$$L2 = LK$$

$$C2 = 2/(RQWo)$$

and

$$L = R(Q^2+1)/(2QWo)$$

$$K = (Q^2-1)/(1+Q^2)$$

Note that when Q = 1, L2 becomes zero. When an all pass with a Q of one is installed, FilterSolutions removes L2 from the all pass.

Balanced Passive Circuits

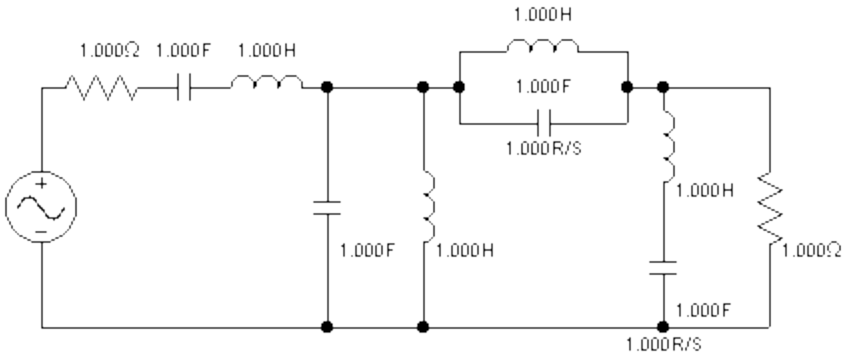
FilterSolutions supports both balanced and unbalanced passive filters. Many applications require balanced circuits due to the better noise immunity characteristics. To translate an unbalanced circuit to a balanced circuit, cut the series inductors in half and place an identical inductor in the bottom, and double series capacitors, and place an identical capacitor on the bottom. All parallel elements should be left alone.

The entire passive circuit may be balanced by checking the "Balance" box above each passive filter schematic. Individual elements, LC tank pairs, or all pass sections may be balanced or unbalanced with the use of the right mouse key.

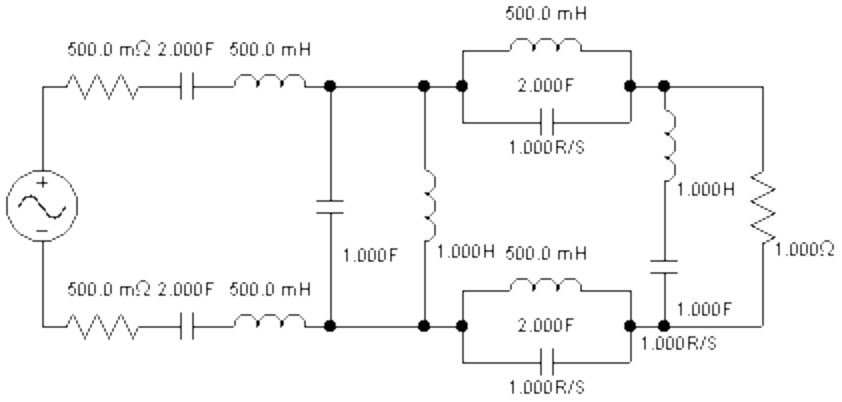
Example:

A FilterSolutions unbalanced passive circuit:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



An equivalent balanced circuit:



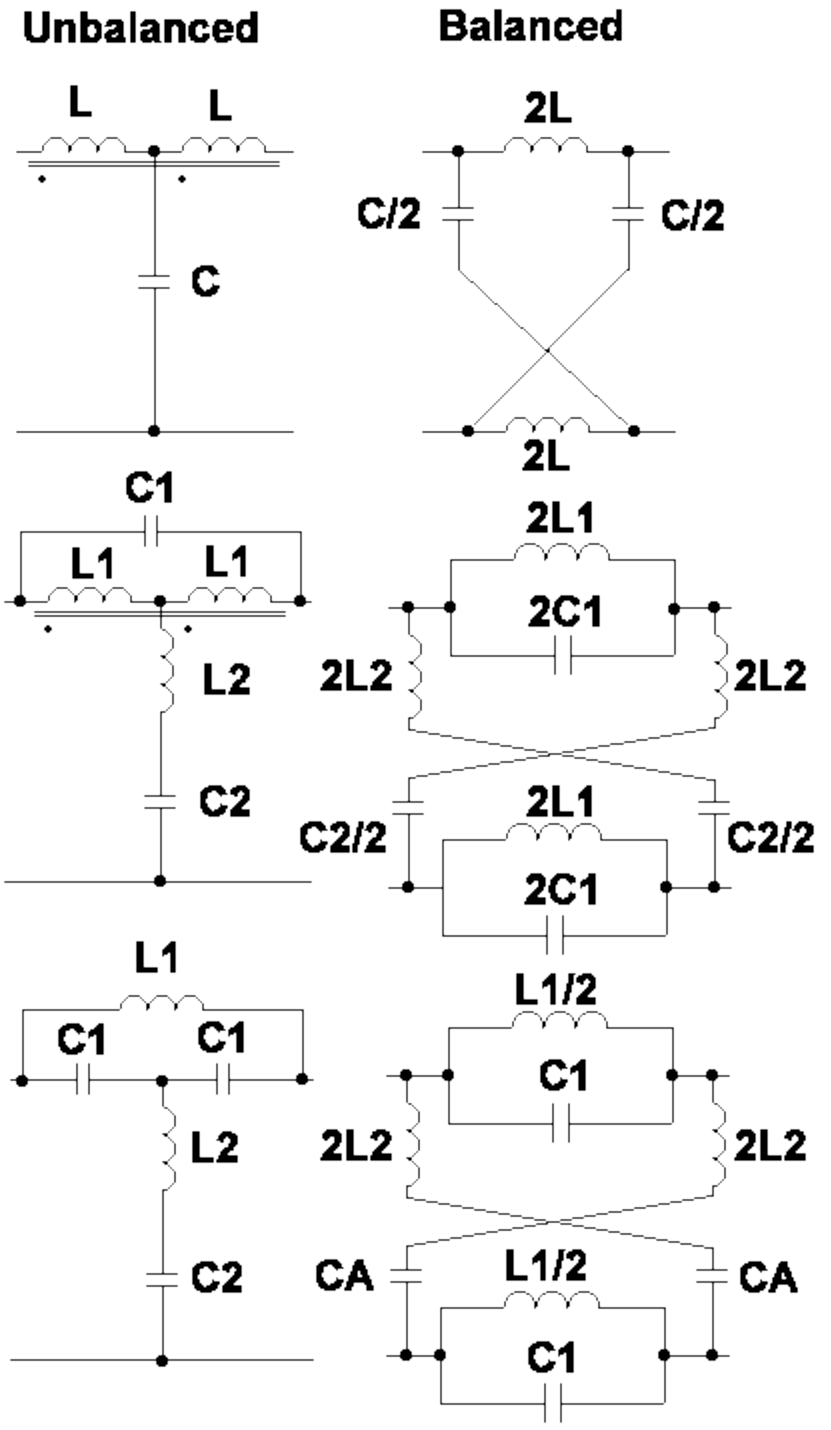
The voltage signal across the load resistance is identical in both circuits above.

Balanced All Pass Sections

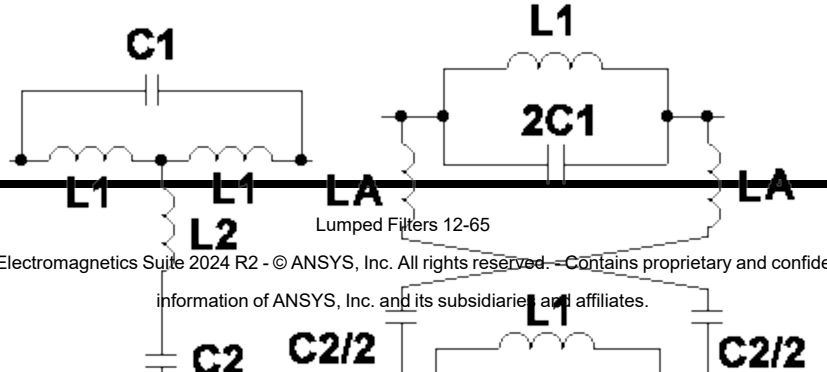
The above techniques may be used to create balanced all pass networks as well. However, balanced all pass networks may also be created more without the use of coupled coils. FilterSolutions offers the "Balance 1" option to select balanced T stages, and "Balance 2" to select balanced lattice stages without coupled coils. Balanced T stages are generally more immune to element value error than balanced lattice stages. Monte Carlo analysis should be conducted to determine filter performance prior to selecting a particular topology.

The following diagram shows the translation from unbalanced T all pass circuit sections on the left to equivalent balanced lattice all pass circuit sections on the right. The translations also hold for dual real and quadruplet zeros when the upper T elements are of equal value.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



CA = L1/(2*RL*RL) RL = Load Resistance



Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Working With Lumped Filters

This chapter contains the following topics:

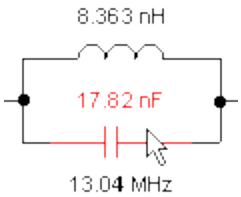
- Modify Elements
- S-Parameter and Spice Model Imports
- S-Parameter and Spice Model Exports
- Inductor and Capacitor Tuning
- Add, Delete, or Split Circuit Elements
- Balancing Individual Elements
- Add, Delete, or Split Circuit Elements
- Balancing Individual Elements
- Norton, Pi, T, and LC Transformations
- Richards Transforms
- Kuroda Transforms
- Standard Parts
- Complex Terminations
- NetLists

Modify Lumped Circuit Elements

Passive Circuit elements may be modified, and the effect displayed in real time. Element values may be updated to selected values, by percent change, to the nearest standard part, to the nearest standard industrial value, or random value. The random values feature may be used to perform Monte Carlo analysis.

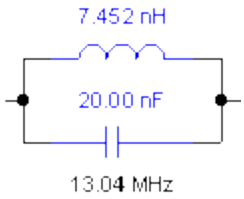
To modify a passive circuit element:

Pass the cursor over the element you desire to modify. It will turn red. Left click the mouse to change the element value.

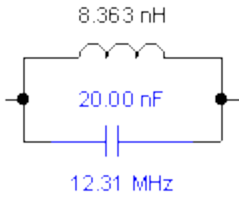


Elements and resonant frequencies are displayed in blue after they are modified.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



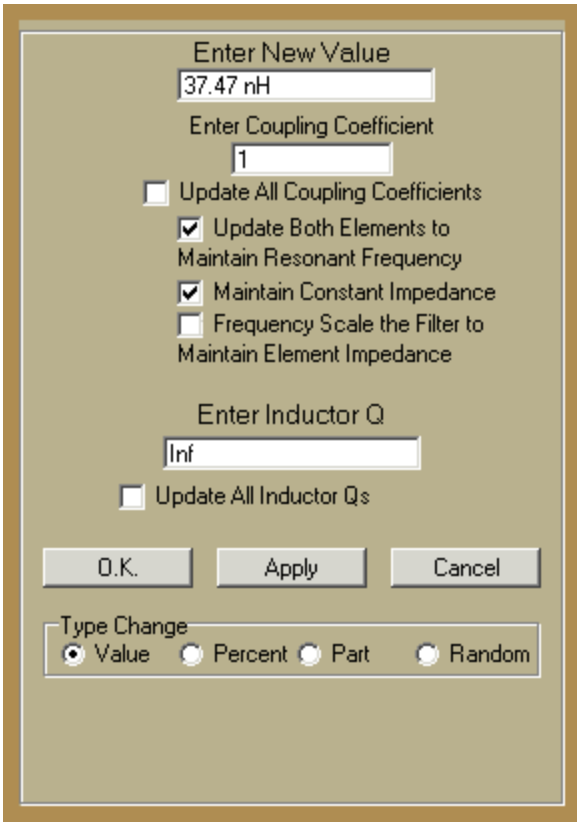
Unmodified resonant frequencies are displayed in black.



The colors of red and blue may be changed with the Color Control Panel.

Click the left mouse key to bring up the change control panel

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The Change control panel entries are defined in the following table:

Enter New Value	Enter the desired new value of the component. If a diplexer noncompensation element is being modified, both of the compensation elements will automatically update to account for the new compensation requirements. This automatic compensation update discontinues after either one of the compensation elements are manually modified.
Enter	Displayed for coupled inductors only. Enter

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

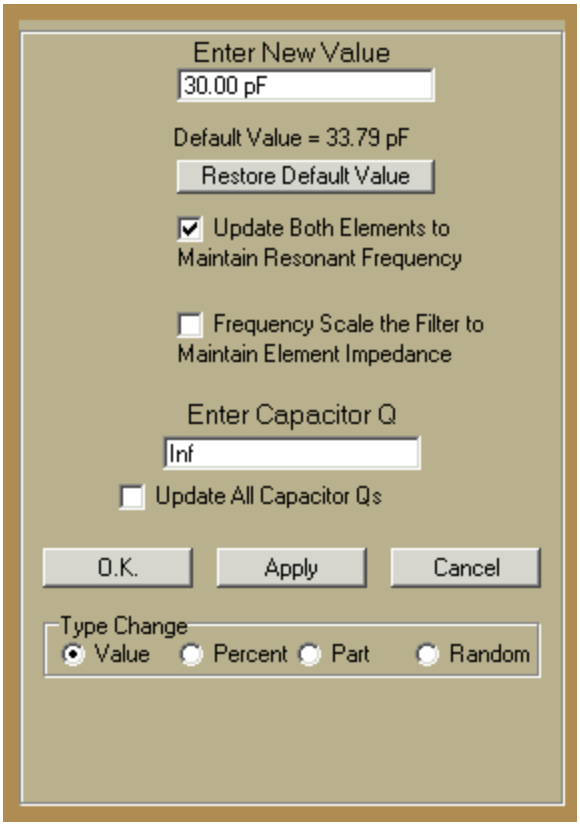
Coupling Coefficient	the desired actual coupling coefficient for analysis computations.
Update All Coupling Coefficients	Check to update the coupling coefficients of ALL coupled inductors in the filter.
Update Both Elements to Maintain Resonant Frequency	Displayed only for LC tank elements When the one of the two LC tank elements are selected, checking this box will cause the other element to automatically update to a value that will maintain the resonant frequency of the tank. Uncheck this box to update the tank elements independently.
Maintain Constant Impedance	Displayed for constant impedance amplitude equalizers only. Check to update other elements in the amplitude equalizer to maintain the constant impedance property.
Frequency Scale the Filter to Maintain Element Impedance	Check this box if you wish to frequency scale the filter to maintain the impedance of the selected element at the new element value. FilterSolutions will change all other reactive elements on the filter to maintain filter characteristics at the new frequency.
Enter Inductor Q	Enter the desired inductor or capacitor Q. Infinite Q may be represented by "Inf", "i" or "0".

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Update All Inductor Q's	Check this box to update all inductors or capacitors in the circuit to the value entered in the "Enter Inductor Q" box.
OK	Closes the control panel and updates the element value in the circuit.
Apply	Update circuit and graphical displays in real time without closing the Change Control Panel.
Cancel	Closes the control panel and does not update the element value.
Type Change	Specify to update element to a fixed value, percent change in value, nearest standard part, or random value.

When a previously changed component is selected for change, the default value of the component is displayed, and the Change control panel looks like this.

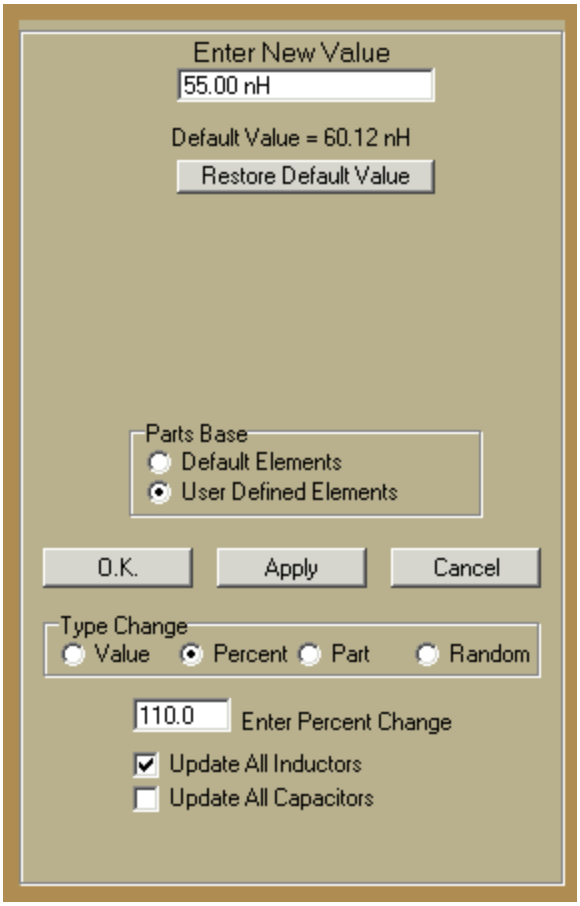
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Restore Default Value sets the component back to the original value and turns it black.

It is possible to specify the percent change in the element value by selecting "Percent" at the bottom of the panel. The checkbox at the bottom will update all like element by the same percentage. The Parts Base selection determines whether to base the percent change computation on the default part value, or the current part value.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

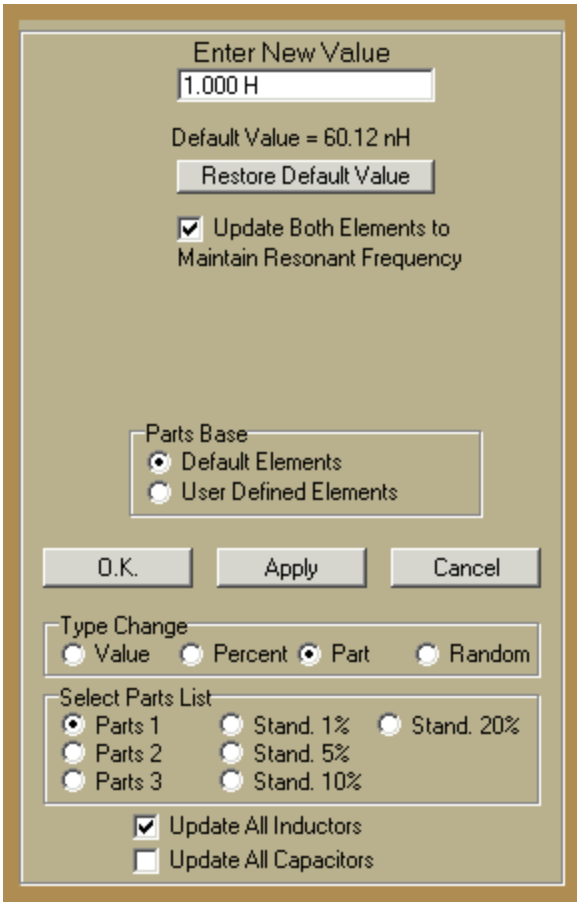


It is possible to update the element value to the nearest standard part by selecting "Part" at the bottom of the panel, and then selecting which standard parts list to use. The checkbox at the bottom will update all like element to their nearest standard part. The Parts Base selection determines whether to base the percent change computation on the default part value, or the current part value. The parts lists are maintained on the Main Control Panel. The Stand. 1%, 5%, 10% and 20% selections use standard industrial 1%, 5%, 10%, and 20% part values.

You will find more information about standard parts lists under **Standard Parts** Help Section.

Standard parts modifications are also accessible in the upper right of the passive, and active schematics.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



1%, 5%, 10%, and 20%Standard Parts

The values for 1%, 5%, 10%, and 20% selections are standard industrial values. These values are listed below:

20% Parts	1.00, 1.5, 2.2, 3.3, 4.7, 6.8, 10.0, 15.0...
10% Parts	1.00, 1.20, 1.50, 1.80, 2.20, 2.70, 3.30, 3.90, 4.70, 5.60, 6.80, 8.20, 10.00, 12.00...
5% Parts	1.00, 1.10, 1.20, 1.30, 1.50, 1.60, 1.80, 2.00, 2.20, 2.40, 2.70, 3.00, 3.30, 3.60, 3.90, 4.30, 4.70, 5.10, 5.60, 6.20, 6.80, 7.50, 8.20, 9.10, 10.00, 11.00...

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1% Parts	1.00, 1.02, 1.05, 1.07, 1.10, 1.13, 1.15, 1.18, 1.21, 1.24, 1.27, 1.30, 1.33, 1.37, 1.40, 1.43, 1.47, 1.50, 1.54, 1.58, 1.62, 1.65, 1.69, 1.74, 1.78, 1.82, 1.87, 1.91, 1.96, 2.00, 2.05, 2.10, 2.15, 2.21, 2.26, 2.32, 2.37, 2.43, 2.49, 2.55, 2.61, 2.67, 2.74, 2.80, 2.87, 2.94, 3.01, 3.09, 3.16, 3.24, 3.32, 3.40, 3.48, 3.57, 3.65, 3.74, 3.83, 3.92, 4.02, 4.12, 4.22, 4.32, 4.42, 4.53, 4.64, 4.75, 4.87, 4.99, 5.11, 5.23, 5.36, 5.49, 5.62, 5.76, 5.90, 6.04, 6.19, 6.34, 6.49, 6.65, 6.81, 6.98, 7.15, 7.32, 7.50, 7.68, 7.87, 8.06, 8.25, 8.45, 8.66, 8.87, 9.09, 9.31, 9.53, 9.76, 10.0, 10.2...

Monte Carlo Analysis

A Monte Carlo sensitivity analysis may be performed with the use of the "Random" selection. Enter the desired standard deviation in percent, and FilterSolutions will automatically update the selected element, or all like elements, by a random number from a Uniform or Gaussian (Normal) distribution. The mean of the distribution is 0%, and the maximum tolerance of the Uniform distribution or the standard deviation of the Gaussian distribution is entered by the user.

Manufactured parts are usually specified as a maximum not-to-exceed percentage error without any specifications for the shape of the error distribution between the upper and lower limits, so neither Uniform nor Gaussian are totally accurate models of the true error, but can be used to produce reasonably accurate results.

All graphical analysis traces are automatically updated each time the "Applied" button is depressed. Check the "Maintain Old Traces" checkbox to prevent old graphical traces from being overwritten. Enter the number of desired simulation runs per button click on the "No. of Runs" box.

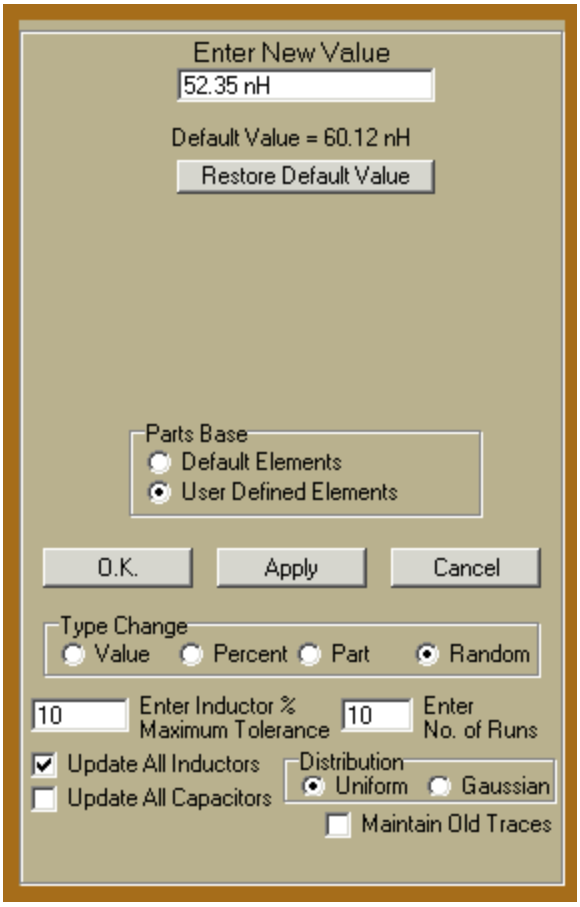
Monte Carlo analysis may be performed on the original ideal parts, or the user selected part values.

Part Tolerances

Tolerance percentages may be assigned to each part by clicking on the desired part and entering the desired tolerance in the Change Control Panel. When a tolerance has been

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

assigned to a part, **the assigned tolerance value will override the tolerance value selected in the Monte Carlo analysis**. In other words, the Monte Carlo tolerance entry only applies to parts that have not been assigned a tolerance.



Random updates are particularly useful in investigating the robustness of a filter to element errors.

S-Parameter and Spice Model Imports

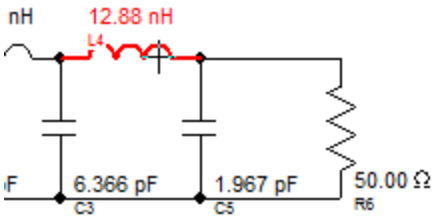
S-Parameter from Touchstone format files and Spice model files in standard Spice format can be imported, both by individual file selection and by automatic nearest match. When imported, the original element impedance is replaced with the S Parameter or Spice model defined impedance for each frequency. Interpolation is employed between the frequencies for S Parameter files. Any passive linear element S-Parameter or Spice model file is acceptable. Active element S-Parameters or Spice models, such as a two pin amplifier, will not be accepted for importation. Elements defined with S-Parameter or Spice model files will continue to use the nominal value for sensitivity analysis and Monte Carlo analysis.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

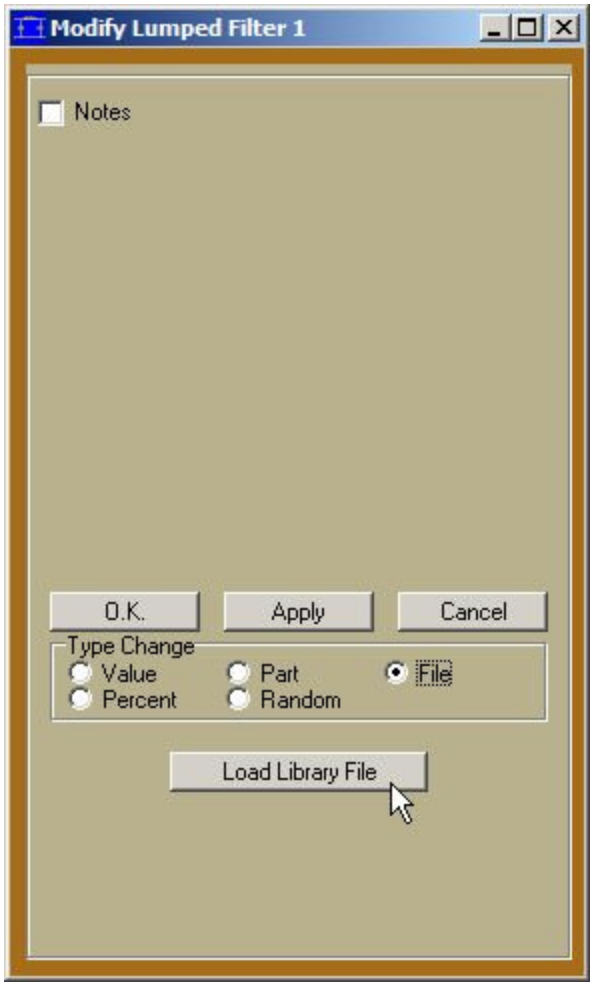
S-Parameter Files and Spice parameters for Coilcraft inductor downloads may be found at: www.coilcraft.com/models.cfm

Once an S Parameter or Spice model is imported into an element, all analysis performed on the associated filter will use the imported S-Parameters or Spice models instead of the element impedance. S-Parameter or Spice model files that are electrically asymmetrical will post pin numbers on the schematic.

To import a specific S Parameter or Spice models files, left click on the desired element, select, "File" in the pop up Change Control Panel, select the desired file, and "OK". The name of the S Parameter file then appears above the element, as shown below in Figure 1.



Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



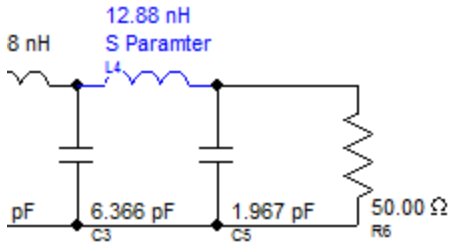


Figure 1: Selecting a Desired S-Parameter or Spice Model File

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Update to Nearest Matching S-Parameter or Spice Model File

To update to the nearest matching S-Parameter or Spice model file, select "Part" and "File", as shown below, then select, "Browse" to select the desired directory containing the S-Parameter or Spice model files, as shown below in Figure 2. Select, "Menu" to bring up the manufacturer and family pop-up menu. Select any file in the Browse directory to select and define the desired directory of S-Parameter files. Once selected, FilterSolutions will remember the directory until another directory is selected by the user. Check, "All Inductors" to update all the inductors in the filter. Care should be taken to insure that all S-Parameter and Spice model files are from components valid for the frequency and power range of the designed application.

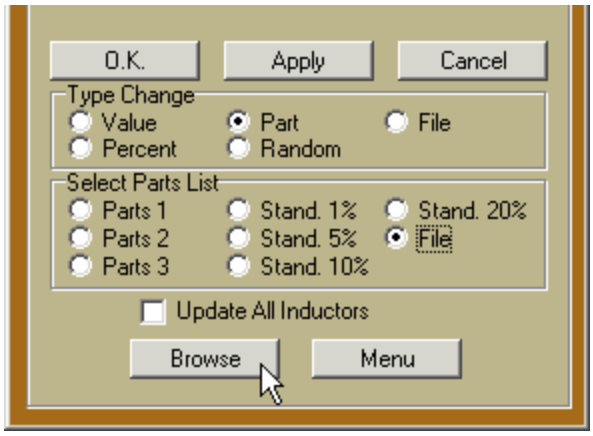


Figure 2: Update to the Nearest Matching S-Parameter or Spice Model File

Automatic Design to Nearest Matched S-Parameter or Spice Model File

To design to the nearest S-Parameter or Spice model file, select "File" or "Browse" in the upper right corner of the schematic, as shown in Figure 3 below. If a directory containing the desired S-Parameter files is already defined, select, "File". Otherwise, select, "Browse" and select any file in the Browse directory to select and define the desired directory of S-Parameter or Spice Model files.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

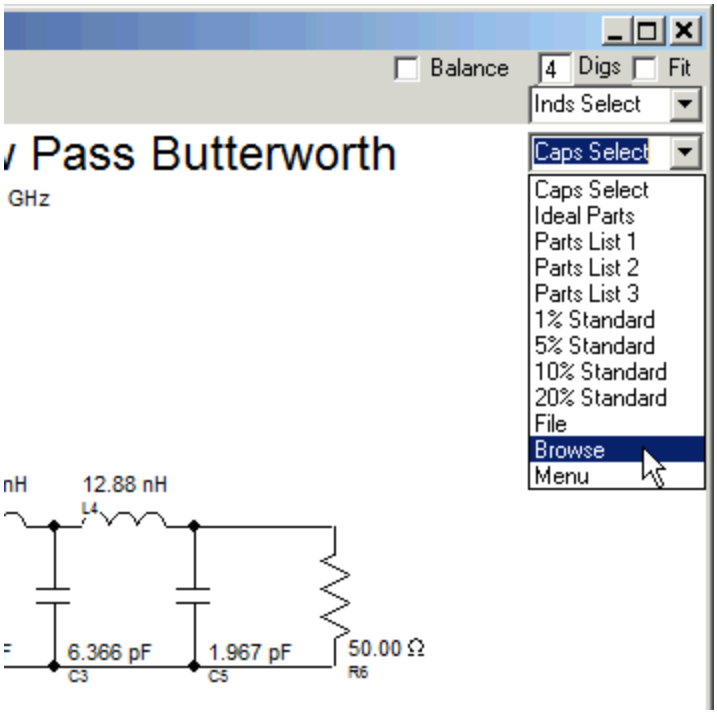


Figure 3: Design to the Nearest S-Parameter File

Parts Library Menu

Use the "Menu" option to select a family of parts from the internal parts libraries. Please refer to, "Parts Libraries" in the Table of Contents for more on this subject.

S-Parameter and Spice Model Exports

S-Parameter or Spice model files that have previously been imported can be exported. Automatic S-Parameter and Spice model selection may make it difficult to determine which S-Parameter or Spice model file is used in any particular element. Left clicking an element that is loaded with a S-Parameter or Spice model file pulls up the element Change Control Panel which may be used to export the specific S-Parameter or Spice model file that is loaded into the selected element. Select **File** to bring up the Touchstone features, then save the element S-Parameter or copy it into the Windows clipboard for pasting elsewhere, as shown in Figure 1.

Additionally, S-Parameter and Spice model files may be saved or viewed by selecting the appropriate button.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In all cases, the original S-Parameter or Spice model file will be saved or copied, including comments, header, and numerical resolution.

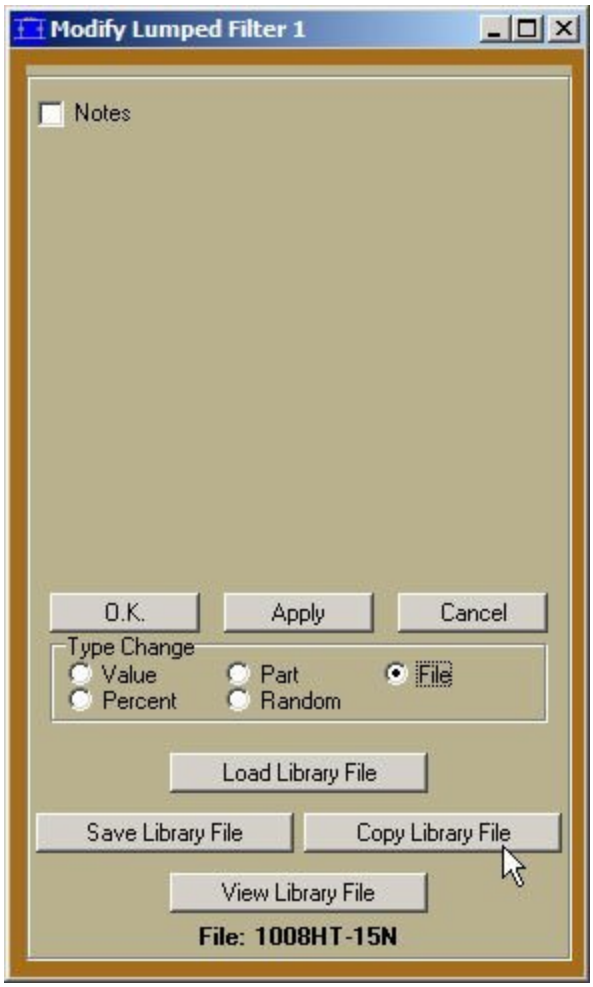


Figure 1. Saving and Copying an Element S-Parameter or Spice Model file

Inductor and Capacitor Tuning

When passive filters ideal elements are replaced with real parts with nonideal values, an expected degradation in filter performance occurs. It is possible to tune the circuit inductors and/or capacitors such that some, or even most, of the degradations are transferred from the filter pass band to the stop band. The effect of this translation is that degradation in the pass band is minimized.

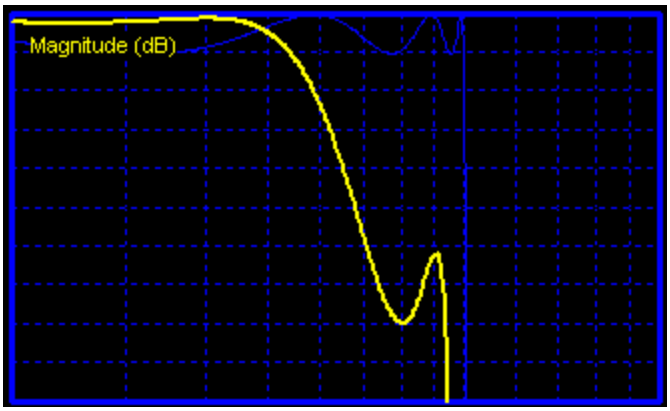
Whenever any inductor or capacitor value in a passive LC filter is modified, an "L tune", "C tune", or "Tune" button is available in the passive filter tool bar, as shown below. Depressing this button causes all filter inductors or capacitors to automatically assume new values that will minimize the pass band error in the filter.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

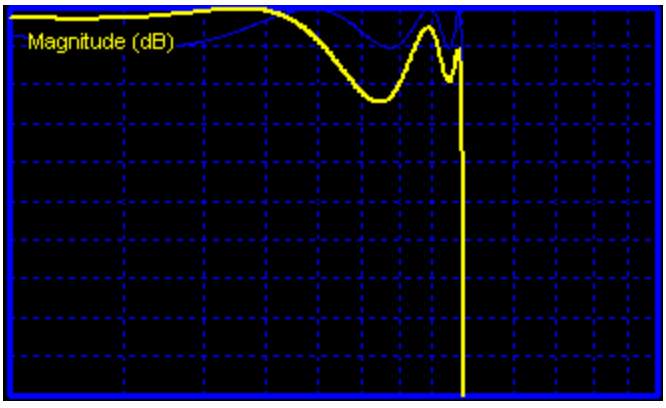
"L tune" Button on Passive Filters

Example:

The following pass band and stop band error is the result of inserting non ideal capacitor values into a low pass filter:



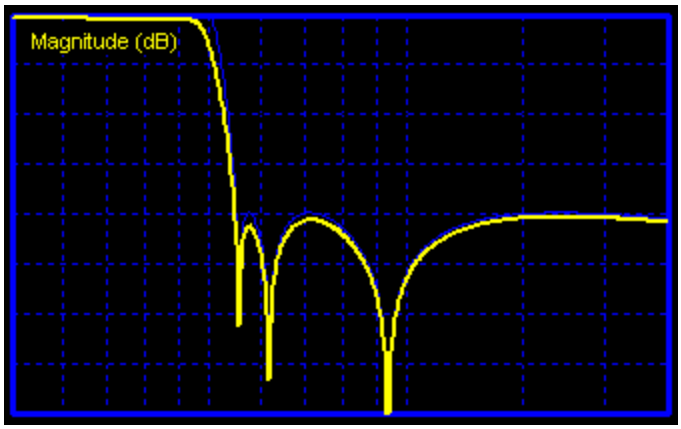
Pass Band Error Resulting From Nonideal Capacitor Values



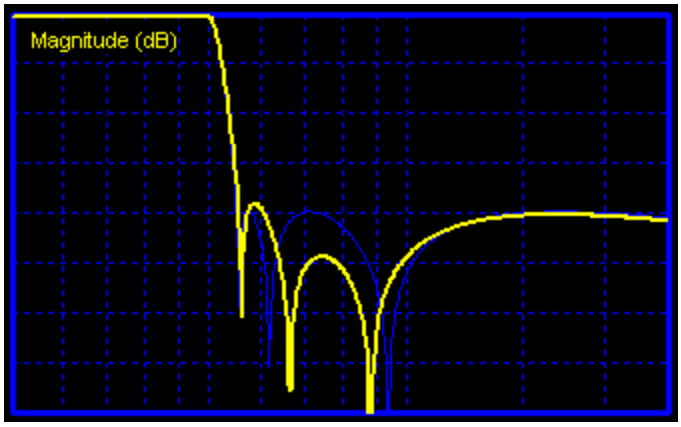
Stop Band Error Resulting From Nonideal Capacitor Values

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

After inductor tuning, the passband error is minimized at the expense of the stop band error.



Pass Band Error After Inductor Tuning



Stop Band Error After Inductor Tuning

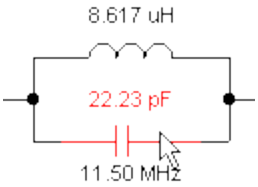
Add or Delete Circuit Elements

You may add or delete elements from FilterSolutions. One possible use of this is to model parasitic effects by placing a capacitor in parallel to a load.

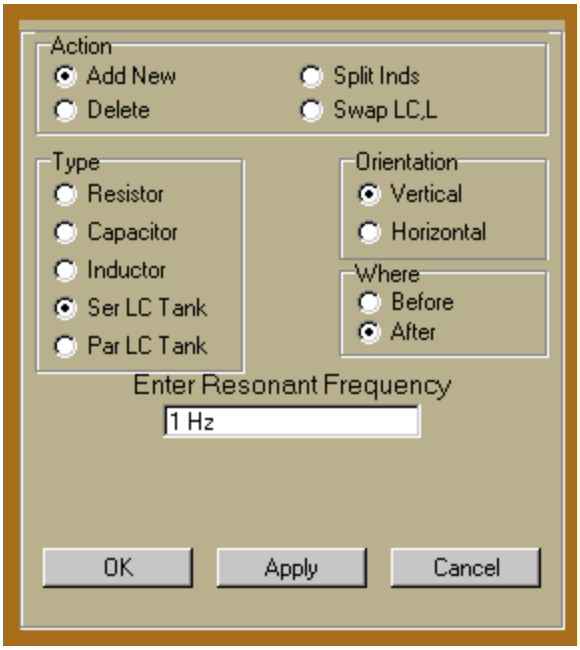
An element in a FilterSolutions passive filter is an isolated capacitor, inductor or resister; an LC tank pair; or an all pass section. All pass, if they are to be added, must be placed adjacent to the load or source resistor.

To add an element to a passive circuit, place the cursor over any element, except for a load or source resistor, adjacent to where you wish to add a new element. The element will highlight as shown:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Right click the mouse to pop up the following control panel:



The "Balance" selection in the upper right does not add or delete elements, but balances and unbalances elements. All pass stages offer "Balance 1" and "Balance 2" selections to choose between balanced T and lattice stages.

Enter the selections and select **OK**. You may update the new element by left clicking on it after it has been added. Real frequencies may be entered by following the entry with "re". FilterSolutions will implement a real frequency LC tank with coupled coils if possible. If **All Pass** is selected, there will be a prompt for Q as well. Enter a Q of 0.5 to place a first order all pass on the real axis. The All Pass option is only available when it is next to a load or source resistor.

To delete the selected element, simple select "Delete" and OK.

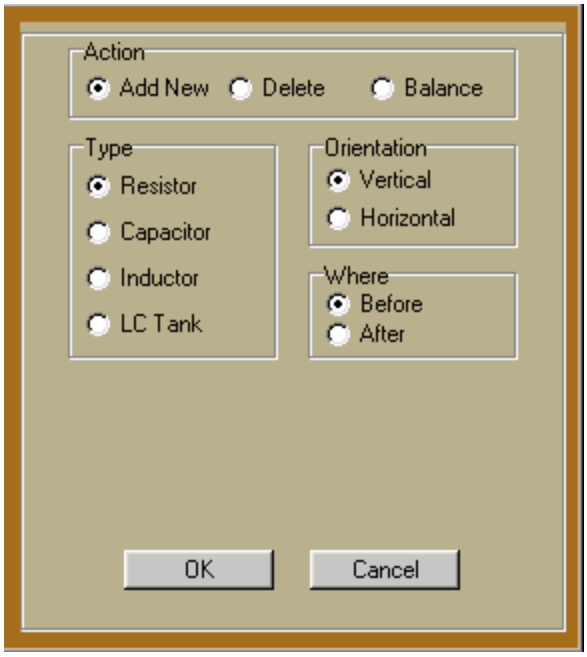
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Splitting Elements

Inductors and capacitors may be split into two series or two parallel elements. The default split ratio is one, for two replacement elements of equal size. A split ratio text entry box allows the user to enter any desire split ratio, including negative split ratios, such that the two new element values, V1/V2, will be equal to the user entered split ratio.

Balancing individual elements

FilterSolutions allows one to balance all or some of the elements. It may be advantageous to use balanced all pass sections in an unbalanced all pass filter. To change the balance on any horizontal element, LC tank or all pass section, **right click** on the desired element to balance. The following change control panel is displayed.



Balancing an Unbalanced Element

Select **Balance** followed by **OK** to balance the desired element. All pass stages offer **Balance 1** and **Balance 2** selections to choose between balanced T and lattice stages.

To unbalance a horizontal element, LC tank, or all pass section, **right click** the element to display the control panel:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Unbalancing a Balanced Element

Select **Unbalance** and **OK** to unbalance the desired element.

Norton, PI, and T Transformations

When LC filters contain an L, PI, T, or M configuration of like elements, as shown in Figure 1 below, it is possible to alter the impedance level of either side of the L, PI, T, or M without changing the frequency response shape of the filter. This is advantageous when specific element values are desired to be used one side of the L, PI, or T, or when the termination resistance of the filter needs to be changed.

Directly Nortonable LC resonators require a PI or L of parallel resonators, or a T or L of series resonators. A T of parallel LC resonators or a PI of series LC resonators require two Norton transformations, one on each side, and consequently two selectable values are presented in the pop-up Norton control panel when this selection is made.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

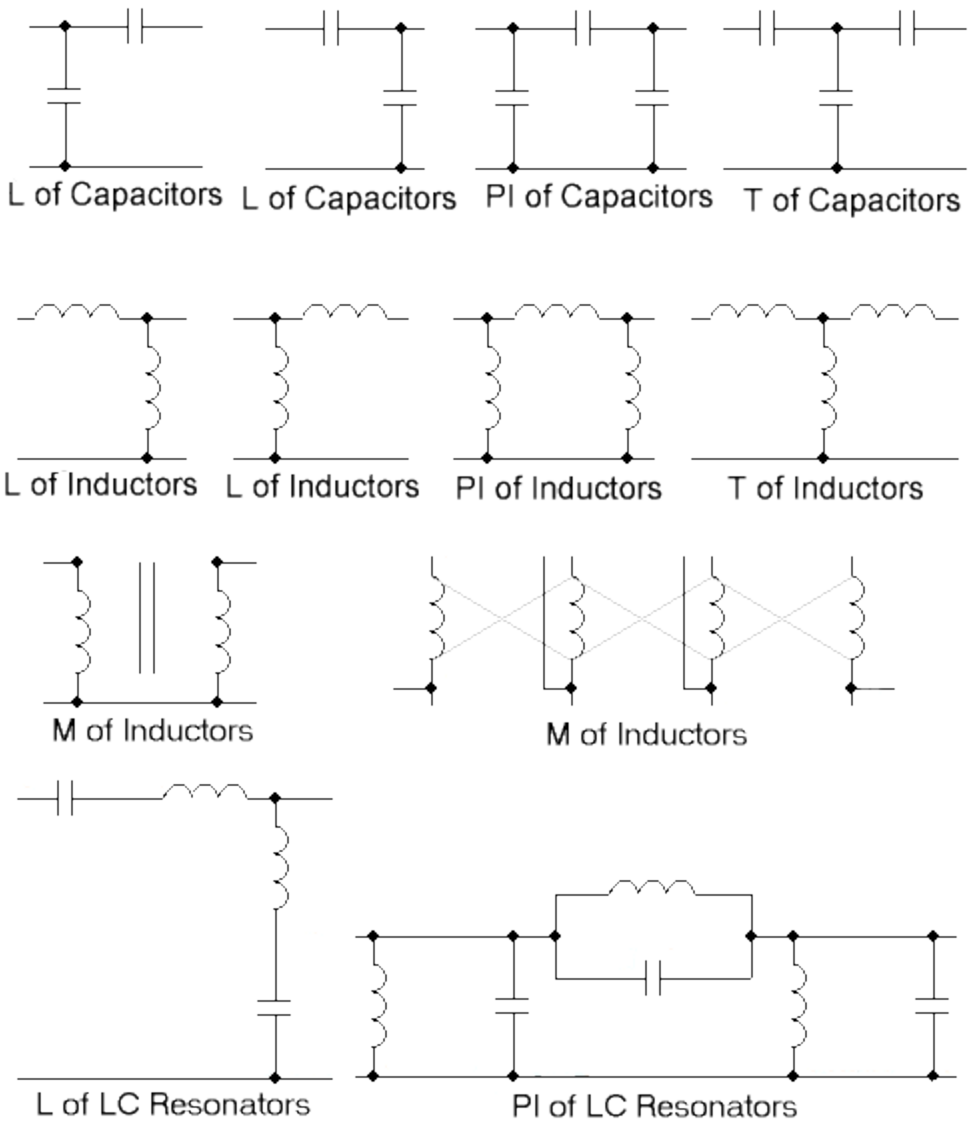
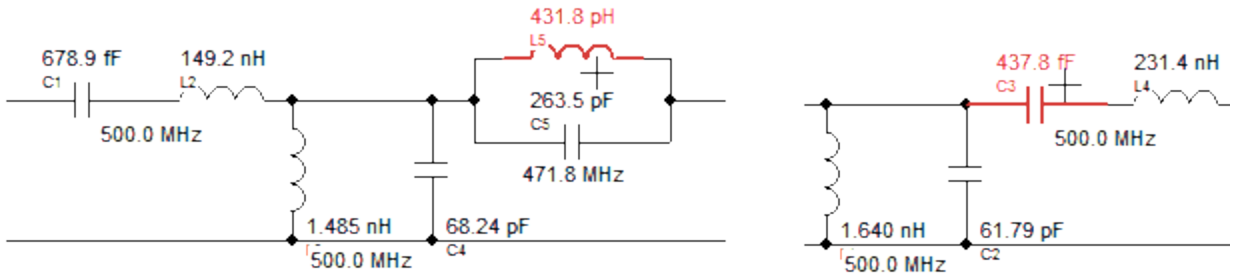


Figure 1: L, Pi, T, and M Examples

To perform a Norton Transformation:

To do a Norton transformation, **right click on the desired L, PI, T, or M combination of like elements,** as shown below. The change control panel will appear with a list of desired Norton effects, and a selection of which side (source or load) to alter the impedance of. Make the desired selection and hit "OK" or "Apply".

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Pass Cursor Over PI, Tee, L, or M elements to Select Them for Norton Transformation

To change a left L to a right L, select "Set Left Element Value" in the Norton selection menu, and enter "0" or "Inf". The left element will be removed, and a new right element will be installed, having the effect of changing the left L to a right L. The same hold for right L to left L.

M inductor combination includes any cross coupled resonator set in a cross coupled resonator filters. One Norton is permitted for the entire cross coupled set of elements.

Negative Element Values

Norton transformations work in part by installing negative element values in parallel with existing positive elements. If the existing positive element value is too small of a capacitor or too large of an inductor, new negative element values are created in the filter. When this happens, the specific Norton transformation values cannot be used in a real filter.

Pi to T and T to PI Transformation

All PI combinations of elements have a unique equivalent T combination, and all T combinations have a unique and equivalent PI combination. To convert between PI and T combinations of element, right click on the desired PI or T elements, and select PI to T or T to PI in the pop-up Change control Panel. Only single element types may be converted. LC resonators do not have T to PI or PI to T transformation.

Norton Theory

The theory behind Norton transformation is very simple and well documented. It is based on the equivalence of the following two circuits:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

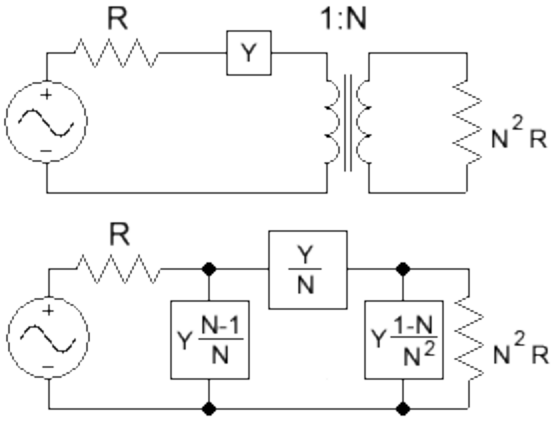


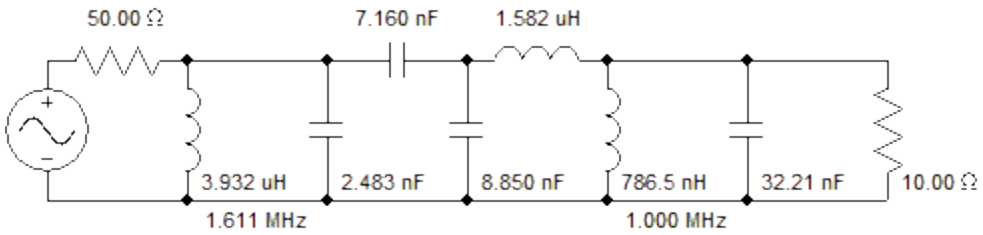
Figure 2: Norton Admittance Equivalence

Admittance is used in the above illustration because Nortons are usually performed on capacitors. It is easily seen above why Norton transformations are sometimes referred to as "Norton Transformers" or "Capacitor Transformers" in that they may substitute for an ideal coupled inductor transformer.

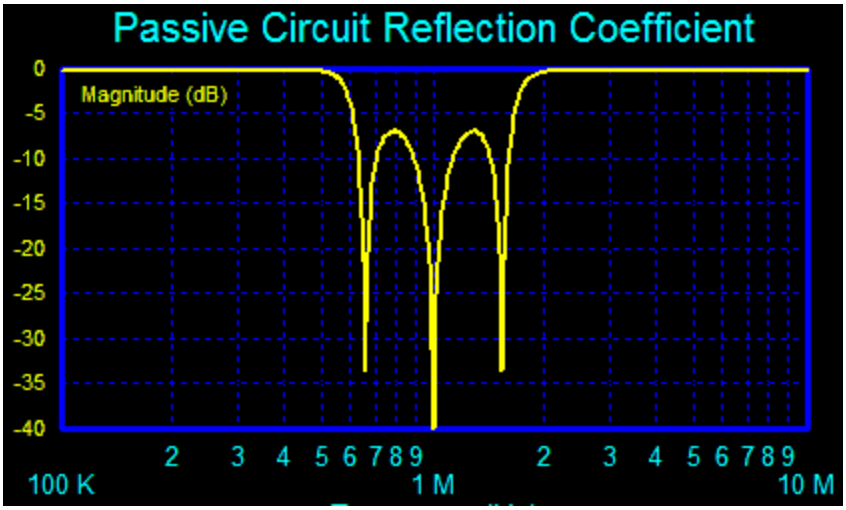
Impedance Matching

Norton transformations may be used to match band pass filters with unequal resistors when it is desirable to achieve a matched reflection response with unequal resistors. To do this, design an equally terminated band pass filter, then Norton the load or the source to the desired value. The result will be a filter with unequal terminations and a matched reflection response.

The following example depicts a 50 ohm source and 10 Ohm load matched with the use of a Norton transformation. Note that the reflection coefficient is the same as that for a matched load:



Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Norton Transformation to Match Unequal Loads

Note that this function may be performed automatically with the use of the "Match Impedance" Checkbox in the Lumped Control Panel.

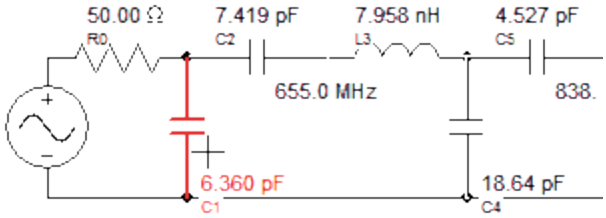
LC Transformations

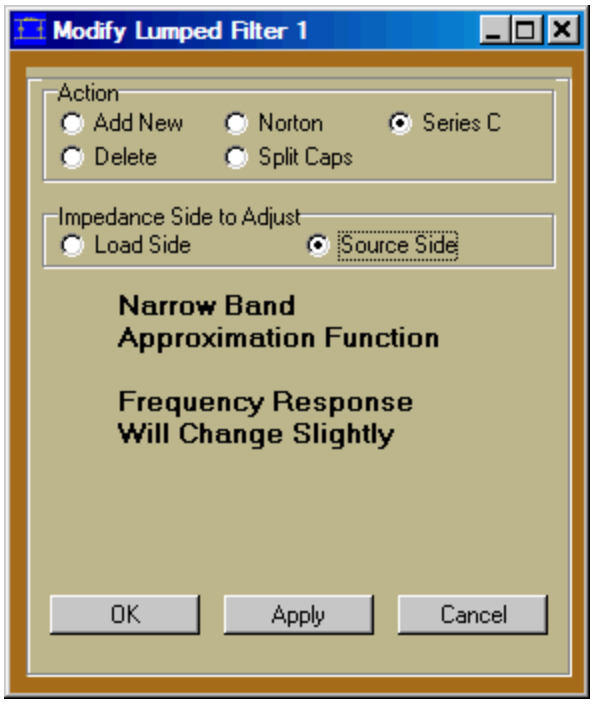
Using the right mouse key to click on outer reactive element of a band pass filters that is of differing type and orientation than the next outer element (an "L" of LC") permits the swapping of the L and C, and applying a transform that restores an approximate frequency response. In the case of coupled resonator filters, the new approximate frequency response is generally more symmetrical than the original frequency response, and applying this transformation is useful in cases where a more symmetrical frequency response is desired than what is obtained from direct coupled resonator or tubular filters.

The LC transformation will change one of the termination values. However, performing the same transformation at both ends of the band pass filter will restore the termination value in most cases. When it does not, a Norton transform will almost always restore the value.

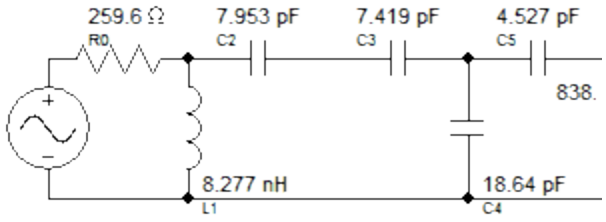
The below example illustrates the process of selecting an outer capacitor with the right mouse key, selecting, "Series C" to transform it to a series capacitance, and the adjacent inductor to a shunt inductance, and applying the subsequent impedance scale to the source resistance.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



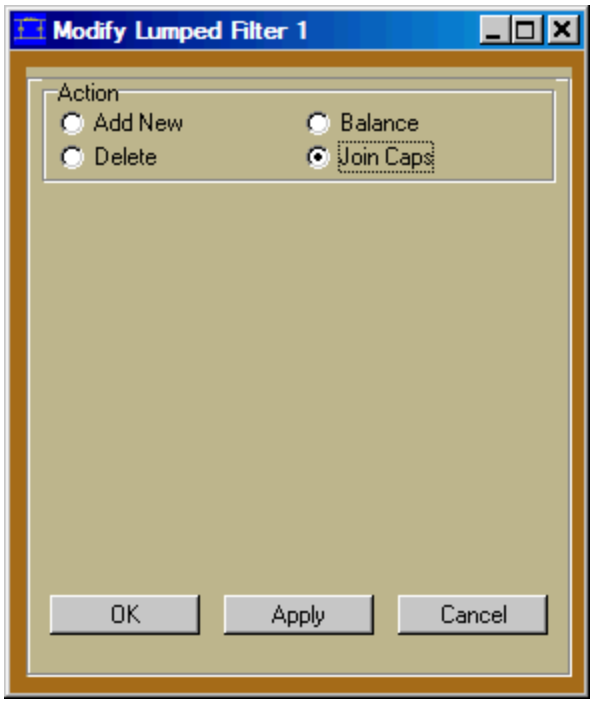


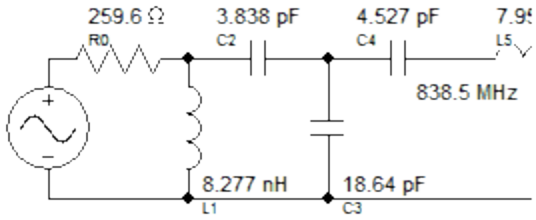
Selecting an LC Transform of outer LC Elements in an "L" configuration



Resulting Elements and Termination Designed for a More Symmetrical Frequency Response

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





Selecting and Joining the Series Capacitors to Make One Capacitor

Richards Transforms

Lumped elements may be transformed to transmission lines of selected phase or impedance with the use of Richards Transforms. This may be desirable for high frequency implementations when lumped elements are not easily available in the desired size or geometry. To convert lumped elements to transmission line equivalents, **right click the desired elemen**t and select, "To Tx Line" in the pop-up edit panel. An example is illustrated in Figure 1 below.

FilterSolutions support the following lumped to transmission line conversions:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Lumped Element	Equivalent Transmission Line	Supported Conversions
Vertical Inductor	Shorted Stub	Select Desired Angle Select Desired Impedance
Vertical Capacitor	Open Stub	Select Desired Angle Select Desired Impedance
Horizontal Inductor	Series Segment	Select Desired Angle Select Desired Impedance
Vertical Capacitor	Series Segment	Select Desired Angle Select Desired Impedance
Vertical Series LC Resonator	Open Stub	90º at Resonance
Vertical Parallel LC Resonator	Shorted Stub	90º at Resonance
Horizontal Series LC Resonator	Series Segment	180º at Resonance
Vertical Parallel LC Resonator	Series Segment	180º at Resonance

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

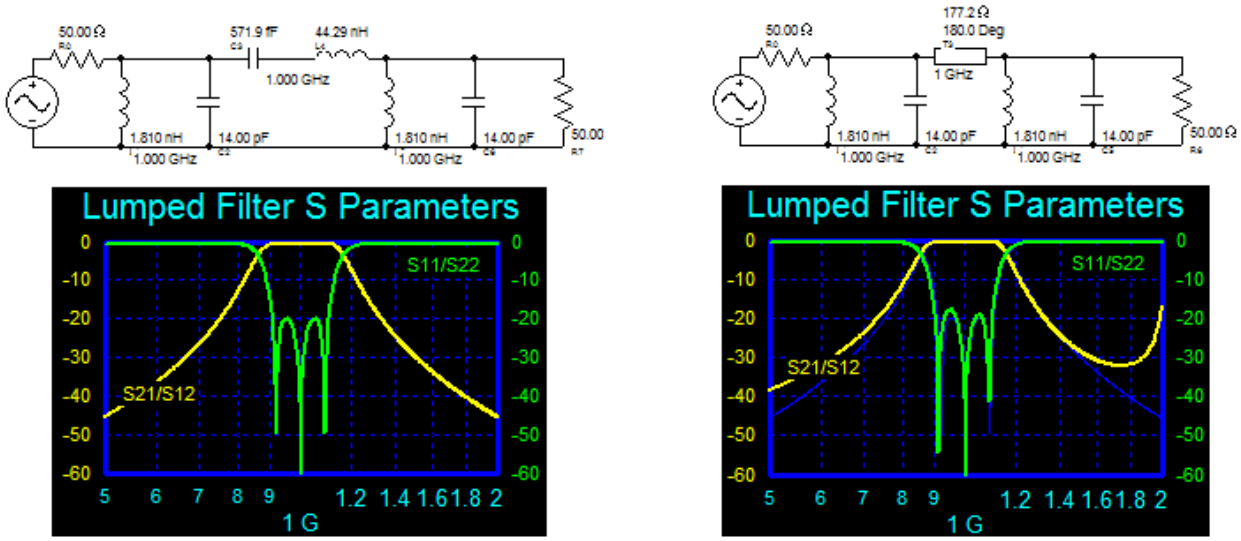


Figure: Richards Transform Equivalents.

Kuroda Transforms

Kuroda Transforms convert vertical elements and resonators to horizontal elements and resonators and vice versa with the use of additional 90º segments at the cutoff or center frequency and impedance level of the filter on each side of the element. The 90º segments serve to adjust the phase angle as needed to maintain proper element impedance at the cutoff or center frequency. 90º segments are omitted at the first and last element in the filter, due to the lack of need. 90º segments are deleted when they occur adjacent to another 90º segment in that they cancel. An example is illustrated in Figure 1, below.

Kuroda Transforms are useful when it is desired to use different element values or change the orientation of elements..

To perform a Kuroda transform, use the right mouse key to select an element or resonator, and select, **to Vertical** or **to Horizontal** in the pop-up edit menu, as needed.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

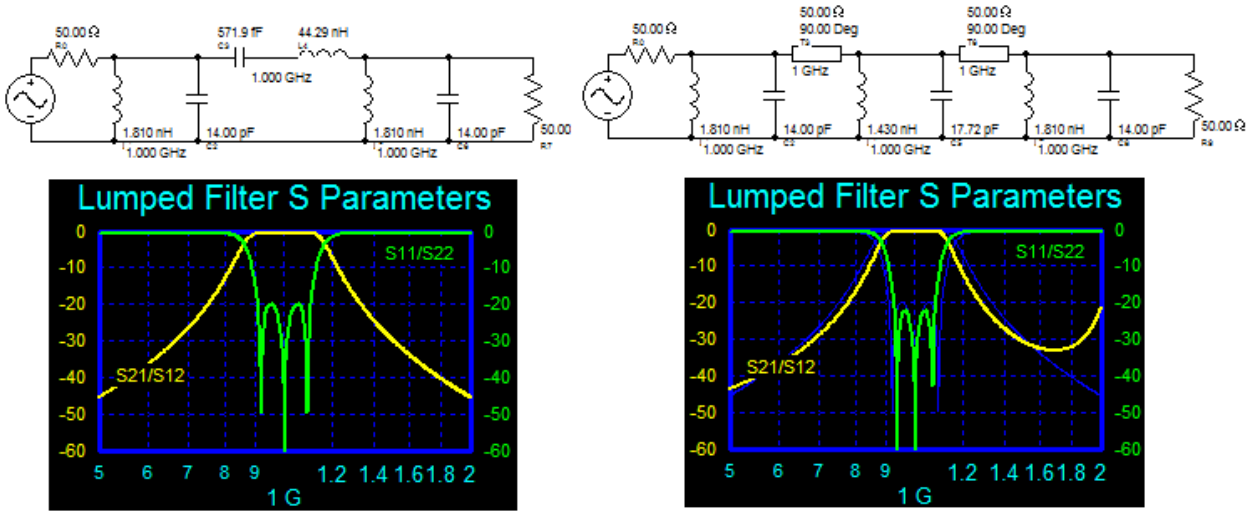


Figure 1: Kuroda Transform Equivalents

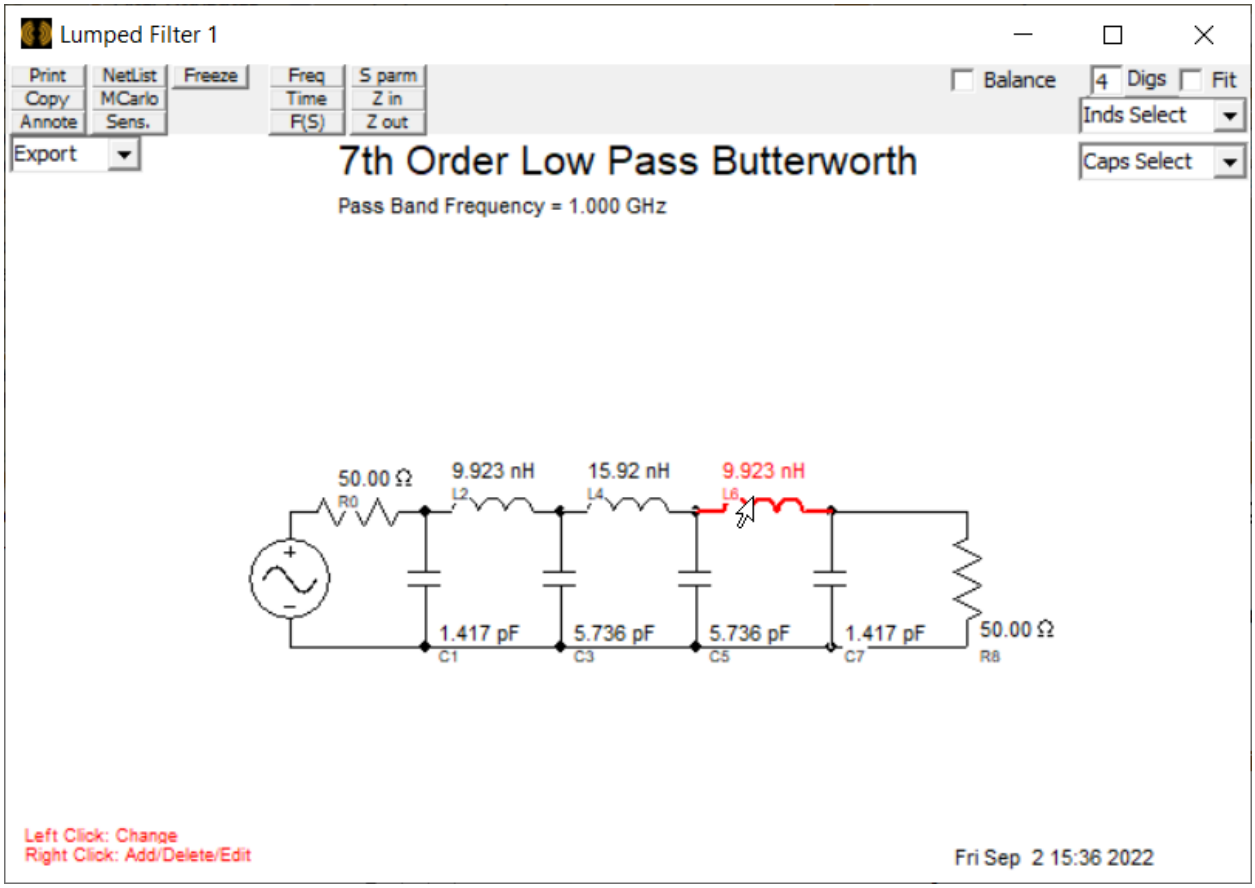
Standard Parts

FilterSolutions provides a flexible means to update the lumped or active filter elements values with the nearest value from the available standard parts, or the nearest 1%, 5%, and 10% standard part value. Up to three standard parts databases are maintained by **FilterSolutions**. If you need more than three, they may easily be maintained in another text or word processing document and copied or pasted in and out of FilterSolutions.

To update filter parts to the nearest value in the parts list:

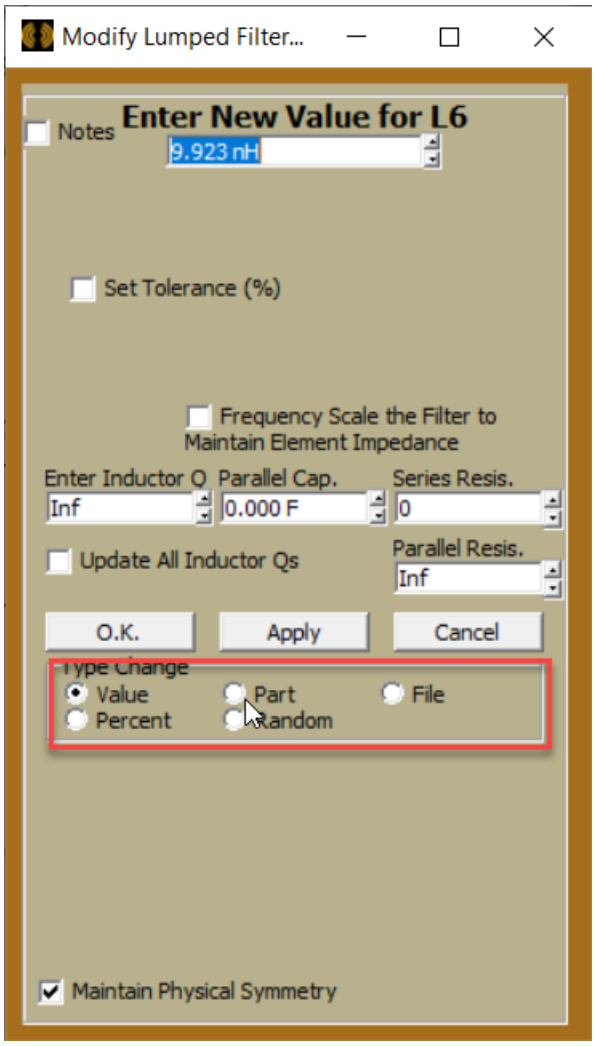
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select an element in the filter display.



2. Select **Parts** in the **Type Change** group box.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Select **OK**

Standard part updates are also available in the upper right corner of passive and active schematics.

Passive and active filters may have individual element or all like elements set to the nearest standard value from the selected database. The checkbox at the bottom of the Change control panel determines if one or all elements are updated.

You may select the default element value or the currently assigned element value as the base for the nearest part. A selection box in the change control panel determines the choice.

The upper part of the FilterSolutions standard parts window is shown below with some sample standard parts.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting the Q (Lumped filters only)

To associate a specific part value with a specific Q, add "Q=50", where 50 is an example Q value to the part line. For example:

Setting Series and Parallel Parasitic Resistance (Lumped filters only)

To associate a specific part value with a specific resistance, add "Rs=5" and "Rp=525", where 5 is an example series resistance value, and 525 is an example parallel resistance. The standard parts form is shown in Figure 1 below.

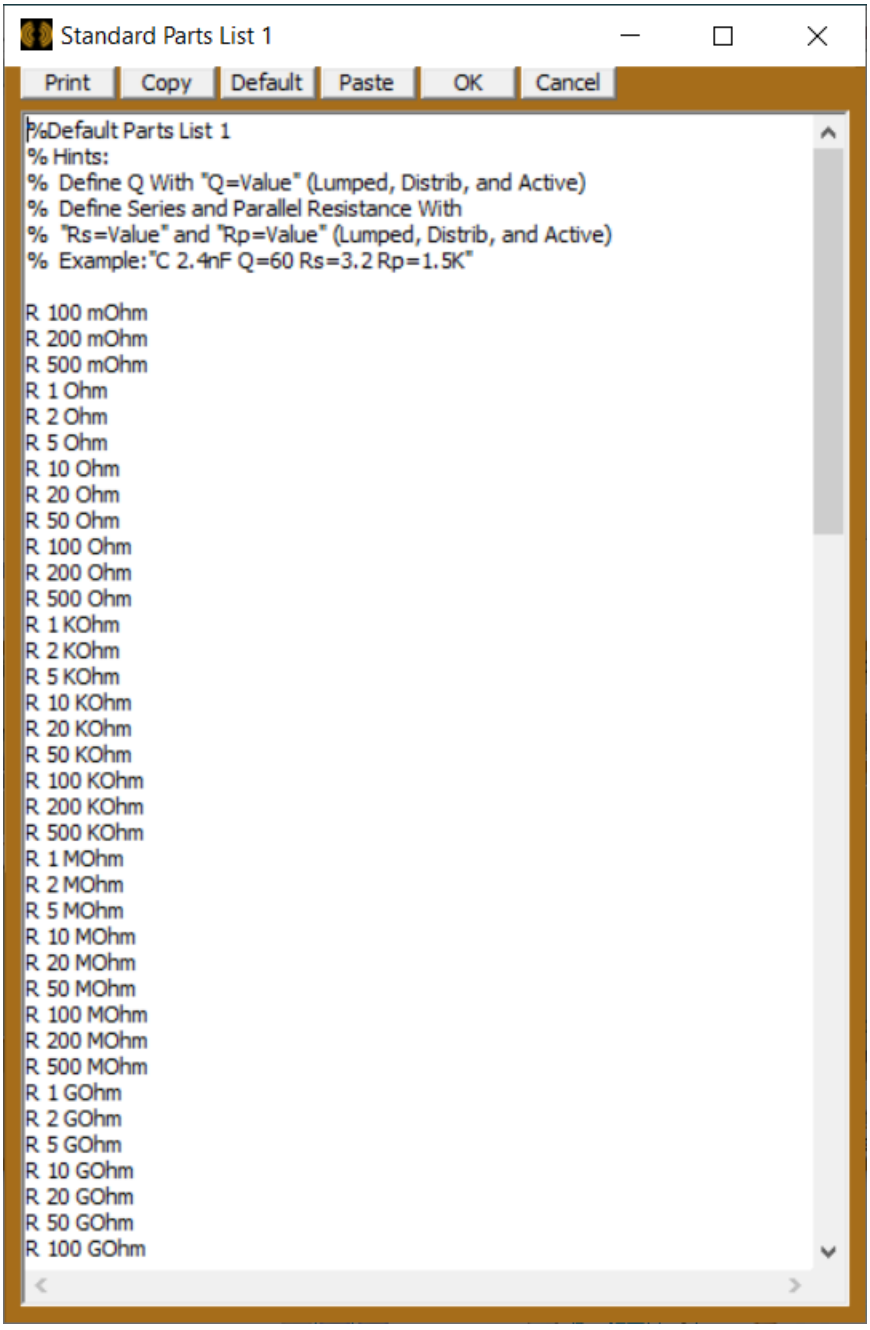
For Example:

"L 32uH Q=40 Rs = 3.5 Rp=7.25K" will have the effect of setting all inductors whose nearest part value is 32 uH to a Q of 40, a series resistance of 3.5 Ohms, and a parallel resistance of 7.25K Ohms.

Saving in an External File

FilterSolutions saves the user defined parts list in its own file. However, if it is desired to save the parts list in an external file, press Ctrl+C to copy the parts list and Ctrl+P to paste in the external file.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

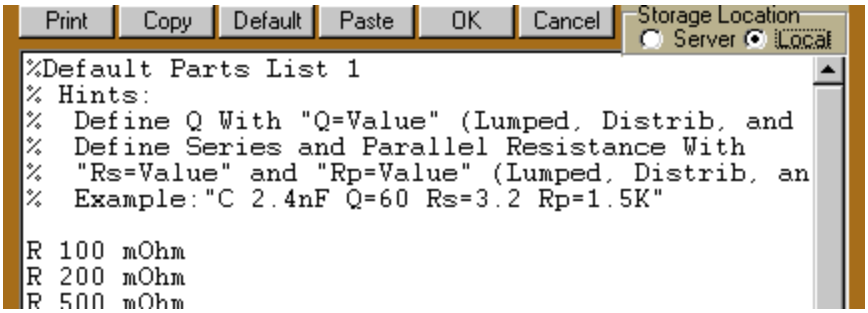


Saving on a common network server (Floating licenses only)

Network users of FilterSolutions may save standard parts on the network manager with the use of an Administrator's password. Network users may retrieve this common standard parts lists for use in their projects. This helps insure that all users on a project are using the same standard parts list. Users and administrators may select in the upper right where their part lists come

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

from, as shown in Figure 2 below. Server parts are read-only for users, and read/write for administrators. The license key issued by Nuhertz determines user/administrator status.



Parts Files

If administrators do not wish to use the administrative passwords to upload standard parts to Nuhertz Server, parts files may be manually copied into their respective locations instead. Client FilterSolutions parts files are located in the FilterSolutions_ini directory Default: (C:\Program Files (x86)\Nuhertz Technologies\FilterSolutions_ini\) under the Nuhertz Technologies program directory. They are names, parts0.dat, parts1.dat and parts2.dat for standard parts lists 1, 2, and 3 respectively. These files may be manually copied into the server Nuhertz Server program directory (Default: C:\Program Files (x86)\Nuhertz Technologies\Nuhertz Server\) as well, removing the necessity of providing an administrators password to the individual(s) charged with maintaining the parts list.

The buttons across the top perform the following functions:

Print	Print the displayed standard parts
Copy	Copy the displayed parts to the windows clipboard. Select the portion to be copied, or select all or nothing to copy all text in the text box.
Default	Sets the standard parts to the default parts list.
Paste	Paste text from the windows clipboard into the standard parts text box.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

OK	Save you standard parts list in the FilterSolutions database and exit.
Cancel	Exit without saving the parts list.
Storage Location	Determines where the standard parts lists is read from. Administrators may write to the server parts lists. Users may only read from it.

Parts List Formats:

Comments	Precede the comment line with a "%" character. Up to 10 comments are allowed.
Numerical Entries	Integer, floating, and engineering notation are all acceptable. Six significant digits are stored in the database. Example: 12, 0.00047, 470n, 50pF are all acceptable numerical entries. All numerical entries are sorted in ascending order in the database. Up to 500 numerical entries are allowed per element type.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Capacitors	Precede the capacitance numerical entry with a "C" or "c" character.
Inductors	Precede the inductance numerical entry with a "L" or "l" character.
Resistor	Precede the capacitance numerical entry with a "R" or "r" character.

1%, 5%, 10% and 20% Standard Parts

The Change Control Panel has selections for 1%, 5%, 10%, and 20%. When these options are selected, the nearest part value is selected from a list of standard industrial values. These values are listed below:

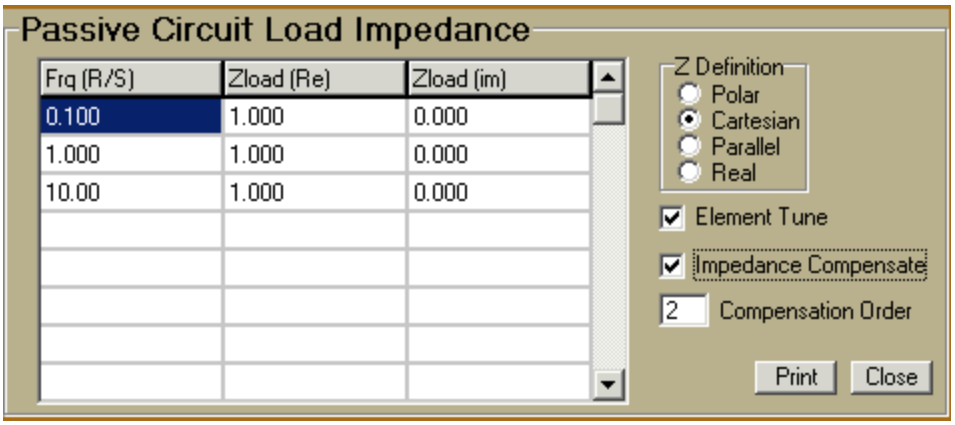
20% Parts	1.00, 1.5, 2.2, 3.3, 4.7, 6.8, 10.0, 15.0...
10% Parts	1.00, 1.20, 1.50, 1.80, 2.20, 2.70, 3.30, 3.90, 4.70, 5.60, 6.80, 8.20, 10.00, 12.0...
5% Parts	1.00, 1.10, 1.20, 1.30, 1.50, 1.60, 1.80, 2.00, 2.20, 2.40, 2.70, 3.00, 3.30, 3.60, 3.90, 4.30, 4.70, 5.10, 5.60, 6.20, 6.80, 7.50, 8.20, 9.10, 10.00, 11.00...
1% Parts	1.00, 1.02, 1.05, 1.07, 1.10, 1.13, 1.15, 1.18, 1.21, 1.24, 1.27, 1.30, 1.33, 1.37, 1.40, 1.43, 1.47, 1.50, 1.54, 1.58, 1.62, 1.65, 1.69, 1.74, 1.78, 1.82, 1.87, 1.91, 1.96, 2.00, 2.05, 2.10, 2.15, 2.21, 2.26, 2.32, 2.37, 2.43, 2.49, 2.55, 2.61, 2.67, 2.74, 2.80, 2.87, 2.94, 3.01, 3.09, 3.16, 3.24, 3.32, 3.40, 3.48, 3.57, 3.65, 3.74, 3.83, 3.92, 4.02, 4.12, 4.22, 4.32, 4.42, 4.53, 4.64, 4.75, 4.87, 4.99, 5.11, 5.23, 5.36, 5.49, 5.62, 5.76, 5.90, 6.04, 6.19, 6.34, 6.49, 6.65, 6.81, 6.98, 7.15, 7.32, 7.50, 7.68, 7.87, 8.06, 8.25, 8.45,

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	8.66, 8.87, 9.09, 9.31, 9.53, 9.76, 10.0, 10.2...

Complex Terminations

Passive and transmission line filters may be terminated with user defined complex impedances by checking the "Complex Terminations" box in the passive and/or transmission line control panels and then selecting either load or source impedance selection buttons. Complex impedances are defined in table form as a function of frequency. The form below pops up with all the controls necessary to define the load or source impedances.



As is easily ascertained from the panel layout, the user enters each impedance as a function of frequency. Impedances may be entered as Polar, Cartesian or Parallel coordinates. Parallel coordinates place the real and imaginary part of the complex impedance in parallel rather than series. Selecting "Real" has the effect of using the original resistive termination value.

Element Tuning

The Element Tuning function adjusts the LC element values in an attempt to minimize the magnitude RMS error between the ideal frequency response and the actual frequency response with the complex terminations.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

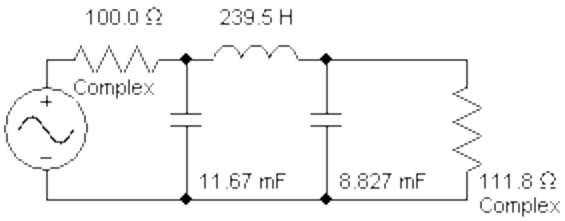
Impedance Compensate

The Impedance Compensation function adds matching element to the complex termination in an attempt to force the termination to appear resistive to the filter. Sometimes this improves the performance of the filter. Since this function also may have the effect of depredating performance, the user should carefully inspect the filter performance of this function prior to using the results.

Filter Display

The termination resistor in the filter display shows the impedance magnitude at the filter cutoff or center frequency accompanied by the word "Complex". The filter is synthesizes using resistive terminations of the same value in Ohms as displayed on the filter.

In the example below, the filter is synthesized using 100 and 111.8 Ohms for the source and load resistance respectively, then element tuned to achieve the desired filter response.



Filter with complex terminations

Lumped Netlists

To help retrieve the designed passive circuits, FilterSolutions will output the filter in a netlist. From the control bar of any displayed passive circuit, select **Netlist**, and to display the Netlist box on the left side of the filter window. The number of significant digits used on the netlist will match the number in the "Digs" box on the right side of the controls bar. The netlist contains entries for AC and Transient analysis, including graph scaling. An AC source is used for the AC analysis. A Pulse source is installed, but commented out for the transient analysis. To run a transient analysis it is necessary to comment out the AC source, and remove the comment from the Pulse source.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Finite Q

If the elements in the passive circuit have finite Q, FilterSolutions will model this by adding a series resistor to inductors and a parallel resistor to capacitors. If **Q** is selected in the upper right of the circuit display, the resistor value is a variable function of frequency. If **Res** is selected, then the size of the resistance is determined by the dominant frequency of the component. For LC tanks, the dominant frequency is taken to be the resonant frequency. For all other components, it is the cutoff or center frequency of the filter. **Not all spice programs recognize component values as a function of frequency.** It is therefore, sometimes necessary to use fixed resistors.

Since FilterSolutions does all analysis with resistance values corresponding to each frequency, there will be some slight difference in the netlist AC and Transient analysis and FilterSolutions' frequency and time analysis when finite Q is used with fixed resistor values.

Coupled Coils

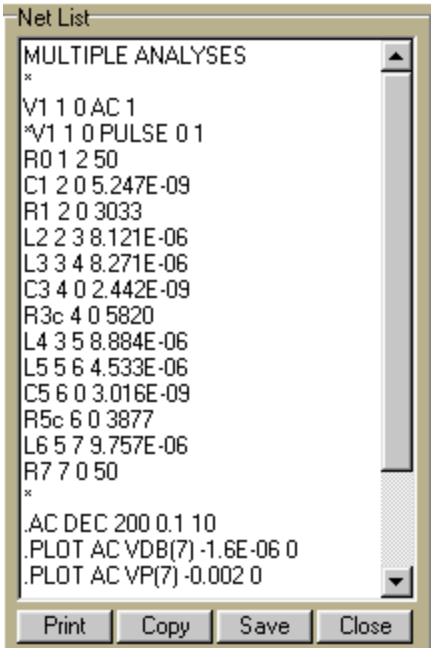
The netlist models coupled coils with three uncoupled inductors in a T arrangement. The center inductor in the model is a negative inductor of the same magnitude as the coupled coils. The side inductors in the model are positive and are of twice the value as the coupled inductors.

Dummy Resistors

Some SPICE simulations do not allow one to build a circuit without a DC path to ground. Dummy resistors are, therefore, inserted into the netlist when necessary to provide a DC path to ground. The sizes of the dummy resistors are large enough such that the error introduced into the SPICE simulation is insignificant.

The following is an example of a FilterSolutions netlist:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note that the bottom parts of the netlist are hidden below the text window. The scroll bar may be used to scroll up and down the netlist to observe these hidden parts.

Print	send the netlist to a printer.
Copy	copy all or part of the net list to the windows clipboard. Partial text may be copied by highlighting desired text. All text may be copied by selecting all the text or none of the text.
Save	save the netlist to a text file.
Close	close the netlist window.

Sensitivity

This chapter contains the following topics:

- Monte Carlo Analysis
- Element Sensitivity Tables
- Element Sensitivity Plots

Monte Carlo Analysis

Lumped LC filters may be statistically analyzed through Monte Carlo analysis. The per cent tolerance of each inductor and capacitor may be set individually, or a default value for all

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

inductors and all capacitors may be used.

To execute a Monte Carlo on a passive LC circuit, open the desired graph to be analysis, such as frequency response, time response, reflection response, etc., use the left mouse key to select any inductor or capacitor to open the change control panel, select **Random** in the change control panel, and fill out the remainder or the information at the bottom of the Change Control panel. If **Maintain Old Traces** is selected, the graphical display will accumulate all the traces for easy visual interpretation.

Only schematic analysis graphs are updated with Monte Carlo analysis traces. Ideal analysis graphs are not updated.

Elements defined with S-Parameter files will continue to use the nominal value for Monte Carlo analysis.

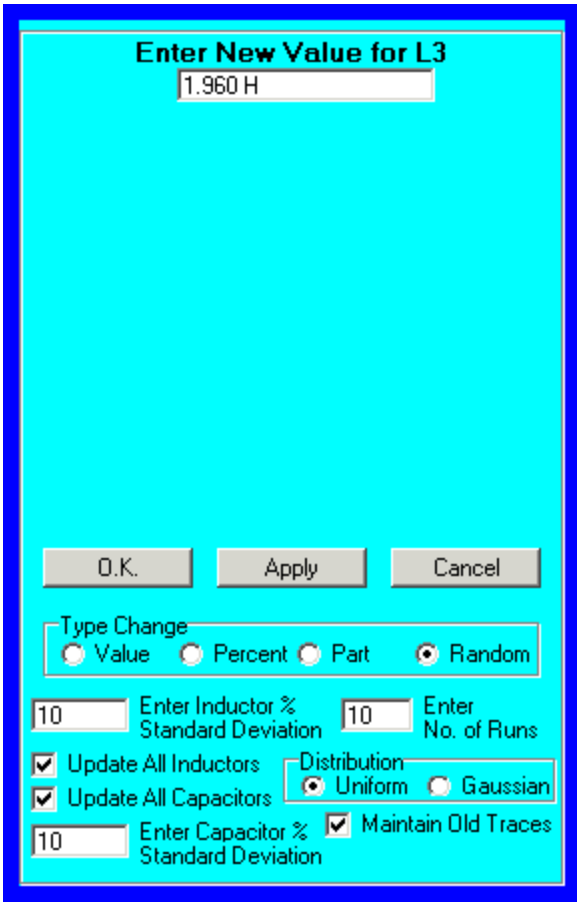
Selecting element tolerance

The change control panel permits a global selection of tolerance values for all inductors and all capacitors. **The global value is only applied to inductors and capacitors that do not have a tolerance assigned.** To assign a specific tolerance value to a specific inductor or capacitor, use the left mouse button to select the inductor or capacitor, check the "Set Tolerance (%)" and enter the desired tolerance in the text entry to the right of the checkbox. The percent tolerance value will appear in the schematic near the element. If a specific percent tolerance is not assigned to an element, the global value in the change control panel is used for Monte Carlo analysis.

Example

A third order Butterworth filter is analysis with a Monte Carlo analysis. All inductors and capacitors are set to 10% tolerance. 10 runs are selected to be generated, and old traces are selected to be remembered, as seen in the example below.

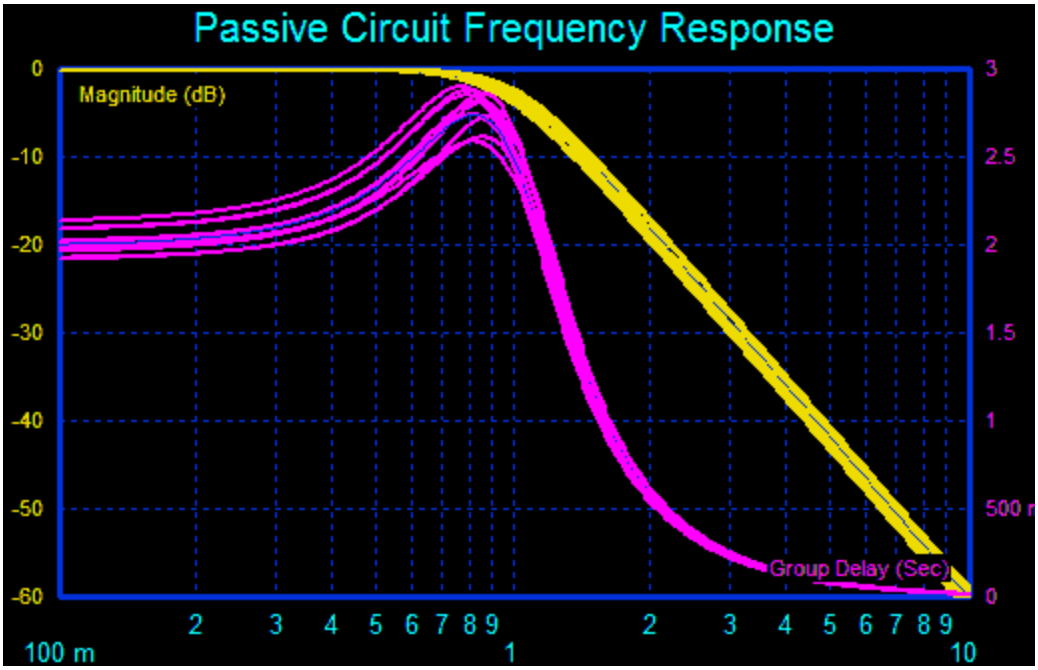
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Monte Carlo Setup

When the Change Control Panel above is filled out, select **Apply** or **OK** to generate the Monte Carlo analysis graphs on all graphs currently open on the selected filter schematic. The frequency response should look like the graph below.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Monte Carlo Graph

Element Sensitivity Tables

The effect of element value errors on magnitude, phase, and group delay filter performance may be analyzed for all elements simultaneously by selecting the **Sens.** button on top of the passive filter schematic. The sensitivity table showing numerical sensitivity values at a specific frequency appears on the schematic. A quick visual scan of the table may indicate which elements need higher precision, and which ones may be low cost loose precision parts.

Due to limited display space, diplexer and triplexer sensitivity tables only display elements from one diplexer or triplexer leg. The user must select which leg to display.

Elements defined with S-Parameter files will continue to use the nominal value for sensitivity analysis.

Sensitivity Definitions

To obtain meaningful sensitivity numbers with reasonable values for all filters, FilterSolutions has adapted the following definitions to measure sensitivity of parts with respect to frequency response aspects:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Magnitude in dB	Arithmetic change in dB per % change in element value
Magnitude in Arithmetic Volts	% change in magnitude per % change in element value
Phase Angle in Degrees	Arithmetic change in degrees per % change in element value
Phase Angle in Radians	Arithmetic change in radians per % change in element value
Group Delay	% change in group delay per % change in element value

Frequency Selection

Low pass and high pass filters default to the cut off frequency, otherwise known as pass band frequency, to perform the sensitivity analysis. Band pass and band stop filters default to the center frequency. Band pass and band stop filters permit the selection of the lower and upper cutoff frequencies, and all filter permit the selection of a user defined frequency.

Specific element sensitivity analysis

Click any element in the sensitivity table to see more information of the effect of filter performance on that specific element. More information on this subject is available in the Help under "Element Sensitivity Plots."

Element Sensitivity Plots

Left click any element in the element sensitivity tables to generate an element sensitivity frequency plot. (Select the **Sens.** button in the passive filter control bar to generate element sensitivity tables.)

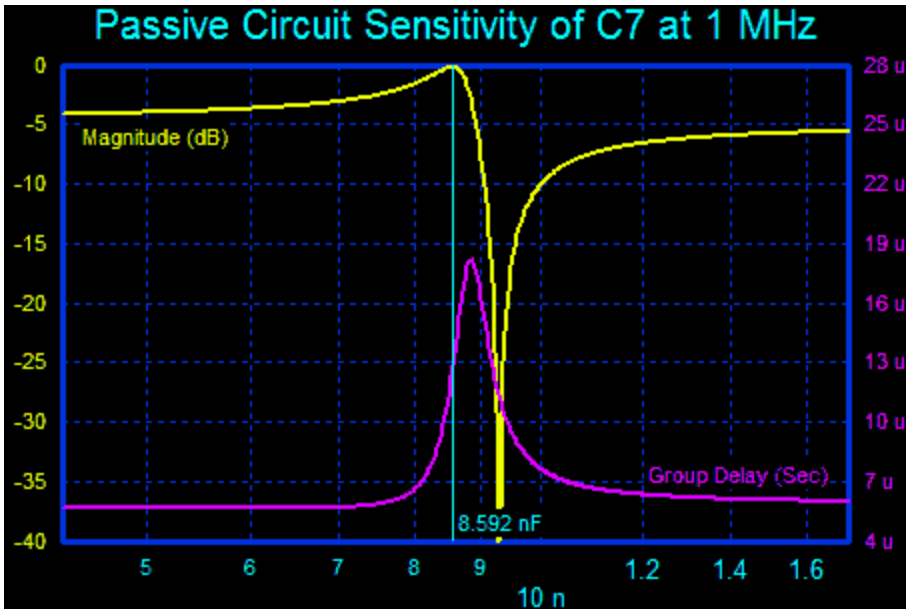
Element sensitivity tables only provide a sensitivity information about the element value. Since sensitivity is not linear, this information has its limits in usefulness. Element sensitivity plots provide magnitude, phase, and group delay frequency information about a range of element values. FilterSolutions provides provide frequency information as a function of element value in the range 1/2 to 2 times the actual value for semi-log plots, and 1/2 to 1.5 times the actual value for arithmetic plots. The "Log" switch setting in the Main control panel determines which type, semi-log or arithmetic, of plot to display.

Elements defined with S-Parameter files will continue to use the nominal value for sensitivity plots.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Example:

The example below shows a case where the sensitivity table would show what might be considered a good low magnitude sensitivity response at the cutoff frequency of 1MHz, whereas the component sensitivity plot shows a potentially significant degradation of performance as the component value increases.



Element Sensitivity Graph

Modelithics Exports to Electronic Desktop

This chapter contains the following topic:

- Modelithics in FS and EDT
- Automated Discrete Design with Modelithics Tutorial

Modelithics in FS and EDT

Lumped element Filters exported to Ansys Circuits have the option of being simulated with highly accurate Modelithics models. The filter layout, including device pad geometries, can then be generated in Circuits. FilterSolutions generated Modelithics/Circuits Schematics may be optimized by the part value and/or layout interconnect widths and lengths. The user may wish to customize the layout geometries to suit his or her specific application. Optimized or tuned Modelithics part values are confined to discrete manufacturer value through the use of Design Array variables loaded with manufacturer values supplied by Modelithics.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Loading Modelithics into Circuits

Modelithics media and licensing must be obtained from Modelithics and installed into the specific version of Electronic Desktop that it is being used on. After Modelithics is installed, it must be loaded into FilterSolutions with the use of the **Load Modelithics Models** button shown in Figure 1. The loading operation only needs to be performed once following the installation of Modelithics. FilterSolutions will store the results internally for later use.

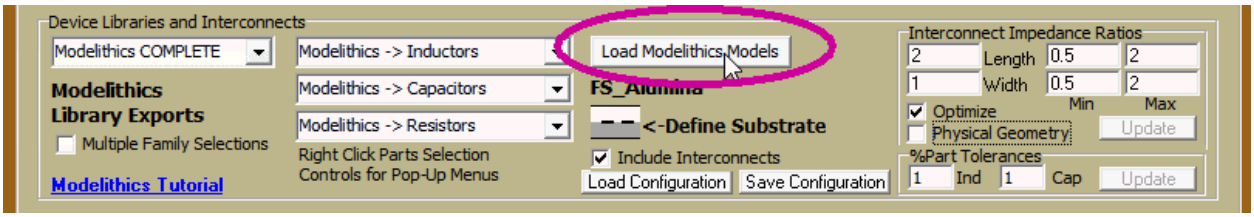


Figure 1: Loading Modelithics models into Ansys Nuhertz FilterSolutions

Selecting Modelithics Models

Modelithics inductors, capacitors, and resistors are easily selectable with the use of the right mouse click Pop up Menus, as shown below in Figure 2. Each selection shows the range of the model family for easy visual reference. Family selection may also be made directly with the use of the drop down selectors, if desired.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

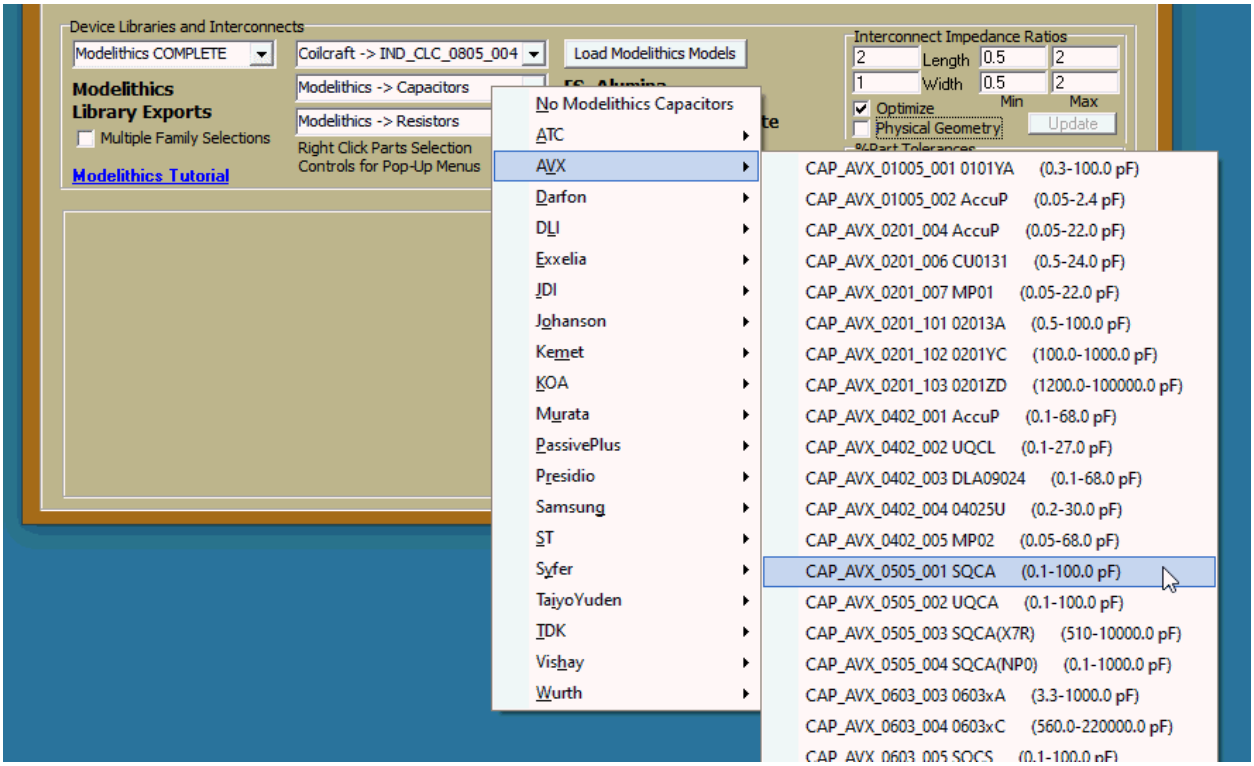
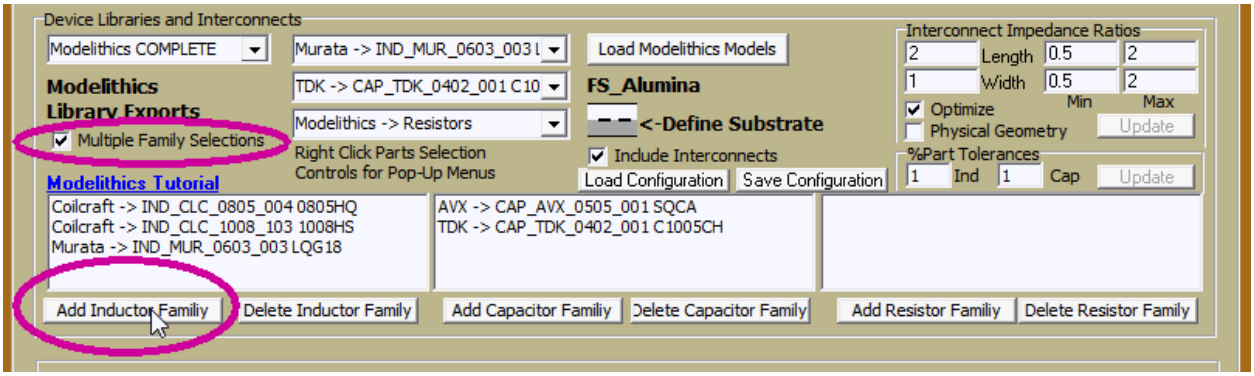


Figure 2: Selecting a Modelithics Manufacturer and Family

Selecting Multiple Families

If more than one family of parts is desired **Multiple Family Selections** may be selected, and individual families may then be selected and entered in to the multiple family tables as shown in Figure 3 either through the **Add**... button or through a drag and Drop operation. At export time, the part with the closest value to the synthesized value from all families entered will be chosen for export.

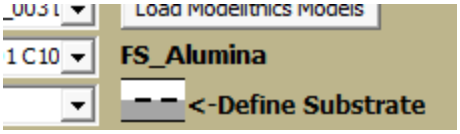


Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

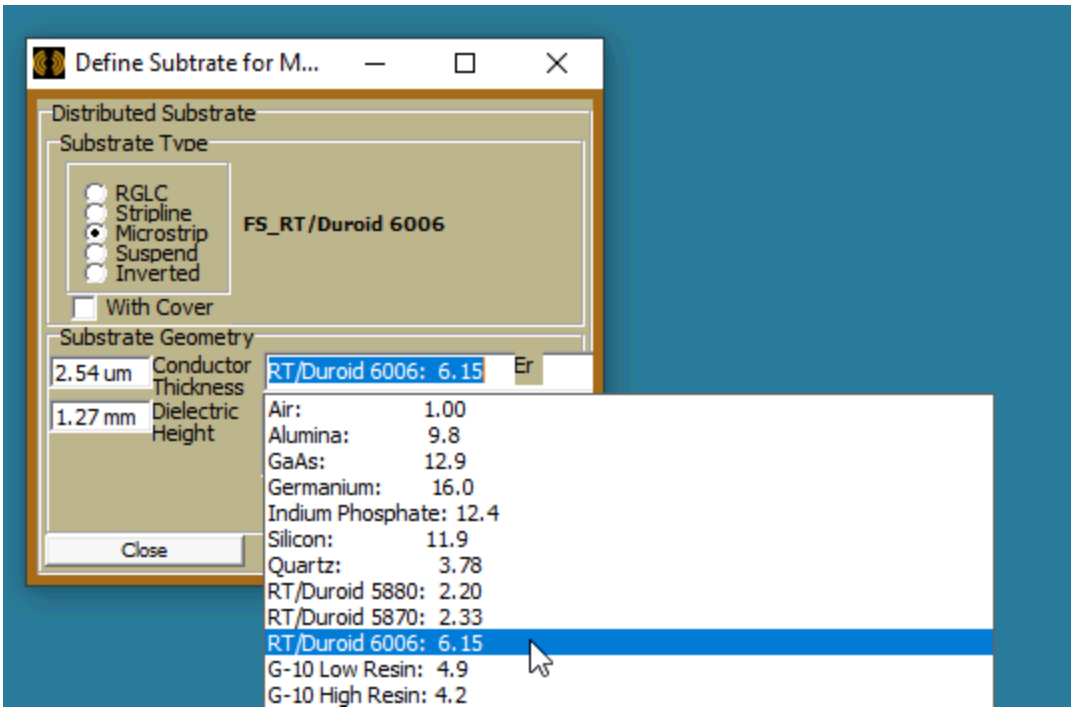
Figure 3: Selecting Multiple Part Families

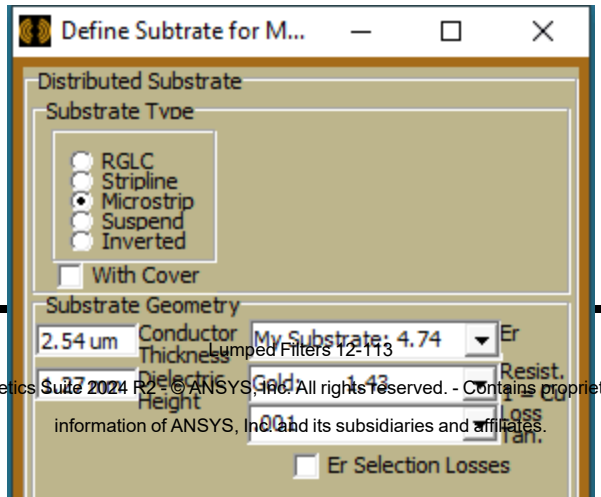
Defining the Substrate

Select **Define Substrate** in the export control panel to define the substrate in the Modelithics schematic to be exported to Circuits.



The substrate definition page shown in Figure 4 below will appear. Any built-in substrate material may be selected, or a custom name and Er value may be entered by typing it in, as shown in Figure 4.





Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Built-In Material Selection Hand Entered Material Selection

Figure4: Defining the Modelithics Substrate

Circuits Schematics and Layouts

Modelithics elements in Circuits schematics support layout cells with ports to attach to micro strip interconnect elements.

Interconnect Length Ratio and **Impedance Width Ratio** are used to set initial length and width geometry of the interconnects. Geometry may be defined in terms of substrate height ratios or physical dimensions, as desired, shown in Figure 5. Circuits/Modelithics Schematics are designed to produce valid layouts, as shown with this five pole Zigzag filter below in Figure 6, where 4 inductors and 11 capacitors may easily be seen to be included in the layout. Initial layout geometry is set to be optimized with the use of Circuits "S" and "W" equations. All Circuits equations are designed to insure a valid layout when reasonable geometry setting are entered.

The first export generally with Optimize unchecked so that only the discrete element values are optimized. Layout geometry Circuits optimization may be turned on and off remotely from the FS export panel by checking or unchecking Optimize and then selecting the **Update** button to the immediate right of Optimize.

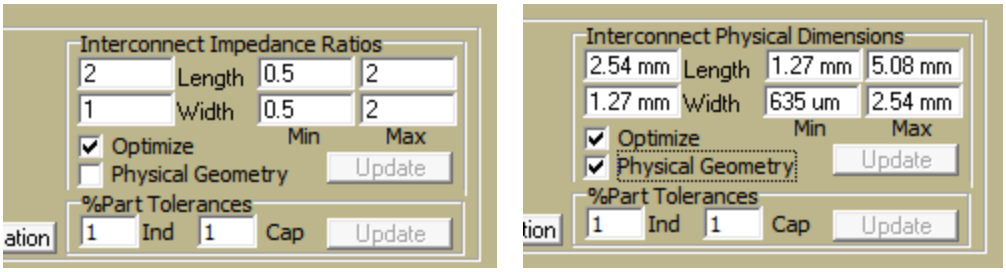


Figure 5: Interconnect geometry Defined in Terms of Substrate Height Ratios or Physical Dimensions

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

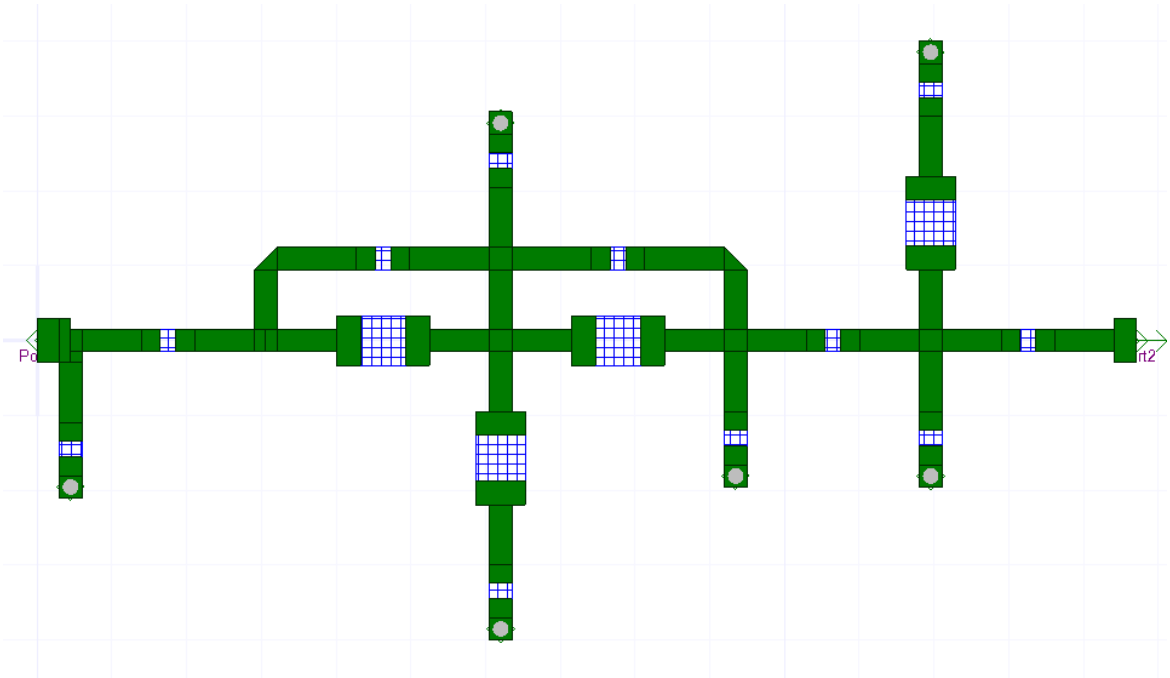


Figure 6: Valid Circuits Schematic Layout of a Modelithics Parts Filter Design

Tuned/Optimized Layouts

Modelithics layouts are designed to remain valid for reasonable width and length adjustment. All equations are embedded into the design variables and layout geometry that are needed to keep the layout valid in the presence of interconnect width and length adjustments. Figure 7 below shows the circuit layout from Figure 6 above after the interconnects have been adjusted through tuning and/or optimizing.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

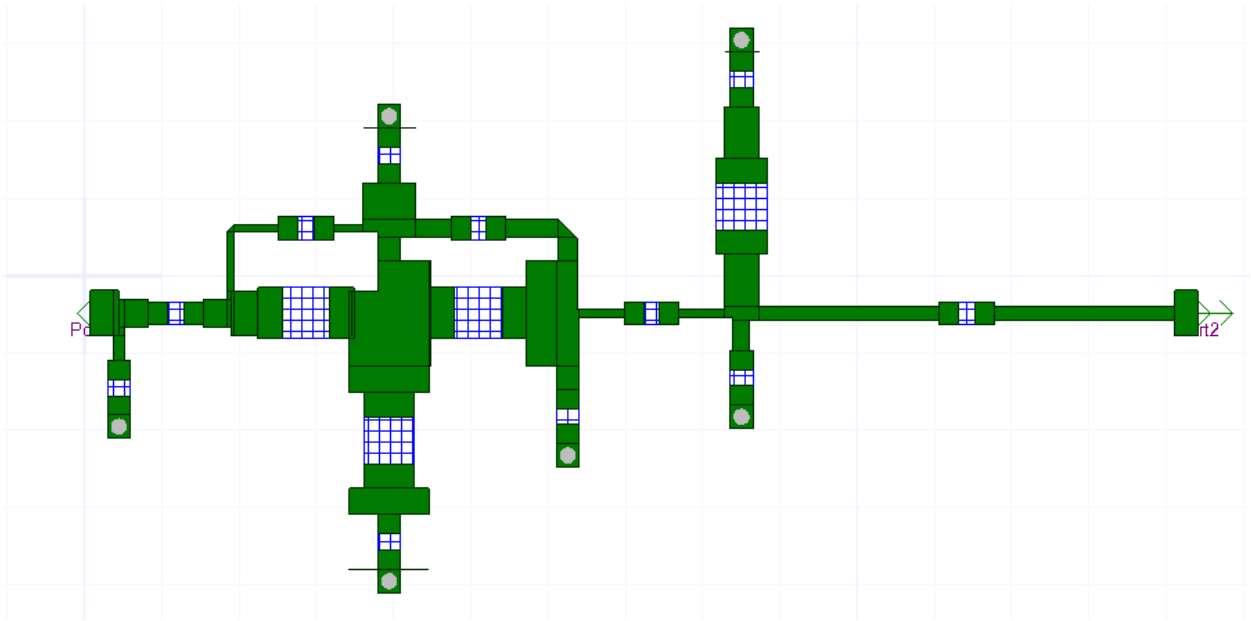


Figure 7: Valid Circuits Schematic Layout With Tuned/Optimized Interconnects

Automated Discrete Design with Modelithics Tutorial

This tutorial reviews information about designing with FilterSolutions and Modelithics. It also contains a step-by-step guide on the AEDT Optimization Process.

About designing with FilterSolutions and Modelithics:

- Highly accurate models.
- Easy to work with.
- Measurement based and well documented.
- Advanced pad features.
- Models for thousands of components for leading manufacturers.
- Good nodes and leads simulation.
- Footprint plugs right into Ansys' Circuits Layouts.

In this guide:

"AEDT Export Model Selection:" on the facing page

"AEDT Substrate Definition" on the facing page

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

"Initial and Updated Interconnect Geometry" on page 12-119

"Multiple Family Model Selection" on page 12-120

"About AEDT Optimization" on page 12-120

"AEDT Schematic Selection" on page 12-120

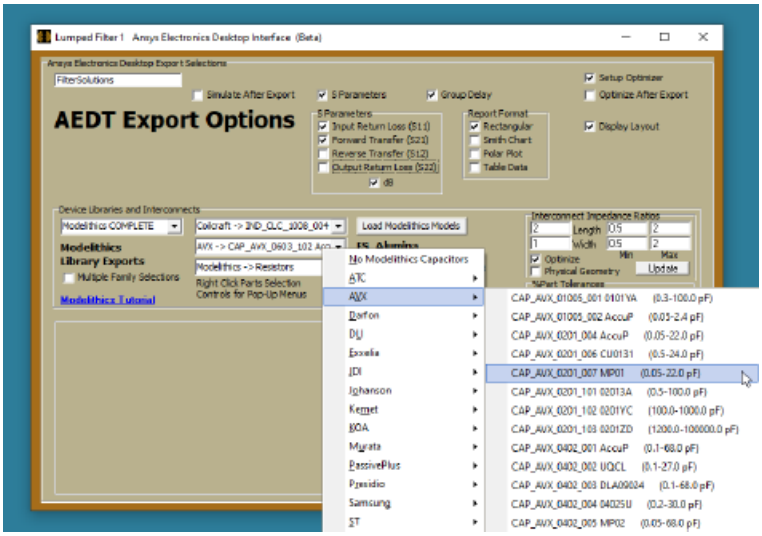
"AEDT Optimization Process " on page 12-121

AEDT Export Model Selection:

The **Device Libraries and Interconnects** group box contains drop down menus for Inductors, Capacitors, and Resistors part selections.

Click **Load Modelithics Models**- only needs to be done once.

Right-click drop down menus to view Modelithics family selections- or open drop downs directly.



AEDT Substrate Definition

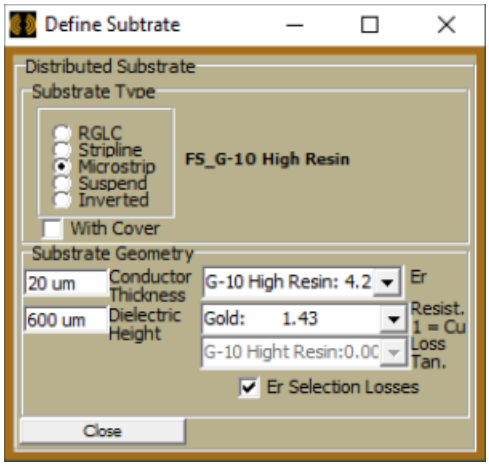
Modelithics models consider substrate parasitic effects.

Select Define Substrate graphics in the AEDT Export Panel

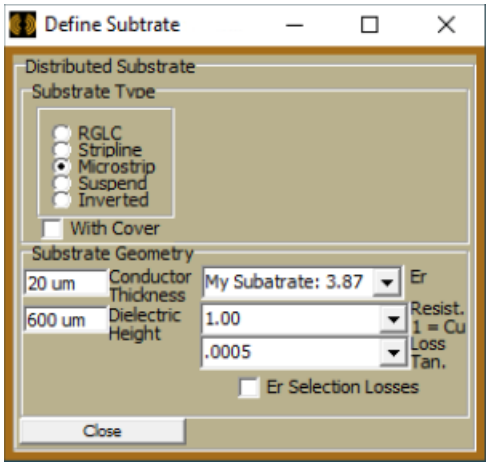
Fill out the pop-up Define Substrate dialog.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Select built-in substrate material.



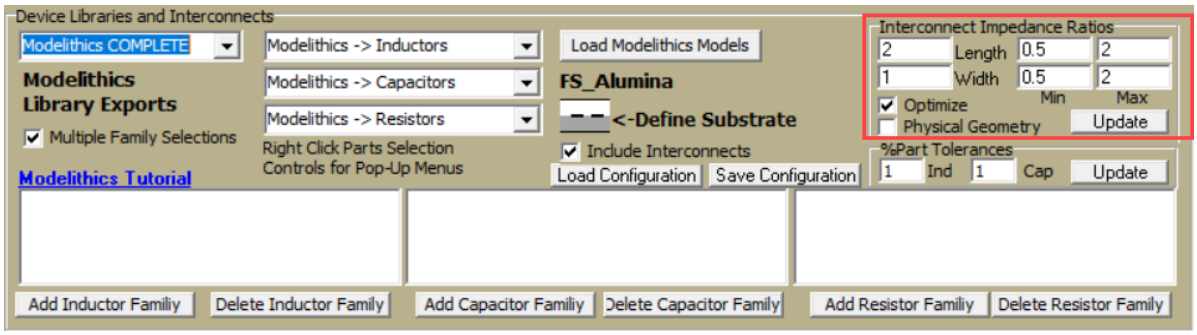
- Define your own substrate material.



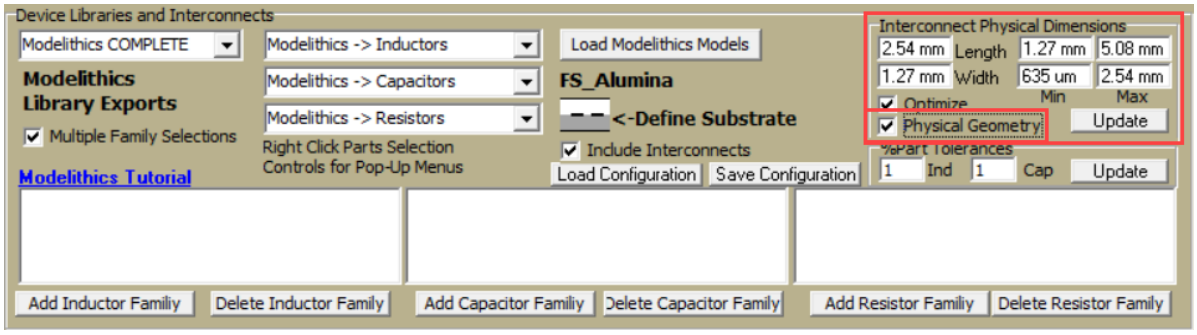
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Initial and Updated Interconnect Geometry

- Set initial interconnect width and length geometry, and optimization limits.



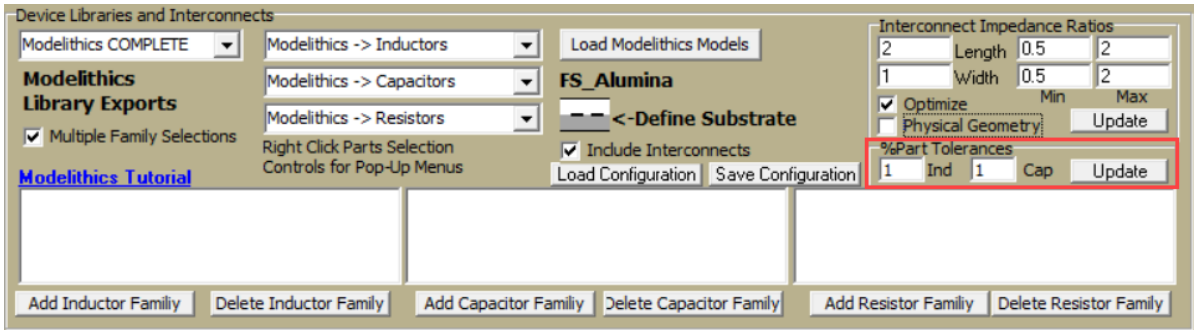
- AEDT Optimization may be used to refine interconnect geometries.
- Interconnect geometry optimization is optional and may be changed in real time:
  - Check or Uncheck **Optimize** at any time and select **Update** to change AEDT optimization include state for geometry variables.
- Interconnect width, length, and limits also changeable by selecting **Update**.
- **Physical Geometry** checked to display geometry in physical dimensions.



- Enter **% Part Tolerance** and **Update** to update Modelithics part tolerance values in real

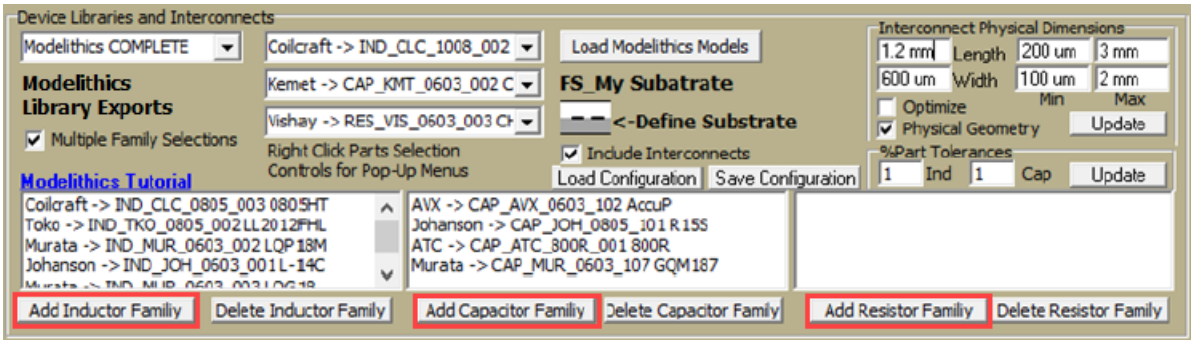
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

time.



Multiple Family Model Selection

Parts are selected from the multiple family listings through **Add Inductor Family**, **Add Capacitor Family**, **Add Resistor Family**, or drag and drop.



About AEDT Optimization

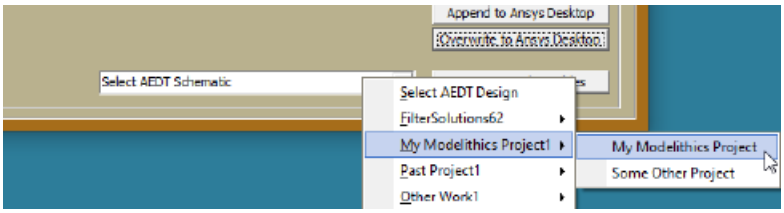
- Modelithics discrete values defined in array variables.
- Discrete optimization implemented through array indexing variables.
- AEDT performs an internal nearest integer function to assure a discrete selection.
- Geometry optimization implements through Width (W...) and Length (S...) ratio variables. (Defaults =1).

AEDT Schematic Selection

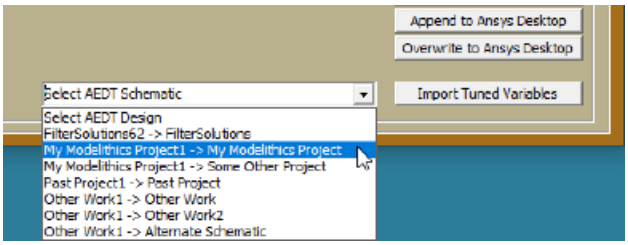
AEDT project and design may be selected for remote editing at any time.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Right-click Select **AEDT Schematic** to use the pup-up menu selection

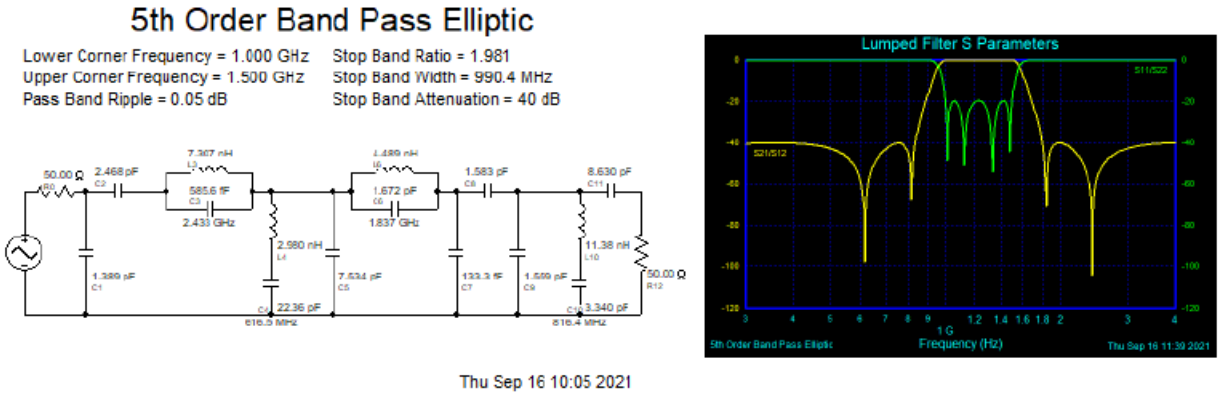


or just open the drop-down and select a schematic.



AEDT Optimization Process

1. **Synthesize with ANFS**
   - 5 Pole minimum inductor zigzag used as an example:
     - 0.05 dB Pass Band, 40 dB Stop Band.
     - Passband from 1 GHz to 1.5 GHz (41% BW)
   - Only 4 (N-1) inductors needed to synthesize the LC Network.

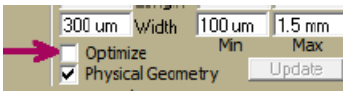


Thu Sep 16 10:05 2021

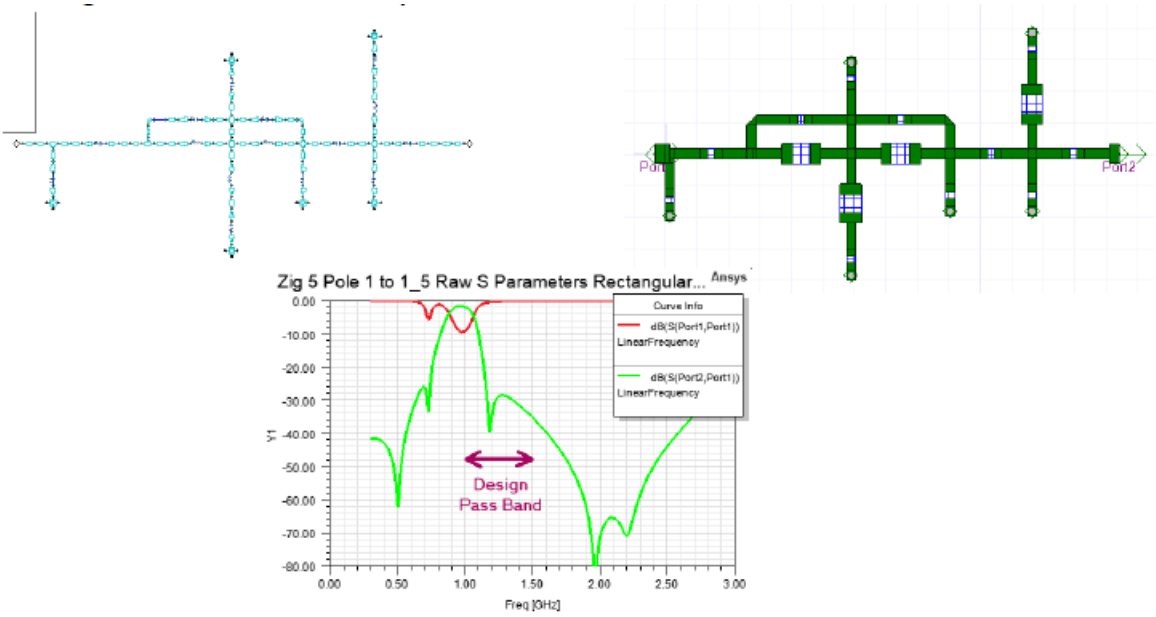
2. **Export to AEDT Circuit**

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Initial export with optimization for discrete elements only.
- Geometry **Optimize** left unchecked



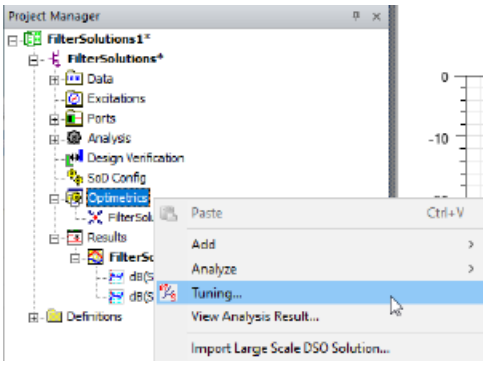
- Significant error due to parasitic effect is observed.



3. **Optional: Tuning Process: Manual Tune with AEDT Tuner**

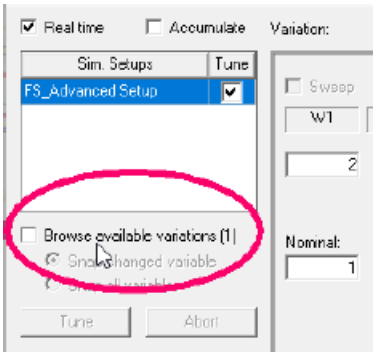
   Apply manual Circuit tuning to bring the actual simulation response closer to the desired simulation response.

   i. Right-click **Optimetrics**, select **Tuning...**

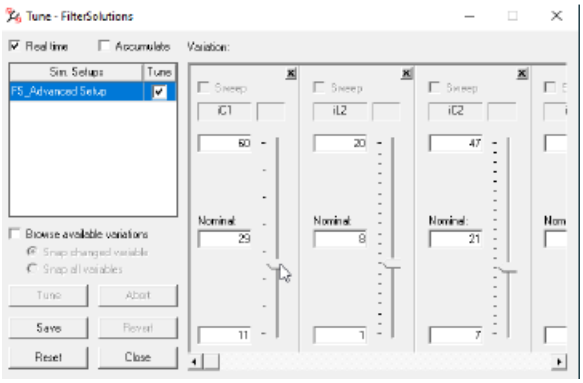


   ii. IMPORTANT: Deselect **Browse available versions** in the Tune Panel.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



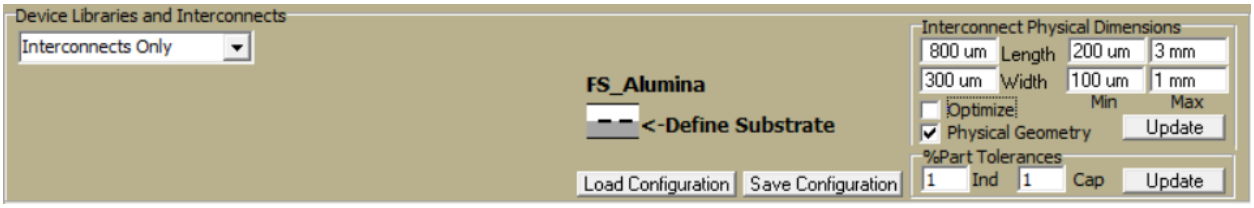
iii. Slide Tuning bars as needed to obtain the desired simulation response. Usually, tuning bars need to slide down to obtain smaller part values that compensate for the interconnect parasitic effects.



iv. Select **Save** and **Close** when optional Circuit tuning is complete to save the tuned variables and close the tuner.

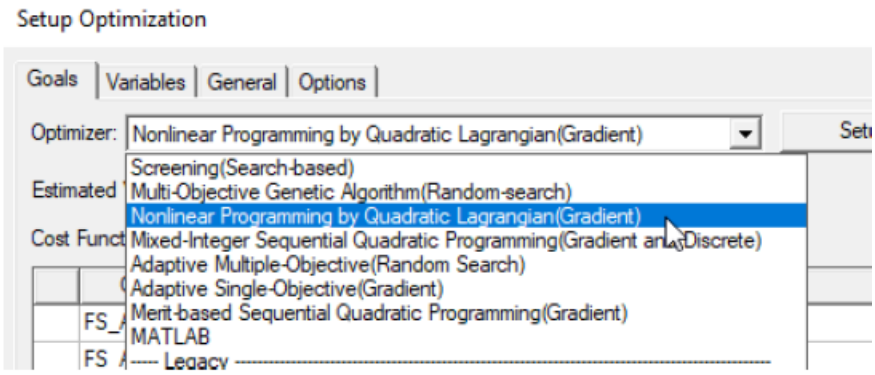
4. **Optional: Preoptimization Process: Optimize Element Values using Nonlinear Programming Quadratic Lagrangian or Other Optimizer**

i. Select **Interconnects Only** in the **Export Panel** and uncheck **Optimize** in the interconnects box to optimize only the continuous element values. Export to AEDT.

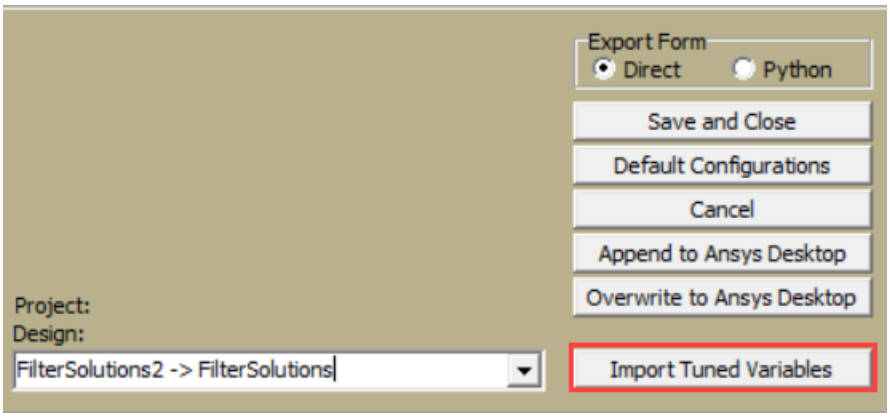


ii. Select **Nonlinear Programming by Quadratic Lagrangian** Optimizer, or another optimizer if desired.

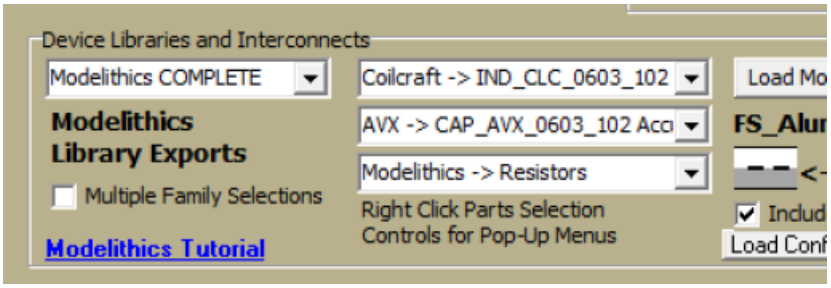
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



iii. Optimize and import optimized variables back into FilterSolutions.



iv. Select **Modelithics COMPLETE** and desired parts families, export to AEDT to begin the Automated Discrete Optimization process.

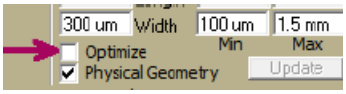


5. **Optimize Discrete Elements**

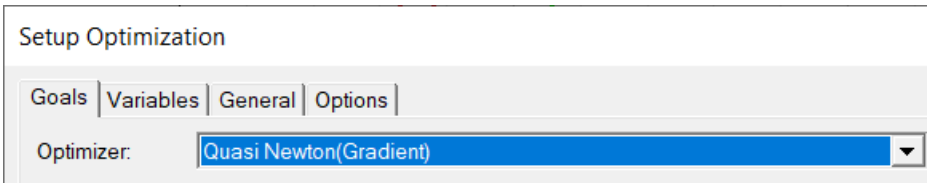
Initial optimization generally with discrete elements only- discrete optimization is realizable with the Mixed-Integer Sequential Quadratic optimizer.

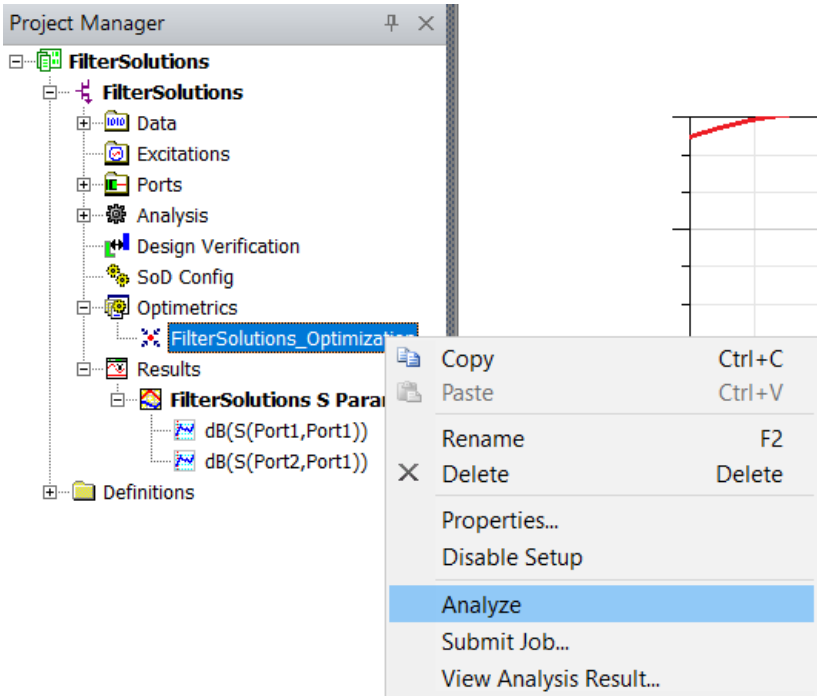
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

i. Geometry Optimized remains unchecked.



ii. Select **Quasi Newton** optimization algorithm and then **Analyze** in the right-click menu.





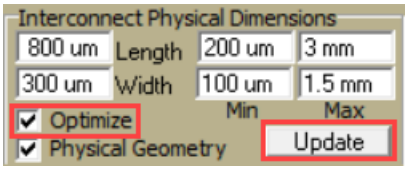
iii. Optional: discrete variables may optionally be imported into FilterSolutions with the use of the FilterSolutions **Export Panel** window, **Import Tuned Variables** button.

6. **Optimize Interconnect**

Next optimization generally with discrete elements and interconnect.

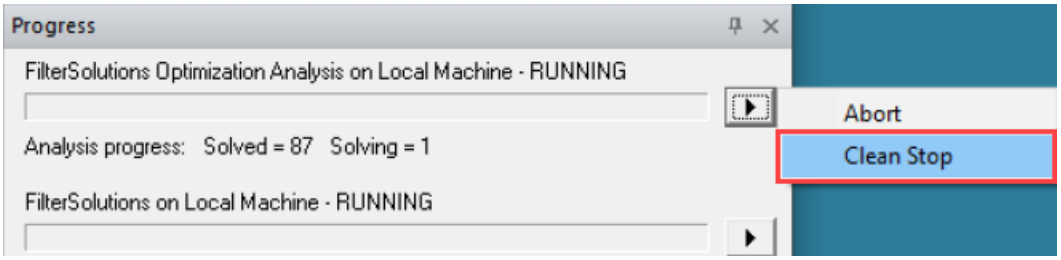
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

i. Check **Optimize** and select **Update**.

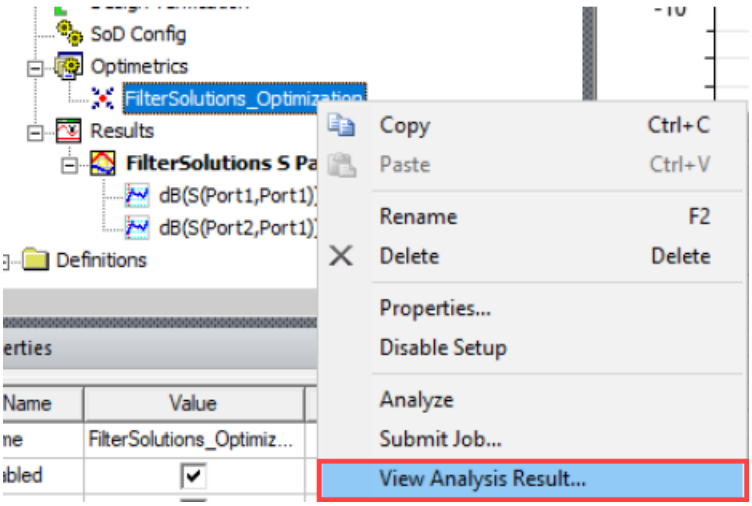


7. **To stop optimization process and retrieve partially optimized variables**

i. Select **Clean Stop** or **Abort** in the **Simulation Progress** window to stop the optimization process and retain the partially optimized variables.



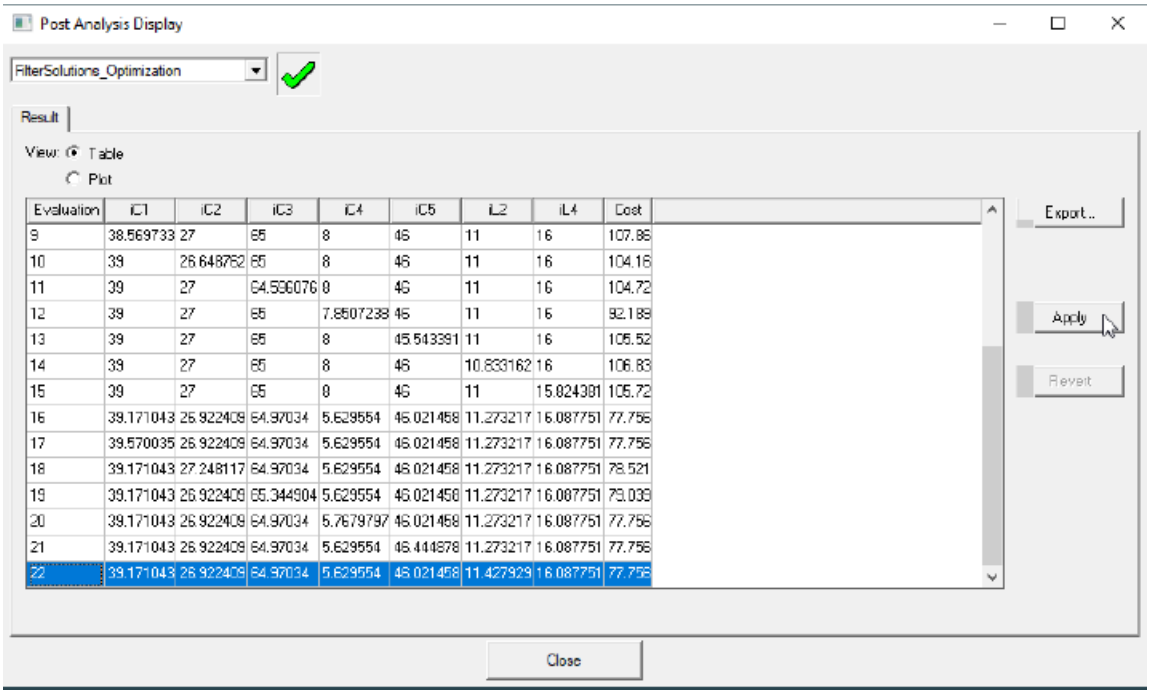
ii. Select **View Analysis Results** in the **Optimetric** task menu.



iii. In the Post Analysis Display, select Table, the evaluation row desired (usually the last),

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and Apply.



Evaluation	iC1	iC2	iC3	iC4	iC5	iL2	iL4	Cost
9	38.569733	27	65	8	46	11	16	107.86
10	39	26.648762	65	8	46	11	16	104.16
11	39	27	64.536076	8	46	11	16	104.72
12	39	27	65	7.8507238	46	11	16	92.188
13	39	27	65	8	45.543391	11	16	105.52
14	39	27	65	8	46	10.833162	16	106.83
15	39	27	65	8	46	11	15.024381	105.72
16	39.171043	26.922409	64.97034	5.629554	46.021458	11.273217	16.087751	77.756
17	39.570035	26.922409	64.97034	5.629554	46.021458	11.273217	16.087751	77.756
18	39.171043	27.248117	64.97034	5.629554	46.021458	11.273217	16.087751	78.521
19	39.171043	26.922409	65.344904	5.629554	46.021458	11.273217	16.087751	79.033
20	39.171043	26.922409	64.97034	5.7679797	46.021458	11.273217	16.087751	77.756
21	39.171043	26.922409	64.97034	5.629554	46.444878	11.273217	16.087751	77.756
22	39.171043	26.922409	64.97034	5.629554	46.021458	11.427929	16.087751	77.756

Automated Discrete Optimization

This chapter contains the following topic:

- Automated Discrete Optimization

Automated Discrete Optimization

Automated Discrete Optimization provides an easy way to complete high frequency lumped element designs that have available to them a complete set of discrete manufacturer element values. The Automated Discrete Optimization process installs the discrete element values from the selected parts families provided by the manufacturer. Specific Array entries are selected by an indexing variable that may be tuned and/or optimized. Although the indexing variable is technically a continuous variable, an internal Nearest Integer function insures the computed value from the array always matches a discrete array entry. Minimum step size is set to 1 to insure that any tuning and/or optimizing processes results in a discrete process.

Automated Discrete Optimization completes the Ansys Circuit schematic and layout with all design variable equations and interconnect geometry equations that are needed to keep the interconnect layout valid in the presence of varying part size and interconnect geometry. Generally, part values are discrete optimized first from the array variables. If more optimization

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

is desired, the interconnect geometry may be included in the tuning/optimization process and the design may be further optimized by varying the layout geometry.

Modelithics

Modelithics supports discrete parts selection, and makes manufacturer part values available that can be used to perform a discrete optimization. Modelithics simulations are also extremely accurate and account for pad geometry and substrate parasitic effects, making Modelithics ideal for use in an accurate discrete optimization process. For more information, see the section on Modelithics in AEDT.

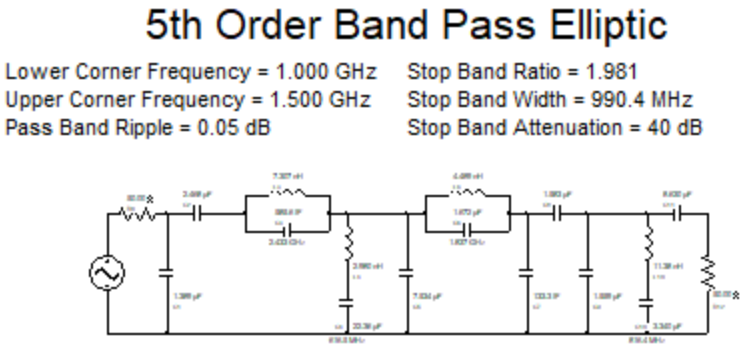
Steps for Automated Discrete Optimization:

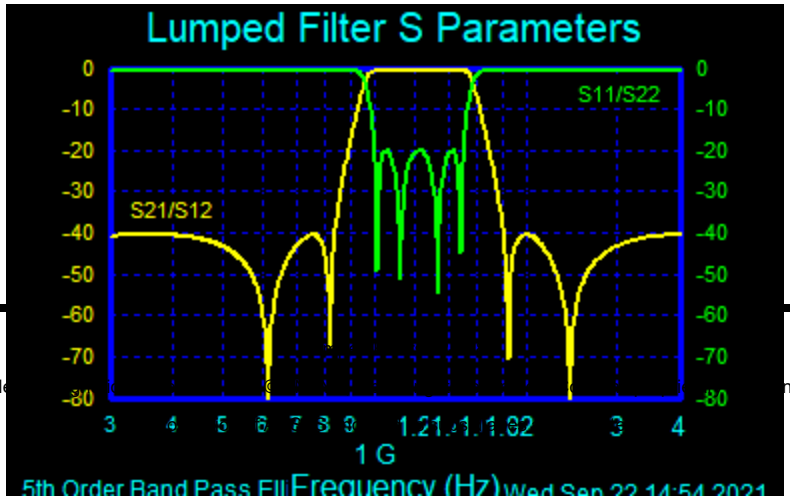
5 pole band pass, 1GHz to 1.5GHz band pass, .05dB pass band ripple, 40 dB stop band ripple

Ideal Design Synthesis

Performed with Ansys Nuhertz FilterSolutions

Generally computed from equations and represents a good stating point





nfidential

Figure 1: 1 GHz to 1.5 GHz Ideal Minimum Inductor Zigzag Design

Optional Preoptimizing With Nonlinear Programming Quadratic Lagrangian or Other Optimizer

Mixed-Integer Sequential Quadratic optimizer is used for discrete optimization due to its ability to optimize both integer strings and continuous variables. However, other optimizers are available that may be more suited for continuous optimization of the specific project that may be used to preoptimize prior to any attempt at discrete optimization. Doing so may (but is not assured to) improve discrete optimization speed and quality following the continuous preoptimizaiton.

To preoptimize using a continuous optimizer, export with **Interconnects Only** in the export panel and uncheck **Optimize** in the Interconnect box, as shown in Figure 2. This will set up the Circuits schematic in AEDT to do a continuous optimization.

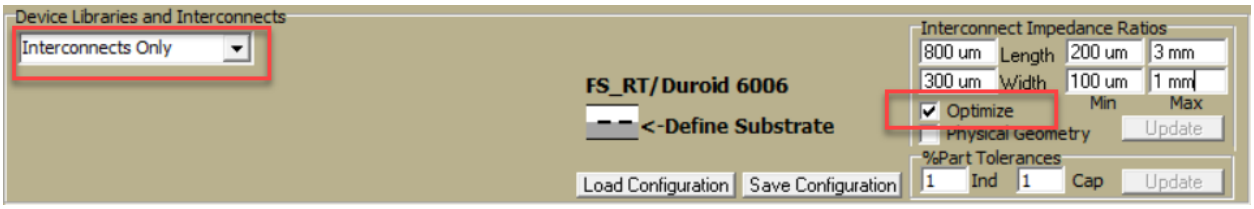


Figure 2: Exporting lumped design with the intent of preoptimizing with the Nonlinear Programming Quadratic Lagrangian or Other Optimizer

In the AEDT project optimizer selection, select the Nonlinear Programming Quadratic Lagrangian or another optimizer desired to use, and optimize the Circuits elements using this optimizer. When complete, import the optimized variables back into FilterSolutions using the normal tuned variable importation process, which is to select your Circuits project in the FilterSolutions export panel, and select **Import Tuned Variables** as shown in Figure 3.

Figure 3: Importing preoptimized variables back into FilterSolutions

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Once the preoptimied variables have been imported, proceed as normal would with the next step.

Raw Modelithics Export to Ansys Circuits

Performed with the FilterSolutions Ansys Export Panel

Includes parasitic effects of finite Q and other internal resistance, internal reactance, pads, and substrate.

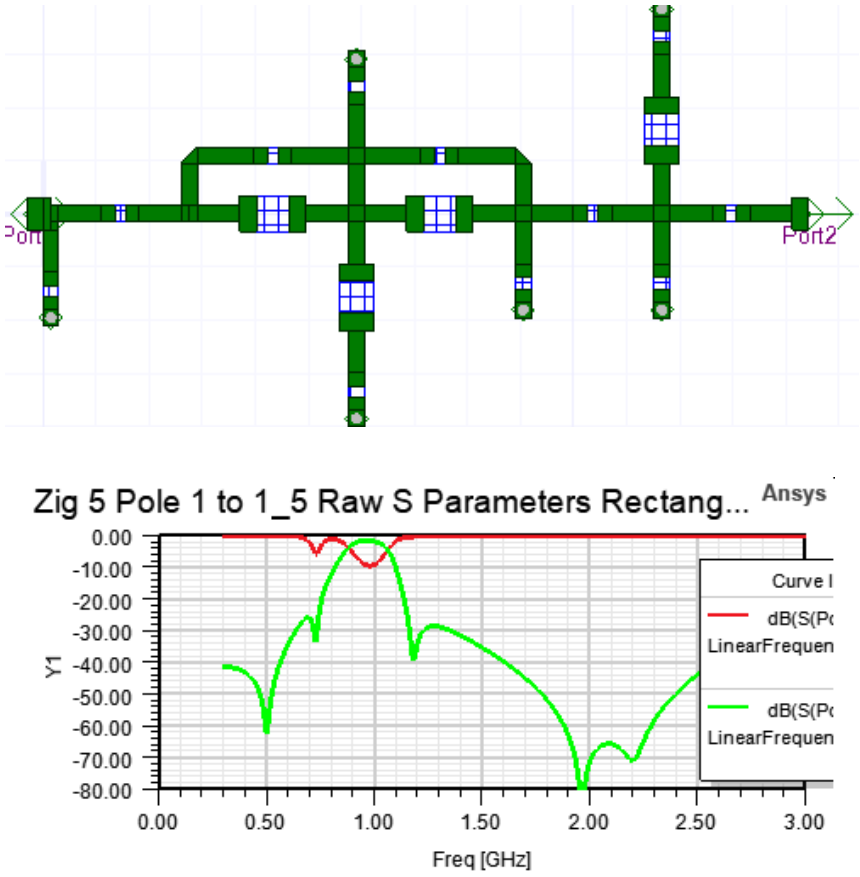


Figure 4: 1 HGz to 1.5 GHz Modelithics With Interconnects Export to Circuits

Optional Manual Tuning

The AEDT Tuning function may optionally be used to tune exported filter designs prior to optimizing. Doing so may ease the optimization process by manually placing the tuned simulation results closer to the desired result so that the optimizer can start with a smaller error.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

It is important to uncheck **Browse Available Versions** prior to tuning. When tuning is complete, select **Save** and **Close** to save the tuning variables for optimization.

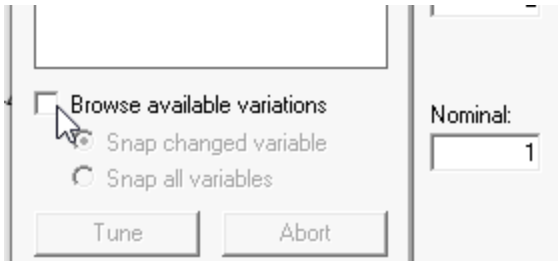
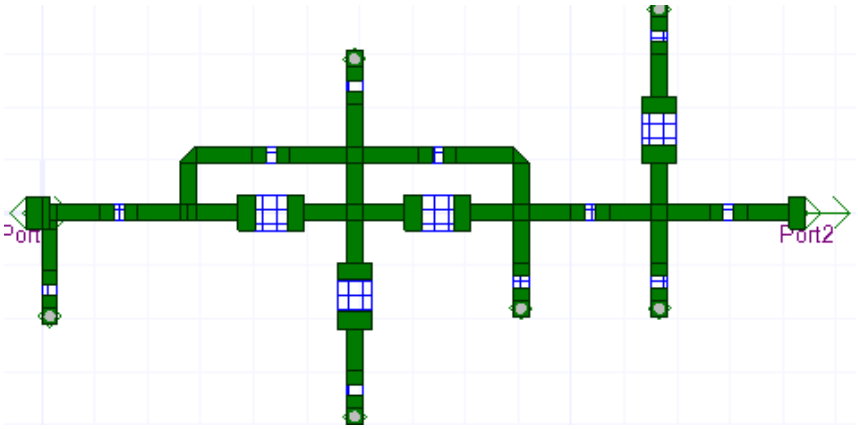


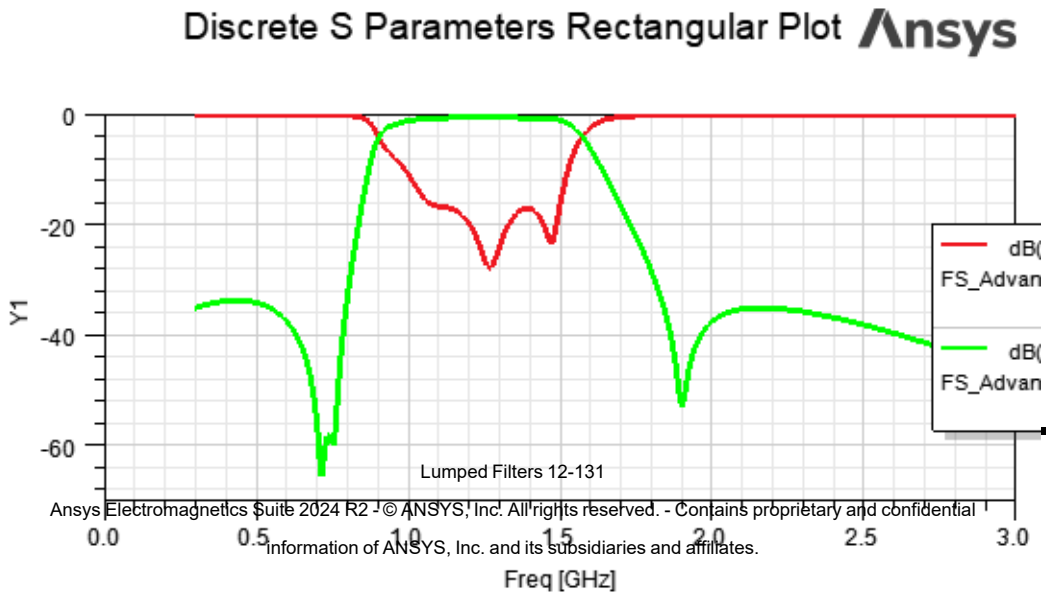
Figure 5: Uncheck Prior to Tuning

Discrete Optimize with Modelithics Parts

Performed by Ansys Electronic Desktop optimizer using the default Mixed-Integer Sequential Quadratic optimizer.

Error is minimized by optimizing for the best manufacturer parts from the selected families.





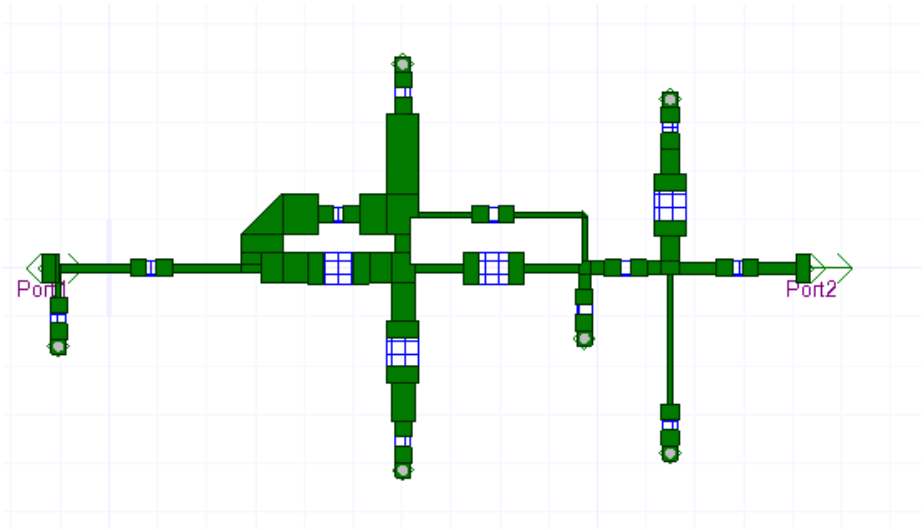
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

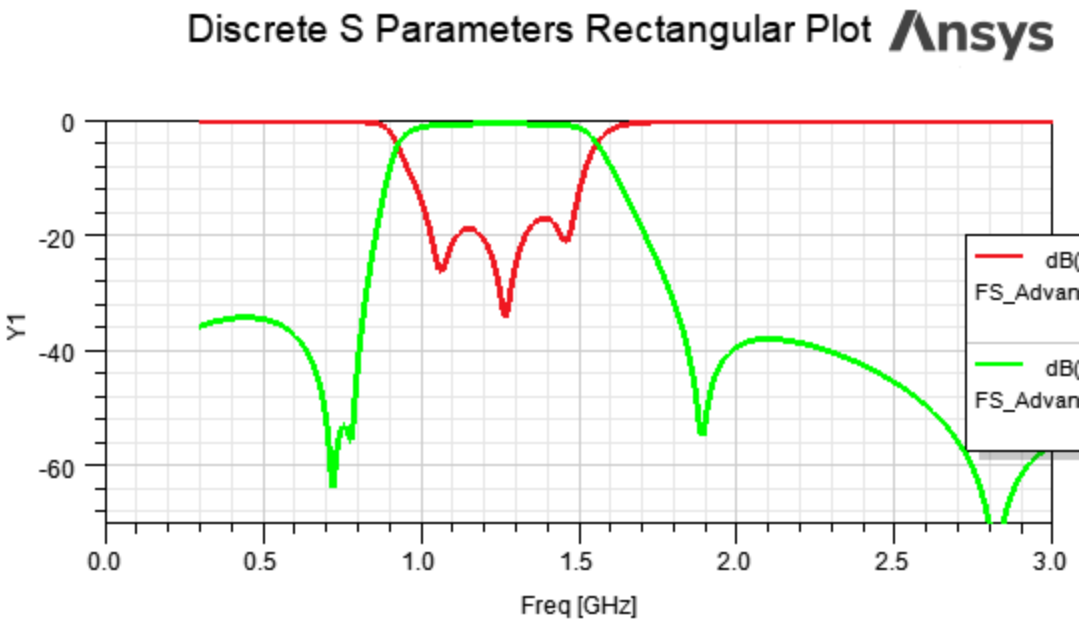
Figure 6: 1 GHz to 1.5 GHz Discrete Optimized for Element Values in Circuits

Full parts and Interconnect Optimization

Performed by Ansys Electronic Desktop optimizer

Select **Optimize** and **Update** in the Interconnect box. Error is further minimized by optimizing the interconnect geometry





Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7: 1 GHz to 1.5 GHz Discrete Fully Optimized for Discrete Element Values and Interconnect Geometry in Circuits

Retrieving Partially Optimized Variables

If it is desired to terminate the optimization process prior to completion and use partially optimized variables, select **Select**> **Clean Stop** or **Abort** in the optimization progress bar, and then select **View Analysis Results** in the Optometric Task Menu. In the Post Analysis Display, select **Table**, the evaluation row desired (usually the last), and **Apply**.

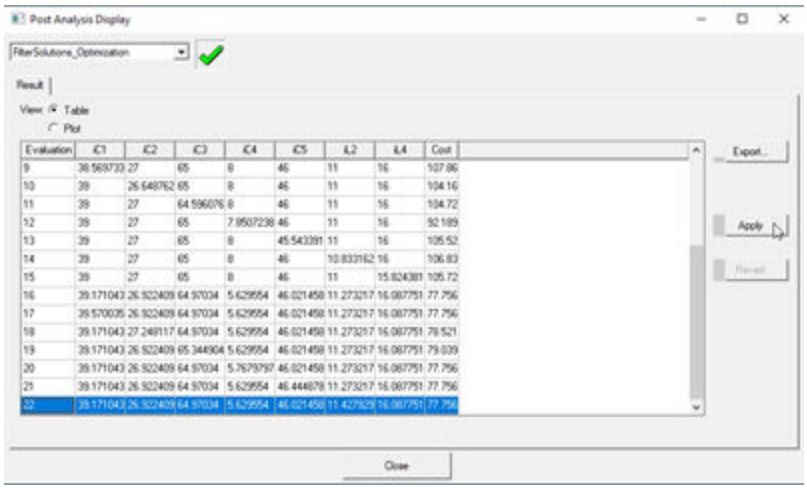


Figure 8: Table of Partially Optimized Variables to Select From

Exporting to Other Software

This chapter contains the following topics:

- Modelithics in AEDT
- 3D Data
- DXF AutoCAD
- Touchstone Files
- Sonnet
- Modelithics in Sonnet

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modelithics in FS and EDT

Lumped element Filters exported to Ansys Circuits have the option of being simulated with highly accurate Modelithics models. The filter layout, including device pad geometries, can then be generated in Circuits. FilterSolutions generated Modelithics/Circuits Schematics may be optimized by the part value and/or layout interconnect widths and lengths. The user may wish to customize the layout geometries to suit his or her specific application. Optimized or tuned Modelithics part values are confined to discrete manufacturer value through the use of Design Array variables loaded with manufacturer values supplied by Modelithics.

Loading Modelithics into Circuits

Modelithics media and licensing must be obtained from Modelithics and installed into the specific version of Electronic Desktop that it is being used on. After Modelithics is installed, it must be loaded into FilterSolutions with the use of the **Load Modelithics Models** button shown in Figure 1. The loading operation only needs to be performed once following the installation of Modelithics. FilterSolutions will store the results internally for later use.

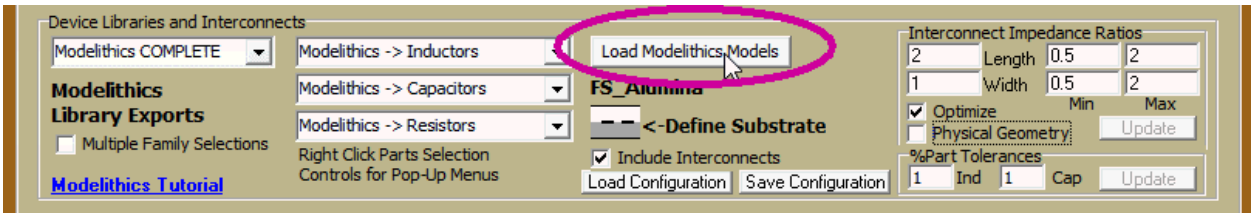


Figure 1: Loading Modelithics models into Ansys Nuhertz FilterSolutions

Selecting Modelithics Models

Modelithics inductors, capacitors, and resistors are easily selectable with the use of the right mouse click Pop up Menus, as shown below in Figure 2. Each selection shows the range of the model family for easy visual reference. Family selection may also be made directly with the use of the drop down selectors, if desired.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

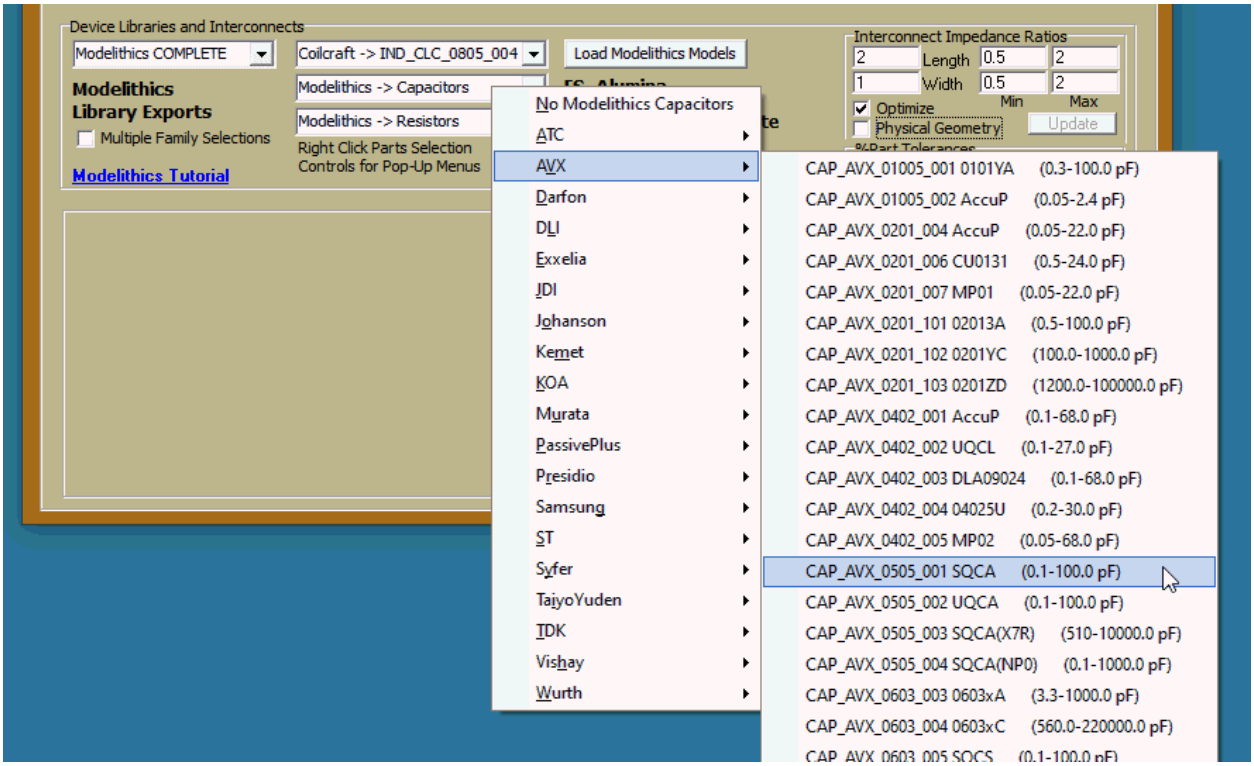
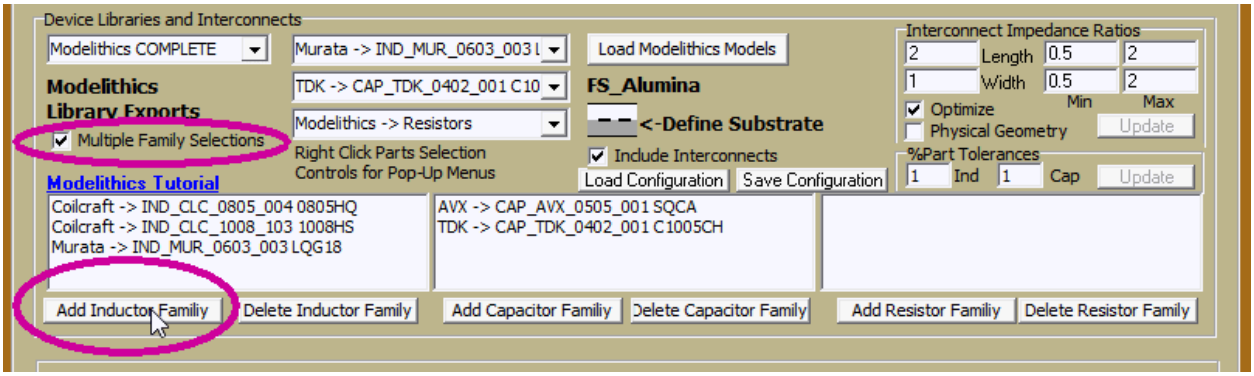


Figure 2: Selecting a Modelithics Manufacturer and Family

Selecting Multiple Families

If more than one family of parts is desired **Multiple Family Selections** may be selected, and individual families may then be selected and entered in to the multiple family tables as shown in Figure 3 either through the **Add**... button or through a drag and Drop operation. At export time, the part with the closest value to the synthesized value from all families entered will be chosen for export.

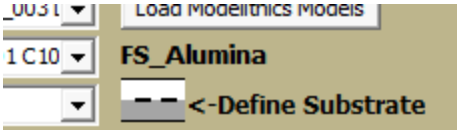


Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

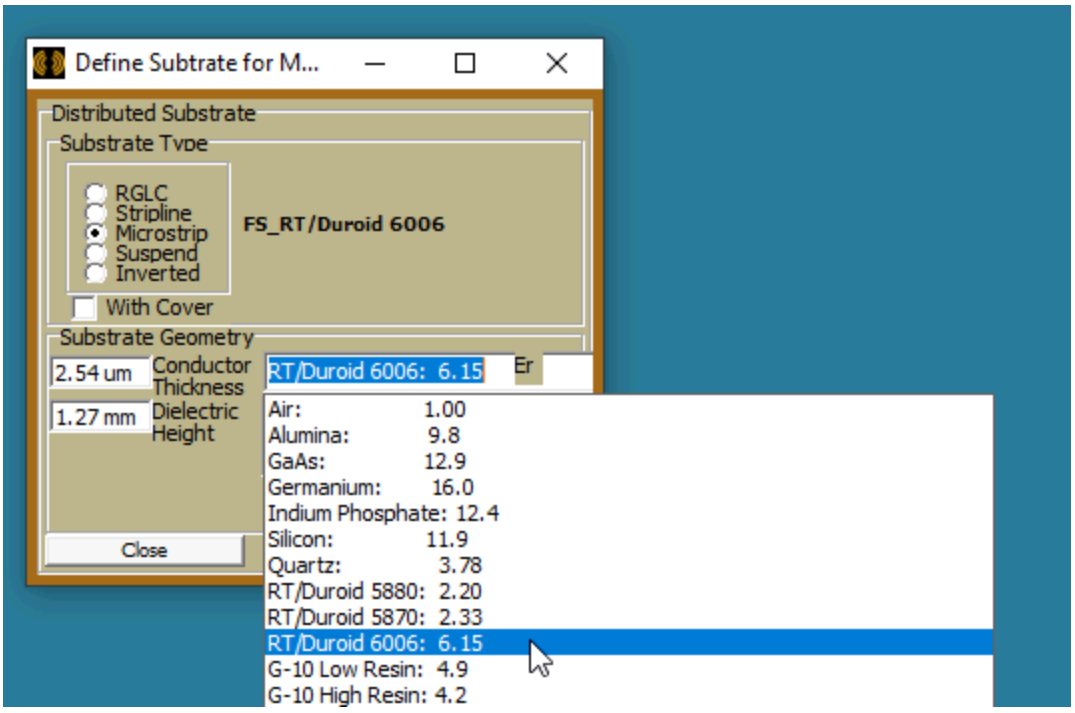
Figure 3: Selecting Multiple Part Families

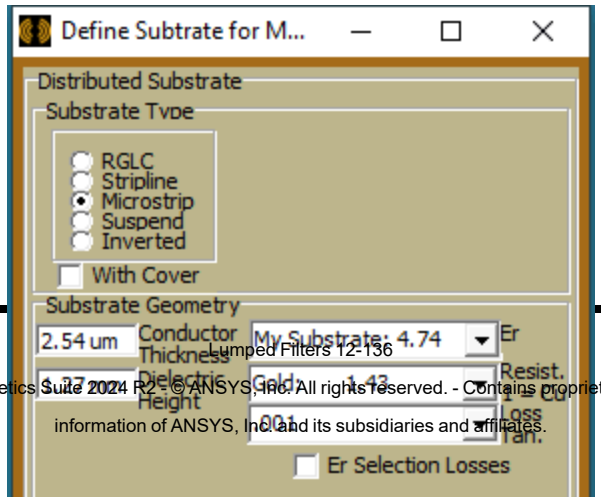
Defining the Substrate

Select **Define Substrate** in the export control panel to define the substrate in the Modelithics schematic to be exported to Circuits.



The substrate definition page shown in Figure 4 below will appear. Any built-in substrate material may be selected, or a custom name and Er value may be entered by typing it in, as shown in Figure 4.





Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Built-In Material Selection Hand Entered Material Selection

Figure4: Defining the Modelithics Substrate

Circuits Schematics and Layouts

Modelithics elements in Circuits schematics support layout cells with ports to attach to micro strip interconnect elements.

Interconnect Length Ratio and **Impedance Width Ratio** are used to set initial length and width geometry of the interconnects. Geometry may be defined in terms of substrate height ratios or physical dimensions, as desired, shown in Figure 5. Circuits/Modelithics Schematics are designed to produce valid layouts, as shown with this five pole Zigzag filter below in Figure 6, where 4 inductors and 11 capacitors may easily be seen to be included in the layout. Initial layout geometry is set to be optimized with the use of Circuits "S" and "W" equations. All Circuits equations are designed to insure a valid layout when reasonable geometry setting are entered.

The first export generally with Optimize unchecked so that only the discrete element values are optimized. Layout geometry Circuits optimization may be turned on and off remotely from the FS export panel by checking or unchecking Optimize and then selecting the **Update** button to the immediate right of Optimize.

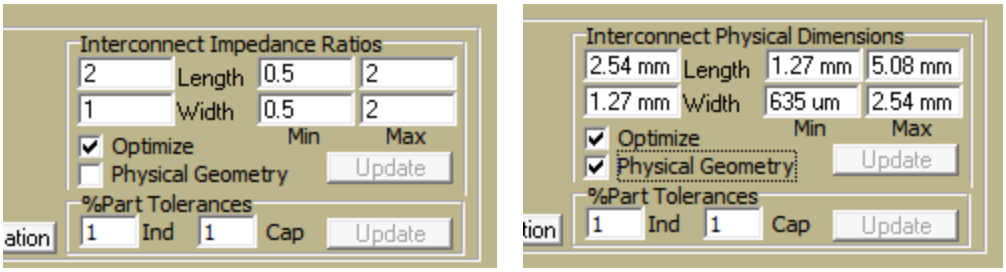


Figure 5: Interconnect geometry Defined in Terms of Substrate Height Ratios or Physical Dimensions

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

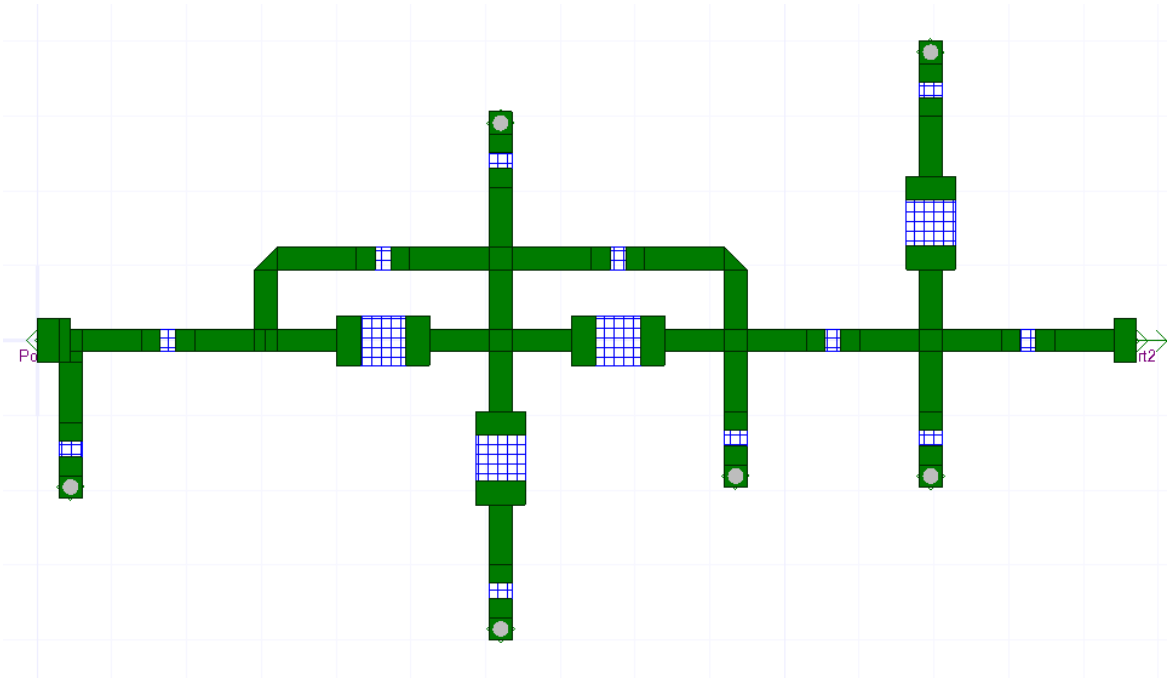


Figure 6: Valid Circuits Schematic Layout of a Modelithics Parts Filter Design

Tuned/Optimized Layouts

Modelithics layouts are designed to remain valid for reasonable width and length adjustment. All equations are embedded into the design variables and layout geometry that are needed to keep the layout valid in the presence of interconnect width and length adjustments. Figure 7 below shows the circuit layout from Figure 6 above after the interconnects have been adjusted through tuning and/or optimizing.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

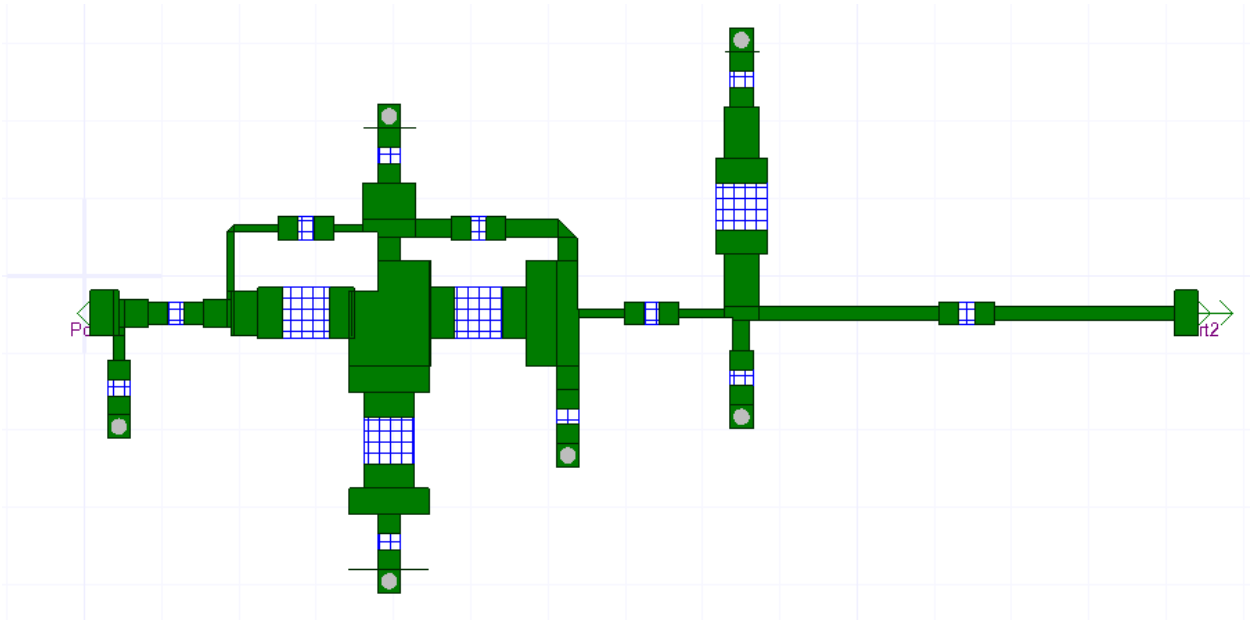


Figure 7: Valid Circuits Schematic Layout With Tuned/Optimized Interconnects

Touchstone Files

You may easily export designed filters to Y, Z, or S parameter Touchstone files that me be imported into other circuits analysis programs. Each filter schematic contains an **Export** function in the upper left that, when selected, offers Touchstone Z, Y, and S parameter file exports.

Circuits analysis programs usually include an element called "N-Port", "Ideal element", etc. that have the capability to read or import Touchstone files that contain Z, Y or S parameter data, and effectively model the designed filter when they read a FilterSolutions generated Touchstone file.

Port Definitions:

2 port filters:

Port 1 = Input.

Port 2 = output

Diplexers:

Port 1 = Input

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Port 2 = Low pass output

Port 3 = High pass output

Triplexers:

Port 1 = Input

Port 2 = Low pass output

Port 3 = High pass output

Port 4 = Band pass output

Tips:

Z and Y parameters model only the filter network, and do not include any source or load terminations.

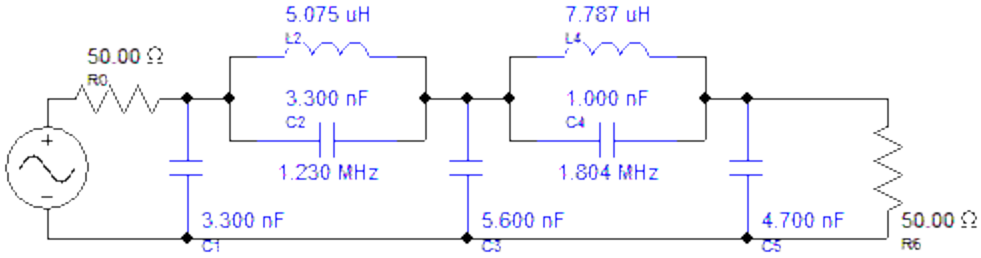
S parameters Touchstone exports are available only for equally terminated filters. This is due to restrictions in standard Touchstone file data format requirements.

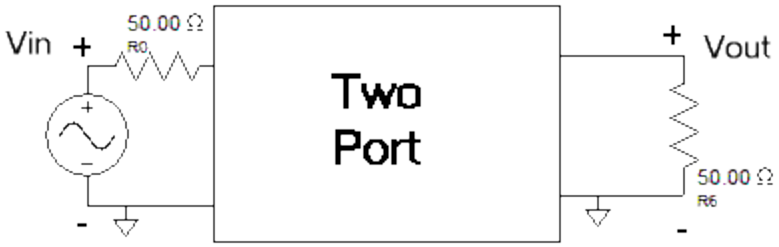
Beginning and ending frequencies are determined by the FilterSolutions Min Freq and Max Freq entries.

If the Touchstone file frequency response contains noise or spikes, add a touch of loss to the reactive elements to smooth it out.

Illustrations:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





LumpedTouchstone Model

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13 - Distributed Filters

This chapter contains the following topics:

- Transmission Line Concepts
- Coupled Resonator and MultiConductor Filters
- Working With Distributed Filters
- Exporting to Ansys Electronic Desktop
- Exporting to HFSS 3D Layout
- Exporting to HFSS
- Exporting to Ansys Planar Circuits

Transmission Line Concepts

This chapter contains the following topics:

- Transmission Line Equations
- SRLGC, Micro Strip, and Strip Lines
- Substrate Geometries
- Inductor Approximations
- Capacitor Approximations
- Radial and Delta Stubs
- Parallel Resonator Approximation
- Coupled Lines
- Series Resonator Approximation
- Stepped Impedance Filters
- Spaced Stub Filters
- Open and Short Terminations
- Transmission Line Prewarping
- Signal Propagation
- Lumped Filter Approximation
- Lumped Element Models

Transmission Line Equations

The physics behind the derivation of transmission line filters is described with a well known set of transmission line equations, two of which are prominent. One prominent equation is useful in calculating the input impedance of a line. The other is useful in describing a voltage signal as it propagates down the line.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The input impedance, Zin, of a line may be described according to the following equations:

$$Zin = Zo\frac{Z1 + Zo * \tanh(M * len)}{Zo + Z1 * \tanh(M * len)}$$

Where:

$$M = \sqrt{(R + J\omega L)(G + J\omega C)}$$

$$len = \text{Transmission Line Length}$$

$$Zo = \sqrt{\frac{R + J\omega L}{G + J\omega C}}$$

$$Z1 = \text{Load Impedance}$$

$$L = \text{Inductance per unit length}$$

$$C = \text{Capacitance per unit length}$$

$$R = \text{Resistance per unit length}$$

$$G = \text{Conductance per unit length}$$

$$J = \sqrt{-1}$$

Lossy Transmission Line Input Impedance

If a line is lossless, meaning the resistance and conductance are both zero, then the equations reduce to a simpler form as seen below

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$Zin = J * Zo * \tan(\varpi * len / V)$$

Where:

len = Transmission Line Length

$$Zo = \sqrt{\frac{L}{C}}$$

$$V = \frac{1}{\sqrt{LC}} = \text{Speed of Propagation}$$

L = Inductance per unit length

C = Capacitance per unit length

$J = \sqrt{-1}$

Lossless Transmission Line Input Impedance

Striplines and Microstrips

Losses for striplines and microstrips are generally a function of frequency, and are calculated with attenuation using conductor resistivity dielectric loss tangents, frequency, and geometry rather than by resistance and conductance.

RLGC, Microstrips, Striplines, and Suspended Substrates

FilterSolutions supports the implementation of distributed filters in RLGC, microstrip, and stripline forms. Microstrips may be suspended above a substrate and may optionally have a cover. Striplines may be balanced or unbalanced.

In the Distributed control panel, select **Real Parameters** and then the physical type. If microstrip, stripline, or suspended is selected, enter the conductor thickness and dielectric height in the spaces provided. For microstrips, enter the cover height, and for suspended enter the suspended height above the substrate. FilterSolutions will calculate the conductor width for each stub and segment in the filter and display this information in the transmission line schematic in addition to all other information already displayed. Segment lengths are adjusted in microstrip filters to account for the dielectric of air above the line.

Microstrip and stripline filters do not implement series stubs. Consequently, **Use Segments** is automatically selected and locked, and balanced filters are not available.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Microstrip and Stripline filters do not support all pass functions. Consequently, **delay equalization, delay filters, and analog raised cosine filters are disabled.**

Microstrip and stripline widths are limited. Widths computed to be less than the half the thickness or greater than 1.e3 heights are displayed as "Invalid"

Coupled Lines

Microstrip and stripline coupled lines are implemented at a width and a gap, both of which are displayed on the schematic. Each of the two individual lines are of the same width. If the width and gap are excessively wide or narrow, or are calculated to be negative or complex, then "Invalid" is displayed on the schematic, and the microstrip or stripline filter is not physically realizable. Realizable width Invalid definitions are the same as for uncoupled lines. Gaps may be as wide as 1.e3 Heights or as narrow as 1.e-4 thicknesses.

Losses

Transmission lines losses are accounted for in analysis simulations. RLGC lines consider the resistance and conductance of each segment and stub when performing simulations. Stripline and microstrips consider the resistivity of the conductor and loss tangent of the dielectric. Line attenuations are calculated as a function of frequency for both the conductor and the dielectric. All resistance, resistivity, and loss tangents are zero for ideal transmission line filter simulations. Resistivity is entered with respect to copper, i.e. enter a "1" for a copper conductor, "0.95" for silver, "1.42" for Gold, "1.64" for Aluminum, etc.

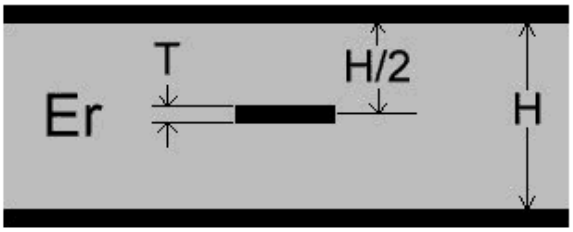
Attenuation

Attenuation (Ao) at the filter center or cutoff frequency are displayed in dB/length on stripline and microstrip stubs and segments.

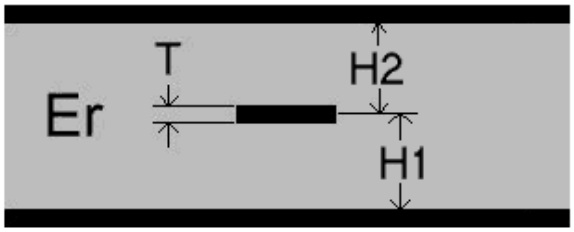
Substrate Geometry Definitions

Definitions for stripline and microstrip substrate geometries are shown below:

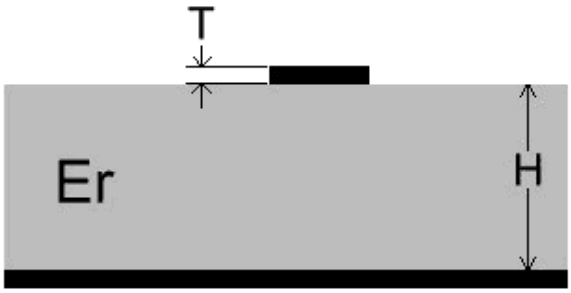
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



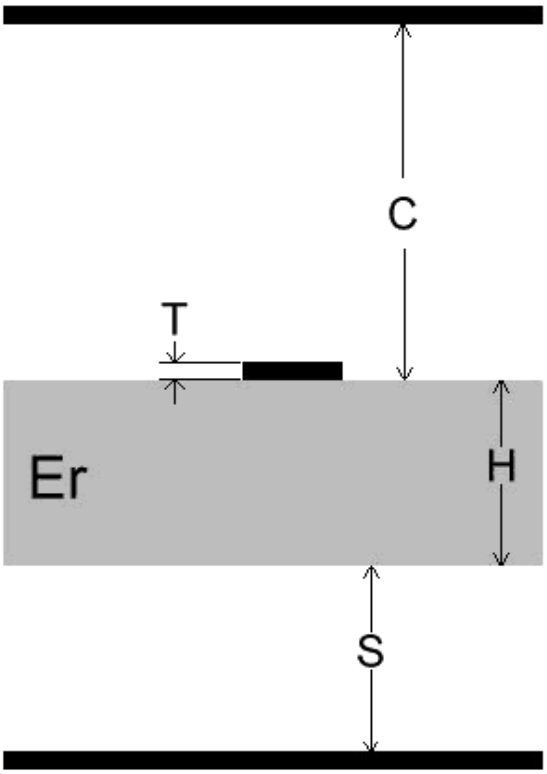
Stripline



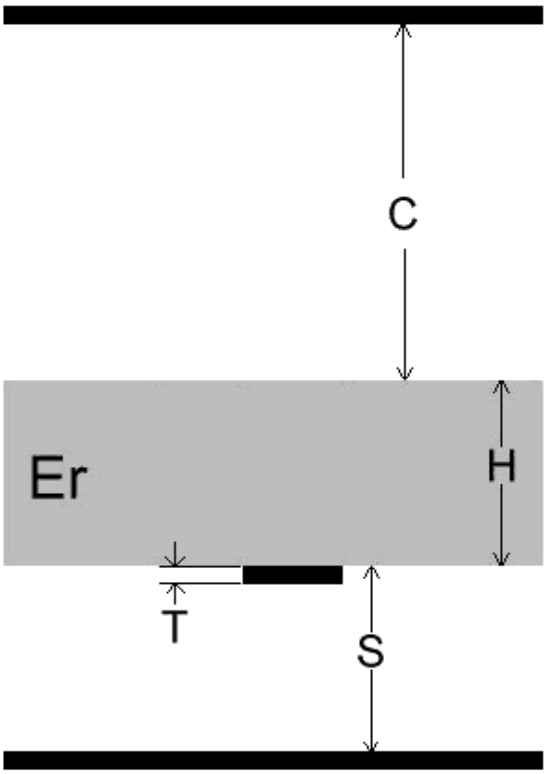
Unbalanced Stripline



Microstrip



Microstrip With
uspended Subtrate and Cover



Inverted Microstrip With
Suspended Subtrate and Cover

Substrate Layer With
Overlay Capacitors and
Interlaced Resonators

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential
information of ANSYS, Inc. and its subsidiaries and affiliates.

Variable Definitions:

T	Conductor Thickness
H	Dielectric Height
S	Substrate Elevation over Ground Plane
C	Cover Height above Substrate

Inductor Approximations

Transmission lines with shorted loads and transmission line segments may be used to approximate inductors. To approximate inductors, a null frequency may be selected such that the impedance of the transmission line is zero or infinity. Frequencies above the null frequency are aliases of frequencies below the null frequency.

Shorted lossless line input impedance may be described mathematically below.

$$Zin = J * Zo * \tan(\varpi * len / V)$$

Where:

$$\text{len} = \text{Transmission Line Length}$$

$$\text{Zo} = \sqrt{\frac{\text{L}}{\text{C}}}$$

$$V = \frac{1}{\sqrt{LC}} = \text{Speed of Propagation}$$

$$\text{L} = \text{Inductance per unit length}$$

$$\text{C} = \text{Capacitance per unit length}$$

$$\text{J} = \sqrt{-1}$$

Shorted Load Lossless Input Impedance

It may be seen with this equation that the infinite impedance length of the stub is calculated by setting the length of the line such that

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\frac{\varpi_N \text{len}}{V} = \frac{\pi}{2}$$

Or

$$\text{len} = \frac{\pi V}{2\varpi_N}$$

Where ϖ_N is the desired null frequency

Stub Length Inductor Equation

At some desired frequency, w, Zo of the line may then be calculated such that

$$\text{Zin} = J\omega_T L = JZ_0 \tan\left(\frac{\omega_T \text{len}}{V}\right)$$

Or

$$Z_0 = \frac{\omega_T L}{\tan\left(\frac{\omega_T \text{len}}{V}\right)}$$

Where:
L = Inductance of the lumped element
ϖ_T = Desired frequency where inductor impedance matches transmission line impedance.

Tx Segment Zo

If it is desire to use a transmission line segment to simulate an inductor, the equation for Zo is,

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$Zo = \frac{\varpi_c L}{\sin\left(\frac{\varpi_c len}{V}\right)}$$

Where

$L = \text{Inductor Value}$

$\varpi_c = \text{Design Frequency}$

Transmission Line Segment Inductor Equations

Capacitor Approximations

Transmission lines with open loads may be used to approximate capacitors. To approximate capacitors, a null frequency may be selected such that the impedance of the transmission line is zero at that frequency. Frequencies above the null frequency are aliases of frequencies below the null frequency.

Open lossless line input impedance may be described mathematically below.

$$Zin = Zo \frac{1}{J * \tan(\varpi * len / V)}$$

Where:

$len = \text{Transmission Line Length}$

$$Zo = \sqrt{\frac{L}{C}}$$

$$V = \frac{1}{\sqrt{LC}} = \text{Speed of Propagation}$$

$L = \text{Inductance per unit length}$

$C = \text{Capacitance per unit length}$

$J = \sqrt{-1}$

Open Load Lossless Input Impedance

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

It may be seen with this equation that the zero impedance length of the stub is calculated by setting the length of the line such that

$$\frac{\varpi_0 \text{len}}{V} = \frac{\pi}{2}$$

Or

$$\text{len} = \frac{\pi V}{2\varpi_0}$$

Where ϖ_0 is the LC Resonant frequency

Stub Length Capacitor Equation

At some desired frequency, w, Zo of the line may then be calculated such that

$$Zin = \frac{1}{J\omega_T C} = \frac{Z_0}{J\tan\left(\frac{\omega_T \text{len}}{V}\right)}$$

Or

$$Z_0 = \frac{\tan\left(\frac{\omega_T \text{len}}{V}\right)}{\omega_T C}$$

Where:
C = Capacitance of the lumped element
ϖ_T = Desired frequency where capacitor impedance matches transmission line impedance.

Zo Capacitor Equation

Tx Segment Zo

If it is desired to use a transmission line segment to simulate a shunt capacitor, generally for use in Stepped Impedance filters, the equation for Zo is,

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$Z_0 = \frac{\varpi_C L}{\sin\left(\frac{\pi \varpi_C}{2\varpi_N}\right)}$$

Where

$L = \text{Inductor Value}$

$\varpi_C = \text{Cutoff Frequency of the Filter}$

$\varpi_N = \text{Desired Null Frequency}$

Transmission Line Segment Shunt Capacitor Equations

Radial and Delta Stubs

Open stub capacitor approximation geometry can sometimes be improved with the use of radial and delta open end stubs in place of straight open end stubs. Radial and delta stubs are generally shorter than straight stubs, and sometimes have improved performance in higher order modes. FilterSolutions radial and delta stubs normally attach at a base width of H/2 , where H is the total dielectric height, and protrude out at a specified angle. Excessively fat stubs widen the base width beyond H/2, however. For shunt capacitor approximations, the synthesis algorithms attempt to maintain the impedance at the cutoff frequency and stop band null frequency. The constant angle selection only maintains the cut off frequency impedance, and lets the stop band null frequency meander from stub to stub. LC resonant capacitors maintain impedance at the LC resonant frequency.

The impedance computations for micro strip radial stubs and all delta stubs use standard text book equations, with a few customized improvements to aid in accuracy. Strip line radial stubs are a product of internal Nuhertz research.

The angle range of a radial stubs is 15 deg to 140 deg. For delta stubs it is 15 deg to 120 deg. Fixed angle stubs may use angles less than 15 deg, but not greater than the upper angle limit.

Radial Stub Geometry

Radial stubs are defined by the angle, width, inner radius, and outer radius, as shown in Figure 1 below. Since only three dimensions are needed, one of the four must be computed as a function of the other three.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

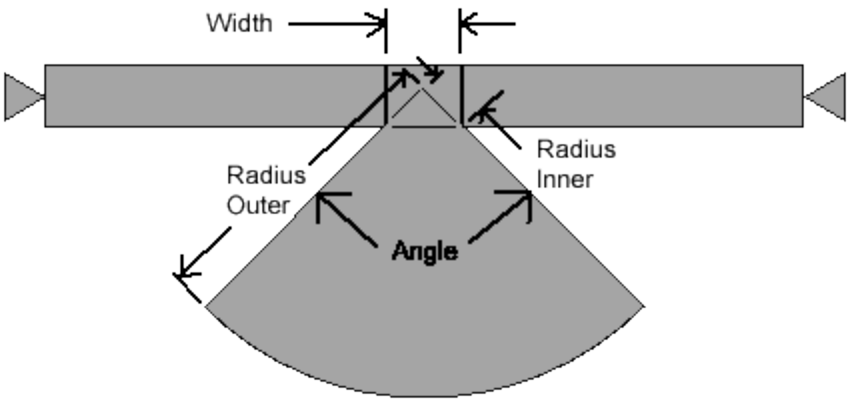


Figure 1: Radial Stub Geometry

Delta Stub Geometry

Delta stubs are defined by the angle, width, and length, as shown in Figure 2 below.

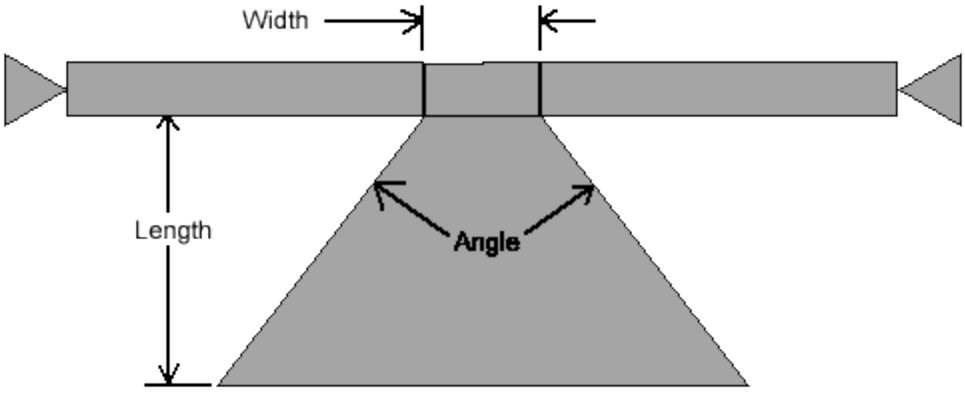


Figure 2: Delta Stub Geometry

Parallel Resonator Approximations

Transmission lines with shorted loads may be used to approximate parallel LC resonators. To approximate LC resonators, the null frequency of the transmission line must be set to match the LC resonant frequency. Aliasing occurs at twice the null frequency. Therefore, it is not practical to approximate a wide band LC resonator with a single transmission line.

Shorted lossless line input impedance may be described mathematically below.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$Zin = J * Zo * \tan(\varpi * len / V)$$

Where:

len = Transmission Line Length

$$Zo = \sqrt{\frac{L}{C}}$$

$$V = \frac{1}{\sqrt{LC}} = \text{Speed of Propagation}$$

L = Inductance per unit length

C = Capacitance per unit length

$$J = \sqrt{-1}$$

Shorted Load Lossless Input Impedance

It may be seen with this equation that the infinite impedance length of the stub is calculated by setting the length of the line such that

$$\frac{\varpi_0 len}{V} = \frac{\pi}{2}$$

Or

$$len = \frac{\pi V}{2\varpi_0}$$

Where ϖ_0 is the LC Resonant frequency

Parallel Resonator Stub Length Equation

The impedance of the line that approximates a parallel LC resonator is stated below without derivation

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$Zo = \frac{4\varpi_0 L}{\pi}$$

Where

L = Inductor Value of the LC Resonator

ϖ_0 = Resonant Frequency of the LC Resonator

Parallel Resonator Impedance Equation

Coupled Transmission Lines

Coupled transmission lines consist of a pair of lines over a ground plane. The even mode characteristic impedance, Zoe, refers to the combined impedance of both lines with respect to the ground plane. The odd mode characteristic impedance, Zoo, refers to the impedance between the two lines. FilterSolutions supports two band pass topologies of coupled lines to synthesize band bass and high pass filters, as shown in Figure 1 below, along with standard coupled line definitions and ideal equivalent tx line circuits. High pass filters are synthesized as band pass filters with a relatively narrow aliasing gap in the frequency response.

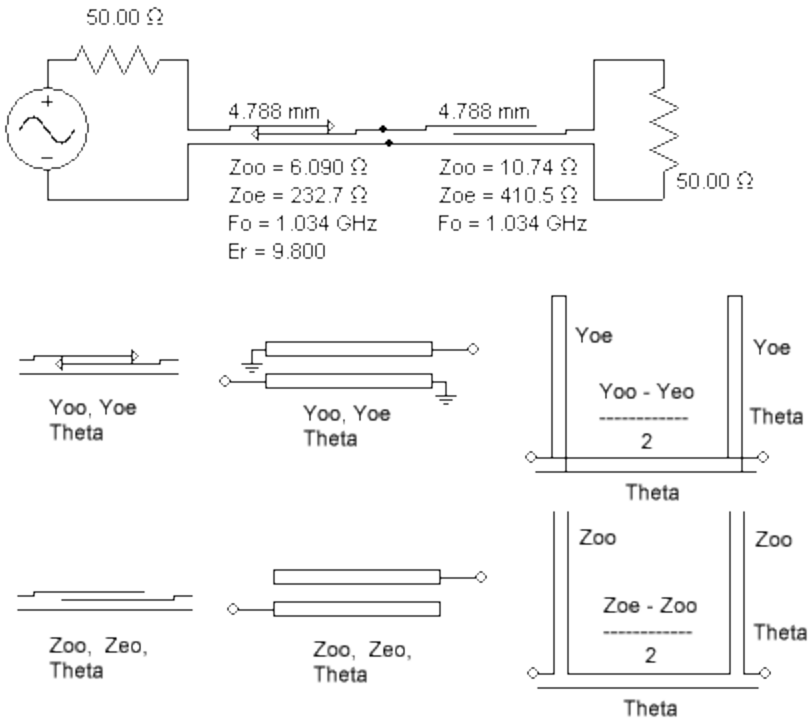


Figure 1 Coupled Lines

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To synthesize filters with coupled lines, select **Use Coupled Lines** and preferably **Combine Stubs** in the Tx Line control panel. **Use Segments** will automatically be selected. If **Combine tubs** is not selected, only the series coupled lines are synthesized.

Series Resonator Approximations

Transmission lines with open loads may be used to approximate series LC resonators. To approximate LC resonators, the null frequency of the transmission line must be set to match the LC resonant frequency. Aliasing occurs at twice the null frequency. Therefore, it is not practical to approximate a wide band LC resonator with a single transmission line.

Open lossless line input impedance may be described mathematically below.

$$Zin = J * Zo * \tan(\varpi * len / V)$$

Where:

$$\text{len} = \text{Transmission Line Length}$$

$$\text{Zo} = \sqrt{\frac{\text{L}}{\text{C}}}$$

$$V = \frac{1}{\sqrt{LC}} = \text{Speed of Propagation}$$

$$\text{L} = \text{Inductance per unit length}$$

$$\text{C} = \text{Capacitance per unit length}$$

$$\text{J} = \sqrt{-1}$$

Open Load Lossless Input Impedance

It may be seen with this equation that the zero impedance length of the stub is calculated by setting the length of the line such that

$$\frac{\varpi_0 \text{len}}{\text{V}} = \frac{\pi}{2}$$

Or

$$\text{len} = \frac{\pi \text{V}}{2\varpi_0}$$

Where ϖ_0 is the LC Resonant frequency

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parallel Resonator Stub Length Equation

The impedance of the line that approximates a parallel LC resonator is stated below without derivation

$$Zo = \frac{\pi \varpi_0 L}{4}$$

Where

L = Inductor Value of the LC Resonator

ϖ_0 = Resonant Frequency of the LC Resonator

Series Resonator Impedance Equation

Transmission line segments may be used to approximate in line series LC resonators. The equations are:

$$Z_0 = \frac{2\varpi_0 L}{\pi}$$

Where

L = Inductor Value

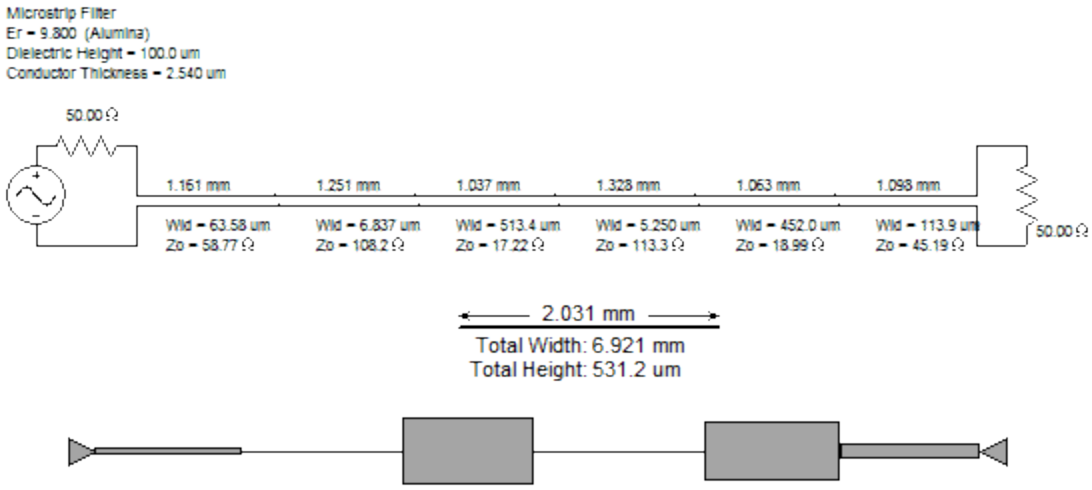
ϖ_0 = Cutoff Frequency of the Filter

Transmission line Segments Series Resonator Equations

Stepped Impedance Filters

Distributed stepped impedance filters are low pass filters that consist of alternating high and low impedance stages. The high impedance stage simulated a series inductor, and the low impedance stage simulates a shunt capacitor. The microstrip and stripline implementation appears as alternating thick and thin segments as shown below:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



10 GHz, 6th Order Stepped Impedance Filter

> **Note:** There is no requirement for equal length stages. Unequal length is usually required when the filter is optimized.

To create a stepped impedance filter, select **Stepped Impedance** in the tx-line control panel when low pass filters are selected.

Spaced Stub Filters

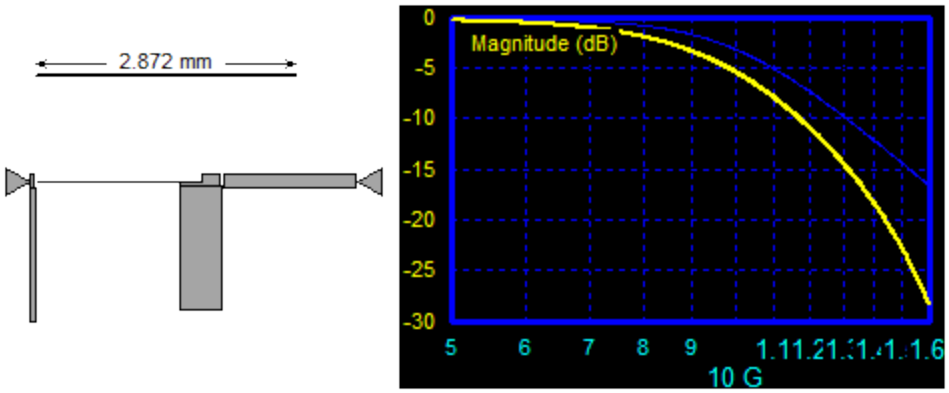
Low and high pass filters may be composed of quarter length stubs with half wave segments spaced in between. Low pass filters use open stubs, and high pass filters use grounded stubs. Such a filter design begins frequency aliasing at twice the cutoff frequency, so no user selectable alias frequency is available to the design control panel. Low pass filters tend to produce spurious modes at twice the cutoff frequency, but these undesired responses may be minimized by adjusting the spacing between the stubs. A highly accurate EM simulator, such as HFSS, is recommended to make a good accurate adjustment.

Micro strip and strip line spaced stub filters require more parts than their direct lumped equivalent counterparts (or approximate lumped equivalent in the case of high pass filters) in that one stub per filter order is always required, but they also tend to be more accurate about the cutoff frequency, especially for high pass filters, and are less likely to result in invalid or unrealizable geometries. Optimizing the smaller lumped equivalent designs may improve the accuracy and minimize the advantage of Spaced Stub filters. It is noteworthy that a low pass spaced stub filters may be converted to a high pass simply by grounding the stubs.

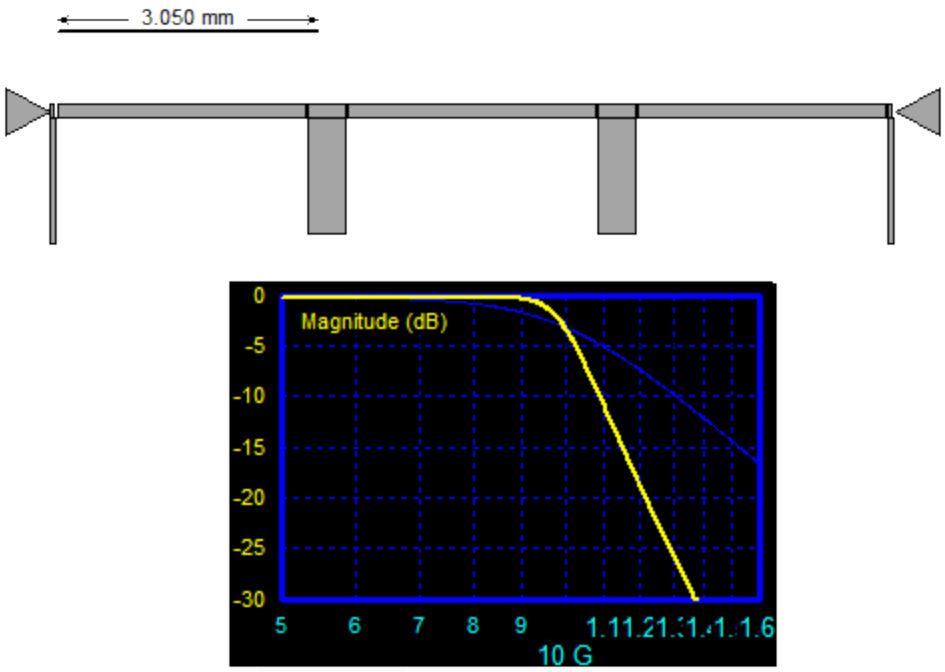
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Spaced stub filters normally produce zero error at the cutoff frequency when RLGC and strip lines are used. Micro strip spaced stub filters will produce a small error due to the differing transmission speed among the elements, however, micro strip cutoff error will generally be less for the spaced stubs than the lumped equivalent.

Examples:

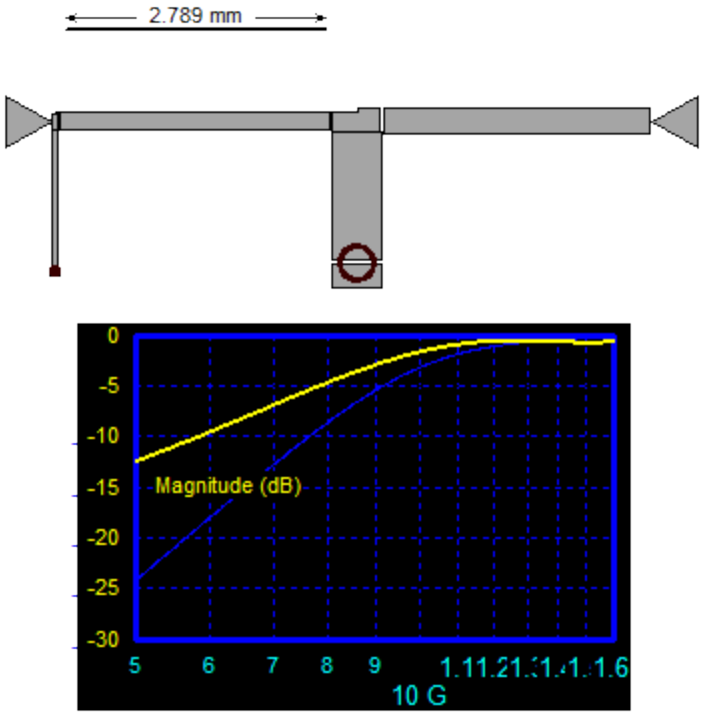


10GHz strip line low pass lumped equivalent filter with slight error at the cutoff frequency



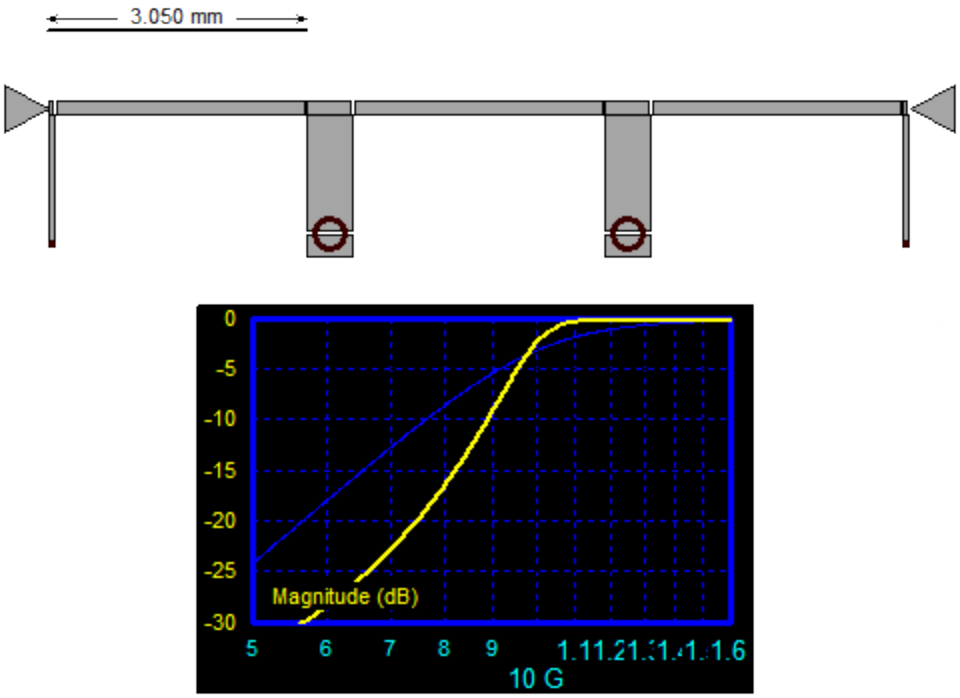
10GHz Butterworth strip line low pass spaced stub filter with exactly 3.01dB cutoff frequency

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



10GHz strip line high pass approximate lumped equivalent filter with slight error at the cutoff frequency

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



10GHz Butterworth strip line low pass spaced stub filter with exactly 3.01dB cutoff frequency

Open and Short Terminations

RLGC lines are not modeled with physical materials, so they assume ideal open and short terminates.

All stripline and microstrip open end stubs and parallel segments are terminated with end effect capacitance. All shorted stubs and parallel lines are terminated with via inductance. The synthesis process includes these termination reactances when computing widths lengths, and angles.

Open End Models

Schematic and layout diagrams show nothing for open ends, so the end effect capacitance must be assumed. Capacitance modals are standard textbook models. Layout stub lengths are from the base of the stub to the open end, as shown in Figure 1 below.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

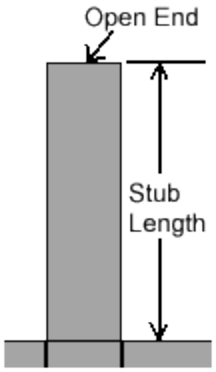


Figure 1: Open Stub Termination Geometry

Shorted End Models

Schematic diagrams show a triangular ground for all shorted ends. Layout views show a via hole. Via hole diameters are 70% of the width. The length of the attached line is taken to be the center of the via hole, as shown in Figure 2 below. Inductance models are standard textbook modals.

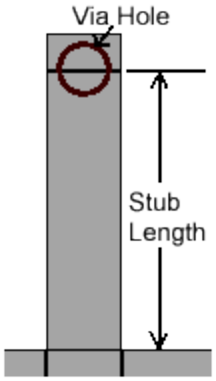


Figure 2: Via Stub Termination Geometry

Transmission Line Prewarping

Just as in digital filters, the inherent cycling of the transmission line frequency response may visibly warp the frequency spectrum if the transmission null frequency is set close to the pass band frequencies. In the case of a band pass or band stop filters, this warping has the effect of reducing the band width of the pass band or stop band. However, just as in digital bilinear prewarping, the frequency scale of a transmission line filter may also be prewarped at as many

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

points as is desired to maintain the band width of the pass band. For low pass and high pass filters, the filter performance at the cutoff frequency is already matched to the ideal filter performance, so no Prewarping is necessary.

When creating named pass or and stop filters with transmission line stubs, FilterSolutions selects the lower band edge frequency point as the "equal" frequency, that is the frequency where the transmission line filter response exactly matches the ideal filter response (See Help for inductor approximations), and then **automatically prewarps** the upper band edge frequency in order to maintain the band width of the filter.

Prewarping Exceptions

Is briefly noted in the above paragraph, no prewarping is performed when transmission line segments are used to simulate inductors. When the "Combine Stubs" option is used to simulate LC resonators, prewarping is performed, but only results in part of the error being removed.

Prewarping Equations

Transmission line prewarping equations are easily derived from the inductor and/or capacitor approximation equations provided in the appropriate sections of Help. The following analysis is based from the inductor approximation impedance expressions for inline element approximate replacements only:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$Z|_{\varpi=\varpi_C} = JZ_0 \tan\left(\frac{\pi\omega_C}{2\varpi_N}\right) = J\varpi_C L$$

and:

$$Z|_{\varpi=\varpi_S} = JZ_0 \tan\left(\frac{\pi\omega_S}{2\varpi_N}\right) = J\varpi_{new} L$$

Therefore:

$$\varpi_{New} = \varpi_C \frac{\tan\left(\frac{\pi\omega_S}{2\varpi_N}\right)}{\tan\left(\frac{\pi\omega_C}{2\varpi_N}\right)}$$

Where:
ϖ_S = User Selected Frequency Point to map. (the "Mapped From" frequency)
ϖ_C = User defined frequency where the Tx line frequency response exactly matches the ideal frequency response
ϖ_N = User Defined Tranmission Line Null Frequency
ϖ_{New} = Calculated "Mapped To" Frequency

Transmission Signal Propagation

FilterSolutions has the capability to simulate transmission line filters with finite length transmission line segments in between transmission line stubs. The impedance of each stub is calculated with the open and short circuit load impedance calculations. The signal propagation in the segments between the stubs is calculated in accordance with standard port impedance and admittance calculations.

Lumped Circuit Approximations

Lumped Element Replacements

Transmission line filters are transmission line circuit filters that approximate lumped circuit filters. Lumped elements are replaced with transmission line stubs, which are opened or shorted at their terminations. Transmission line elements such as this always produce aliasing. If each individual element is replaced with a corresponding transmission line stub, extending the stub lengths and recalculating the stub impedances accordingly can control this aliasing.

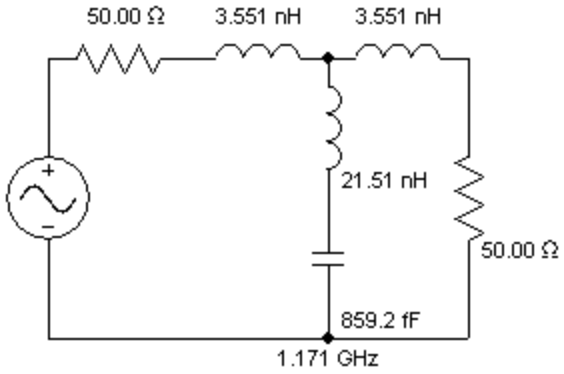
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LC Resonator Replacements

It is possible to replace lumped LC resonators with single transmission line stubs by setting the stub length such that the zero or infinite impedance matches the resonant frequency of the lumped LC resonator. Transmission line filters with this feature require fewer stubs than when simulating each element individually, however significant aliasing is always produced.

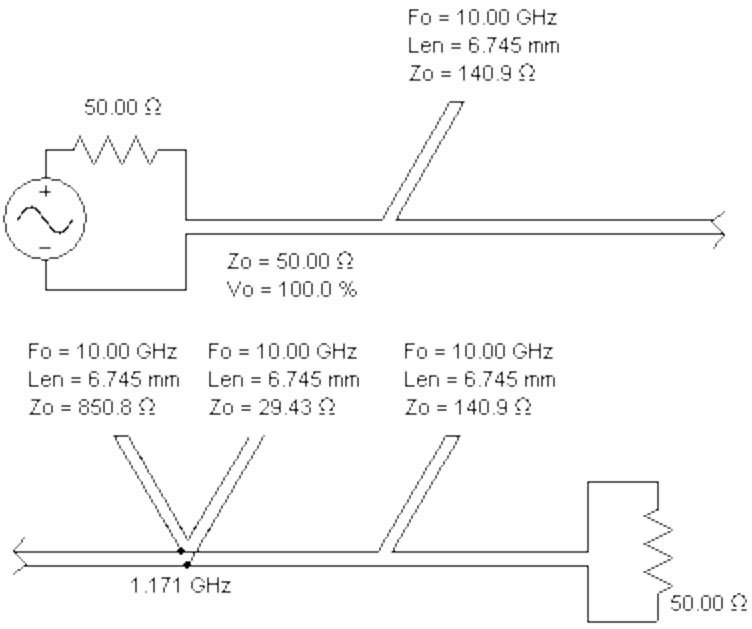
Example 1

The following 1G Hz lumped element filter contains undesirable element values

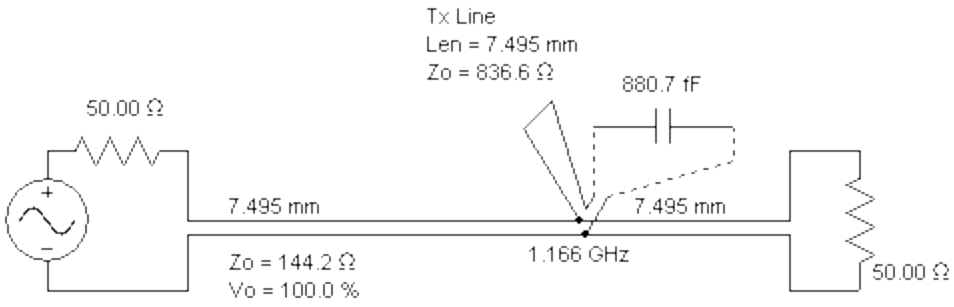


The following transmission line filters are approximately equivalent:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

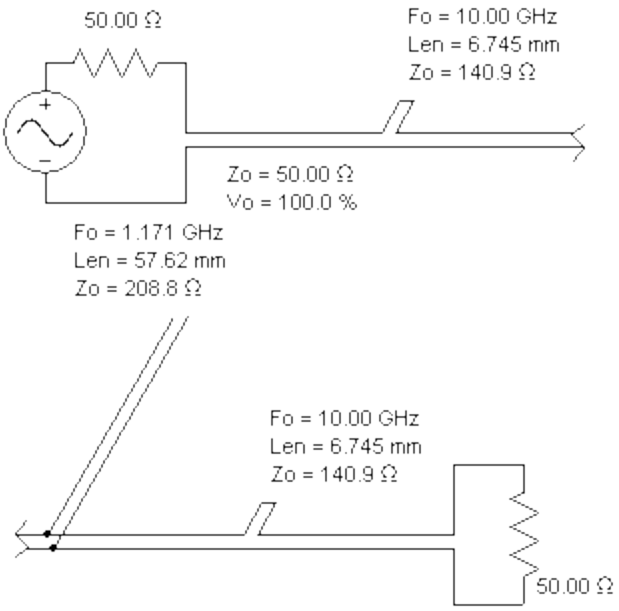


The two shorted stubs may be replaced with transmission line segments as shown below. This implementation may be realized as a micro strip or strip line.



The resonator section may be combined into one stub as shown below. Although this topology is more efficient, it produces significant aliasing.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The segment distance between the stubs should ideally be zero, but may be manually set to any finite value to simulate real world parameters.

Lumped Element Models

Stubs that are used to simulate lumped elements may be converted to the lumped elements they are simulating with the use of the right mouse key. Full lumped parasitic analysis is supported.

In addition to finite Q, series and parallel resistance parasitics and reactive parasitics are also modeled. Select **Parasitics** tab in the main Lumped control panel to enter default values. Series inductor resistance is sometimes referred to as DC series resistance (DCR) and series capacitor resistance is sometimes referred to as equivalent series resistance (ESR). Capacitors support series inductor parasitics, and inductors support parallel capacitance parasitics.

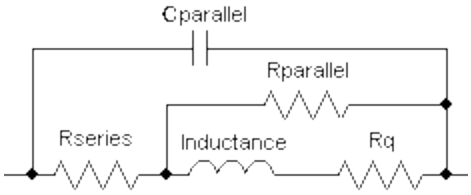
Like finite Q, the analysis is available in the schematic analysis and amplitude equalization is available to mitigate the lossy parasitic effects.

The following models are used for analysis purposes when series, parallel, and/or finite Q effects are entered.

Inductors

The model elements of Q resistance, series resistance, parallel resistance, parallel capacitance and inductance are arranged according to the following inductor model

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

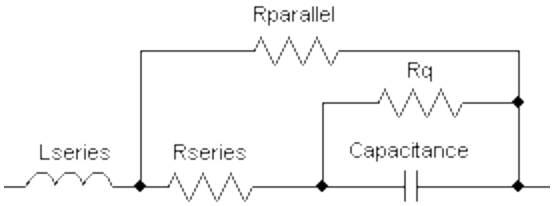


Inductor model with Parasitic Effects

Rq is computed as a function of frequency: Rq = (L*W)/Q where L is inductance in Henries, and W is frequency in R/S

Capacitors

The model elements of Q resistance, series resistance, parallel resistance, and series inductance capacitance are arranged according to the following capacitor model.



Capacitor model with Parasitic Effects

Rq is computed as a function of frequency: Rq = Q/(C*W) where C is capacitance in Farads, and W is frequency in R/S

Coupled Resonator and Multiconductor Filters

This chapter contains the following topics:

- Lumped Coupled Resonator Simulations
- Shunt Stub Resonators
- Parallel Edge Coupled
- Multiconductor Models
- Hairpin Resonator Filters
- Miniaturized Hairpins

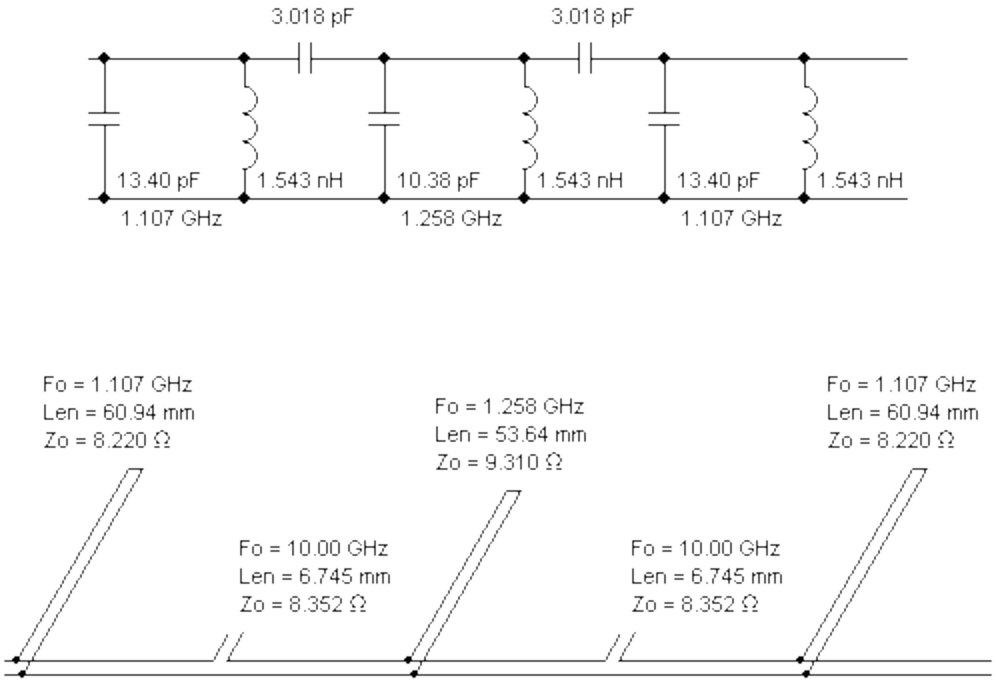
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Ring Resonator
- Cascaded Quadruplet Resonators
- Interdigital Filters
- Combline Filters
- Tapped and Pinned Coupling
- Notch Filters

Lumped Coupled Resonator Simulations

Lumped Equivalent Coupled Resonator Filter may be produced by selecting a band pass filter with **Lumped Translation** or **Inductor Translation** selected, then checking the **Resonator translation** check box at the top of the distributed schematic. **Inductor Translation** will leave capacitors alone and only translate the inductors and stub resonators to a distributed topology.

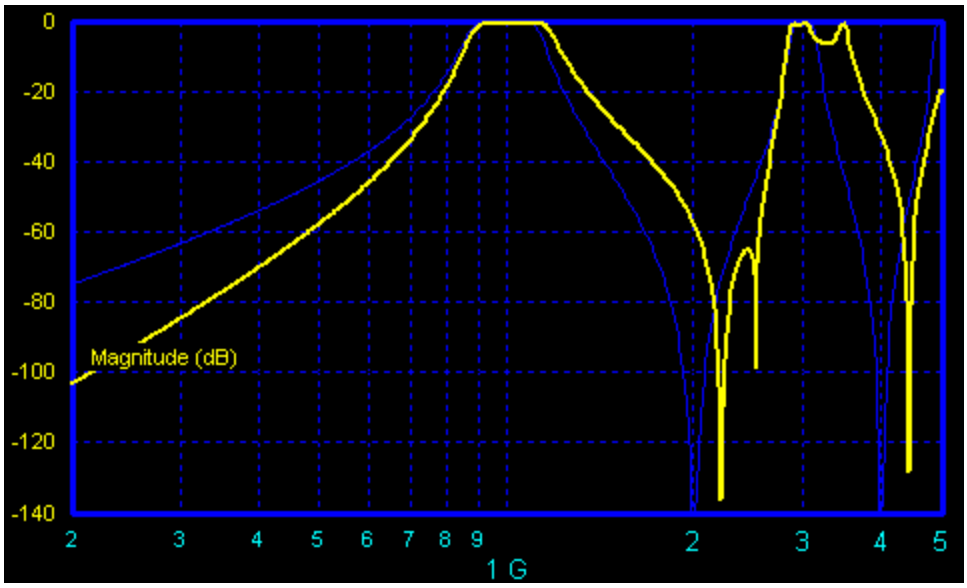
If it is desired to simulate LC resonators with a single stub, select **Combine Stubs** option on the main control panel. The resulting transmission line coupling stubs then behaves like ideal capacitors or inductors. The **Use Coupled Lines** option in conjunction with **Combine Stubs** uses coupled lines to simulate LC resonators.



Coupled Resonator Filter With Lumped Capacitor Couplers

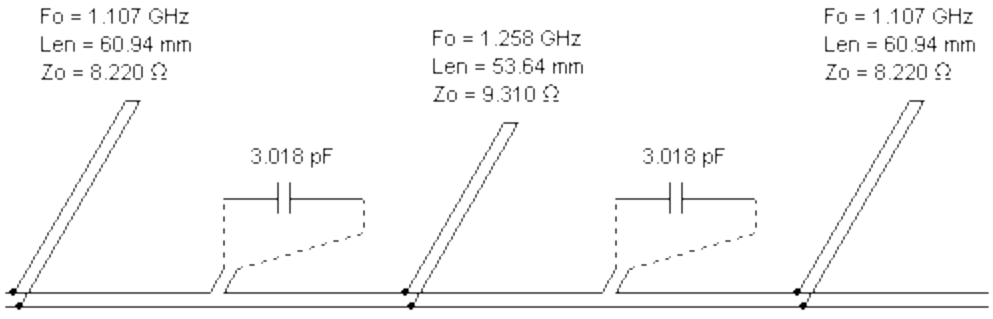
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The short coupling stubs in the 1 GHz transmission line filter above simulate ideal capacitors that alias at 1000 GHz. For all practical purposes, they may be assumed to be ideal capacitors in the simulation. The single stub resonators in this example will produce significant aliasing, as shown below. The background blue trace is the classical band bass transmission line simulation.



Coupled Resonator Aliasing

Alternatively, transmission lines may be replaced with lumped elements. See the section on line/lumped composite filters.



Composite Transmission Line/Lumped Coupled Resonator Band Pass Filter

Using a top series coupling inductor model (or shunt coupling inductor model for series coupled resonator) instead of a capacitor permits the use of a microstrip or stripline segment inductor

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

simulation. Combining the shunt stubs (or series stubs in the case of series coupled resonators) into a resonator reduces the physical size of the filters.

Shunt Stub Resonators

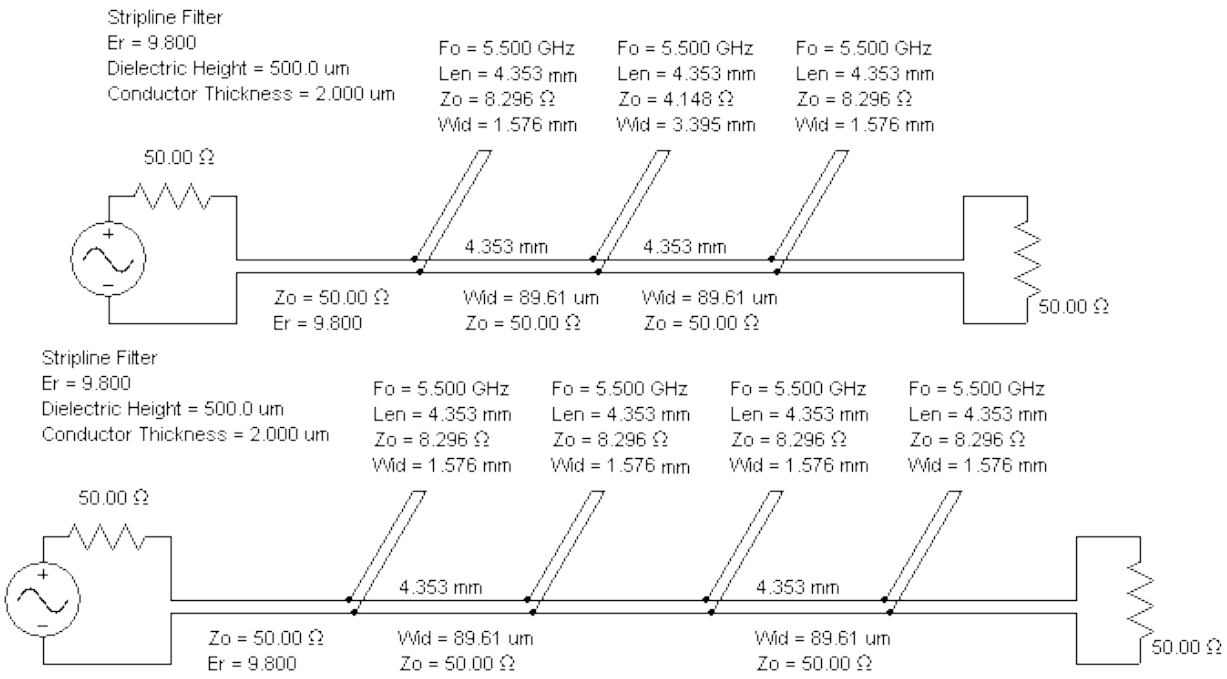
Shunt Stub Resonator Filter may be produced by selecting a band pass filter with **Shunt Stub Resonators** selected from the distributed selection list.

Unlike lumped coupled resonator simulations, shunt stub resonator filters do not depend on lumped element approximations, are not limited to narrow band filters, and may be produced with more easily realizable geometries. If a medium or wide band filters is chosen, then shunt (or series) parallel coupled resonator filters are automatically selected.

Shunt stub resonators may be synthesized with RLGC lines, striplines, microstrips, and suspended substrates.

Butterworth and Chebyshev filters may be composed of shunt stub resonators with all stubs being identical to each other in width and length by checking, **Equal Stub Widths**. However, this is at the expense of requiring more stubs.

The impedance of the center stub resonator may be manually set by checking **Set Center Zo** and entering the desired impedance.

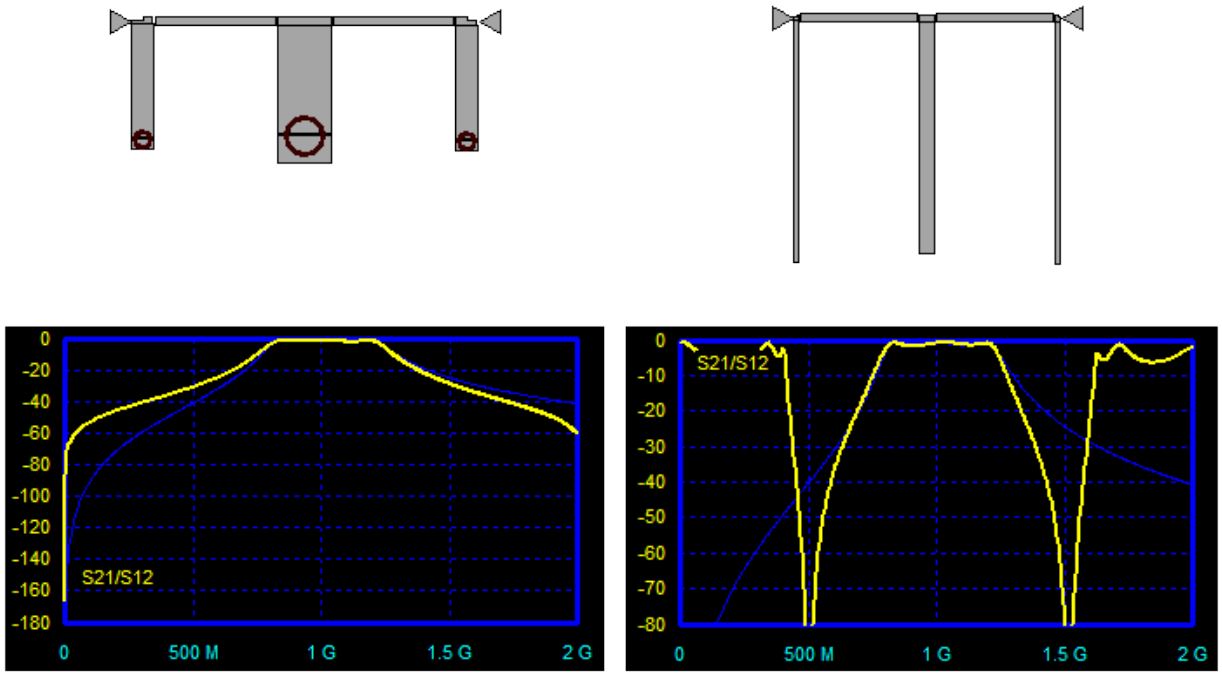


Shunt Stub Resonator Filters
With Unequal and Equal Stubs

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Open End Stubs

From transmission line theory, a shorted stub has the same impedance as an open stub that is 1/4 wavelength longer at one specific frequency. With a few additional impedance modifications, a shunt stub resonator may be constructed with open end half length stubs at the center frequency, and maintain comparable pass band performance. This transformation produces transmission zeros at each side of the pass band, and also passes DC. Strictly speaking, it is a low pass filter rather than a band pass filter. This can be seen in the S21 parameters of the respective topologies below.

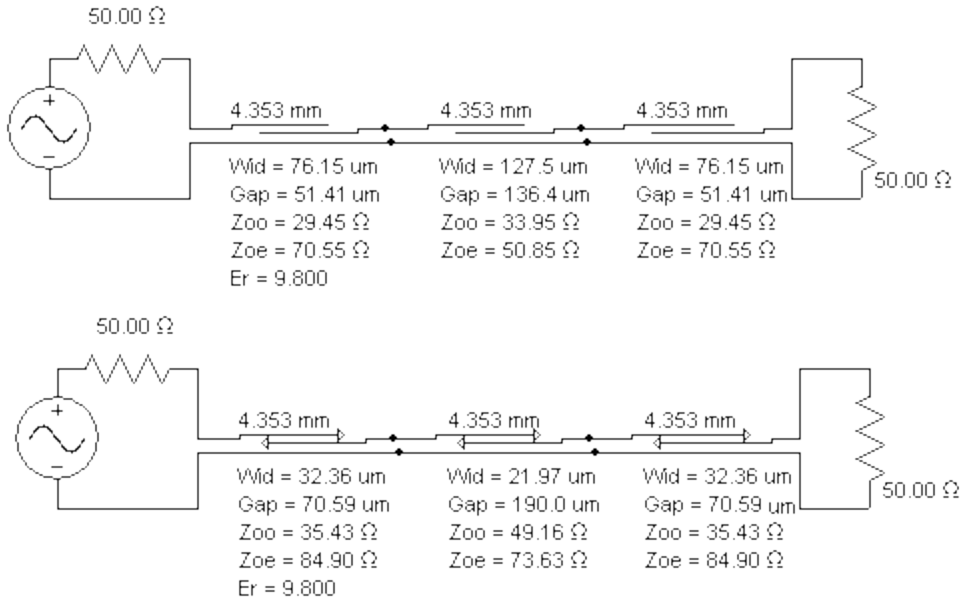


Standard Shunt Stub Resonator and Open Stub Modification, and associated S21.

Parallel Edge Coupled Resonators

Parallel edge coupled filters may be produced by selecting a band pass filter with **Parallel Edge Couple** selected from the distributed selection list. Parallel edge coupled filters use a series of coupled lines to achieve a band pass filter. These resonator filters do not depend on lumped element approximations, are not limited to narrow band filters, and may be produced with more easily realizable geometries. These resonators are composed of coupled lines with open ends or shorted (via) ends. Open ends are more desirable due to simpler construction, but via ends may offer a more flexible geometry selection. Via ends are synthesized by selecting **Via Ends**.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

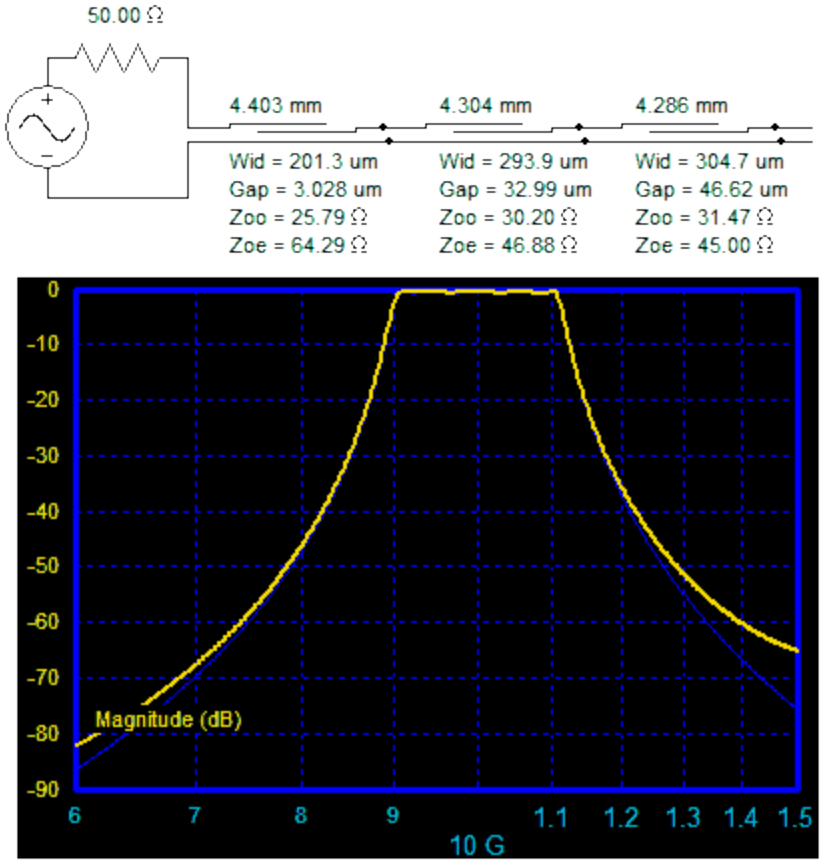


Parallel Coupled Resanator Filters
With Open and Shorted Ends

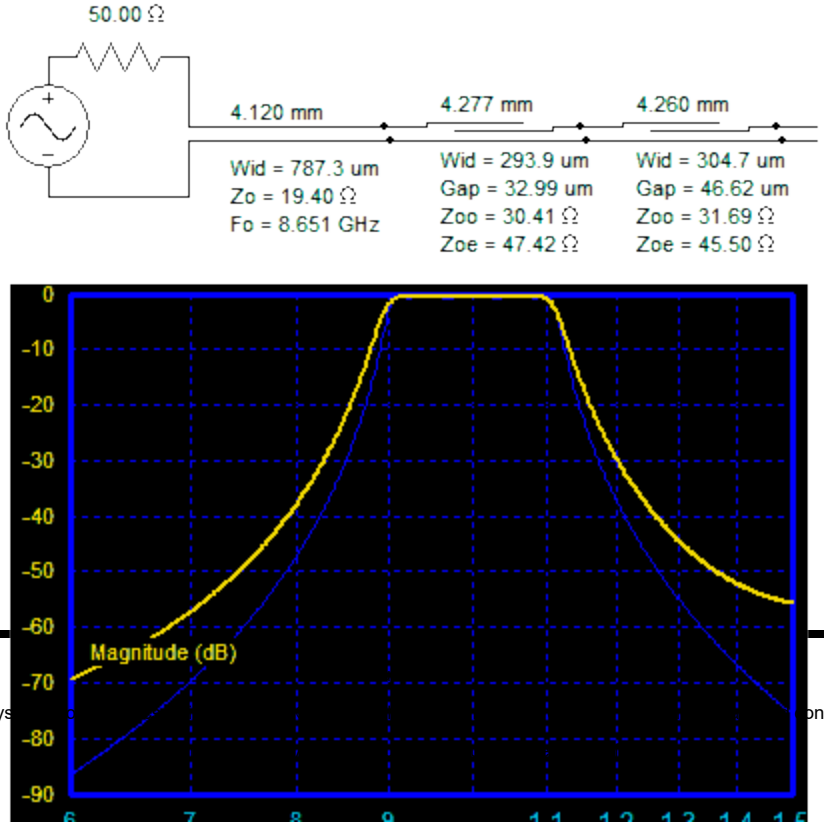
Narrow Gaps:

Frequently, the coupling gaps are too small to be practical or realizable, especially in the outer couplers. When this situation occurs in narrow and mid band filters, the "Tapped" or "Pinned" solutions replaces the coupled line with a single segment that approximates the coupled line impedance over a narrow band. The tapped and pinned approximation usually produces negligible degradation of the frequency response, as seen below. FilterSolutions replaces all unrealizable coupled lines with pinned segments regardless if the "Pinned" option is selected or not.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

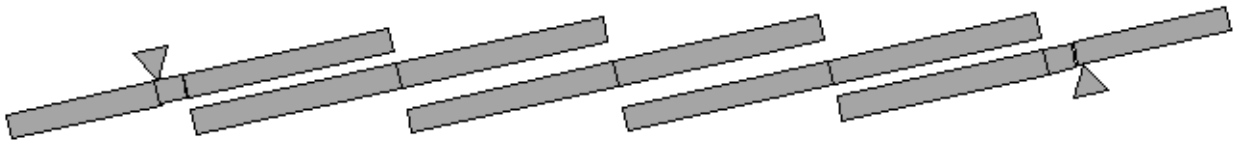


Parallel Coupled Resonator Filter With Narrow Gap And Frequency Response



Equal Width Conductors

It is frequently desirable to synthesize with conductor of all equal widths. FilterSolutions provides the capability of selecting this option by selecting it in the FilterQuick topology scroll menu, or by checking the "Equal Width Conductors" in the Advanced user design panel. Parallel edge coupled topologies synthesized with equal width conductors are tapped by the feed lines part way up the first and last segment. The outer section of the segment may be positioned vertically as a stub by by checking "Stub Taps" in the Advanced user panel.



Parallel Edge Coupled Layout with Equal Width Conductors

Multiconductor Tx Line Models

Multiconductor transmission line models for Hairpin, Interdigital, and Combline filters are supported and made available for selection in the distributed selection list. Multiconductor transmission lines may be modeled with RLGC lines, striplines, microstrips, and suspended substrates.

RLGC:

RLGC filters consist of standard RLGC transmission line models in array of coupled vertical transmission lines. Each vertical line and each coupling between the lines has a specific impedance and effective dielectric constant, or a specific inductance and capacitance, all of which are echoed out on the Nuhertz schematics. Losses are modeled with known resistance and conductance for each line. Hairpin resonators are modeled with equal impedance legs.

Striplines:

Stripline models include actual conductor and dielectric geometries required to realize the RLGC lines. Dielectric geometries and conductor width are supplied by the dielectric definitions, and conductor width and length geometries are calculated. Losses are calculated based upon conductor and dielectric geometries, conductor resistivity (with respect to copper), and dielectric loss tangent. Hairpin resonators are modeled with equal width legs and will result in unequal impedances due to the differing gap widths on each side of the hairpin.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Microstrips and Suspended Substrates:

Microstrip models are similar to stripline models, except that the dielectric constant (Er) is no longer a constant, due to the existence of air (Er=1) above the conductor. Each vertical line and coupling line has a different effective Er which must be factored into the geometry calculations. The differing effective Er's also produce some error in the filter response in the form of frequency distortion and spurious passbands. Hairpin resonators may be modeled with equal width legs and will result in unequal impedances due to the differing gap widths on each side of the hairpin.

Optimization:

The accuracy of the design may be increased by checking the "Quick Optimize" CheckBox. However, this will be at the expense of more computer time. It is good design practice the synthesize the filter without optimization first, then optimize it after an acceptable design is found.

Analysis Calculations:

FilterSolutions produces generally accurate frequency and time response (Advanced Panel required for time response simulations) of the multiconductor filters it synthesizes. This permits a quick inspection to see if the synthesized filter performance is acceptable.

Hairpin Resonator Filters

Hairpin Filter may be produced by selecting a band pass filter with **Hairpin** selected in the distributed selection list.

Hairpin filters are not as compact as Interdigitals, but they also do not require vias, making them potentially advantageous.

Hairpin Options

Hairpin line width and gap between the legs may be user selectable, of the default values are not desired.

Via terminated hairpins are selectable as an option, but are not recommended.

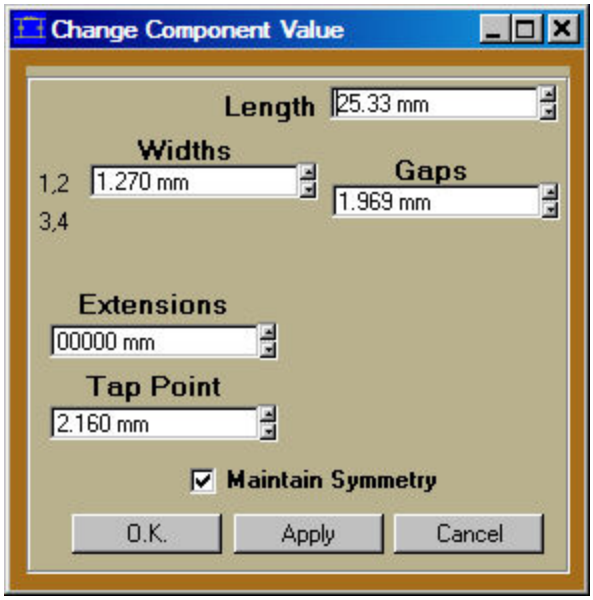
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

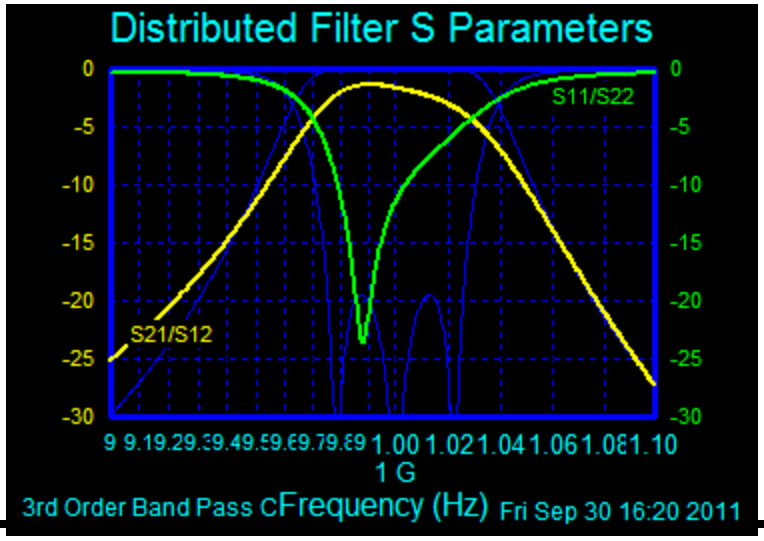
Tuning

Hairpin filters may be tuned by adjusting their tap position, end ring gaps, overall length, and resonator separations. Clicking on the ring resonator schematic or the "Edit" button brings up the Change Control Panel, where values may be easily edited and scrolled to different values while the analysis graphs update in real time. The Change control panel maintains physical symmetry or not as desired.

AEDT Tuning and Optimizing

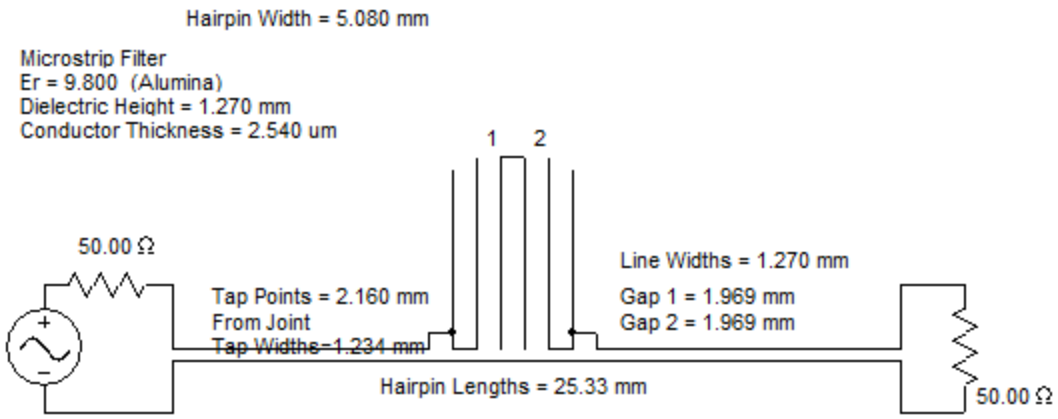
AEDT tuners and optimizers may tune or optimize the exported filter, and the tuned filter may be imported back into Nuhertz.

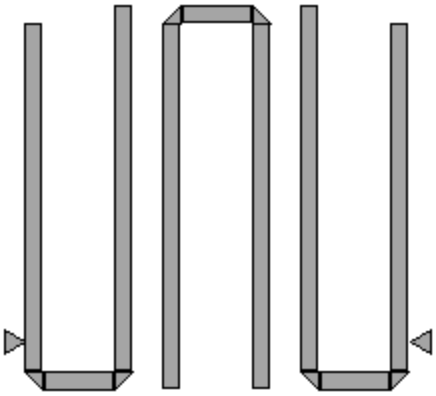




Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Change Control Panel for Tuning the Miniaturized Hairpin Filter Geometry.





Miniaturized Hairpin Resonator Filters

Miniaturized Hairpin Filters may be produced by selecting a band pass filter with **Miniature Hairpin** selected in the distributed selection list.

Miniaturized Hairpin filters are more compact than standard hairpins, making them desirable when board space is limited or expensive.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Miniaturized Hairpin Options

Miniaturized Hairpin line width and gap between the legs may be user selectable, of the default values are not desired.

Manufacturing Geometry Limitations

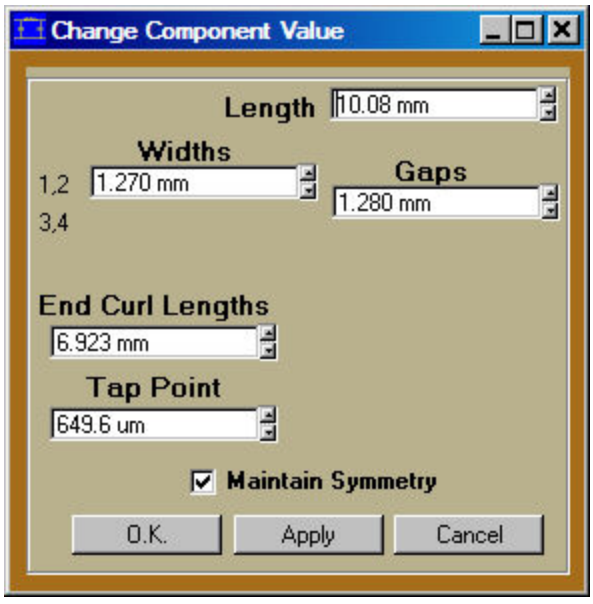
To meet manufacturing geometry requirements, geometry limits under the **Geometry** tab are enforced.

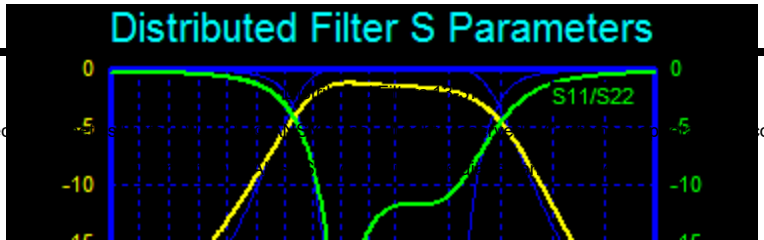
Tuning

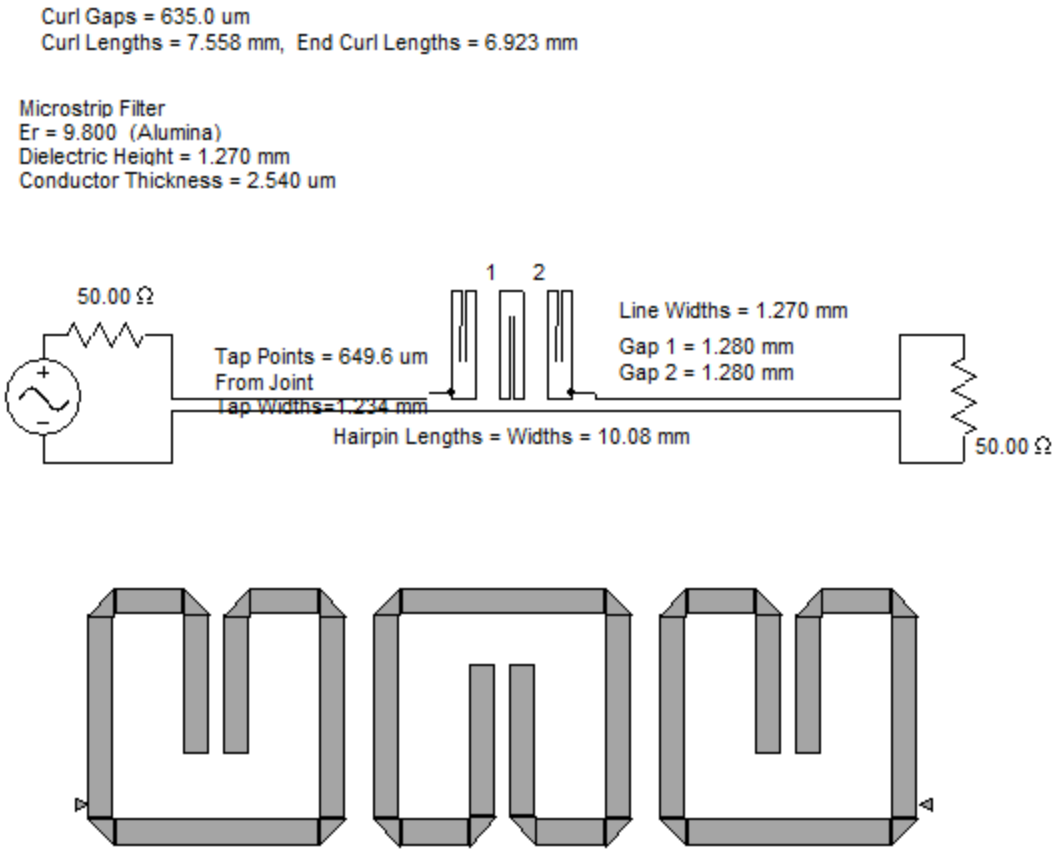
Miniaturized hairpin filters may be tuned by adjusting their tap position, end ring gaps, overall length, and resonator separations. Clicking on the ring resonator schematic or the **Edit** button brings up the Change Control Panel, where values may be easily edited and scrolled to different values while the analysis graphs update in real time. The Change control panel maintains physical symmetry or not as desired.

AEDT Tuning and Optimizing

AEDT tuners and optimizers may tune or optimize the exported filter, and the tuned filter may be imported back into Nuhertz.







Ring Resonators Resonator Filters

Ring resonator filters may be produced by selecting a band pass filter with **Ring resonator** selected in the distributed selection list.

Ring Resonator filters require less physical space than wide pin standard hairpins, making them desirable when board space flexibility is restricted, and wide pin would otherwise be required.

Ring Resonator Options

Ring resonator line width and gap between the legs may be user selectable, of the default values are not desired.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Manufacturing Geometry Limitations

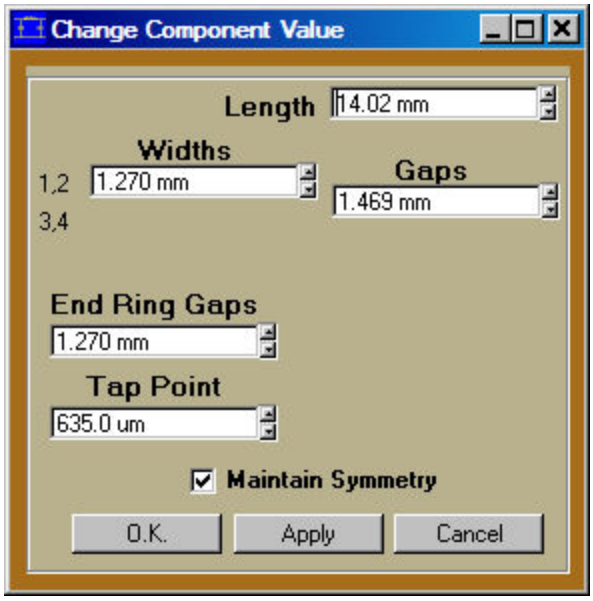
To meet manufacturing geometry requirements, geometry limits under the "Geometry tab are enforced.

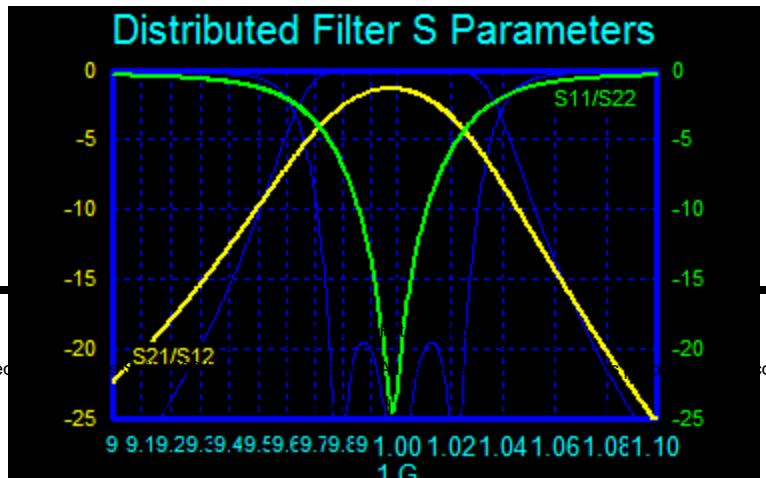
Tuning

Ring Resonator filters may be tuned by adjusting their tap position, end ring gaps, overall length, and resonator separations. Clicking on the ring resonator schematic or the **Edit** button brings up the Change Control Panel, where values may be easily edited and scrolled to different values while the analysis graphs update in real time. The Change control panel maintains physical symmetry or not as desired.

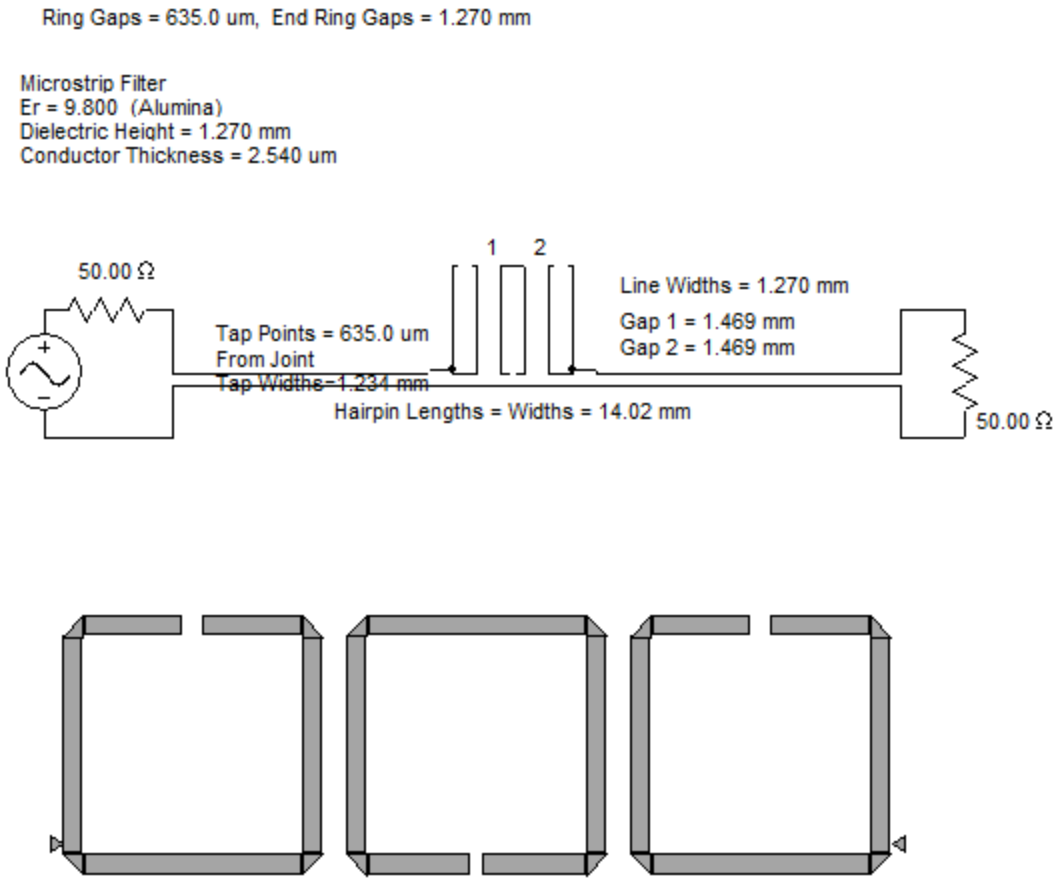
Third Party Tuning and Optimizing

AEDT tuners and optimizers may tune or optimize the exported filter, and the tuned filter may be imported back into FilterSolutions.





Change Control Panel for Tuning the Ring Resonator Filter Geometry.



Series Quadruplet Resonators

Even pole Half Band Elliptic filters may be realized with ring resonators or miniaturized hairpins by cross coupling the first and last resonator in each quadruplet of a cascaded set of quadruplets. Each cross coupling implements a transmission zero in the filter being implemented. Table 1 below shows the coupling matrix of an eight pole cascaded quadruplet filter. Note the cross couplings between resonators 1 and 4 (-0.0250556), and resonators 5 and 8 (-0.0156019).

```
0               0.0809627 0               -0.0250556 0               0 0 0
0.0809627       0               0.0776943 0               0 0 0 0
0               0.0776943 0               0.0514389 0               0 0 0
-0.0250556      0               0.0514389 0               0.0556118 0               0 0
0               0 0 0.0556118 0               0.0542574 0               -0.0156019
```


Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0	0	0	0	0.0542574	0	0.0716122	0	
0	0	0	0	0	0.0716122	0	0.0833026	
0	0	0	0	-0.0156019	0	0.0833026	0	

Table 1: 8 Pole Quadruplet Coupling Matrix

Designing Cascaded Quadruplet Resonator Filters

It is not physically possible to implement all of the transmission zeros in Elliptic filters above 4 poles in a cascaded quadruplet topology. Therefore, "Half Band Ripple" is automatically selected in order to set the maximum number of transmission zeros possible that can be implemented in the stop band by cascaded quadruplet topology. The "Half" in "Half Band Ripple" refers to utilizing roughly half of the number of transmission zeros that a standard elliptic filter normally requires. FilterSolutions will synthesize a flat stop band with the remaining half of transmission zeros and implement them as cross couplings in each quadruplet of a cascaded quadruplet resonator filter. Table 1 above shows an 8 pole Half Band Ripple filter with two of the normal 4 stop band zeros of a standard Elliptic stop band. Note that equally terminated 8 pole Elliptic filters utilize only 3 transmission zeros.

Single Point Ripple Stop Bands

Cascaded quadruplet resonator filters may be simplified somewhat by selecting, **Single Point Ripple** to place all stop band zeros at the same point above and below the pass band. Doing this forces the filter to be physically symmetric permits the tuning and optimizing to utilize half the number of variables as a flat stop band filter. This in turn makes the tuning and optimizing faster and more reliable. The downside is that single point ripple filters may require more poles to be as effective as flat stop band filters.

Cascaded Quadruplet Topology

Figure 2 below shows the physical topology of a 10 pole, flat stop band cascaded quadruplet filter. Note that resonators 2 and 5 are cross coupled, and 6 and 9 are cross coupled. Since 10 is not divisible by 4, two single resonator end pieces are utilized to bring the total number of resonators up to 10.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

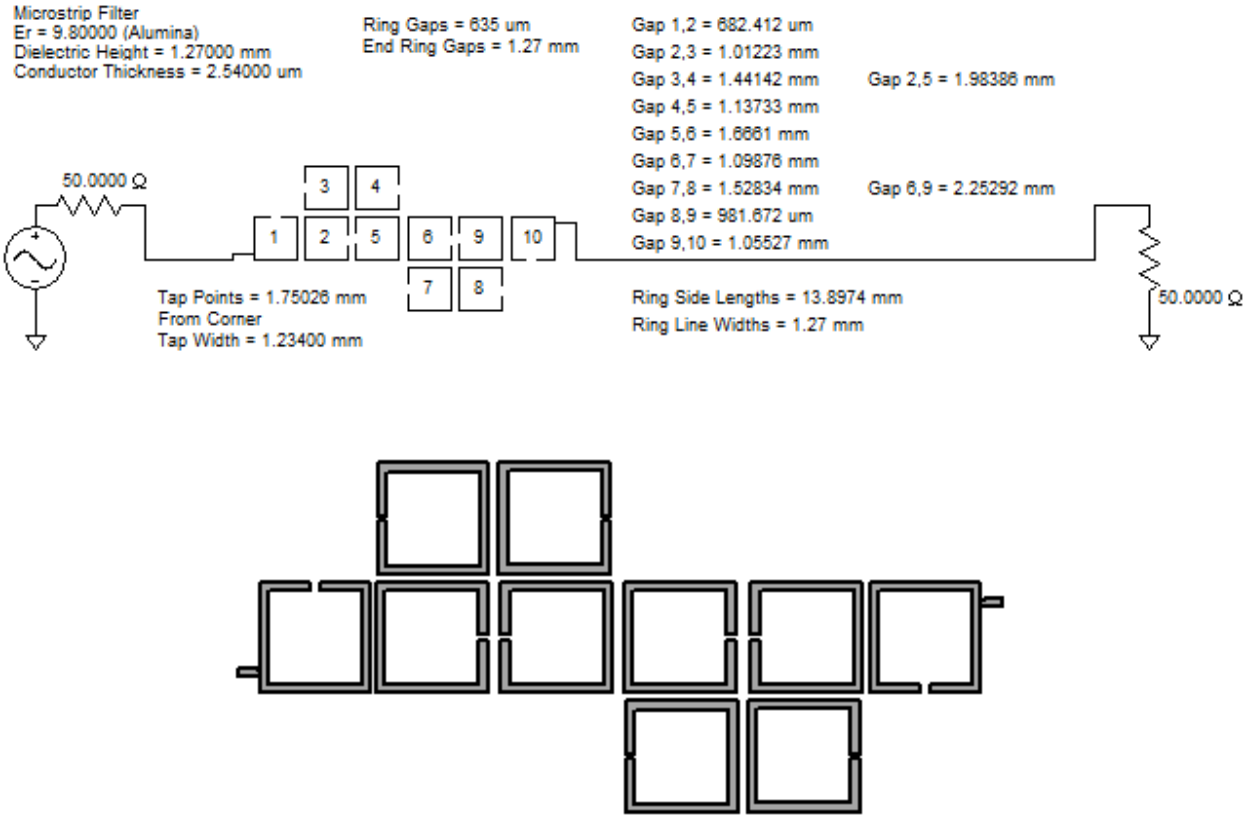


Figure 2: 10 Pole Cascaded Quadruplet Topology

EM and Circuits Analysis

Cross coupled micro strips are notoriously inaccurate to circuits simulate, especially when implemented with miniaturized hairpins. It is therefor very necessary to tune and optimize the electromagnetic response in order to produce a viable design. A 4 pole single stage quadruplet usually produces a good optimized EM response, but 6 poles and above may produce less than desirable optimized EM response, especially when miniature hairpins are used. The designer should consider the quality of the optimized EM response when choosing a topology to implement.

EM Considerations

Higher order cascaded quadruplet resonator filters generally utilize deep stop bands. If deep stop bands are a design priority, be sure to use an EM simulator that accurately simulates deep stop bands. If deep stop band accuracy is not a priority, then the designer may consider

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

optimizing the pass band only, and let the stop bands fall wherever they land. Stop band goals may be removed by manually removing them following the export, or in the FilterSolutions export page, if goal editing is supported.

Interdigital Filters

Interdigital Filter may be produced by selecting a band pass filter with **Interdigital** selected in the distributed selection list.

Wideband Interdigital Filters

Wide band Interdigital filters are synthesized with open end on the input and out couplers. This option works much better for wideband filters, or for application that require open end couplers.

Tapped Interdigital Filters.

Interdigital filters (without the "Wideband" selection) are prone to having narrow gaps at the input and output couplers, or not synthesizing at all due to unrealizable geometries. This situation may be corrected with the use of taps by selecting the **Tapped** check box. This option removes the outer couples and taps the remaining outer couplers.

Pinned Interdigital Filters

Interdigital filters with the "Wideband" selection prone to having narrow gaps at the input and output couplers, or not synthesizing at all due to unrealizable geometries. This situation may be corrected with the use of taps by selecting the **Pinned** check box. The pinned option replaces the outer couplers with hairpin resonators..

Equal Width Approximation

It is frequently advantageous to maintain the same width for all conductors. When this is the case, selecting the **Equal Width Approx** option accomplishes this. The resulting increased error is usually negligible.

Interdigital Options

Interdigital line impedance may be user selectable, of the default value is not desired. Standard Interdigital filters use via terminated resonators., and offer relatively high spurious frequencies

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(3X center frequency).

Open End Interdigital

Open end Interdigital termination are selectable as an option. Open end Interdigital offers the advantage of not using vias, but have the disadvantage in requiring larger geometries and lower spurious frequencies (2Xcenter frequency).

Manufacturing Geometry Limitations

To meet manufacturing geometry requirements, geometry limits under the "Geometry tab are enforced.

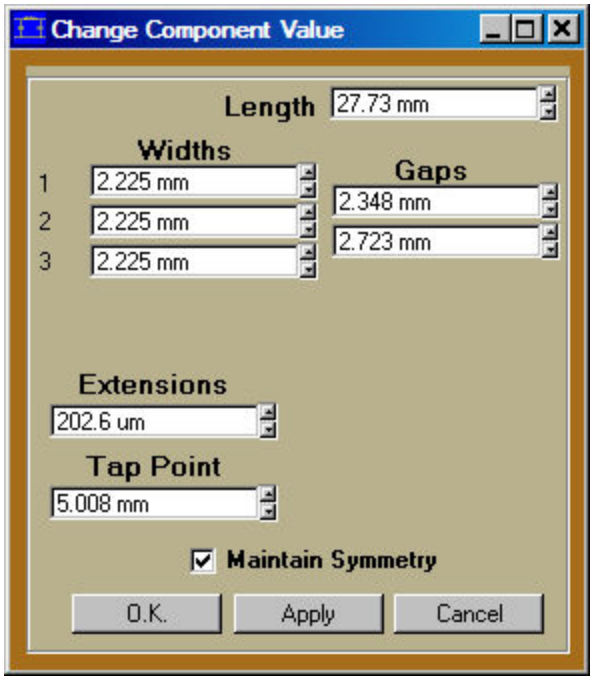
Tuning

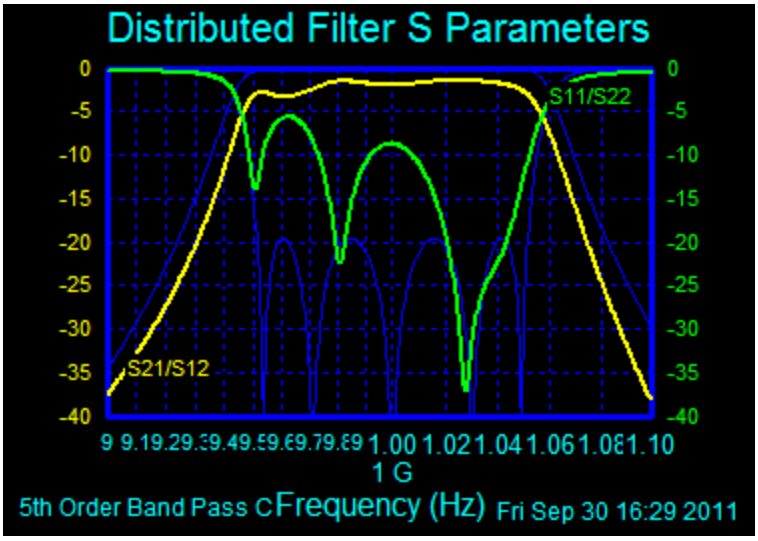
Interdigital filters may be tuned by adjusting their tap position, end ring gaps, overall length, and resonator separations. Clicking on the ring resonator schematic or the "Edit" button brings up the Change Control Panel, where values may be easily edited and scrolled to different values while the analysis graphs update in real time. The Change control panel maintains physical symmetry or not as desired.

AEDT Tuning and Optimizing

AEDT tuners and optimizers may tune or optimize the exported filter, and the tuned filter may be imported back into Nuhertz.

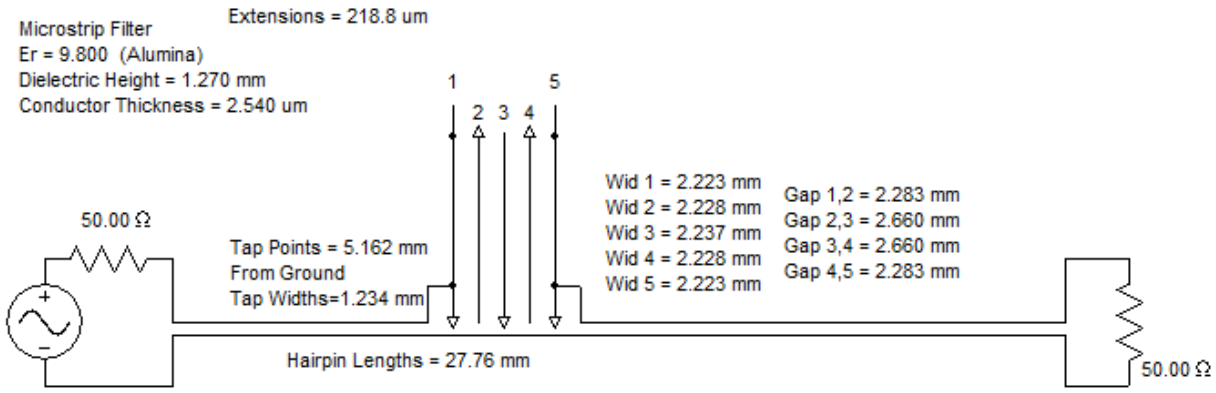
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

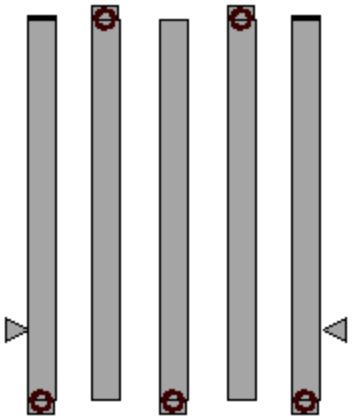




Change Control Panel for Tuning the Interdigital Filter Geometry.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





Combline Filters

Combline Filter may be produced by selecting a band pass filter with **Combline** selected in the distributed selection list.

Quarter Length Frequency

Combline filters require a selection of a quarter length frequency. The standard 2 to 1 ratio above the center frequency will not work, and will prompt the user to increase it if it is selected. Ratios of 4 to 8 times higher than the center frequency are preferred.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Taps

Combline filters are frequently unrealizable with realistic geometries, or have excessively small outer gaps. If either of these situations occur, it is desirable to "Tap" the filter. Tapped Combline filters are generally more frequently realizable and have more desirable outer gap geometries when they are realized.

Combline Options

Interdigital line impedance may be user selectable, of the default value is not desired.

Via terminated hairpins are selectable as an option, but are not recommended.

Equal Width Approximation

It is frequently advantageous to maintain the same width for all conductors. When this is the case, selecting the **Equal Width Approx** option accomplishes this. The resulting increased error is usually negligible.

Manufacturing Geometry Limitations

To meet manufacturing geometry requirements, geometry limits under the **Geometry** tab are enforced.

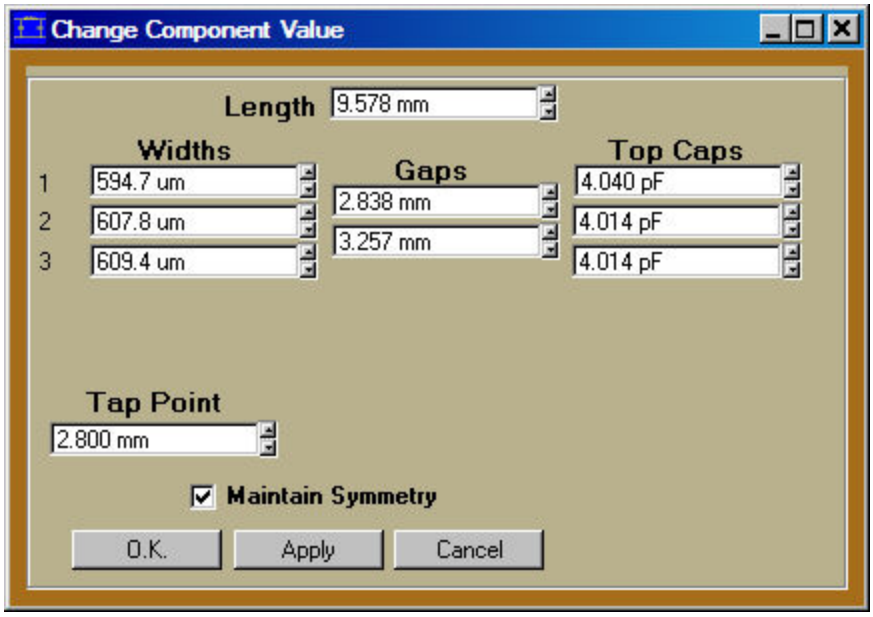
Tuning

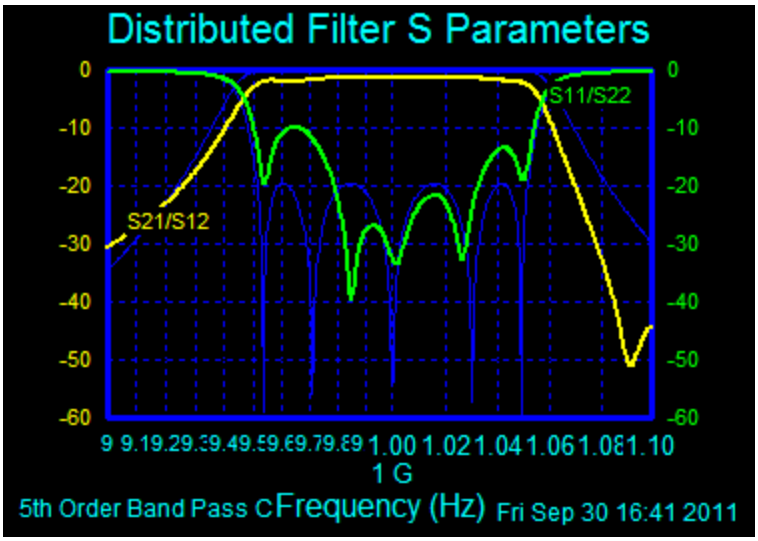
Combline filters may be tuned by adjusting their tap position, end ring gaps, overall length, and resonator separations. Clicking on the ring resonator schematic or the "Edit" button brings up the Change Control Panel, where values may be easily edited and scrolled to different values while the analysis graphs update in real time. The Change control panel maintains physical symmetry or not as desired.

AEDT Tuning and Optimizing

AEDT tuners and optimizers may tune or optimize the exported filter, and the tuned filter may be imported back into Nuhertz.

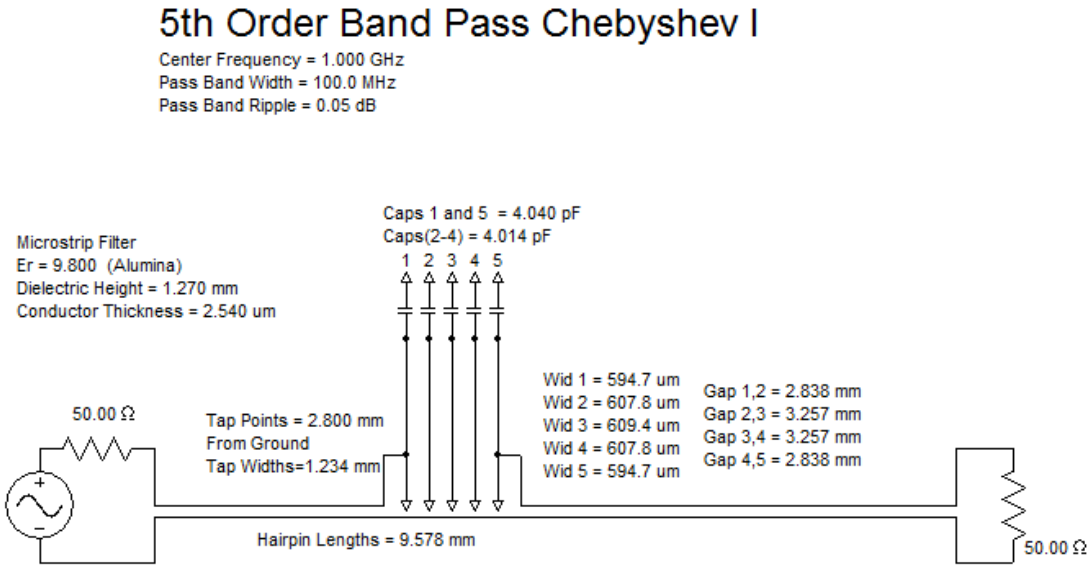
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

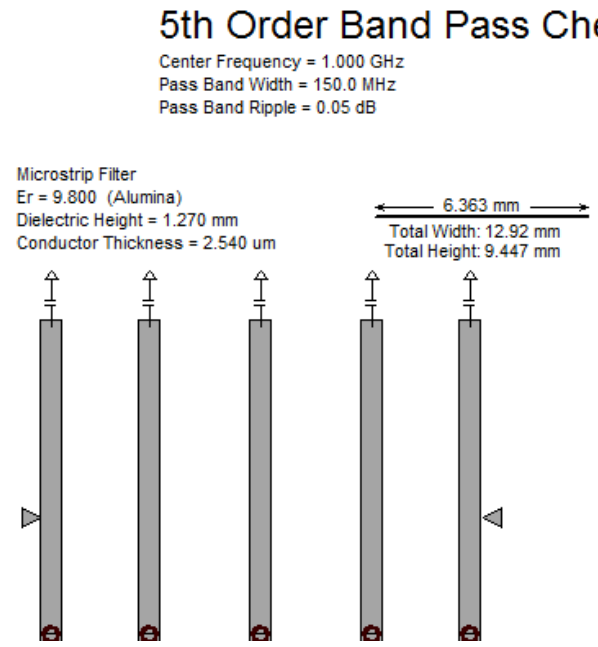




Change Control Panel for Tuning the Combline Filter Geometry.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





Taps and Pins

Multiconductor filters and Parallel coupled resonator filters frequently produce undesirably narrow gaps at their outer resonators. When this happens, the outer resonators may be replaced with a tap or, a hairpin resonator for multiconductor filters, or a narrow band equivalent segment

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

for parallel edge coupled filters. In all cases the gap with is generally much larger and more easily realizable. In the Nuhertz software applications and literature, Tapped coupling is referred to as "**Tapped"**, and Hairpin and segment coupling is referred to as "**Pinned"**.

Examples of Tapped and Pinned coupling may be found under the Help topics for Parallel Coupled, Hairpin, Interdigital, and Combline filters.

Distributed Line Notch Filters

Transmission line notch filters may be implemented with a series of segments and stubs, or with a series of coupled lines. Misrostrip and stripline geometries generally favor the segment and stub approach for wide notches and the coupled line approach for narrow notches.

Wide Notch Filters

For wide notch filters, deselect **Coupled Lines**, and the filter will be synthesized with a series of segments and stubs that approximate LC notch resonators. If the notch is too narrow, microstrip and stripline geometries will be unrealizable, and the coupled line approach will be necessary.

Narrow Notch Filters

For narrow notch filters, select **Notch Resonators** from the band stop selection list, and the filter will be synthesized with a series of coupled lines that approximate a series of LC series notch resonators. If the notch is too wide, microstrip and stripline geometries will be unrealizable, and the uncoupled line approach will be necessary.

Equal Line Widths

Narrow notch micro strip and strip line filters synthesize tracts that are similar in width. Little error is introduced by forcing all the tracts to be of equal width. FilterSolutions provides the option of forcing all of the lines to be of equal width to simply the filter construction at the expense of introducing an error that is usually negligible. The user should the filter response to confirm that the introduced error is acceptable prior to accepting the filter design.

Open Stub Grounds

The narrow notch topology requires one of the coupled lines to be grounded on one side. For microstrip and stripline filters, a hard ground may be costly. FilterSolutions offers the option of simulating a hard ground with an open stub of the same length as the coupled line, which

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

provides an approximate ground at frequencies near the notch frequency. This grounding scheme introduces errors in the bandwidth, center frequency, and passband quality that may need to be considered. **Manual prewarping the bandwidth and center frequency may be required.** Care should be taken to insure that all introduced error are acceptable or are eliminated with manual prewarping of the band width and center frequency.

Working With Distributed Filters

This chapter contains the following topics:

- Simulating Transmission Lines
- Selecting Materials
- Series Overlay Capacitors
- Optimizing the Frequency Response
- Setting Geometry Limits
- Controlling Stub Widths
- Layout and Schematic Views
- Layout Geometry Definitions
- Modify Distributed Filters
- S-Parameter and Spice Model Imports
- S-Parameter and Spice Model Exports
- Alter Distributed Filters Designs
- Tuning Planar Filters
- Composite Lumped/Line Filters
- Coupled Resonator Band Pass Filters
- Net Lists

Simulating Transmission Lines

Transmission line simulations are performed with the buttons located above the schematic. **The buttons on the main control panel are for ideal filter response only and do not simulate the transmission line.** The actual transmission line simulation may differ dramatically from the ideal filter due to the parasitics in the lines and the frequency aliasing effect due to finite length lines.

The following parasitics are accounted for to at least some degree in transmission line simulation:

- Finite line length
- Conductor resistivity

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Dielectric loss tangent
- End effect
- Joints, tees, and crosses
- Corners
- Hairpin cross coupling between the legs.
- Realizable widths and gaps of microstrips and striplines
- Microstrip dielectric constant error.
- Line thickness

Optimizing

Once a design is roughed out considering all of the parasitic effects listed above, it should be optimized. The optimization tool will attempt to force the passband response of the simulated filter to match the ideal pass band response of the filter.

Finalizing

Transmission lines simulations are component simulations only, and are not EMI based. To insure full accuracy, the design should be simulation with a good EM simulation tool, such as HFSS, prior to being finalized.

Distributed Filter Materials

Click on text entries for **Dielectric Constant**, Conductor **Resistivity**, and Dielectric **Loss Tangent** to view a menu of materials and their corresponding values. Select the conductor material or dielectric to **automatically enter the value** into the text entry. If a value is desired that does not appear in the list, simply type it in.

RLGC lines do not use the loss tangent and resistivity entries. Instead the user entries of resistance per length and conductance per length are used for loss computations.

Series Overlay Capacitors

Overlay Concept

High pass and wide band pass filters usually obtain their best performance with lumped series capacitors coupling the shunt inductors or resonators. Lumped capacitors may have undesirable traits such as higher then desired losses, excessive expense, and/or difficult to realize capacitance values. When designing with suspended substrates, these undesirable side effects may be eliminated or minimized by implementing the series capacitance with broad side coupled

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

lines about the suspended substrate material, known as an overlay capacitance. FilterSolutions uses a reasonably accurate EM baed model to synthesize the overlay geometry and perform subsequent simulations. Final optimization should be performed using an EM tool.

Overlay Geometry

Overlay capacitance is realized by sandwiching the substrate layer between broadside coupled lines, taking advantage of associated plate and fringe capacitance. Parasitic shunt capacitance may be minimized with the use of a high dielectric constant substrate, which minimizes lower plate area, and using a relatively thin substrate layer and thick air layer, which has the effect of minimizing lower plate geometry and maximizing lower plate distance to ground, respectively. Planar circuitry may all be placed above or below the substrate dielectric layer, and utilize connecting vias to connect the overlay plates, as shown in Figure 3 below, or the circuitry may be alternated above and below the dielectric layer to implement the overlay plates by interlacing the resonators, as shown in Figure 4 below.

Overlay Controls

Overlay require the Advanced User Panel. Set up a suspended substrate definition under the **Substrate** tab. Define a high pass or band pass filter that normally uses series capacitors. Check the **Use Series Caps** checkbox, then check the **Overlay Capacitance** checkbox. If resonator interlacing is desired, check **Interlaced Resonators**. When you synthesize your next filter, FilterSolutions will compute the necessary series capacitance for each leg, and also the width and length of the overlay geometry needed to realize the desired capacitance. The filter design schematic will annotate each series capacitor with the computed width and length geometry, and will show the physical overlay geometry in the layout view.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

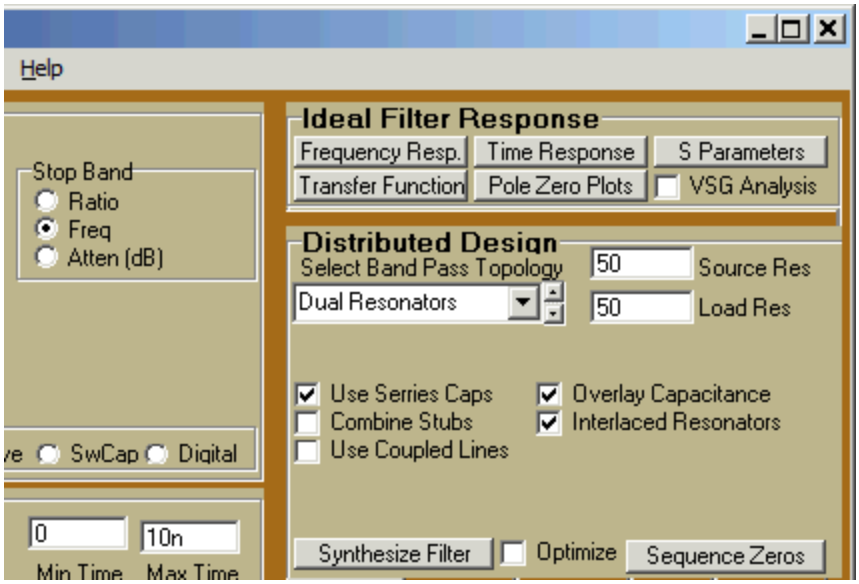


Figure 1: Overlay Setup in the Advanced User Panel

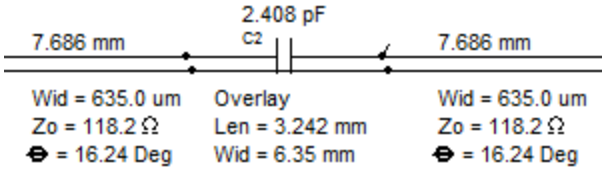


Figure 2: Overlay Width and Length Geometry in the Schematic View

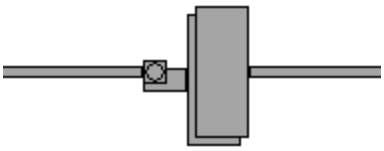


Figure 3: Overlay With Connecting Via Physical Appearance in the Layout View

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

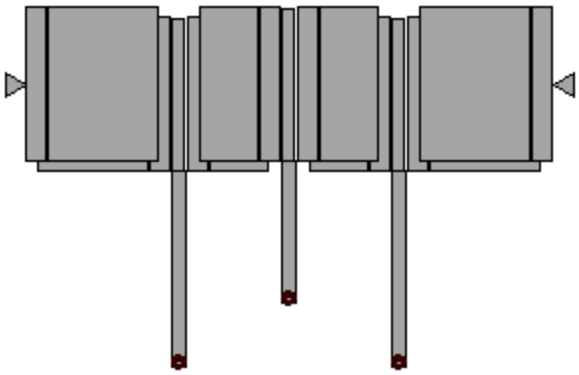


Figure 4: Overlay With Interlaced Resonators Physical Appearance in the Layout View

Optimizing the Frequency Response

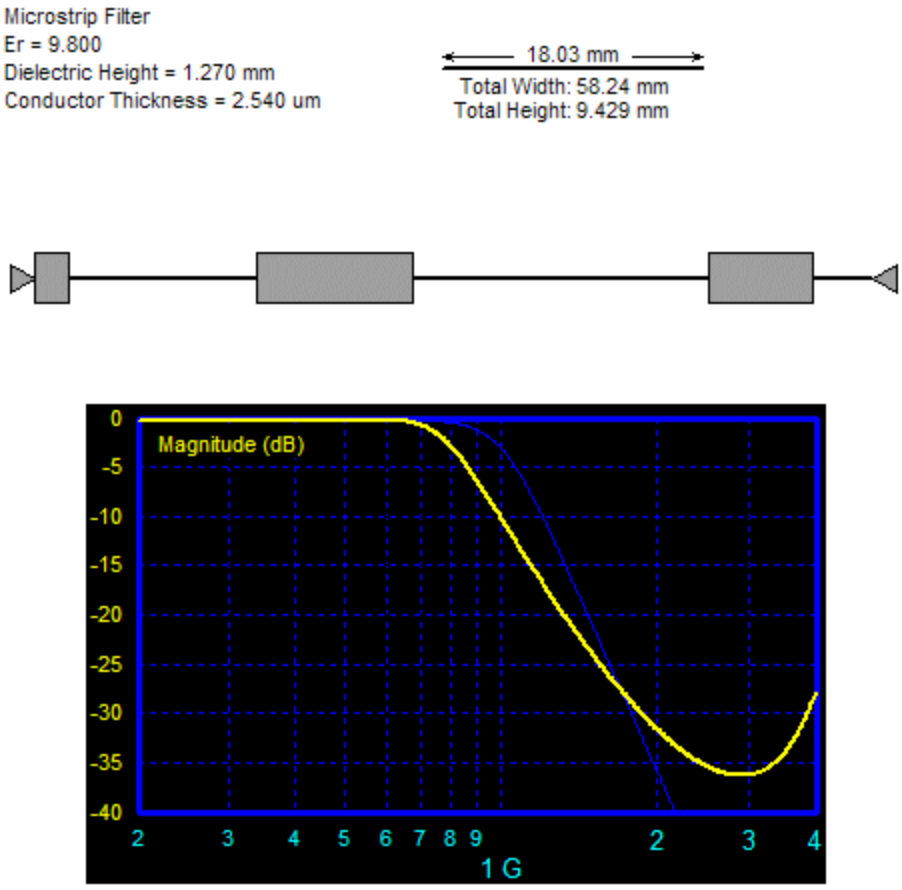
The synthesis process for transmission line filters frequently introduces errors into the frequency response, and strip line and micro strip parasitics further degrade the filter performance. FilterSolutions offers an optimization feature to help correct for this performance degradation. Selecting this option with the Quick Optimization checkbox just above the synthesis button invokes an optimization process that attempts to minimize the RMS error between the actual and ideal frequency responses about the pass band. It should be remembered that the frequency response will never perfectly match the ideal. The error can only be minimized.

> **Note:** Quick Optimization function is superficial and not meant to be robust, but should be used when it improves performance. Use AEDT optimizers for more robust optimization.

Example:

A segment only 1GHz 6^{th} order low pass Butterworth design tends to undercut the -3dB cutoff frequency as shown below.

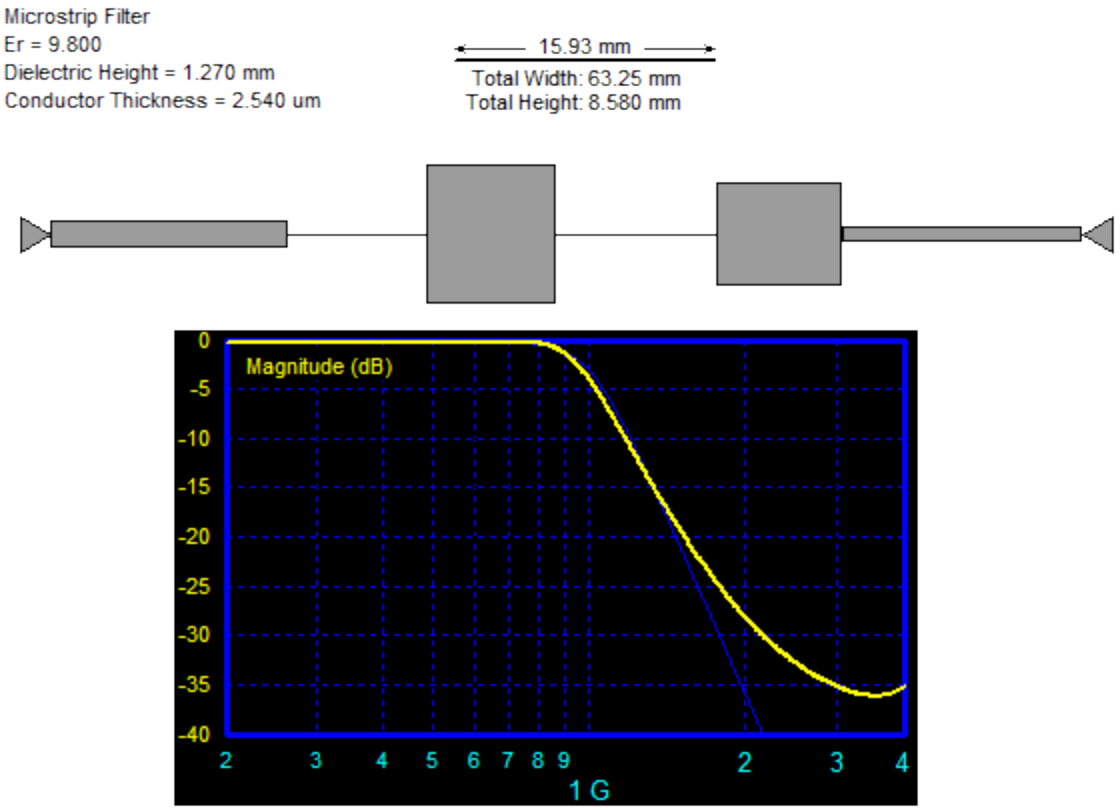
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Segment Only Butterworth Filter With Synthesis and Parasitic Errors

Selecting the Optimization button modifies the filter in such a manner so as to minimize the error in the pass band, and places the cutoff frequency closer to the desired -3dB, as shown below.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Segment Only Butterworth Filter With Optimized Pass Band

Setting Geometry Limits

Frequently, the width and gap computations produce undesirable geometries. For band pass filters, this can be mitigated with the use of taps and pins, but sometimes even this does not produce desired geometries. FilterSolutions provides an option to limit width and gap dimensions. When a width or gap is calculated to be outside the selected limits, FilterSolutions will force them back into the desired range. This, of course, produces errors in the filter frequency response. Some of these errors may be optimized away with the use of the Optimization function.

To invoke the geometry limit function, select **Real Parameters**, and **Limits**. Enter desired limits, and be sure that **Apply Limits** is checked when synthesizing filters.

Adjusting Line Lengths to Maintain Equivalent Impedance

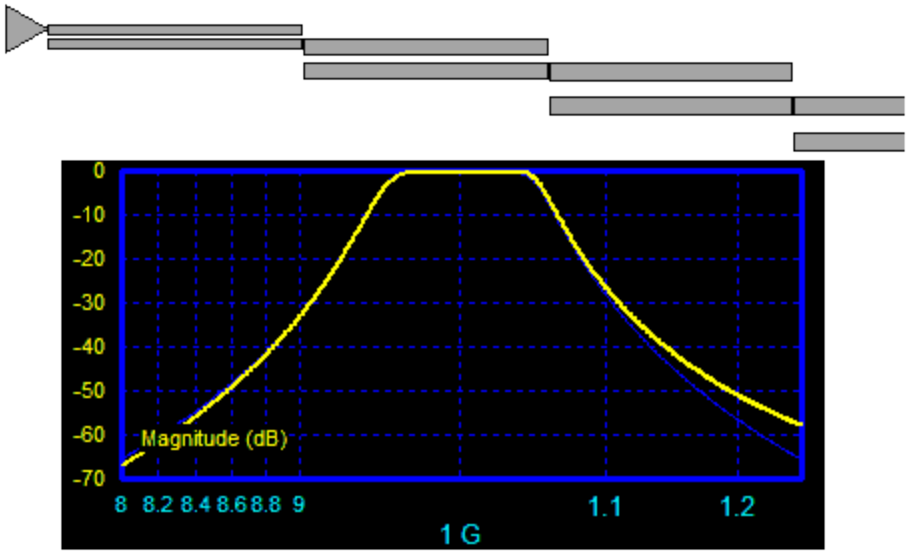
Transmission lines that simulate inductors and capacitors may have their lengths adjusted to maintain equivalent impedance at the cutoff frequency. This has the advantage of minimizing

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

error in the passband, but tends to increase error in the stop band. To invoke this function, simply check the **Adjust length on Limit** check box.

Example:

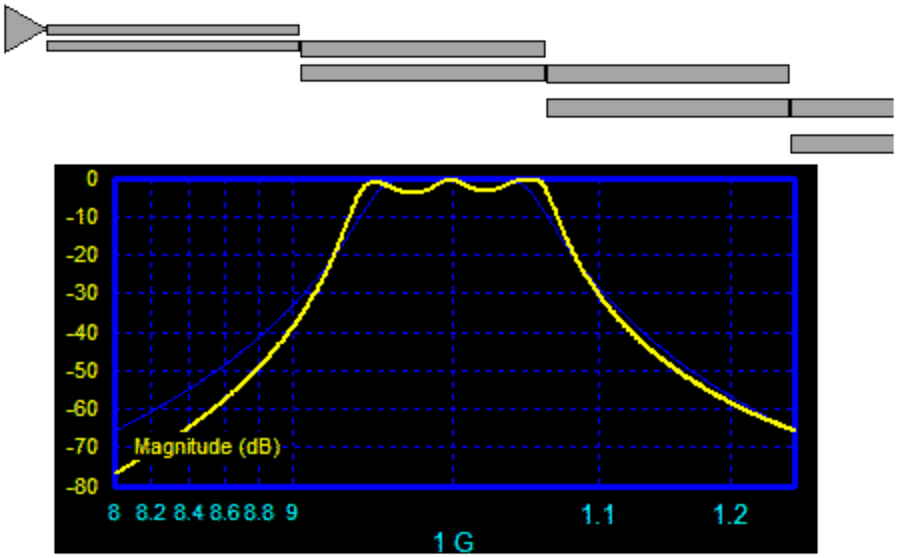
The following 1GHz parallel edge coupled filter has the common problem of narrow outer gaps of only 0.5 um:



Parallel Coupled Filter With Small Outer Gaps

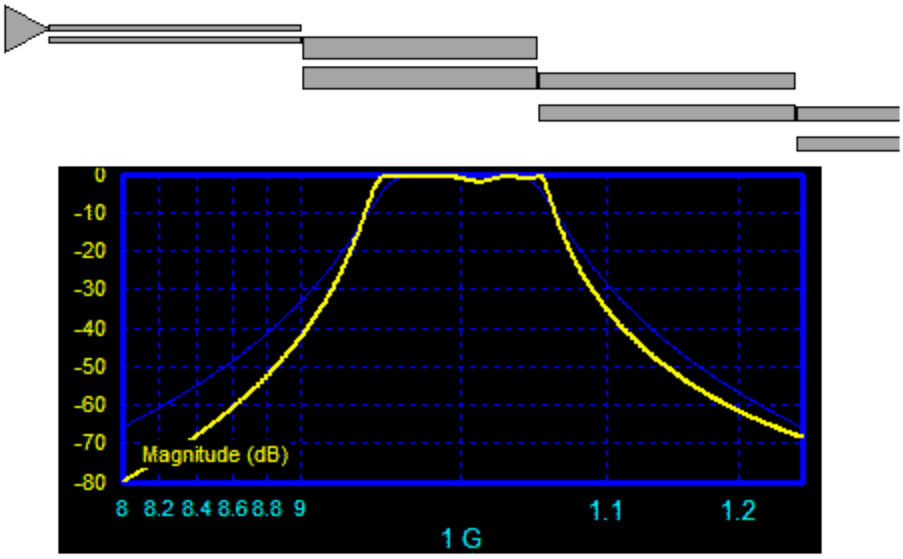
If the minimum gap size you can manufacture is 500 um, you set the minimum gap to 500 um, but the frequency response deteriorates as shown:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Parallel Coupled Filter Outer Gaps Minimum Limits Set

Selecting "Optimize" restores most of the frequency response, as shown below:

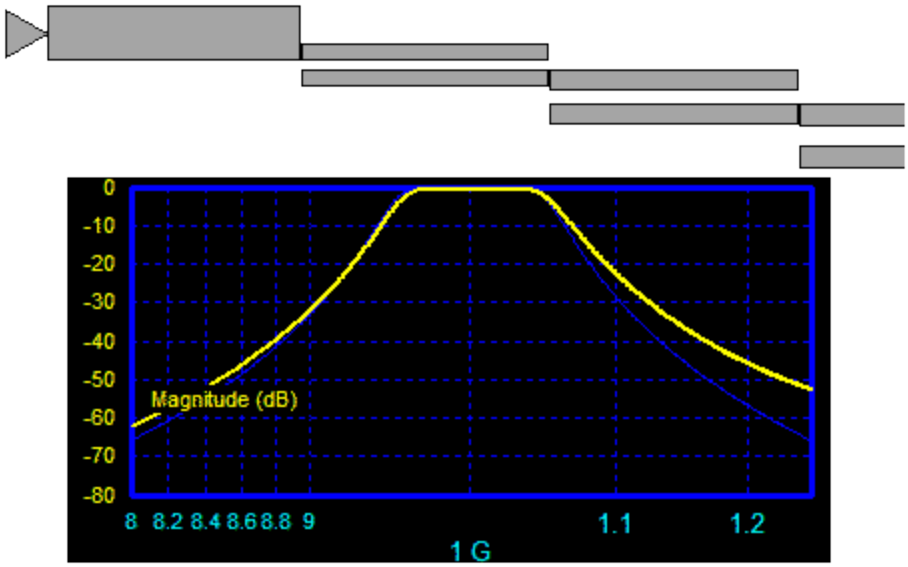


Parallel Coupled Filter Outer Gaps Minimum Limits Set

and Optimized Frequency Response

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Alternatively, the **Tapped** or **Pinned** option could be selected on the schematic control panel to eliminate the small gap by replacing the outer coupled lines with segments. This preserves the pass band fairly well, but reduces the stop band attenuation, as shown below for the Pinned option:



Parallel Coupled Filter With Outer Segments "Pinned"

Controlling Stub Widths

Microstrip and stripline stubs may become unmanageably wide. When this happens, a single wide stub may be split into two thinner parallel stubs with the use of the **Split Stubs** option, which available under the **Real Parameters** button in the tx line control panel when striplines or microstrips are selected.

When **Split stubs** is selected, the user may select the stub width as a ratio of dielectric heights, or stub angle in the case of radial or delta stubs, that the user wishes to split. All stub widths wider than this value will be split into two equivalent thinner parallel stubs. It is important to note that the two equivalent split stubs will perform slightly different that the original wide stub. This is due to parasitic effects resulting from the two parallel stubs will not be exactly the same s the parasitic effects from the original. For example, the Cross line function is not identical to a line Tee function, and the end the length of the end effect of the original stub is not the same as the sum of the length of the two parallel stubs.

The effect of splitting the stubs is easily seen in the layout view when **Layout** is selected at the top of a stripline or microstrip schematic.

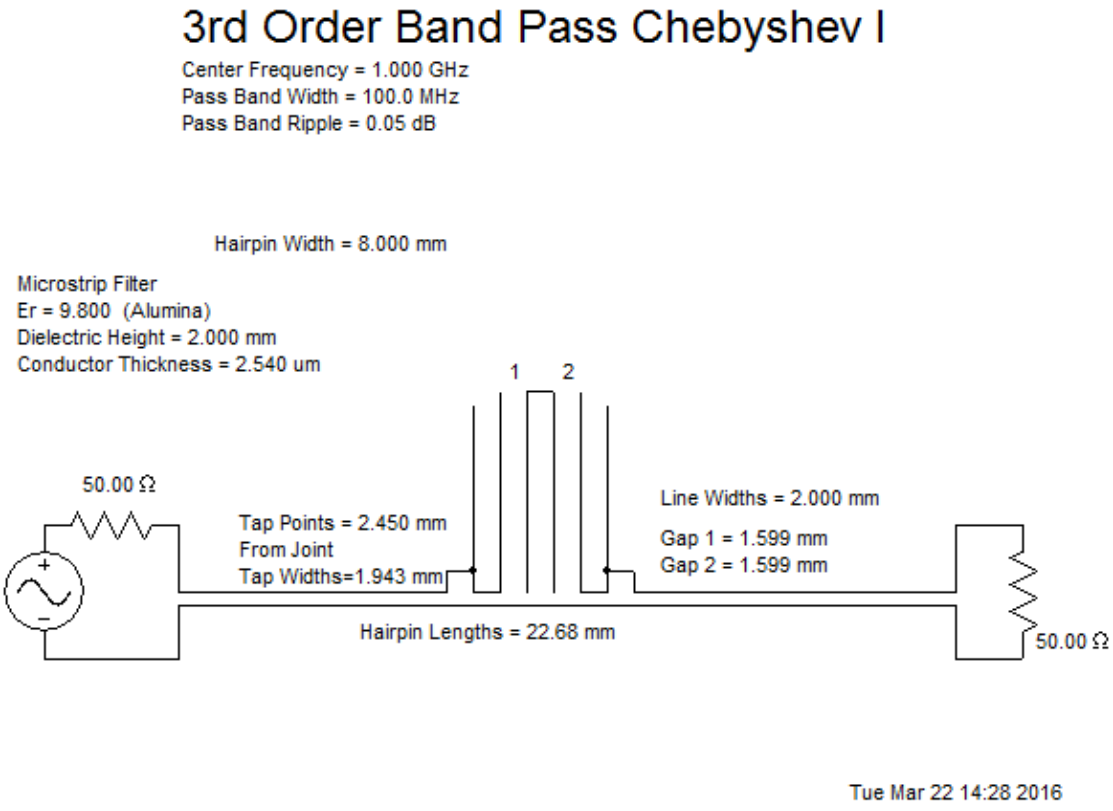
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Layouts and Schematic Views

FilterSolutions offers a layout view and a schematic view to aid in micro strip and strip line design interpretation (single filters only, not diplexers). The schematic view contains all the geometry information necessary to physically construct the filter. The layout view portrays the filter in its actual dimensions, but shows minimal geometric data. Taken together, a relatively easy geometric interpretation of the filter design may be made.

Schematic view:

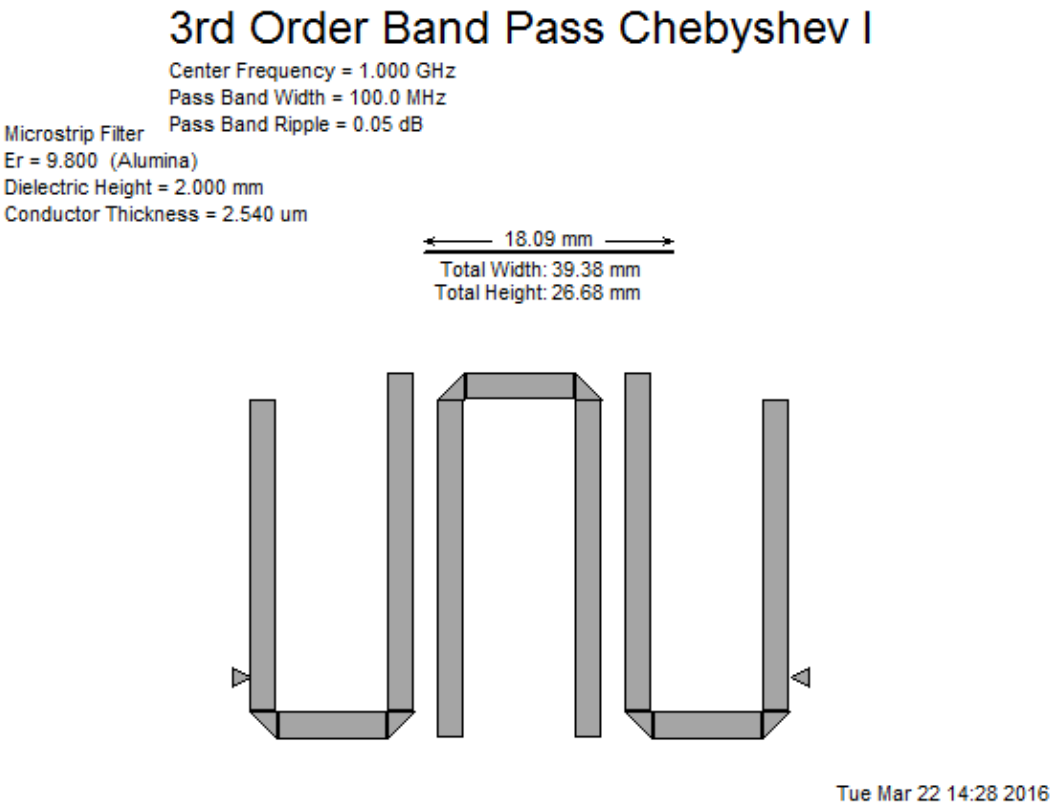
A micro strip hairpin filter will be used as an example. In the schematic view below, all the geometry information needed to physically construct the filter is included in the schematic. Selection of the **Other Info** checkbox replaces width and gap information with transmission line impedances.



Microstrip Schematic View

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If it is desired to view the physical geometry of the filter, **Layout** is selected, and the filter layout is shown, as in the example below. The physical layout of the filter along with total width and height information position of grounds and capacitors (if any), and node position are displayed. Tap points are taken to be the center of the taps.



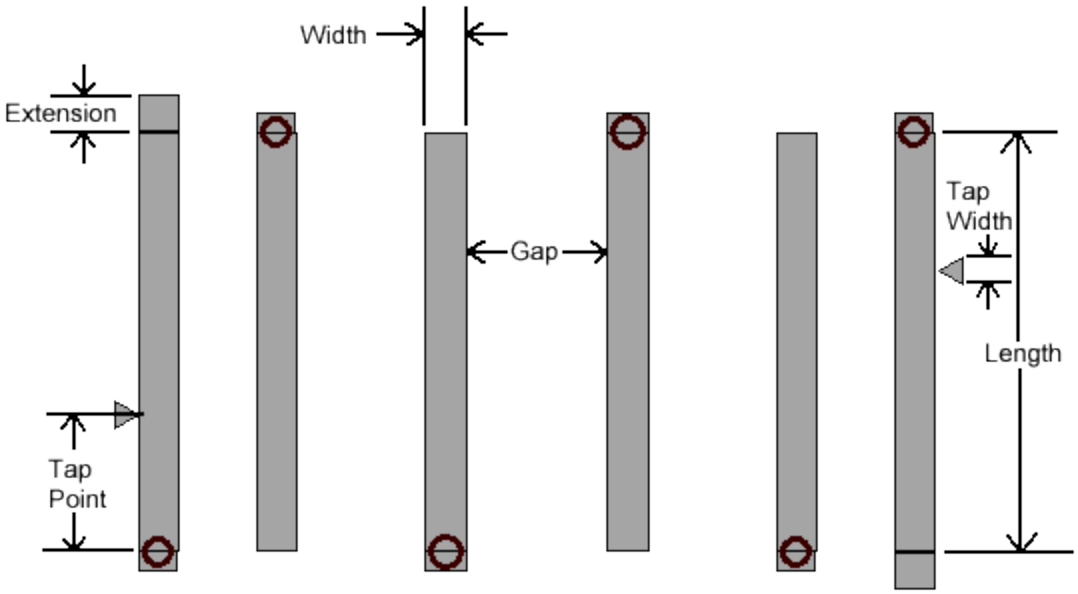
Microstrip Layout View

Layout Geometry Definitions

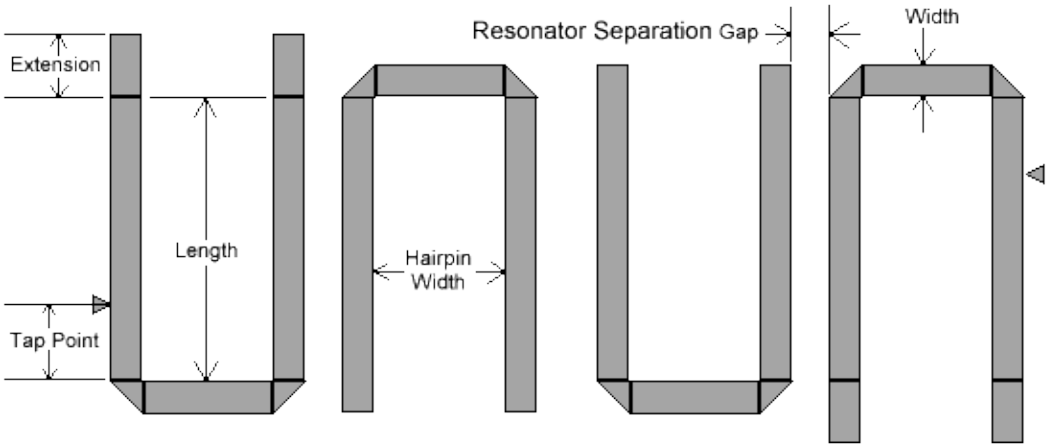
Multiconductors

Widths, gaps, and length definitions are as shown in Figures 1 below.

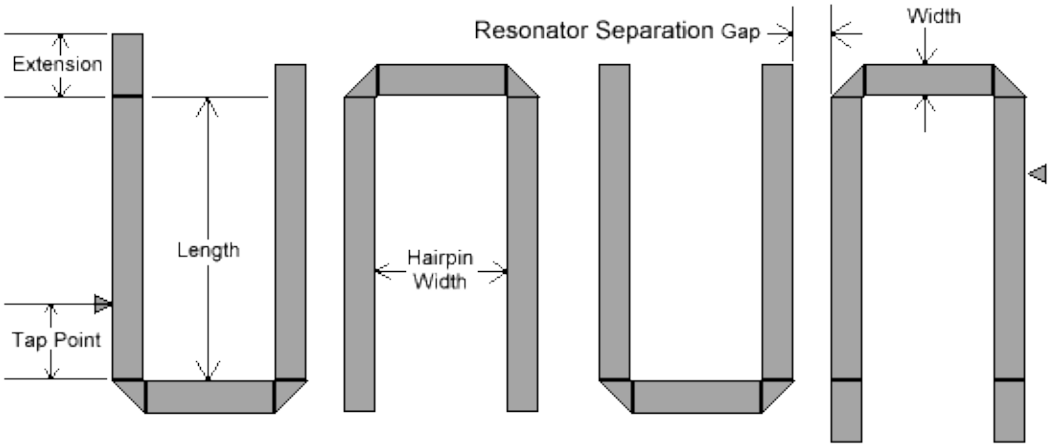
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

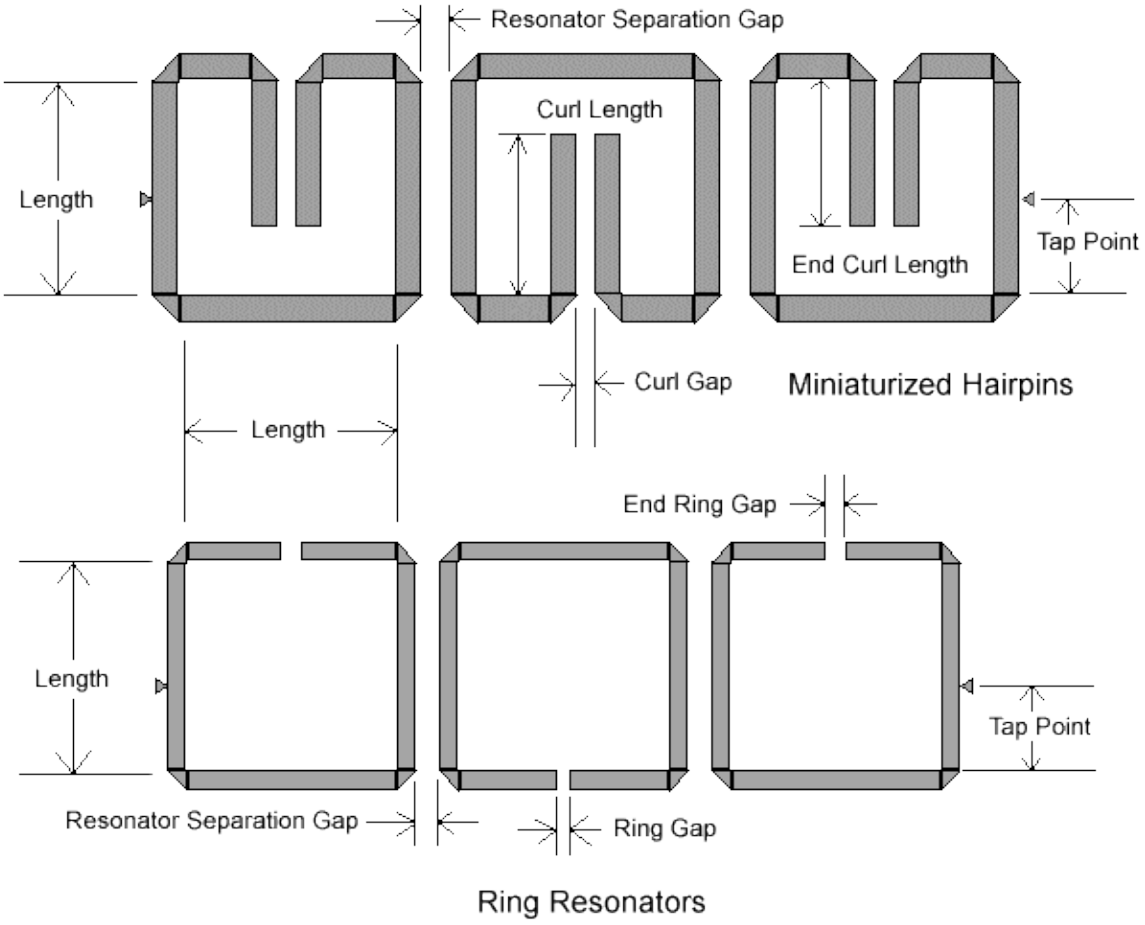


Hairpin Tuning Type 1



Hairpin Tuning Type 2

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

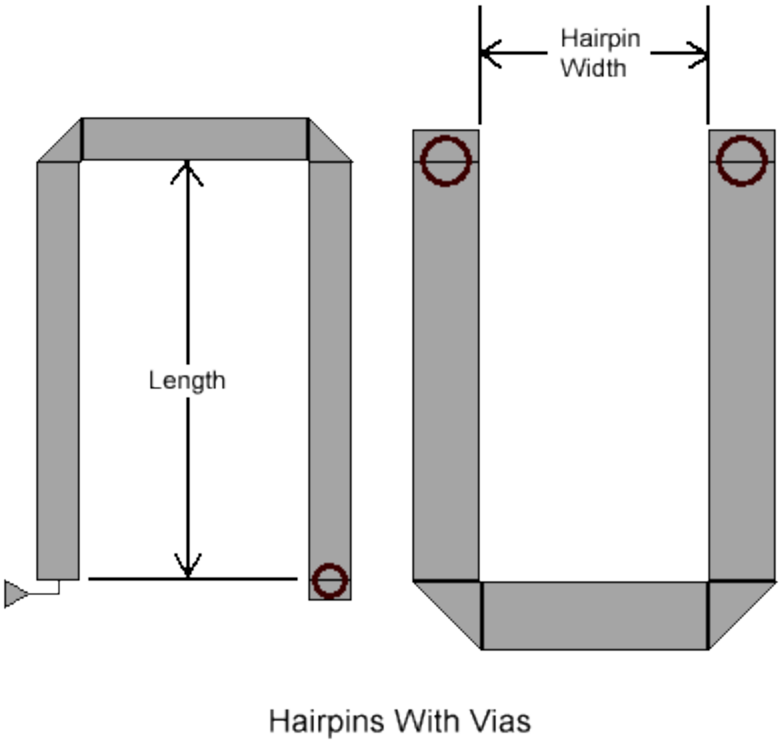


Figure 1: Multiconductor Geometry Definitions

Coupled Resonators

Coupled Resonator geometry definitions are as shown in Figure 2 below:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

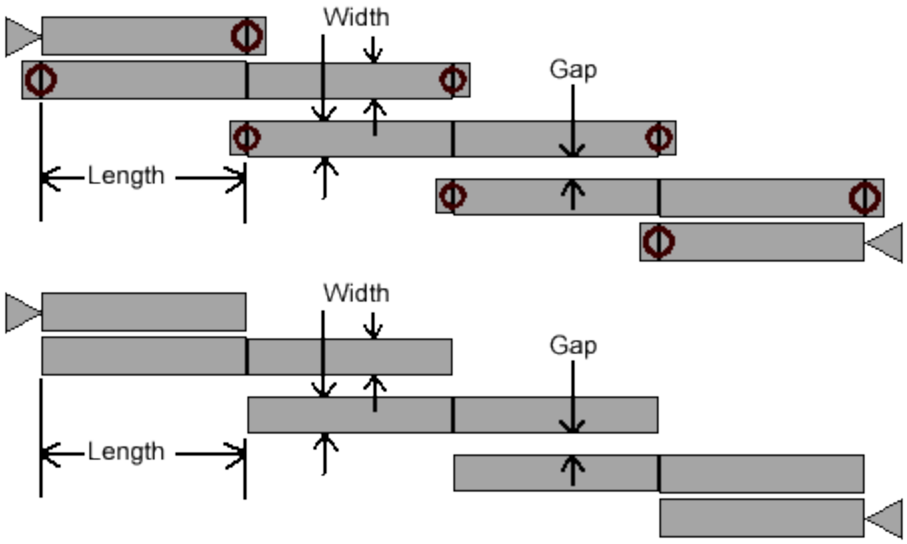
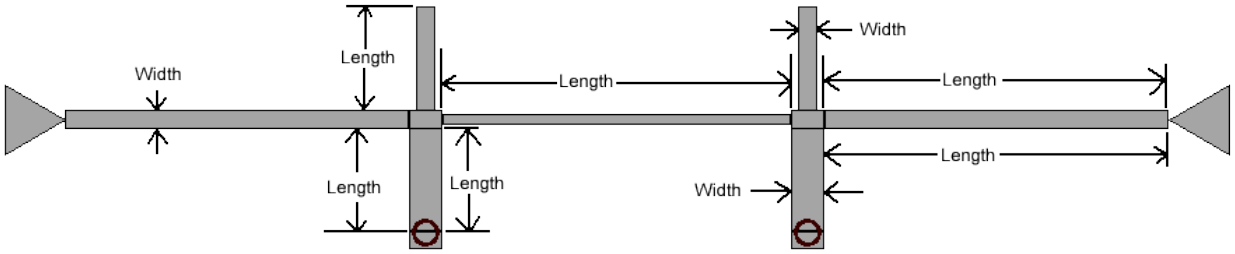


Figure 2: Coupled Resonator Geometry Definitions

Ladder Structures

Width and length definitions are as shown in Figure 3 below:



Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

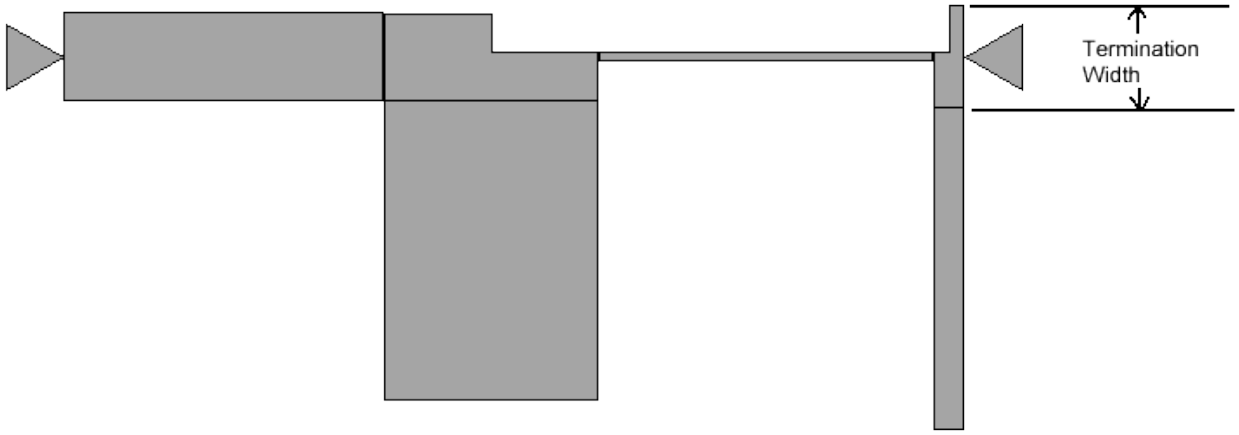
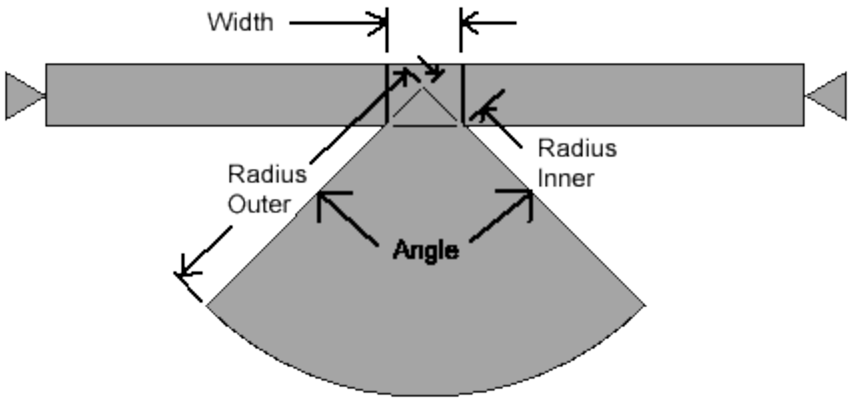


Figure 3: Ladder Geometry Definitions

Radial and Delta Stub Geometry

Radial and delta stub geometries are as shown on Figure 4 below:



Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

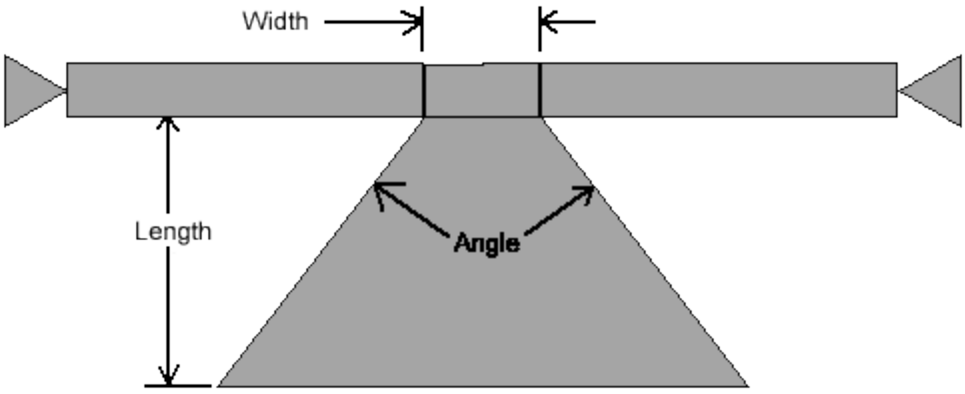


Figure 4: Radialand Delta Stub Geometries

Suspended Substrate Overlay Capacitors

Overlay consists of broadside coupled lines over and under the dielectric layer of a suspended substrate. Overlay geometries are as shown on Figure 5 below:

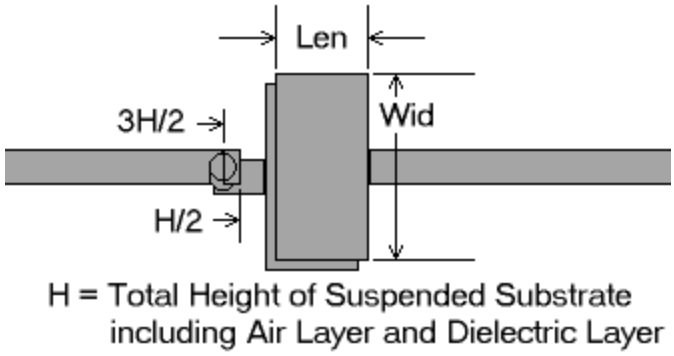


Figure 5: Overlay Capacitor Geometries

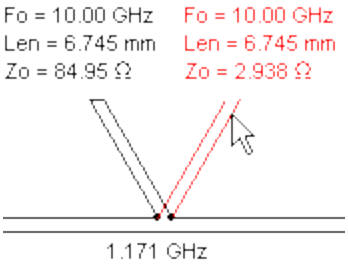
Modify Distributed Filters

Individual distributed filter transmission lines may be modified, and the effect displayed in real time with the use of the left mouse key. Stubs and connecting segments may be updated to selected values, by percent change, or random value. The random values feature may be used to perform Monte Carlo analysis.

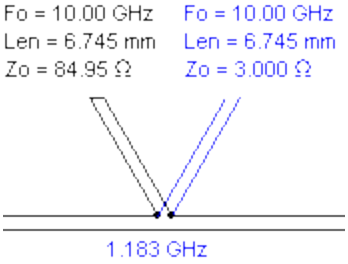
To modify a transmission line stub or connecting segment:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

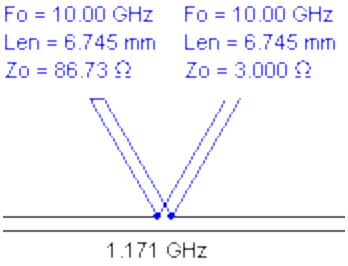
Pass the cursor over the stub or segment you desire to modify. It will turn red. Left click the mouse to change the transmission line or stub parameters.



Elements and resonant frequencies are displayed in blue after they are modified.



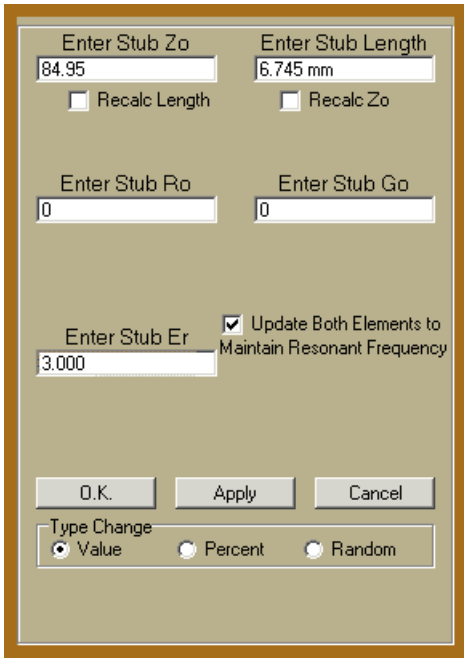
Unmodified resonant frequencies are displayed in black.



The colors of red and blue may be changed with the Color Control Panel.

Click the left mouse key to bring up the change control panel

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The Change control panel entries are defined in the following table:

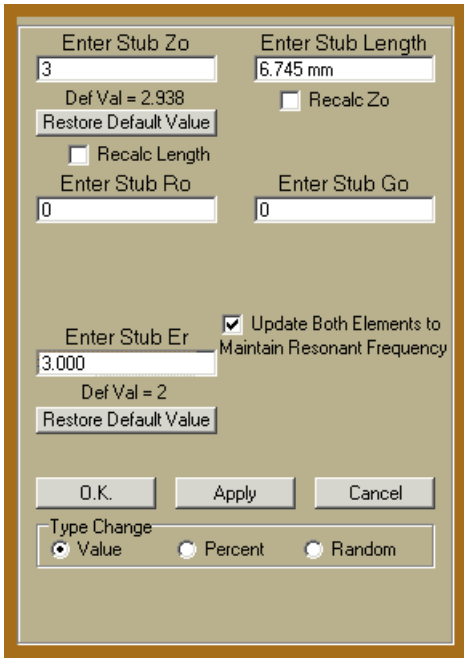
Enter Stub **Zo, Length, Ro, Go, Er**	Enter the desired new value of the selected stub. If a diplexer noncompensation element is being modified, both of the compensation elements will automatically update to account for the new compensation requirements. This automatic compensation update discontinues after either one of the compensation elements are manually modified.
Recalc Length	When Zo of a stub is changed, Check this box to automatically adjust the length of the stub to keep the equivalent capacitance or inductance the same. **Not displayed for Combined Stubs.**

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Recalc Zo	When the length of a stub is changed, Check this box to automatically adjust the Zo of the stub to keep the equivalent capacitance or inductance the same. **Not displayed for Combined Stubs.**
Update Both Elements to Maintain Resonant Frequency	Displayed only for stub pairs derived from LC tank elements When the one of the two stub pairs are selected, checking this box will cause the other stub to automatically update to a value that will maintain the resonant frequency of the pair. Uncheck this box to update the stubs independently.
Update All Inductor Q s	Check this box to update all inductors or capacitors in the circuit to the value entered in the Enter Inductor Q box.
OK	Closes the control panel and updates the element value in the circuit.
Apply	Update circuit and graphical displays in real time without closing the Change Control Panel.
Cancel	Closes the control panel and does not update the element value.
Type Change	Specify to update element to a fixed value, percent change in value, or random value.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

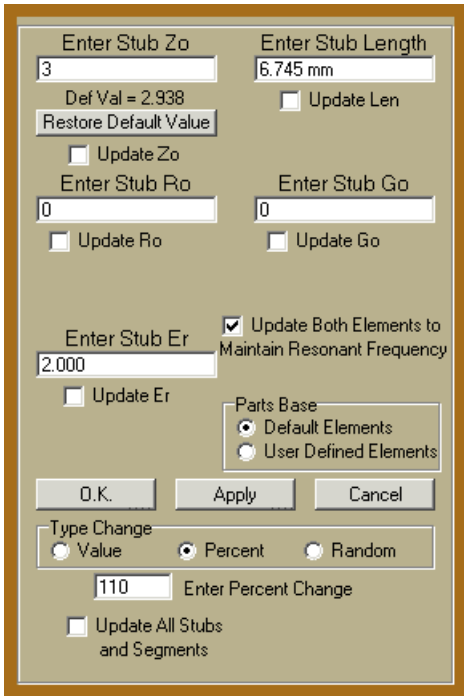
When a previously changed stub is selected for change, the default value of the stub is displayed, and the panel looks like this.



Selecting **Restore Default Value** sets the component back to the original value.

It is possible to specify the percent change in the stub value by selecting **Percent** at the bottom of the panel. Check the **Update** checkbox below each parameter it is desired to update. The **Update** checkbox at the bottom will update all stubs by the same percentage. The **Parts Base** selection box allows one to base the percent computation on the current stub values or the default stub values.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

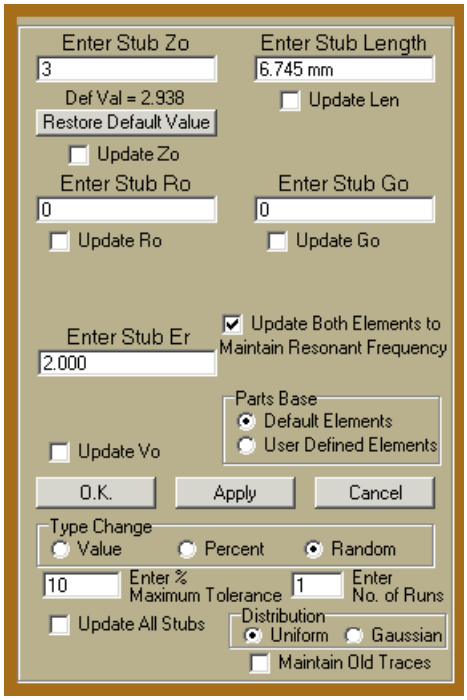


Monte Carlo Analysis

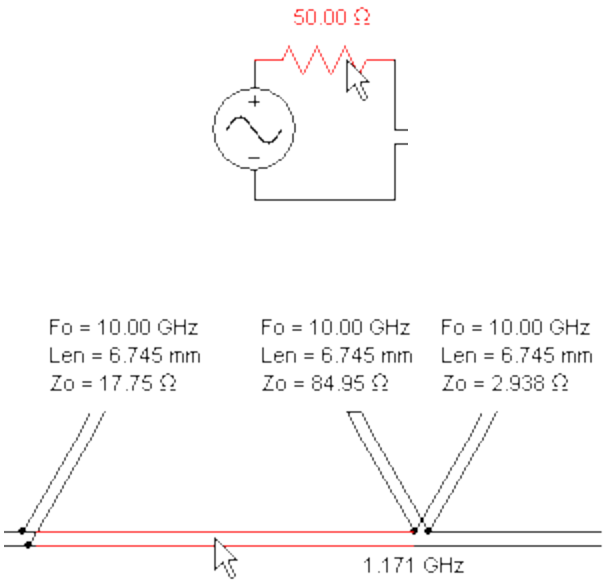
A Monte Carlo sensitivity analysis may be performed with the use of the "Random" selection. Enter the desired standard deviation in percent, and FilterSolutions will automatically update the selected stub parameters, or the selected stub or all stubs, by a random number from a Uniform or Gaussian (Normal) distribution. The mean of the distribution is 0%, and the user enters the maximum tolerance of the Uniform distribution, or the standard deviation of the Gaussian distribution.

All graphical analysis traces are automatically updated each time the **Applied** button is depressed. Check the **Maintain Old Traces** checkbox to prevent old graphical traces from being overwritten.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To modify other transmission line parameters, place the cursor of a segment or resistor, as shown.



Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The change control panel will allow one to change resistor values, transmission line values, or segment lengths. When a segment length is zero, the length of the segment is not displayed.

Multiconductor Filters

Combline, Interdigital, and Hairpin filters may be edited by selecting the multiconductor symbol in the schematic. All bar or hairpin widths (or impedances in the case of RLGC filters) are adjustable. The multiconductor length, extensions (if any) and tap points (if any) are individually adjustable. (Tap points are taken to be the center of the taps.)

Monte Carlo analysis is not supported for multiconductor filters.

S-Parameter and Spice Model Imports

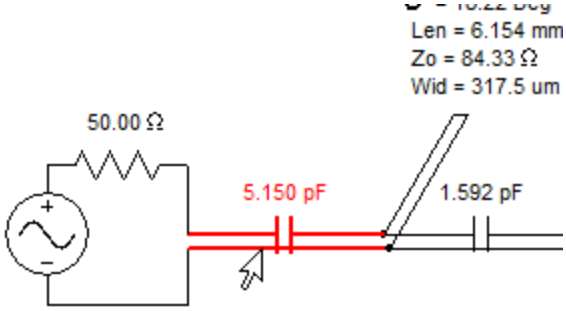
FilterSolutions permits you to import S-Parameter from Touchstone format files and Spice model files in standard Spice format, both by individual file selection and by automatic nearest match. When you do this, the original element impedance is replaced with the S Parameter or Spice model defined impedance for each frequency. Interpolation is employed between the frequencies for S Parameter files. Any passive linear element S-Parameter or Spice model file is acceptable. Active element S-Parameters or Spice models, such as a two pin amplifier, will not be accepted for importation. Elements defined with S-Parameter or Spice model files will continue to use the nominal value for sensitivity analysis and Monte Carlo analysis.

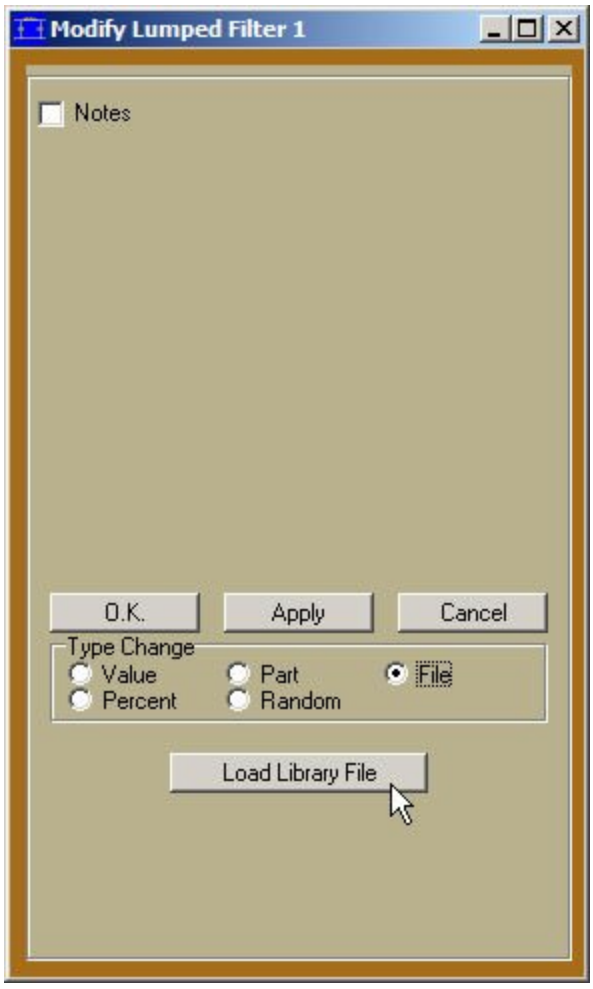
S-Parameter Files and Spice parameters for Coilcraft inductor downloads may be found at: www.coilcraft.com/models.cfm

Once an S Parameter or Spice model is imported into an element, all analysis performed on the associated filter will use the imported S-Parameters or Spice models instead of the element impedance. S-Parameter or Spice model files that are electrically asymmetrical will post pin numbers on the schematic.

To import a specific S Parameter or Spice models files, left click on the desired element, select, **File** in the popup Change Control Panel, select the desired file, and **OK**. The name of the S Parameter file then appears above the element, as shown below in Figure 1.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

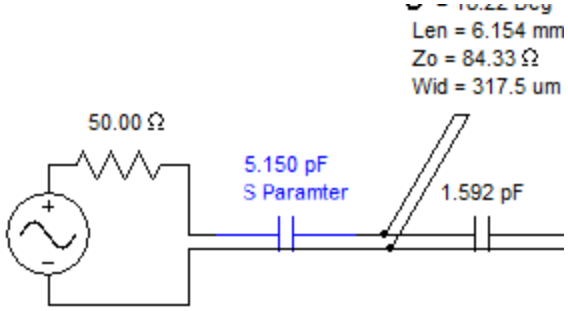


Figure 1: Selecting a Desired S-Parameter or Spice Model File

Update to Nearest Matching S-Parameter or Spice Model File

To update to the nearest matching S-Parameter or Spice model file, select **Part** and **File**, as shown below, then select, **Browse** to select the desired directory containing the S-Parameter or Spice model files, as shown below in Figure 2. Select, **Menu** to bring up the manufacturer and family pop-up menu. Select any file in the Browse directory to select and define the desired directory of S-Parameter files. Once selected, FilterSolutions will remember the directory until another directory is selected by the user. Check, **All Inductors** to update all the inductors in the filter. Care should be taken to insure that all S-Parameter and Spice model files are from components valid for the frequency and power range of the designed application.

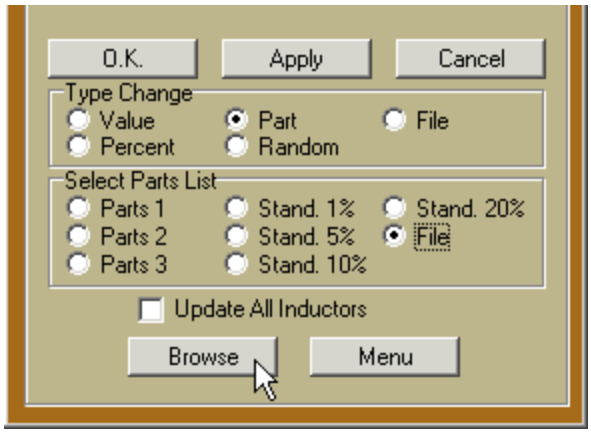


Figure 2: Update to the Nearest Matching S-Parameter or Spice Model File

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

S-Parameter and Spice Model Exports

S-Parameter or Spice model files that have previously been imported can be exported. Automatic S-Parameter and Spice model selection may make it difficult to determine which S-Parameter or Spice model file is used in any particular element. Left clicking an element that is loaded with a S-Parameter or Spice model file pulls up the element Change Control Panel which may be used to export the specific S-Parameter or Spice model file that is loaded into the selected element. Select **File** to bring up the Touchstone features, then save the element S-Parameter or copy it into the Windows clipboard for pasting elsewhere, as shown in Figure 1.

Additionally, S-Parameter and Spice model files may be saved or viewed by selecting the appropriate button.

In all cases, the original S-Parameter or Spice model file will be saved or copied, including comments, header, and numerical resolution.

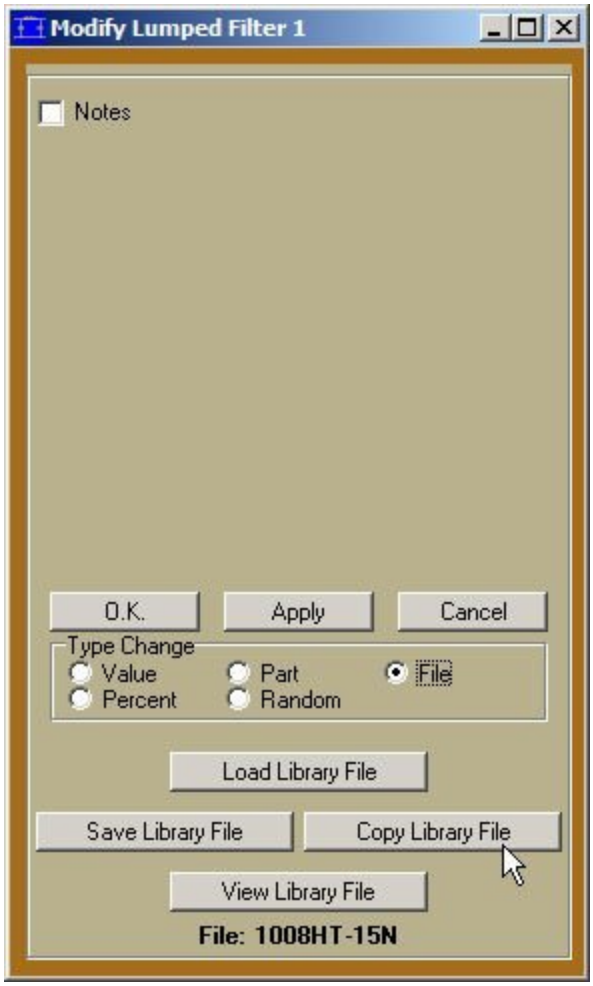


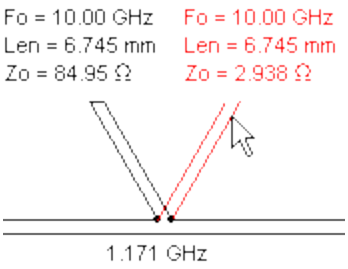
Figure 1. Saving and Copying an Element S-Parameter or Spice Model file

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

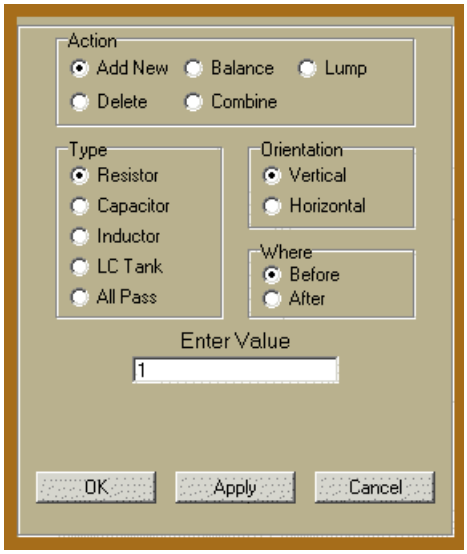
Alter Transmission Line Designs

Transmission lines may be subsequently altered after the initial design with the use of the right mouse key, and the effect of the design alteration displayed in real time. Stubs may be added, deleted, balanced, unbalanced, combined, separated, converted to lumped elements or converted back to stubs.

Pass the cursor over the stub or segment you desire to modify. It will turn red. Right click the mouse to change the transmission line or stub parameters.



Right click the mouse key to bring up the following panel:



Add New	Add a new lumped element to the transmission line filter. The element may be subsequently substituted with a transmission

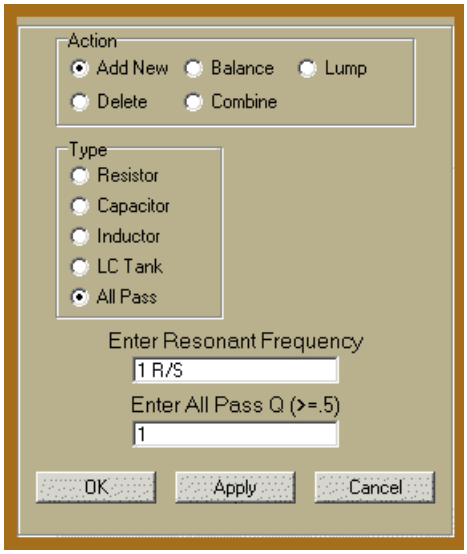
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	line stub. Enter the desired value into the "Add New" box.
Delete	Delete the selected element or stub.
Balance	Balance the selected element or stub. Conversely, unbalance a balanced element of stub.
Combine	Combine LC pair stub equivalent resonators into single stub resonators.
Lump	Convert transmission line stubs into lumped elements. Conversely, convert lumped elements into transmission line stubs.

Adding All Pass Sections

All pass section may be added following a source resistor or preceding a termination resistor. The change panel allows the selection of a resonant frequency and a Q. Enter Q=0.5 to add first order all pass section.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



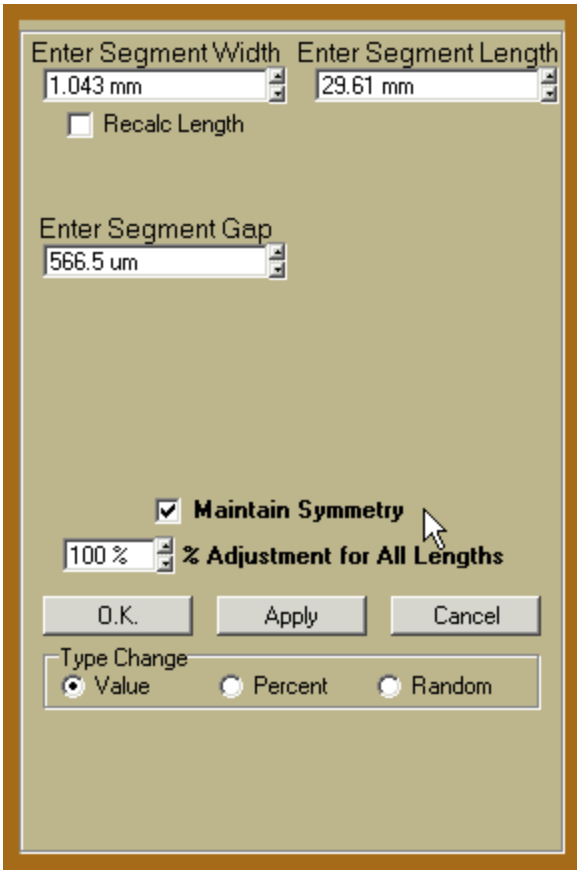
Segments Between Stubs

Right clicking on a segment between stubs may convert the segment to a lumped equivalent model, if the lumped model exists. Generally, a lumped model will exist for all nonzero length segments that are not used for coupling resonator filters. Balance RLGC segments do not support lumped equivalent models.

Tuning Planar Filters

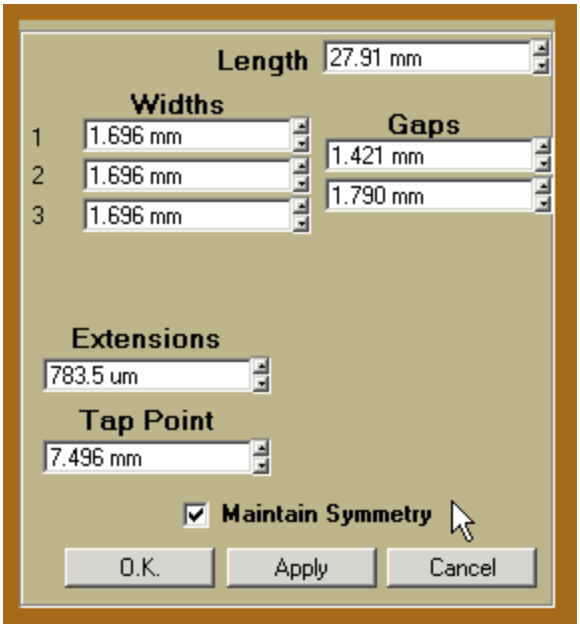
Planar filters may be tuned with the use of the Left-Click to edit or Edit button to bring up the change control panel, shown below:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Tuning Segmented Planar Filters

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Tuning Multiconductor Filters

Updates are applied upon selection of **OK**, **Apply**, or one of the up/down arrows. Physical filter symmetry may be maintained, or not maintained, as desired. **Adjust for All Lengths** will update ALL filter segments and stubs with the effect of tuning the center or cutoff frequency of the filter.

Third Party Tuning

In addition to direct tuning, tuning entries are created in AEDT HFSS, HFSS 3D Layout, and Ansys Circuits exported filters.

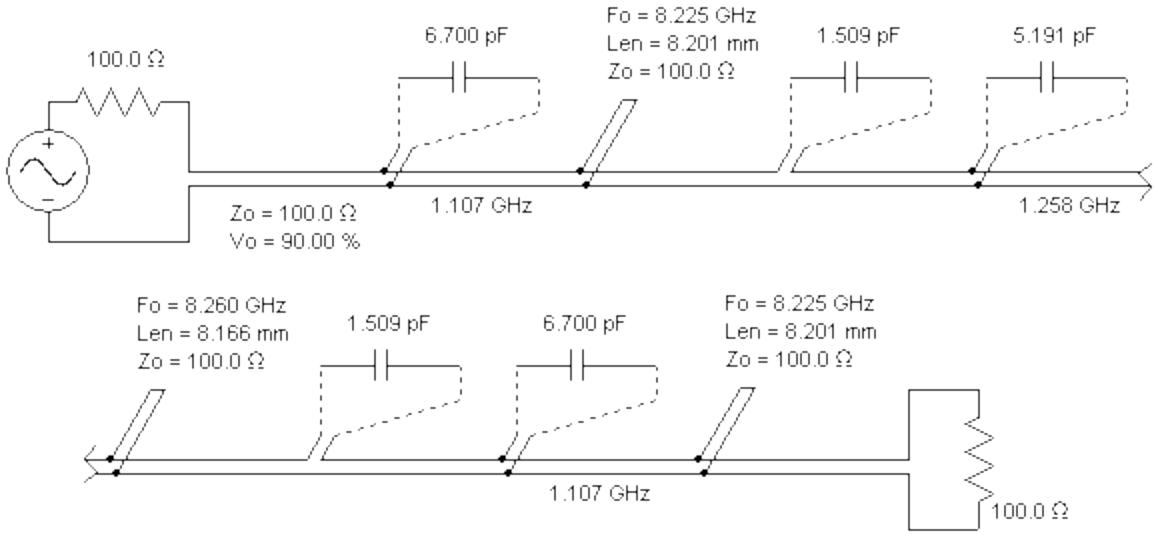
Composite Lumped/Line Filters

FilterSolutions offers the flexibility to create the filters with a mixture of lumped and transmission line stubs. This is a flexible method of substituting only the low Q lumped elements with transmission line stubs, and using lumped elements when high Q parts are available.

To create such a composite filter, create a transmission line filter first that contains the desired filter traits, then **right click the stubs to be replaced with lumped elements**. A selection is available in the change control panel to lump all shorted or all open stubs with a single selection if desired. If a resonator consists of a single stub, it must first be separated into two stubs before one or both elements can be substituted with lumped elements.

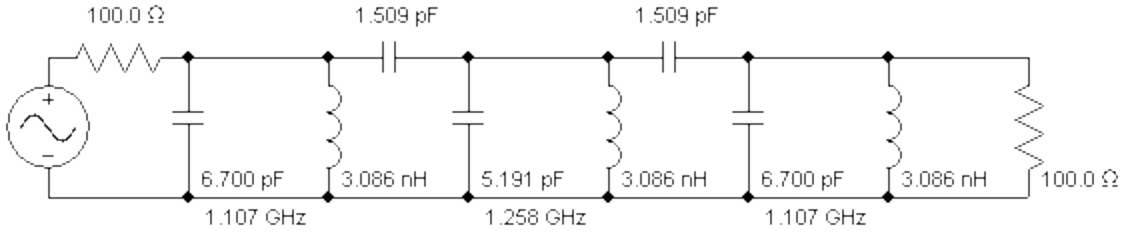
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A coupled resonator band pass filter is shown below with open stubs replaced with lumped capacitors. The shorted stubs remain as substitutions for inductors. The impedance of the shorted stubs was manually set to 100 Ohms with the use of the change control panel.



Coupled Resonator Band Pass Filter With Transmission Lines as Inductors

The Equivalent lump filter is shown below.



Equivalent Coupled Resonator Band Pass Filter with Lumped Elements

Coupled Resonator Band Pass Filters

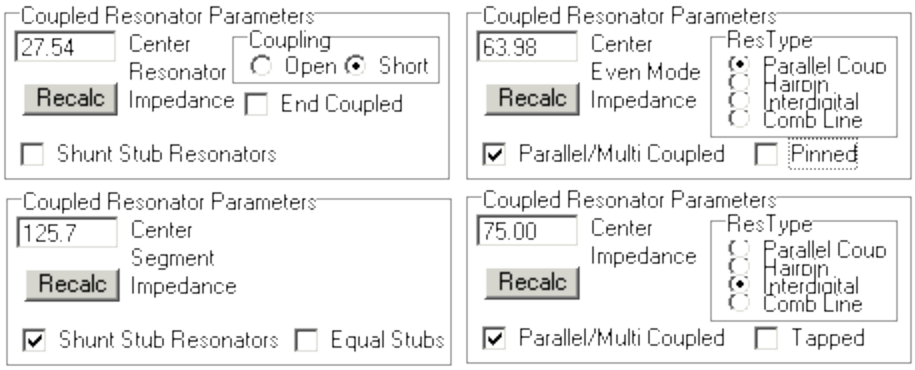
Coupled resonator band pass filters are possible with transmission lines. See the "Passive Filter", "Coupled Resonator Band Pass Filter" section for the theory, synthesis, and use of coupled resonator filters. This section will build upon those concepts.

The process to create a transmission line coupled resonator filter is identical to creating one with passive filters. Create a Gaussian, Bessel, Butterworth, Legendre, or Chebyshev I band pass

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

filter, and select the **Coupled Resonator** checkbox that is located on top of the display. If a coupled resonator filter is realizable, it will be displayed. Otherwise, and error message will result.

The Coupled Resonator Control Panel provides some flexibility with the coupled resonator design. It will be displayed in the top right of the display.



Center Impedance	The value of the center stub impedance, center segment impedance, center even mode impedance, or center multiconductor line impedance.
Parallel/Multi Coupled	Displayed in "Use Coupled Lines" mode only. Check to display parallel coupled resonator filters or multiconductor filters. IF left unchecked, lumped equivalent coupled resonator filters are displayed. This checkbox is disabled and forced checked when lumped equivalent coupled resonator filters cannot be calculated.
Coupling	Displayed for lumped coupled resonator simulations. Select the desired type of line between resonators. "Open" corresponds to passive capacitor. "Short" corresponds to passive Inductors. Not visible when parallel coupled or shunt (or series) resonators are selected.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

End Coupled	Displayed for lumped coupled resonator simulations. Check if you desire the filters to have coupling stubs on each end of the filter.
Equal Stubs	Displayed for Butterworth and Chebyshev filters when Shunt Stub Resonators is checked. Check to create shunt stub resonators with all stubs identical.
Pinned	Displayed for "Coupled Line" checked on "First Element Shunt" schematics or "Parallel Coup" selected. Check to replace narrow end coupling gaps with segments or hairpins.
Recalc	Update the filter with the entered center resonator or inductor impedance.
Res Type	Displayed for "Parallel/MultiCoupled" Checked only. Select parallel edge coupled filters or the multiconductor filters of Hairpin, Interdigital, , or Combline.
Shunt (or Series) Stub Resonators	Displayed in "Use Segments" mode and not "Use Coupled Lines" mode only. Check to display shunt (or series) stub resonator filters.
Tapped	Displayed for "Coupled Line" checked and "Parallel Coup" not selected on "First Element Shunt" schematics. Check to replace narrow end coupling gaps with

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	taps. Tap points are taken to be the center of the taps.

Transmission Line Netlists

To help retrieve the designed transmission line filters, FilterSolutions will output the filter in a netlist. From the control bar of any displayed passive circuit, select "Netlist", and the netlist box will be displayed on the left side of the filter window. The number of significant digits used on the netlist will match the number in the "Digs" box on the right side of the controls bar. The netlist contains entries for AC and Transient analysis, including graph scaling. An AC source is used for the AC analysis. A Pulse source is installed, but commented out for the transient analysis. To run a transient analysis it is necessary to comment out the AC source, and remove the comment from the Pulse source.

Ideal Lines

Ideal transmission lines are modeled in the net list by defining the impedance and quarter wavelength frequency. The quarter wavelength frequency is calculated by: F(1/4) = Vo/(4.*Len) where "F" is the quarter wavelength frequency in Hertz, "Vo" is the speed of propagation in units/second, and "Len" is the length of the transmission line in the same units.

In the netlist, impedance is identified as "Zo=", quarter wavelength is identified as "F=" and the wave length ratio (1/4) is identified as "NL=0.25".

Striplines and Microstrips

No standard textual spice format exists for striplines and microstrips have, so FilterSolutions uses the syntax, "ST" and "MT" for single lines, and "SX" and "MX" for coupled lines. Line geometry is defined by easy to understand labels, such is "Len=" for length, "Thick=" for conductor thickness, etc. The netlist is accurate and executable, but special software is required to execute it due the unique textual definitions that are used. Netlist simulations on standard spice reading software generally must be in RLGC format. Invalid width and gap values are replaced with a "=*****" entry.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Real Lines

Real transmission lines are modeled in the net list by defining the length (Len=), inductance (L=), capacitance (C=), resistance (R=), and conductance (G=) of the line. L, C, R, and G are defined in value per unit of length of the line. The units of length for L, C, R, and G is the same as that used on the length (Len) specification.

Coupled Lines

Coupled transmission lines are shown in the netlist using the syntax, "Xcoup" followed by Zoo and Zoe for odd and even mode impedances, or Roo, Roe, Goo, and Goe for odd and even mode resistance and capacitance. Since "Xcoup" is not a standard spice function, it is commented out and replaced with a spice realizable transmission line model when possible. If this is not possible, then Xcoup is not commented out, and the netlist will not execute on standard spice programs.

Lumped Elements

See the section on **Passive Netlists** for descriptions of lumped element, especially lumped elements with finite Q.

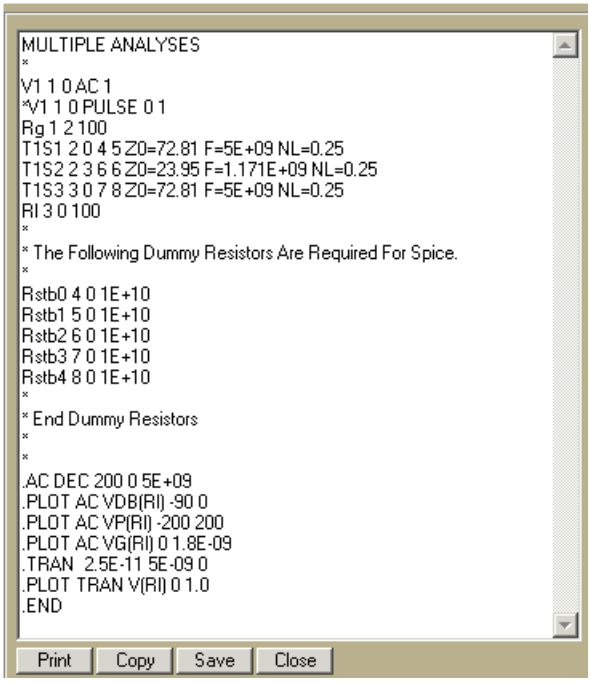
Dummy Resistors

Some SPICE simulation do not allow the entry of a transmission line without a DC path to ground at all four end points. Dummy resistors are, therefore, inserted into the netlist at each transmission line node that needs a DC path to ground. The sizes of the dummy resistors are large enough such that the error introduced into the SPICE simulation is insignificant.

Example:

The following is an example of a FilterSolutions netlist:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



T1S1 and T1S2 are ideal transmission lines. T1S3 is a real transmission line. The R's are dummy resistors and exist to allow SPICE simulations of the netlist.

Print	sends the netlist to a printer.
Copy	copies all or part of the net list to the windows clipboard. Partial text may be copied by highlighting desired text. All text may be copied by selecting all the text or none of the text.
Save	saves the netlist to a text file.
Close	closes the netlist window.

Exporting to Ansys Electronic Desktop

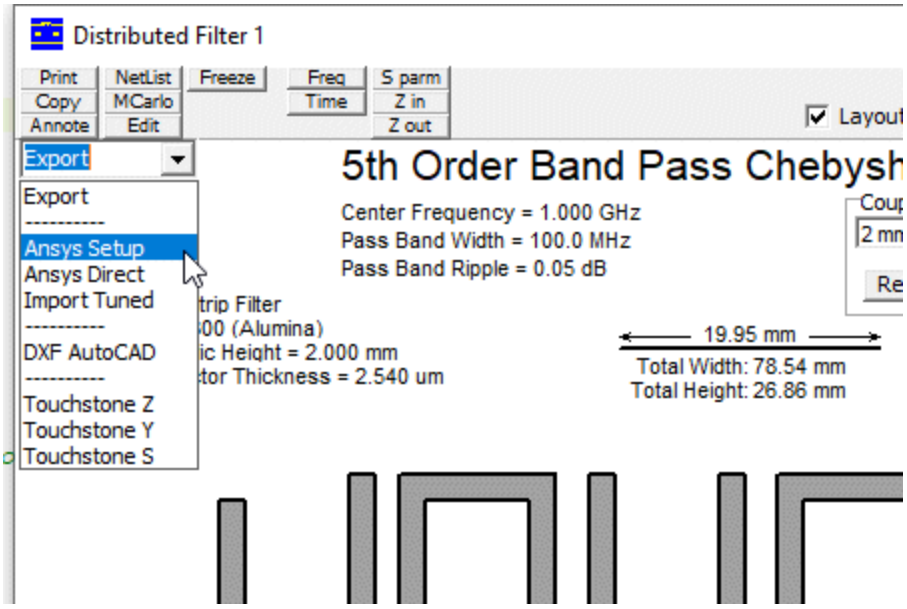
This chapter contains the following topics:

- Ansys Nuhertz Export Panel
- Ansys Iron Python Script
- Retrieving Design Variables
- Optimizing in FilterSolutions and AEDT

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Nuhertz Export/Import Panel

The export panel may be launched for viewing and editing by selecting **Ansys Setup** in the FilterSolutions schematic drop down, as shown in Figure 1. The panel may be automatically invoked and executed by selecting **Ansys Direct**.



FilterSolutions provides an easy, seamless approach to exports and imports to and from Ansys Electronic Deskop (AEDT), including support for a wide variety of design and analysis features. The panel with all its features is shown on Figure 2 below in the default state.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

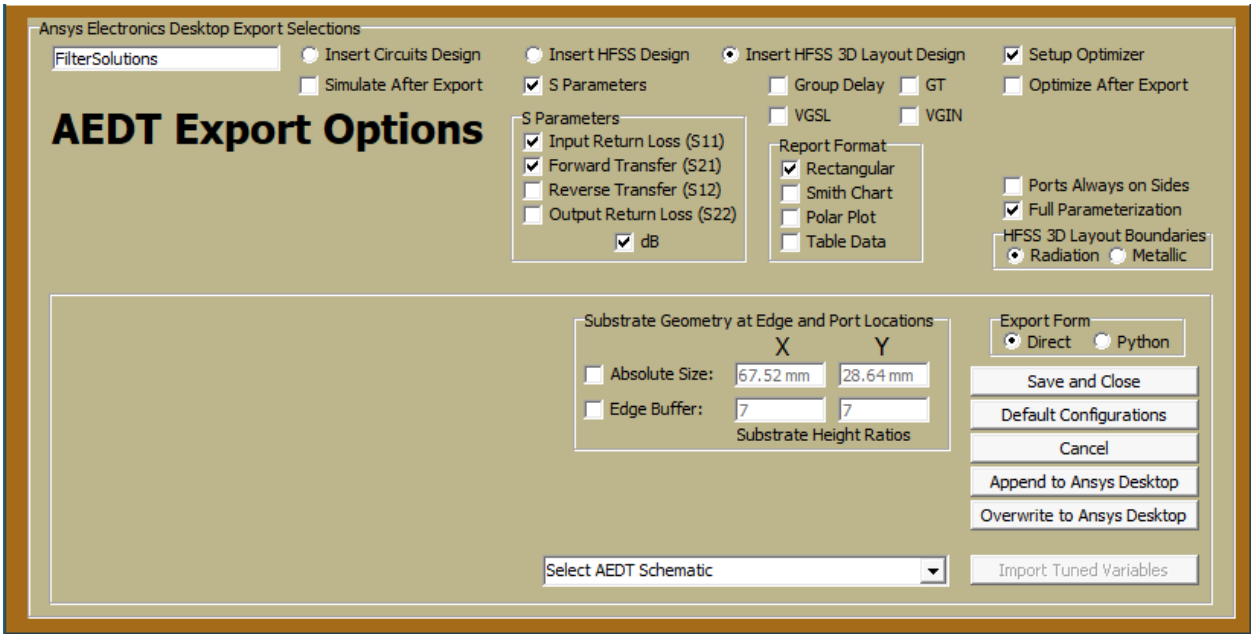


Figure 2: Ansys Nuhertz Export/Import Control Panel

Panel User Entry Definitions

FilterSolutions	Enter the desired name of the projected exported to Electronics Desktop
Insert Circuits Design	Check to export your FilterSolutions project into an AEDT Circuits project
Insert HFSS Design	Check to export your FilterSolutions project into an AEDT HFSS project
Insert HFSS 3D Layout Design	Check to export your FilterSolutions project into an AEDT HFSS project
Simulate After Export	Check to simulate the exported project immediately following the export
S Parameters	Check to setup an S Parameter results report in the exported project
S11, S12, S21, S22	Check the specific S Parameter(s) desired to be setup in the results report

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Group Delay	Check to setup a Group Delay results report in the exported project
Rectangular	Check to setup a rectangular plot in the results report
Smith Chart	Check to setup a smith chart in the results report
Polar Plot	Check to setup a polar plot in the results report
Table Data	Check to setup table data in the results report
dB	Check to display S Parameters in dB
Setup Optimizer	Check to setup the Desktop optimizer in the exported project
Optimize After Export	Check to initiate the Desktop optimizer immediately following the export
Full Parameterization	Check to install the full set of local variable geometry equations needed to define the layout geometry that are needed to optimize the layout. Uncheck to define geometry numerically
Ports Always on Sides	Visible when ports are located on the upper or lower ends of the layout and substrate Check to alter the layout as needed so ports are located on the left or right side of the layout.
HFSS 3D Layout Boundaries	Choose between Radiation (simulated open) and metallic boundaries. Generally, PCB designs use radiation boundaries and enclosed boxes use metallic.
Substrate Geometry at Edge and Port Locations	Define specific substrate geometry, if desired. Geometry definitions are defined at the port locations, if the edge contains a port, and the substrate edge if not.
Absolute Size	Check to define the absolute size of the substrate X and Y dimensions in the selected units (see Options panel for units)
Edge Buffer	Check to define the absolute size of the substrate X and Y dimensions in units of substrate heights
Export Form	Select the form of the export: Direct: Sends the export to Desktop directly Python: Creates an Iron Python text script that can be edited and imported by Desktop at a later time or place.
Save and Close	Save the existing export panel configuration and close the panel

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Default Configurations	Set the export panel settings to the default configuration
Cancel	Close the export panel without saving the existing configuration
Append to Ansys Desktop	Export to a new Desktop project without closing any existing projects
Overwrite to Ansys Desktop	Close any existing Desktop projects that use the same name as the existing exported project
Import Tuned Variables	Enabled following a project export Import the full suite of edited, tuned, or optimized local variables back into FilterSolutions
Select AEDT Schematic	Drop down or right click to observer the available AEDT projects and designs available to import variables from. Imported variables will come from the design selected.

Ansys Iron Python Script

Exports to Ansys Electronics Desktop are performed with Ansys IronPython, either directly, or by saving the IronPython script to a text file for further edit or importation at a later time or place. The selection of the export panel **Export Form** determines which export option takes place, as shown in Figure 1:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

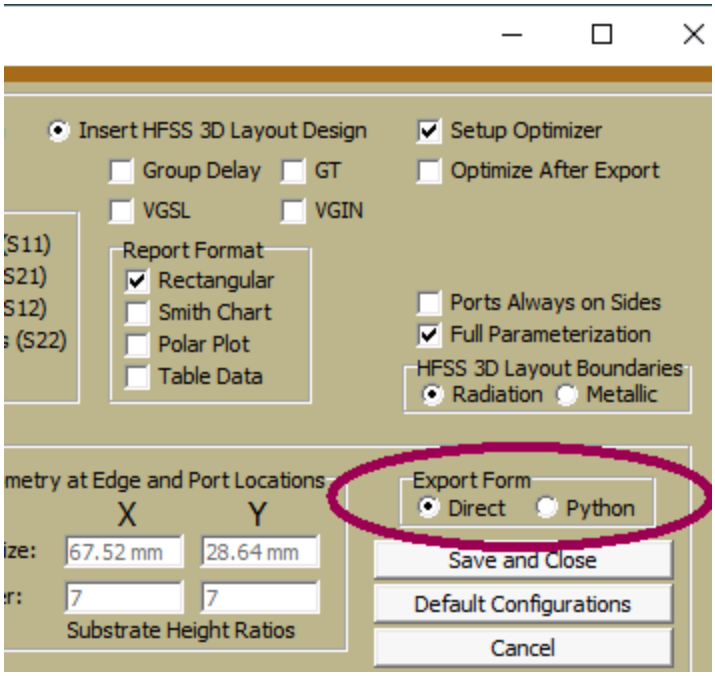


Figure 1: Selecting Direct vs. Iron Python Text Export Form Option

Direct Export

Export Form selection of **Direct** exports the FilterSolutions project into Ansys Electronic Desktop send the FilterSolutions project directly to Desktop with no file save buffering. Desktop communications are bidirectional, and may transmit project data and or receive Desktop data from as needed to insure a viable export takes place upon initiation.

IronPython Script Export

Export Form selection of **Python** exports the FilterSolutions project into an IronPython text file with the same name as the FilterSolutions project and a .py suffix. The Iron Python text file may be manually edited and stored independently of the FilterSolutions an Desktop project, and may be imported into Desktop at any time with the use of the **Tools** > **Run Script** feature in the Desktop Menu, as shown in Figure 2.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

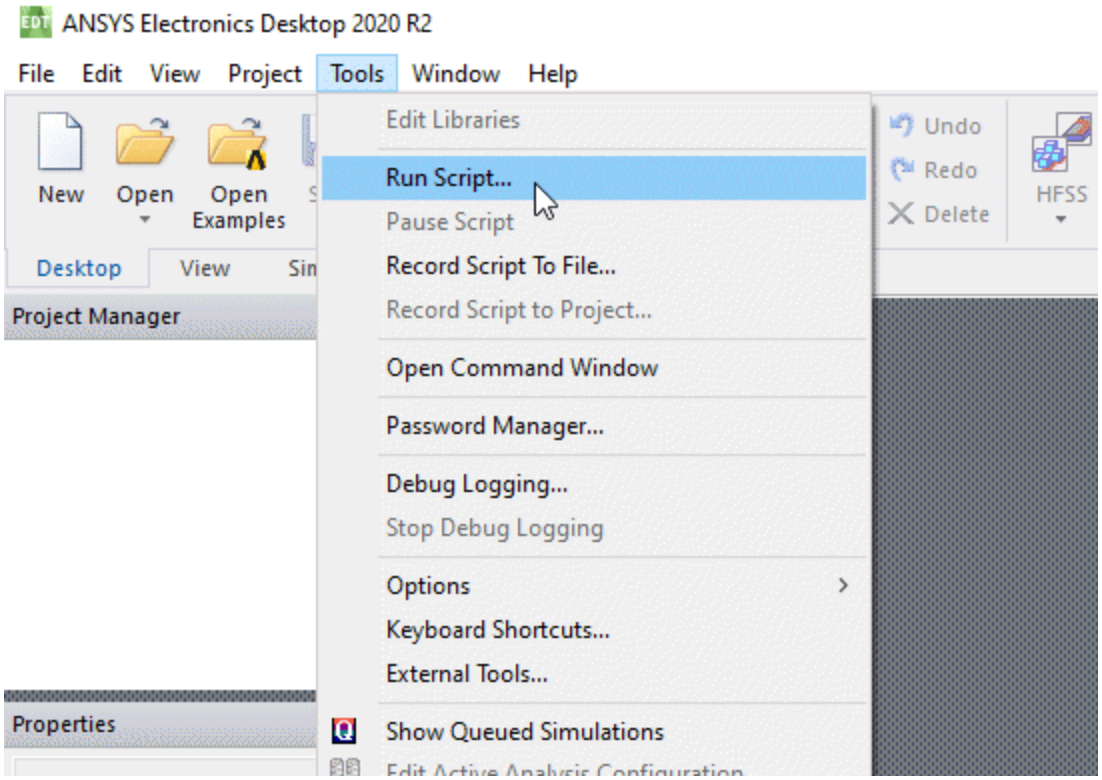


Figure 2: Importing an Iron Python Script into Desktop

Editing the IronPython Script

IronPython uses the Python programming language syntax and standard python libraries and can additionally use .NET classes and objects to give one the best of both worlds. extensive information regarding IronPython is available in the Desktop Help. The saved IronPython file format is a text file that can be edited with any text editor. Once opened, the IronPython script may be edited to change variable values, add or delete components, change the structure of the project, etc. The IronPython script, edited or in original form, may then be saved in a compact format for importation into Desktop at a later date, or on a different PC. A sample snippet of the saved IronPython script is shown in Figure 3 below as it appears in Notepad.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

*FilterSolutions.py - Notepad

File Edit Format View Help

```
# ----------------------------------------------
# Script Generated by FilterSolutions; Ansys, Inc., Nuhertz Product Group (www.ansys.com)
# ----------------------------------------------
import ScriptEnv
ScriptEnv.Initialize("Ansoft.ElectronicsDesktop")
oDesktop.RestoreWindow()
oProject = oDesktop.NewProject("FilterSolutions")
oProject.InsertDesign("HFSS 3D Layout Design", "FilterSolutions", "", "")
oDefinitionManager = oProject.GetDefinitionManager()
oDesign = oProject.SetActiveDesign("FilterSolutions")
oEditor = oDesign.SetActiveEditor("Layout")
oDesign.DesignOptions(
        [
                "NAME:options",
                "CausalMaterials:="     , False,
                "MeshingMethod:="       , "Phi",
                "EnableDesignIntersectionCheck:="       , True,
                "CircuitSparamDefinition:="     , False,
                "CircuitIntegrationType:="      , "FFT",
                "ExportAfterSolve:="    , False,
                "ExportDir:="   , "",
                "BroadbandFreqOption:=" , "AutoMaxFreq",
                "BroadbandMaxNumFreq:=" , 5,
                "UseAdvancedDCExtrap:=" , False
        ],0)
.
.
.
.
.
.
.
```

Figure 3: Snippet of a Saved Iron Python Script hat Can Be Edited in Notepad

Retrieving Design Variables

Parameters that are optimized in HFSS or Circuits may be imported back into FilterSolutions for storage and/or future use. Parameter importation is performed with the use of the **Import Tuned Variables** button in the lower right of the export panel, or with the **Import Tuned** selection of the FilterSolutions schematic export drop down, as shown in Figure 1 and 2 respectively.

A typical use for importing tuned/optimized variables is to optimize an exported design in Circuits and then to imports the optimized variables into FilterSolutions and then to

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

export the Circuits optimized FilterSolutions project HFSS or HFSS 3DL for further EM optimization.

FilterSolutions will remember with AEDT project and design it exported to for as long as FilterSolutions and the AEDT exported schematic remain open. Otherwise, the Select AEDT Schematic drop down may be used to select an AEDT project and design to import the variables from. The "Import Tuned Variables" button is enabled when it becomes possible to import variables.

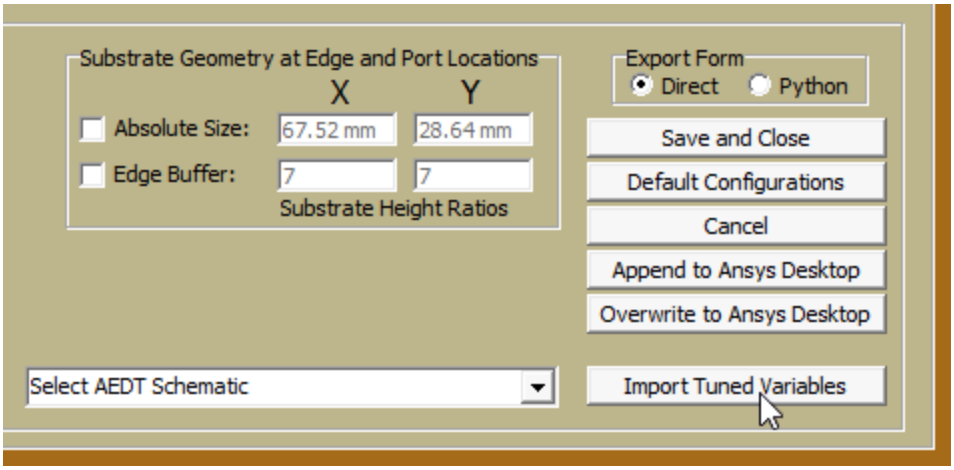


Figure 1: Importing Tuned and Optimized AEDT variables from the FilterSolutions export panel

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

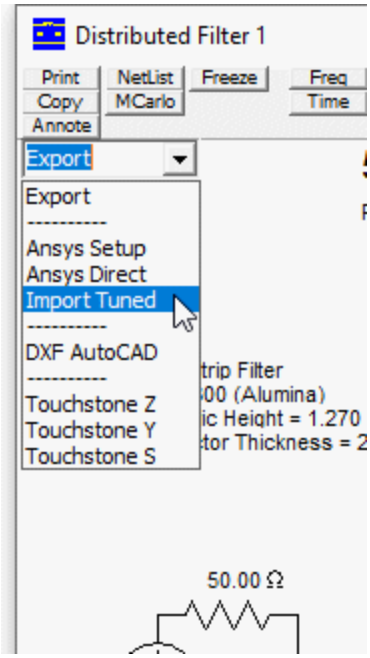


Figure 2: Importing Tuned and Optimized AEDT variables from the FilterSolutions schematic export Drop-Down

Optimizing in FilterSolutions and HFSS

Optimizing in FilterSolutions

FilterSolutions produces a highly accurate circuits simulation with limited optimization through its optional Quick Optimize, capability. Quick Optimize rapidly optimizes the filter frequency response to approach the ideal filter response to the extent needed to make the design suitable for direct EM optimizations. Invoke Quick Optimize during synthesis time by checking the **Quick Optimize** check box. The Quick Optimize check box is located at the top of the FilterQuick design panel, and at the lower right of the Advanced design panel, as shown in Figure 1 and 2 respectively.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

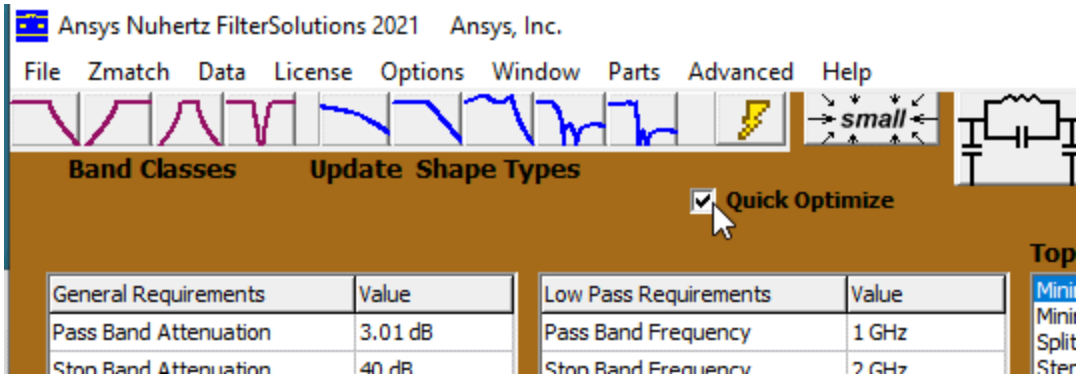


Figure 1: FilterQuick Quick Optimize Selection

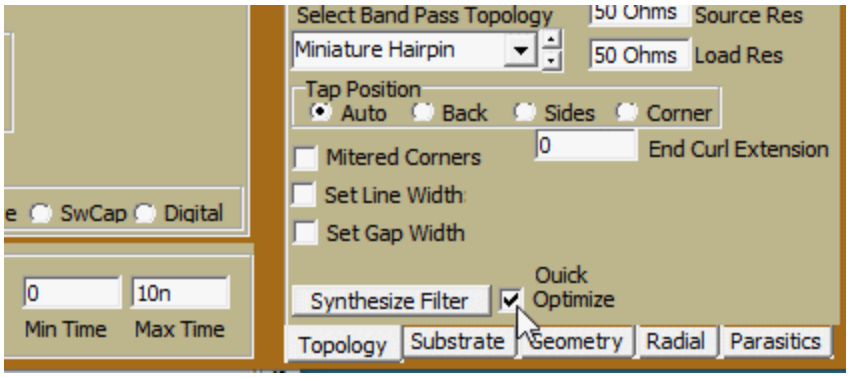


Figure 2: Advanced Panel Quick Optimize Selection

Optimizing in HFSS

FilterSolutions Quick Optimization gets the design simulation response close, but final EM optimization is performed in HFSS. HFSS optimization export selections are located in the upper right of the export control panel, as shown in Figure 3 below. The detailed optimization check box functions are listed in table 1 below. Exported optimizations include all items necessary to perform an optimization, including goals, parameters, and equations. If needed, the exported optimization setup may be manually edited in Ansys Electronic Desktop (AEDT) to make refinements needed to meet specific design requirements.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

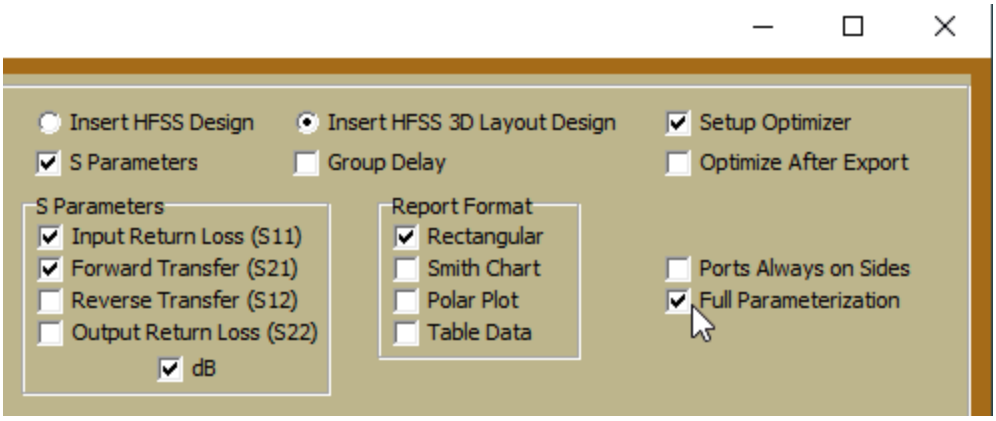


Figure 3: AEDT Export Panel Optimization Selections.

Check Box Name	Check Box Description
Setup Optimizer	Check to set up the optimizer in the export to AEDT.
Optimize After Export	Check to initiate an optimization process in AEDT immediately following the export
Full Geometric Parameterization	Check to define optimizable layout geometry with equations. This MUST BE checked for AEDT optimizations to work.

Table 1: Optimization Check Box Descriptions

Exporting to HFSS 3D Layout

This chapter contains the following topics:

- HFSS 3D Layout Stack Up
- Exporting to HFSS 3D Layout Schematics
- HFSS 3D Layout Port Configurations

HFSS 3D Layout Stackup

FilterSolutions sets up the HFSS 3D Layout stack up and ports automatically in a efficient manner so as to maximize simulation efficiency and accuracy and to minimize manual user

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

burden. Both single ports and dual ports are supported. Supported stack up configurations are listed in table 1 below:

Configuration	Description	Port Type
Strip Line	Conductor equally spaced above and below a ground plate	Wave Port
Unbalanced Strip Line	Conductor unequally spaced above and below a ground plate	Wave Port
Micro Strip	Conductor placed above a dielectric layer and ground plate	Lumped Port
Micro Strip With Cover	Conductor placed above a dielectric layer and ground plate and below an air pace with a ground plate	Lumped Port
Suspended Micro Strip	Conductor place above a dielectric layer, airspace, and ground plate	Lumped Port
Suspended Micro Strip With Cover	Conductor place above a dielectric layer, airspace, and ground plate and below an air pace with a ground plate	Lumped Port (Unless symmetric)
Inverted Micro Strip	Conductor place above an airspace and ground plate, and below a dielectric layer	Lumped Port
Inverted Micro Strip With Cover	Conductor place above an airspace and ground plate, and below a dielectric layer, air pace and ground plate	Lumped Port (Unless symmetric)
Broadside Micro Strips	Conductor placed above and below a dielectric layer with air space above and below, and at least one ground plate	Lumped Port (Unless symmetric)
Symmetric Micro Strips	Conductor above, below, or both placed above and/or below a dielectric with optional equally spaced air layers and required ground plates above and below the dielectric layer	Wave Port

Table 1: List of HFSS 3D Layout Stack Up Configurations

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting up the Substrate In FilterSolutions

FilterQuick Panel

Click the substrate icon to bring up the Substrate Definition panel, as shown in Figure 1:

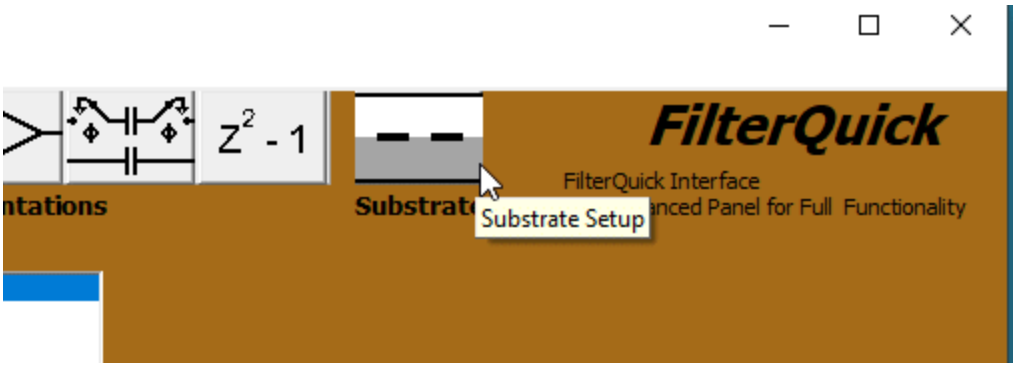


Figure 1: Selecting the FilterQuick Substrate Icon

The Substrate Definition panel will appear as shown below in Figure 2. Select the topology desired, layer dimensions, and materials, and then Close and Exit. Dielectric and conductance values may be entered with a numerical text entry, if desired. i.e., "4.56" for a dielectric constant or conductance of 4.56. Dielectric name may be entered with a name, colon, and number. For example, "MyDielectric:6.78" will create a dielectric name of "MyDielectric" and a dielectric constant of 6.78. At export time, HFSS 3D stack up is created in Overlap mode that conforms to the substrate definition.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

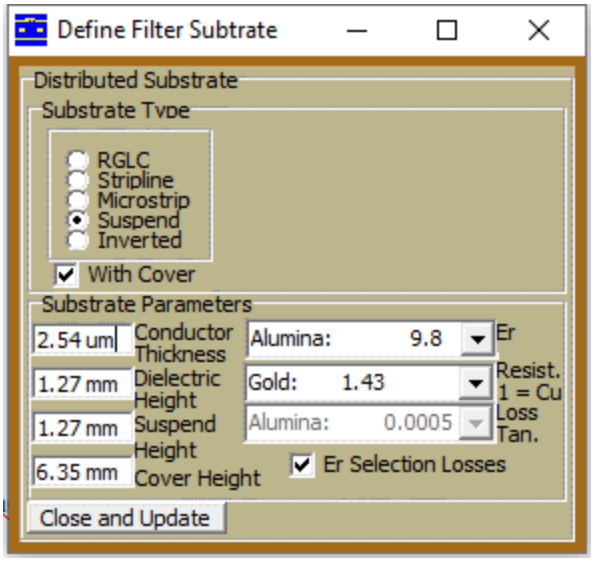


Figure 2; FilterQuick Substrate Definition Panel

Advanced Panel

Click the "Substrate" tab in the lower right of the Advanced panel, as shown in Figure 3 below. From there, substrate setup remains the same as described in FilterQuick above.

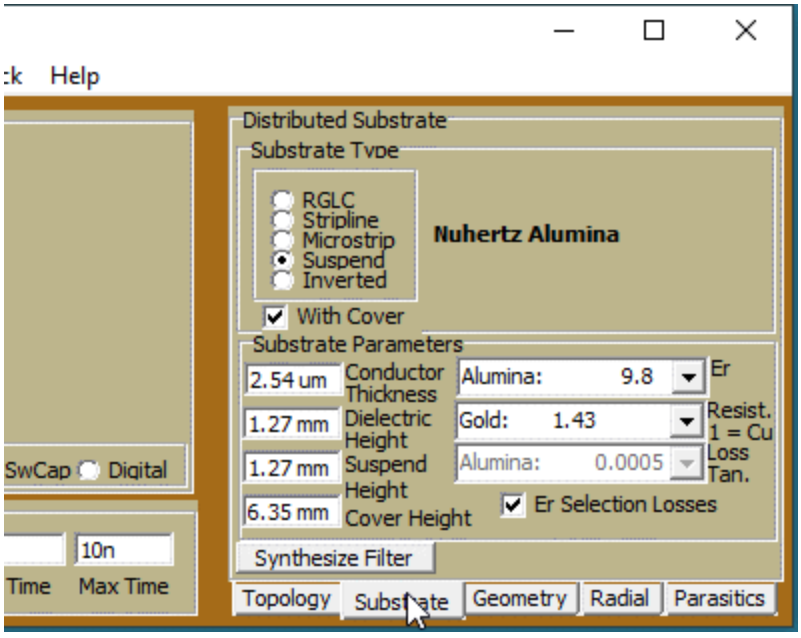


Figure 3: Advanced Panel Substrate Definition

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

HFSS 3D Layout Stack Up

An HFSS 3D Layout stack up is created in Overlap mode that matches the substrate definition. Overlap mode is advantageous in that it permits the addition of additional via layers, if the designer desires to add more parts to the HFSS 3D Layout design that require via layers.

Symmetric Substrates

Strip Lines are by definition symmetric. Micro strips may be made to be symmetric by selecting a suspended or inverted micros strip with cover, and with the cover height distance set to be equal to the suspended height distance. Symmetric substrates are assigned wave ports, just like strip lines.

Stack Up Vias

Nuhertz project via elements that attached to the lower and sometimes upper ground plate are included in all FilterSolutions project exports that contain such vias, including for suspended and inverted substrates. This via grounding through an air layer simulates properly, but the physical construction of the design may have to be physically implemented as a ground plane or box wall.

Exporting to HFSS 3DL Schematics

An export into an AEDT Circuits Schematic is performed by selecting the **Insert Circuits Design** selection in the FilterSolutions export panel, as shown in Figure 1.

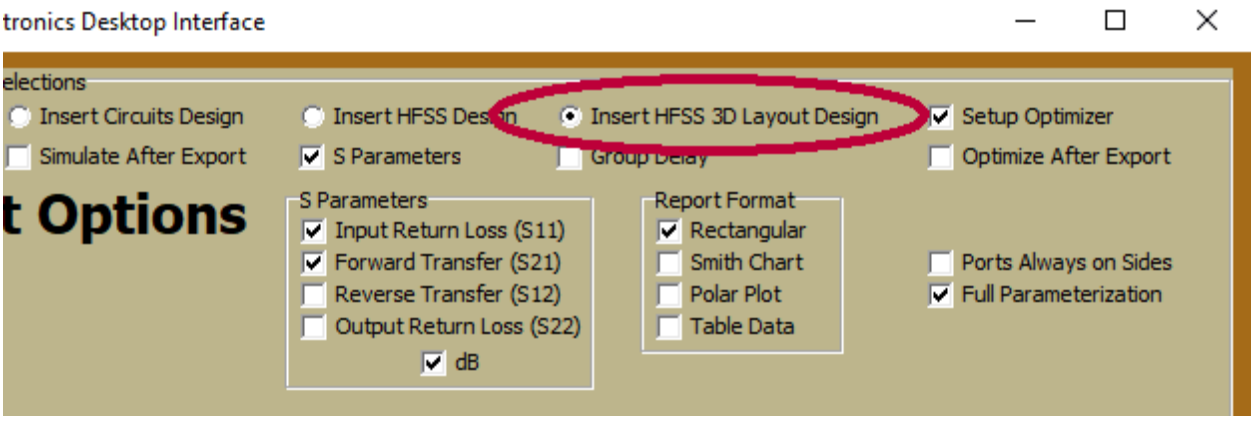


Figure 1: Export into HFSS 3D Layout selection

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Schematic Elements and Layout

Upon export from the FilterSolutions AEDT export panel, an HFSS 3D Layout schematic is created with a matching stack up. The stack up is designed to use metallic ground plates for the base layer and for the top cover, if any.

The AEDT HFSS 3D Layout schematic is populated with all of the necessary planar elements element needed to create a valid layout and execute a simulation. For example, a five pole Parallel Edge Coupled design would utilize a series of rectangles and polygons as shown in Figure 2. Ports, Setup, Results, and variables are all exported with the schematic to enable a valid simulations with good visual esthetics.

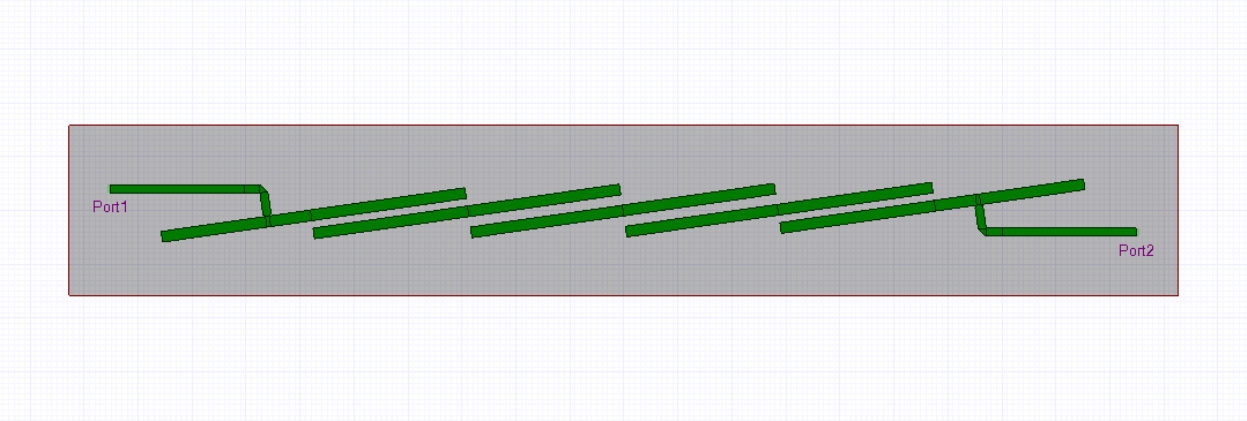


Figure 2: Exported AEDT HFSS 3D Layout Schematic

Composite lumped/line filters with shunt and series elements are exported to HFSS 3D Layout by implementing RLC circuit elements for lumped elements. Figure 3 shows a 5th order bandpass Butterworth with shunt stubs replaced by shunt capacitance and inductance lumped elements. Parallel and series RLC circuit elements represent capacitors and inductors respectively.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

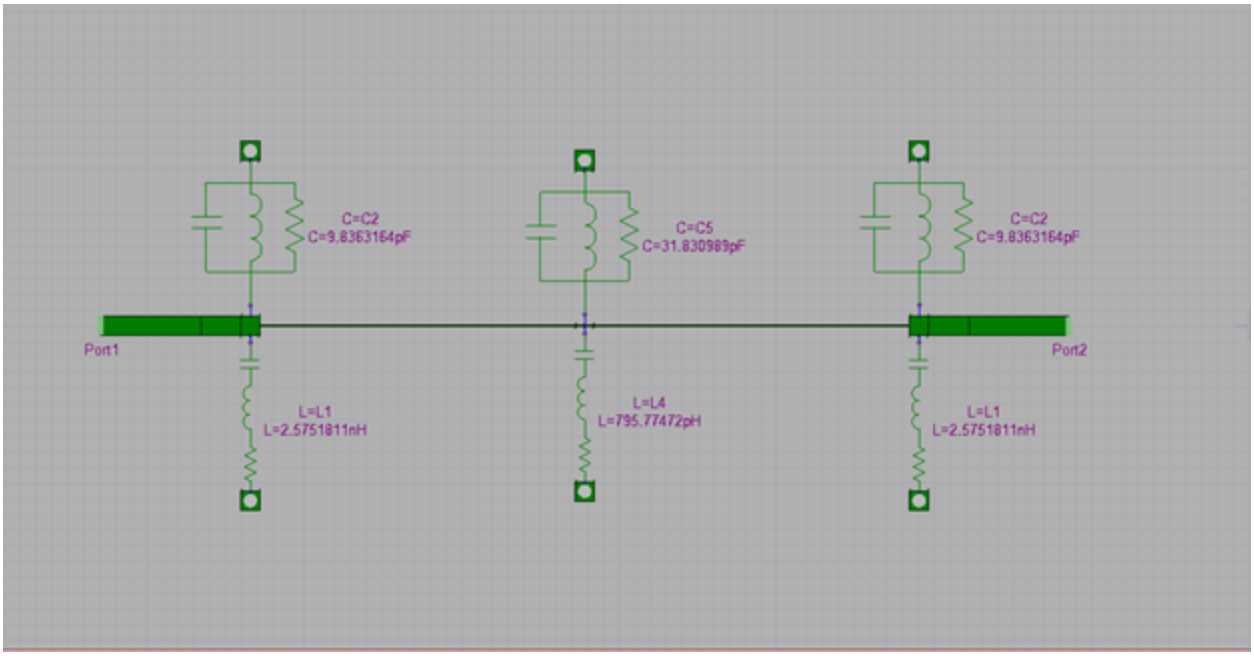


Figure 3: Exported AEDT HFSS 3D Layout Schematic with RLC circuit elements

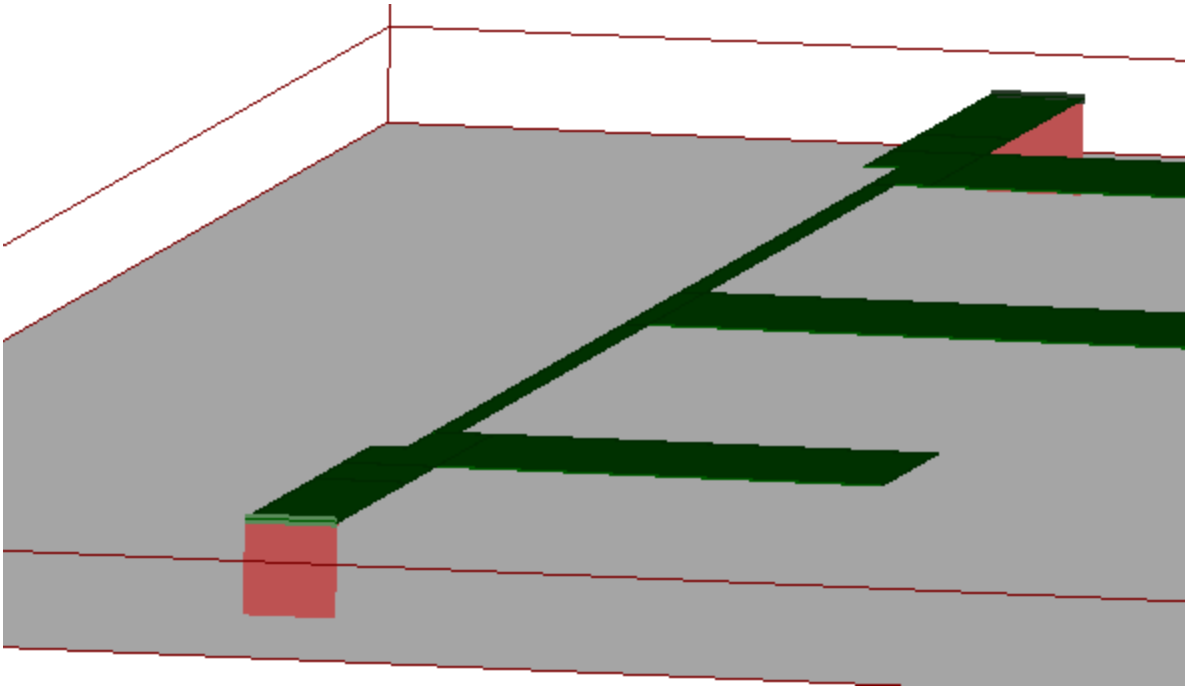
HFSS 3D Layout Port Configurations

FilterSolutions exports include support for lumped ports and wave ports. Lumped ports are chosen for all stack up configurations that do not use a top cover, and for all suspended and inverted substrates that do use a top cover with a cover height that is not equal to the suspended height. Wave ports are used for the remaining stack up configurations which are strip line and symmetric suspended micro strip.

Lumped Ports

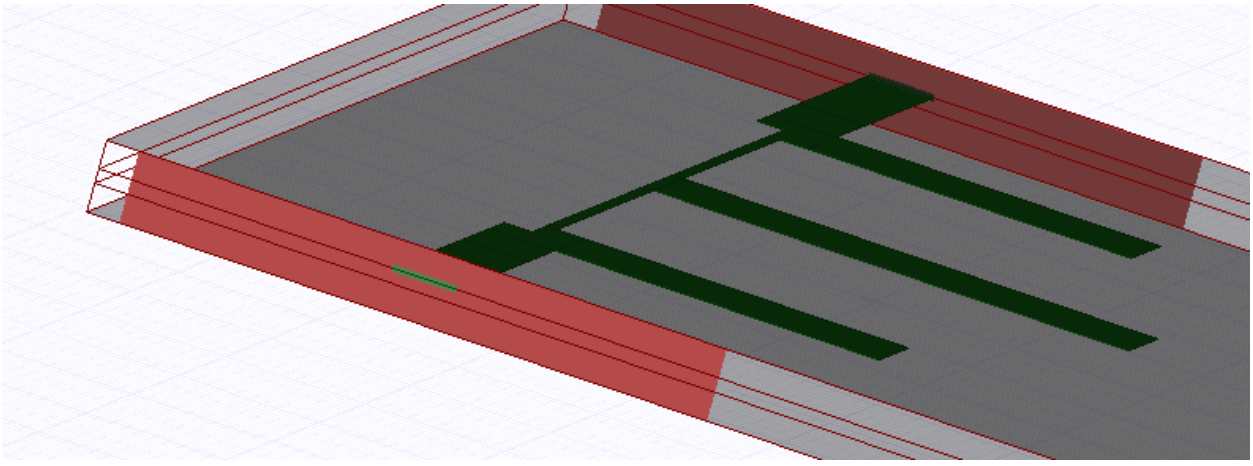
The FilterSolutions export process specifies the lumped port rectangle from conductor to ground plate. The designer is free to further modify the wave port configuration, if desired.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Wave Ports

The FilterSolutions export process specifies the wave port rectangle between the ground plates using a horizontal extent of four times the width of the attached line and vertical extents that extend to the upper and lower metallic covers. The designer is free to further modify the wave port configuration, if desired.



Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Exporting to HFSS

This chapter contains the following topics:

- Exporting to HFSS Schematics
- HFSS Port Configuration

HFSS Schematics

An export into an HFSS Schematic is performed by selecting the **Insert HFSS Design** selection in the FilterSolutions export panel, as shown in Figure 1.

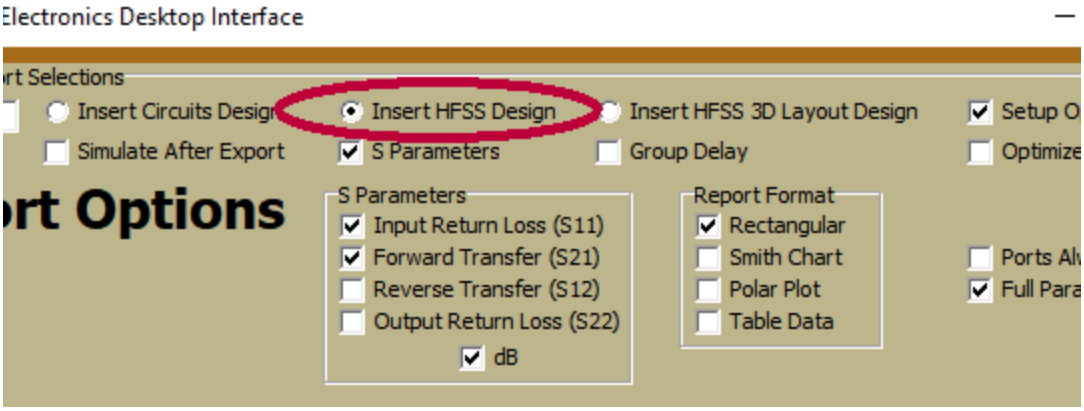


Figure 1: Export into HFSS selection

Schematic Elements

Upon export from the FilterSolutions AEDT export panel, an HFSS schematic in full 3D mode is created with an embedded substrate and planar elements.

The HFSS schematic is populated with all of the necessary substrate and planar element needed to execute an HFSS electromagnetic (EM) simulation and to create a viable layout. For example, a five pole Parallel Edge Coupled design would utilize rectangles, polygons, extents, and boundaries as needed, as shown in Figure 2. Ports, Setup, and Results are all exported with the schematic to enable a valid simulation with good visual esthetics.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

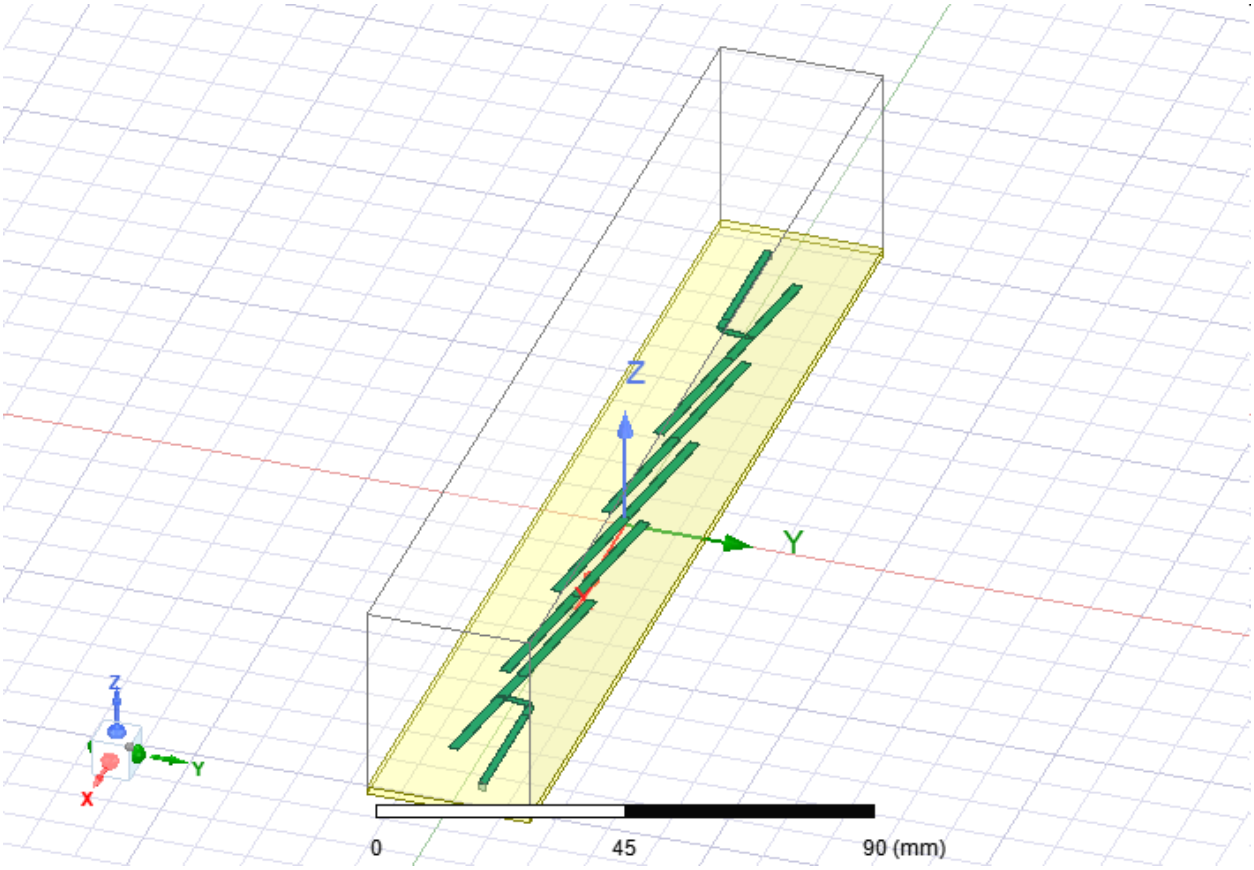


Figure 2: Exported AEDT Circuits Schematic and Layout

Composite lumped/line filters with shunt and series elements are exported to HFSS by implementing RLC boundaries for lumped elements. Figure 3 shows a 5th order bandpass Butterworth with shunt stubs replaced by shunt capacitance and inductance lumped elements. Capacitors and inductors are represented by RLC boundaries.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

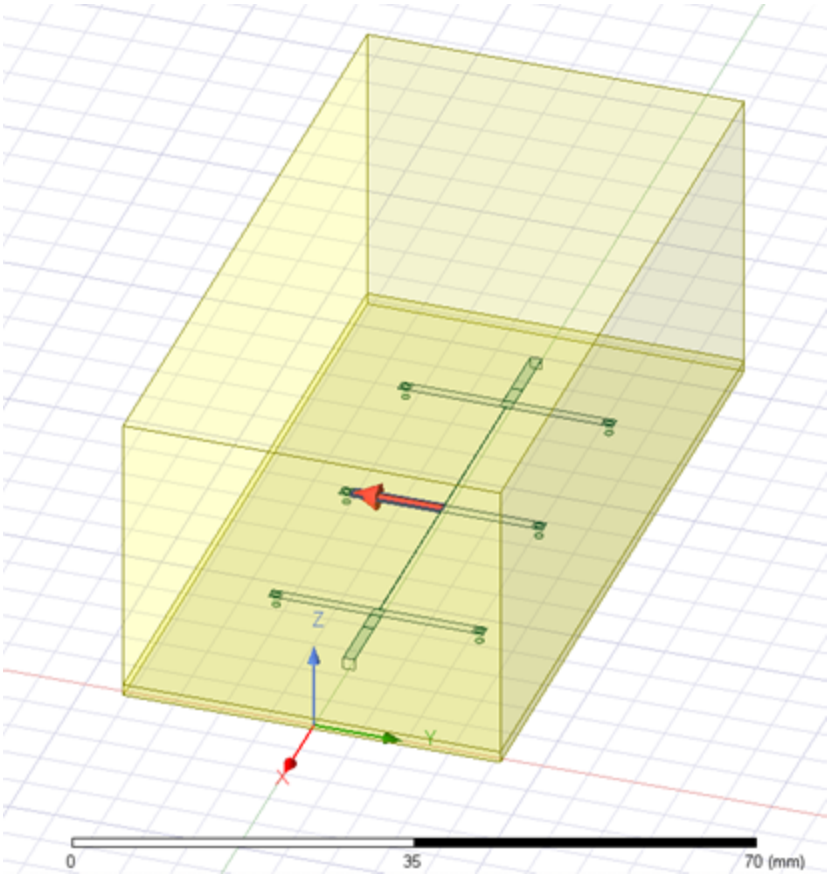


Figure 3: Exported AEDT HFSS Schematic with RLC boundaries

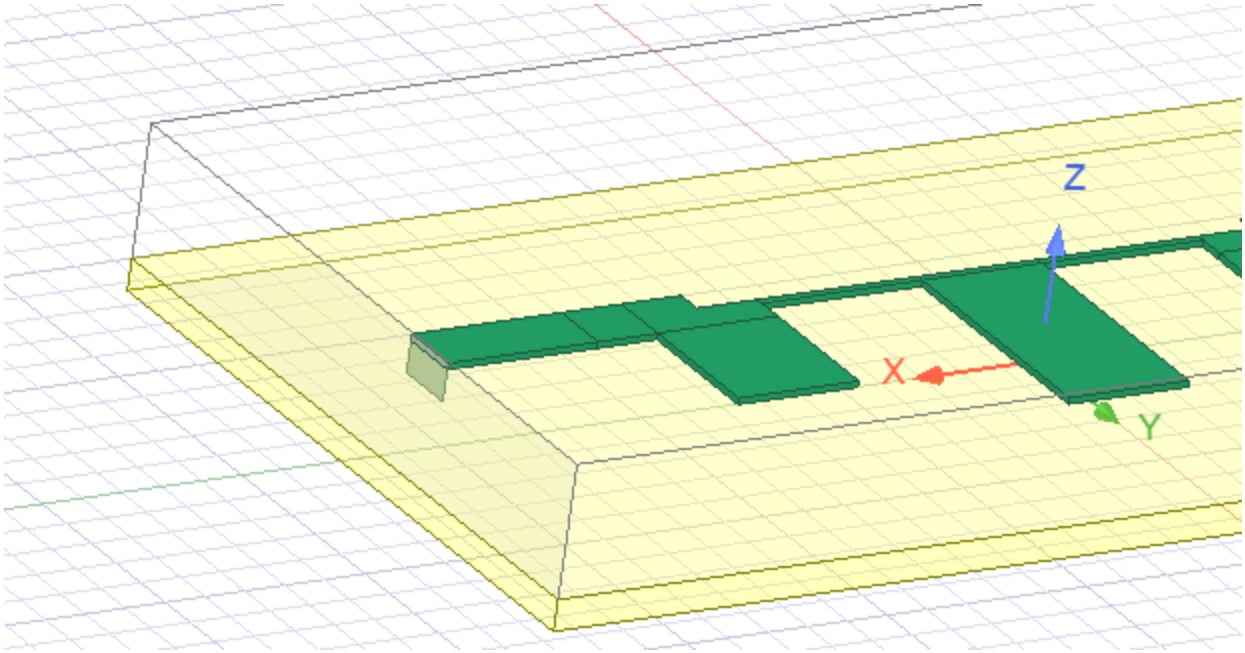
HFSS Port Configuration

FilterSolutions exports include support for lumped ports and wave ports. Lumped ports are chosen for all 3Dconfigurations that do not use a top cover, and for all suspended and inverted substrates that do use a top cover with a cover height that is not equal to the suspended height. Wave ports are used for the remaining stack up configurations which are strip line and symmetric suspended micro strip.

Lumped Ports

The FilterSolutions export process creates the lumped port rectangle from conductor to ground plate and places the port on it. The designer is free to further modify the wave port configuration, if desired.

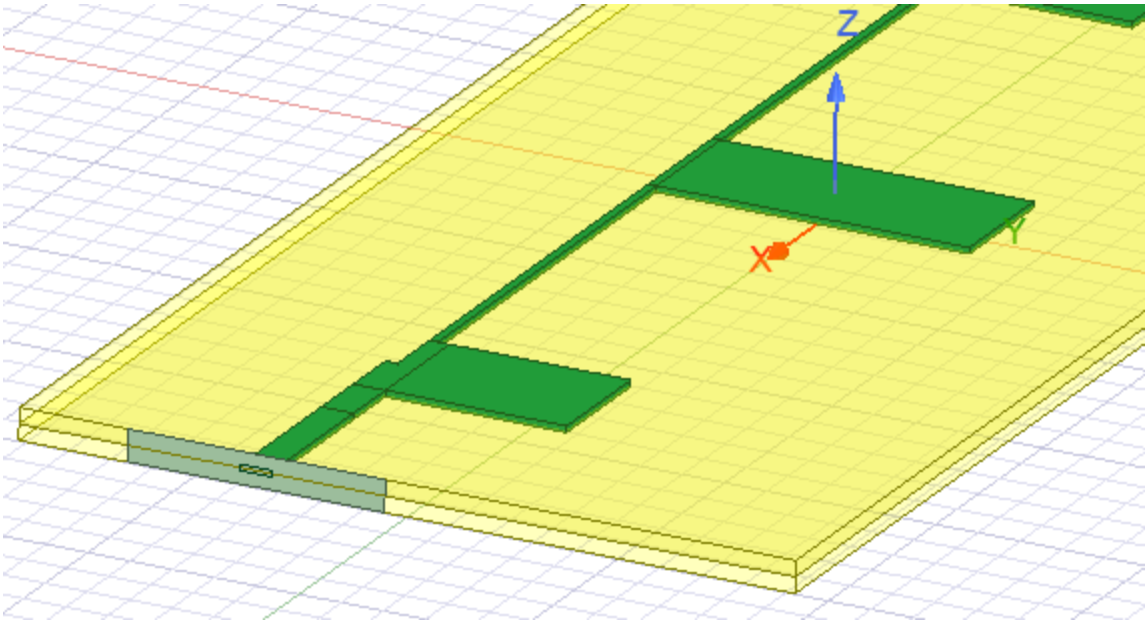
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Wave Ports

The FilterSolutions export process creates the wave port rectangle between the ground plates and places the port on it, using a horizontal extent of four times the width of the attached line and vertical extents that extend to the upper and lower metallic covers. The designer is free to further modify the wave port configuration, if desired.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Exporting to Ansys Planar Circuits

This chapter contains the following topics:

- Planar Circuits Stack Up
- Exporting to Planar Circuits Schematics
- Ansys Planar Circuits Optimization

Planar Ansys Circuits Stack Up

FilterSolutions sets up the Circuits stack up automatically in a efficient manner so as to maximize simulation efficiency and accuracy and to minimize manual user burden, and provides for an accurate circuits layout display and copy into HFSS 3D Layout. FS exports into Ansys circuits supports covered micro strip, uncovered micro strip, and balanced strip line designs.

Setting up the Substrate In FilterSolutions

FilterQuick Panel

Click the substrate icon to bring up the Substrate Definition panel, as shown in Figure 1:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

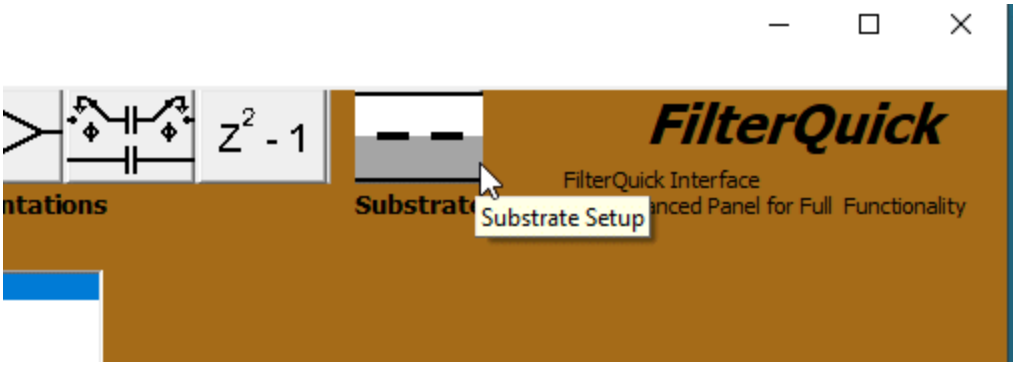


Figure 1: Selecting the FilterQuick Substrate Icon

The Substrate Definition panel will appear as shown below in Figure 2. Select the topology desired, layer dimensions, and materials, and then Close and Exit. Dielectric and conductance values may be entered with a numerical text entry, if desired. i.e., "4.56" for a dielectric constant or conductance of 4.56. Dielectric name may be entered with a name, colon, and number. For example, "MyDielectric:6.78" will create a dielectric name of "MyDielectric" and a dielectric constant of 6.78. At export time, the layout stack up is created in Overlap mode that conforms to the substrate definition.

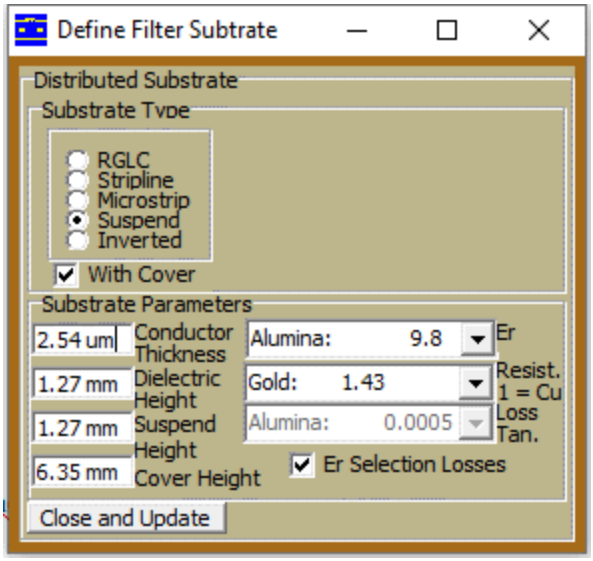


Figure 2; FilterQuick Substrate Definition Panel

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Advanced Panel

Click the "Substrate" tab in the lower right of the Advanced panel, as shown in Figure 3 below. From there, substrate setup remains the same as described in FilterQuick above.

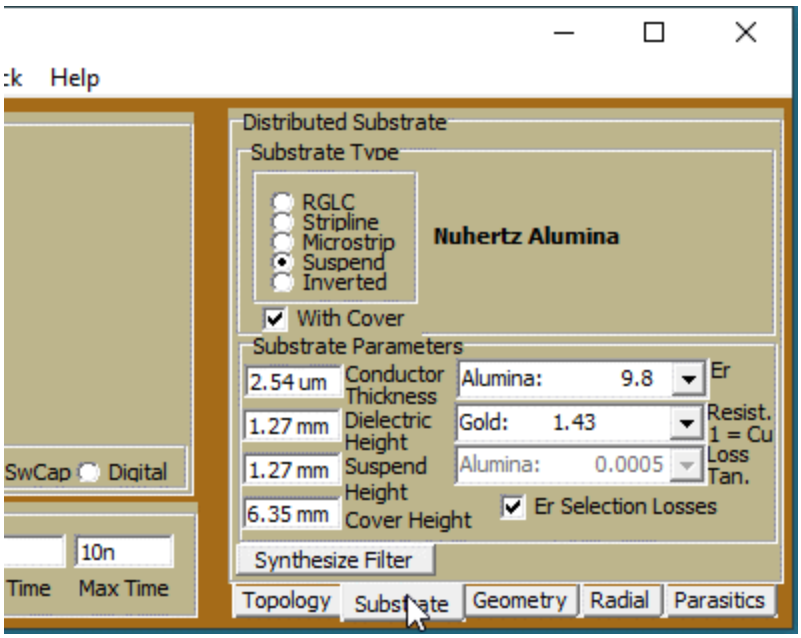


Figure 3: Advanced Panel Substrate Definition

Circuits Stack Up

An Circuits stack up is created in Overlap mode that matches the substrate definition. Overlap mode is advantageous in that it permits the insertion of additional via layers, if the designer desires to add more parts to the circuits design Layout that require via layers.

Planar Circuits Schematics

An export into an AEDT Circuits Schematic is performed by selecting the **Insert Circuits Design** selection in the FilterSolutions export panel, as shown in Figure 1.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

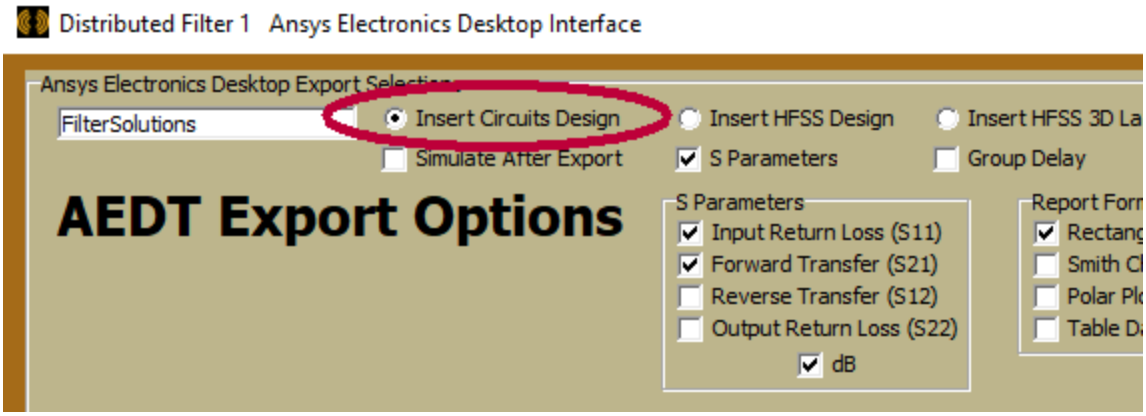


Figure 1: Export into AEDT Circuits selection

Schematic Elements and Layout

Upon export from the FilterSolutions AEDT export panel, a circuits schematic is created with a matching substrate and stack up. The substrate is used by the circuits elements to simulate the schematic, and the stack up is used is used to create the layout.

The AEDT circuits schematic is populated with all of the necessary circuits element needed to execute a circuits simulation and to create a viable layout. For example, a five pole Parallel Edge Coupled design would utilize edge coupled micro strips, and micro strip lines, stubs, tapers, and bends, as needed, as shown in Figure 2. Ports, Setup, and Results are all exported with the schematic to enable a valid simulations with good visual esthetics.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

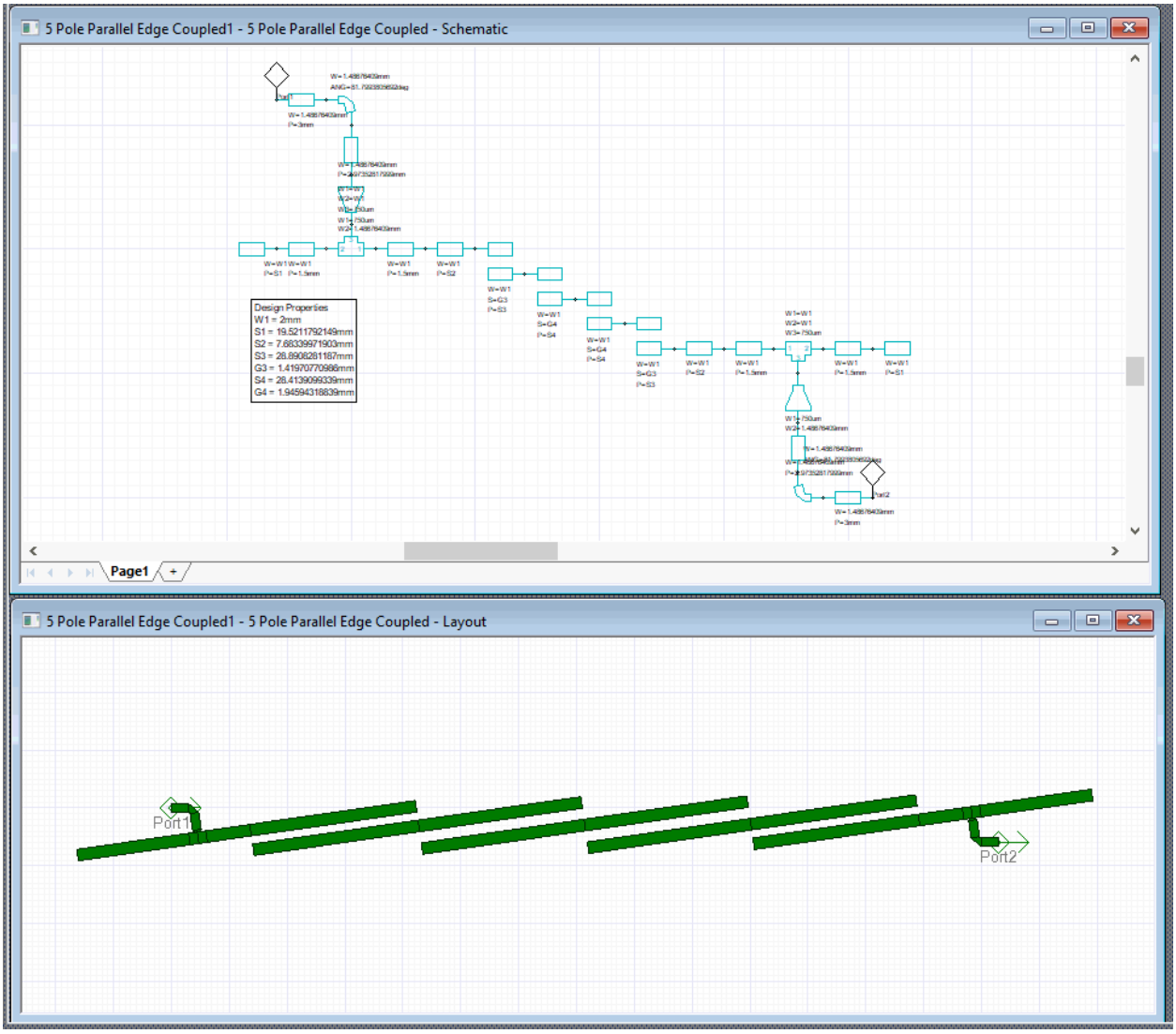


Figure 2: Exported AEDT Circuits Schematic and Layout

Ansys Circuits Optimization

Optimizing in FilterSolutions

FilterSolutions produces a highly accurate circuits simulation with limited optimization through its optional Quick Optimize, capability. Quick Optimize rapidly optimizes the filter frequency response to approach the ideal filter response to the extent needed to make the design suitable for direct EM optimizations. Invoke Quick Optimize during synthesis time by checking the Quick Optimize check box. The Quick Optimize check box is located at the top of the FilterQuick design panel, and at the lower right of the Advanced design panel, as shown in Figure 1 and 2 respectively.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

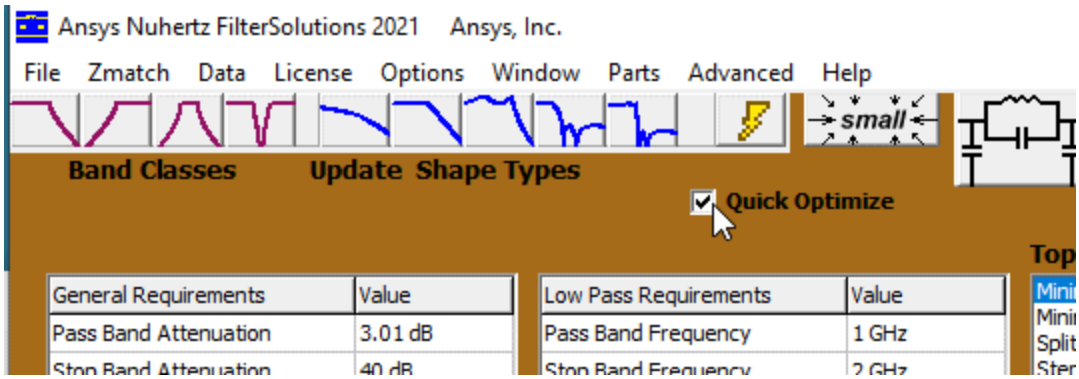


Figure 1: FilterQuick Quick Optimize Selection

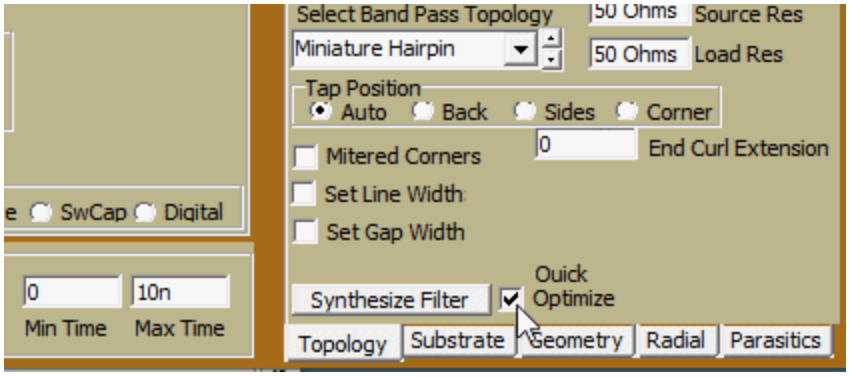


Figure 2: Advanced Panel Quick Optimize Selection

Optimizing in Circuits

FilterSolutions Quick Optimization gets the design simulation response close, but additional optimization may performed in AEDT Circuits. This may be useful, for example, if it is desired to obtain a more accurate circuits optimized design prior to using the design or prior to initiating an electromagnetic (EM) optimization of the design. Circuits optimization export selections are located in the upper right of the export control panel, as shown in Figure 3 below. The detailed optimization check box functions are listed in table 1 below. Exported optimizations include all items necessary to perform an optimization, including goals, parameters, and equations. If

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

needed, the exported optimization setup may be manually edited in Ansys Electronic Desktop (AEDT) to make refinements needed to meet specific design requirements.

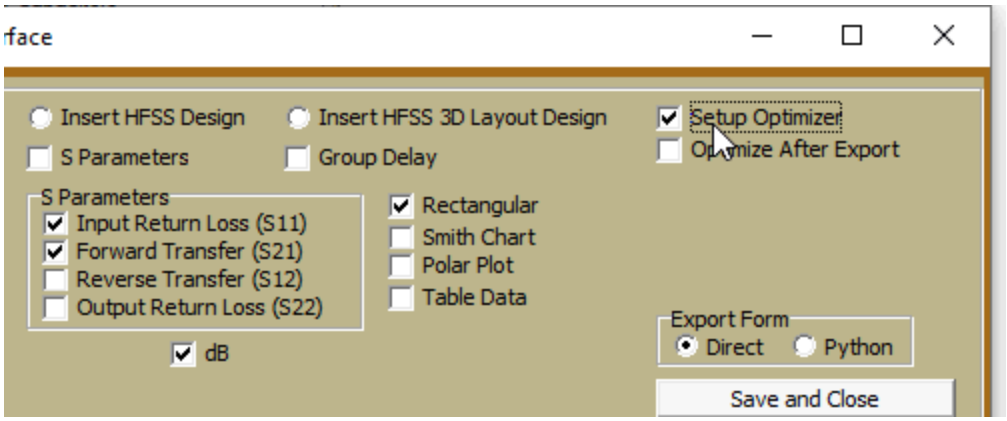


Figure 3: AEDT Export Panel Optimization Selections.

Check Box Name	Check Box Description
Setup Optimizer	Check to set up the optimizer in the export to AEDT.
Optimize After Export	Check to initiate an optimization process in AEDT immediately following the export

Table 1: Optimization Check Box Descriptions

Automated Port Tuning

This chapter contains the following topics:

- Planar Circuits Stack Up
- Exporting to Planar Circuits Schematics
- Ansys Planar Circuits Optimization

General Information

This chapter contains the following topics:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Automated Port Tuning Description
- Port Tuning Goals
- Reconstructing the Filter Design
- Ring Resonator Gap Tuning

Automated Port Tuning Description

General

Automated Port Tuning is a fast and easy way to implement Circuits simulation speed with EM simulation accuracy using co-simulation and GUI automation to provide for fast accurate planar optimization with minimal need for manual interactions. Tuning ports are inserted at strategic places in the planar design, and small updates to the resonator lengths and separations have the virtual effect of updating the full design geometry. Small perturbation updates are required in that small adjustments of the tuning elements induce small errors in the full geometry, and large updates induce large errors in the full geometry. It is generally necessary to rough out te optimization on the first port tuning optimization cycle and fine tune the optimization on subsequent cycles.

Resonator Length Updates

Tuning ports must be inserted into each length of micro strip or strip line that may require a tuning adjustment. The spacing of the tuning ports must as small as possible and port interactions must be eliminated or acceptably minimized to obtain valid results. Short micro strips or strip lines are placed between the two ports, as shown in Figure 1. Sets of closely spaced ports may be used to insert coupled micro strips or strip lines. Horizontal ports containing single micro strip or strip line connections may be substituted for series ports provided they meet this criteria. Small adjustments to the inserted micro strips or strip line lengths have the effect of accurately adjusting the full micro strip or strip line length.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

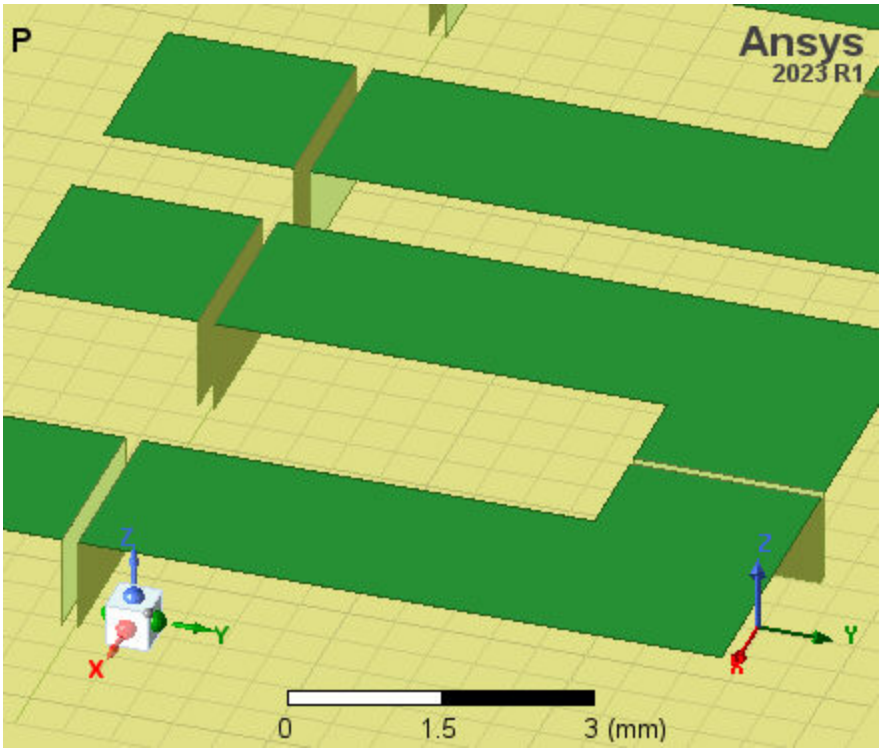


Figure 1: Small Gap Tuning Ports Inserted Into a Micro Strip

Resonator Separation Adjustments

Resonator separation adjustments may be simulated by placing a 45˚ open transmission line stub with infinite impedance (Zo). Since it is difficult to tune Zo around infinity, the necessary equations are substituted to make a capacitor look like a 45˚ transmission line stub at infinite Zo, but using admittance (Yo) at 0. Yo may be easily be tuned above and below 0 to provide a useful value that may be used to adjust the resonator separation. Positive tuned Yo narrows the resonator separation, while negative tuned Yo increases separation. Admittance elements formed from frequency dependent capacitors are shown in Figure 2.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

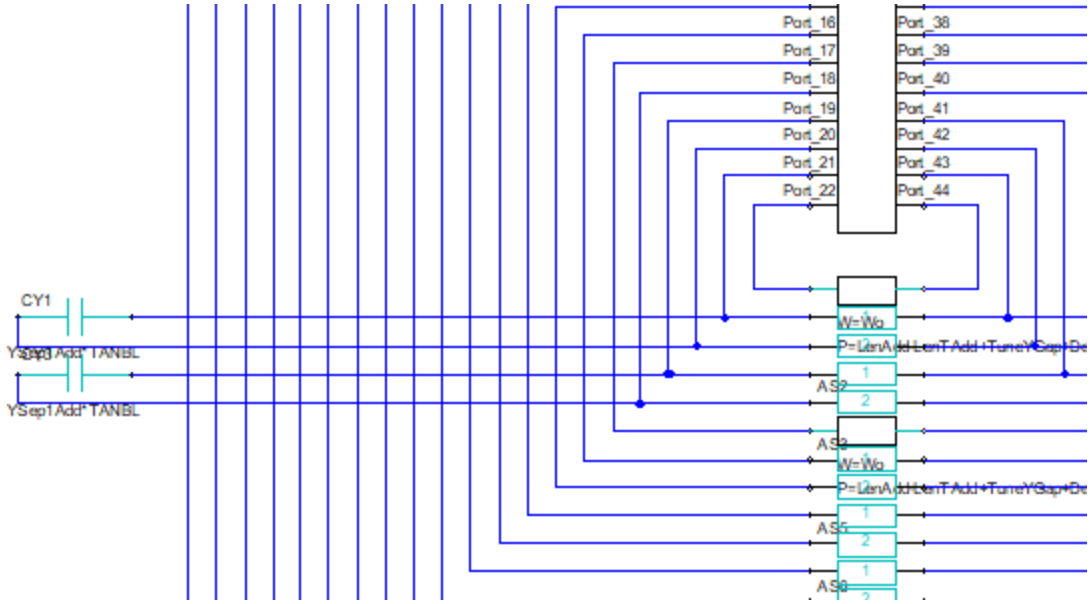


Figure 2: Admittance Elements and Micro Strip Lines Inserted into Tuning Ports

Tuning Port Error

To obtain accurate results, tuning ports must not contribute significant error the tuning port schematic simulation. A 100% parasitic deembeding solution is desired. If not possible, care should be taken to remove as much tuning port error as possible. It should be remembered that the accuracy of the final Automated Port Tuning design cannot be more accurate than the accuracy of the tuning ports of the EM tool in use.

Optimization Goals

FilterSolutions provides a calculated set of design goals for use in the optimization processes. It may be necessary for the designer to modify these goals some to meet design requirements. Since the Automated Port Tuning process may involve several iterations, it is important for to retain the design optimization goals from each optimization process for use in the next optimization iteration. FilterSolutions provides the means to do that which may vary slightly depending on the EM tool in use. In all cases, the FilterSolutions Goal Table may be used.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reconstructing the Planar Geometry

FilterSolutions automates the reconstruction of the filter design such that the user only has to select the appropriate button to import the optimized variables and use them to reconstruct the filter. Since large tuning adjustments produce larger filter reconstruction errors, it is usually necessary for the first port tuning optimization cycle to rough out the design and for subsequent port tuning optimization cycles to fine tune the design. For this reason, FilterSolutions provides a dedicated button to quickly and easily cycle the Automated Port Tuning optimizations by providing a single click button to: import the tuning port variables, reconstruct the filter, and export the filter back out to the tuning port schematic for the next cycle while retaining all optimization goals.

Final Design Optimizations.

The final reconstructed planar design may require further EM optimizations to meet design requirements. When this happens, it is a simple matter to perform a final EM optimization on the full, contiguous design until it meets the design requirements. The final EM optimization process should require substantially less time to perform due to the initial optimizations performed by the FilterSolutions Automated Port Tuning Process.

Automated Port Tuning Goals

Goal Requirements

Automated Port Tuning is an iterative process that requires reexportation using the same design gaols for each iteration cycle. If the internally generated goals are used for the automated port tuning optimization process, this does not present any issues. However, if the goals are created manually, or the internally automated goals are adjusted, it is manually intensive to have to set them for each iteration. Therefor, FilterSolutions provides a means to fix the goals to the desired values for each iteration, gratly reducing the need for manual intervention between the optimization iterations.

Ansys Circuits Optometrics

If Ansys Circuits is used as the circuits simulator, the internally generated goals may be included in the first Automated Port Tuning exported schematic, and then manually edited. Once edited, these goals are be retained by using the "Import and Reexport Over" button to cycle the Automated Port Tuning optimizations, as shown in Figure 1. This button will only export the reconstructed filter into HFSS or HFSS 3D Layout, and will only updated needed parts of Circuits, leaving the Optometrics containing the design goals unchanged.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

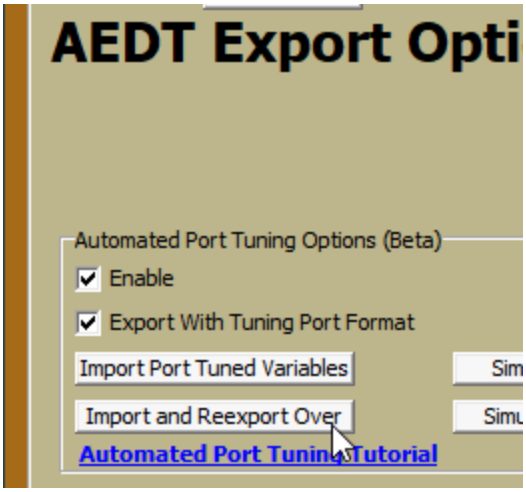
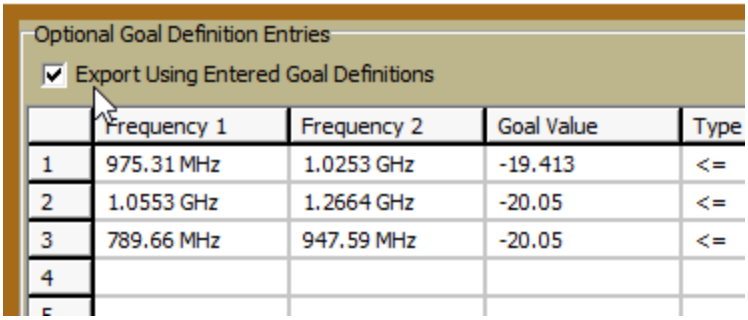


Figure 1: Import and Reexport Method to Retain the Design Optimization Goals

FilterSolutions Goals Table

All supported circuits simulators support retaining goals during optimizations cycles through the use of the FilterSolutions Goals Table, available in the upper right of supported export GUI's. The detailed functionally of the goals table differs some from circuits simulator to simulator, but the overall function is the same, which is to permit an easy way for the designer enter or adjust his/her desired goals and to retain those goals for future use for all exports. Once the goals are edited and set, they will always be used in the export whenever the "Export Using Entered Goal Definitions" check box is checked., as shown in Figure 2.

To export using the Goal Table goals in Ansys Circuits, the "Append to..." or "Overwrite to..." buttons must be used. The "Import and Reexport Over" function in Ansys Circuits always retains the existing Circuits optimetrics, and will ignore the Goal Table.



	Frequency 1	Frequency 2	Goal Value	Type
1	975.31 MHz	1.0253 GHz	-19.413	<=
2	1.0553 GHz	1.2664 GHz	-20.05	<=
3	789.66 MHz	947.59 MHz	-20.05	<=
4				

Figure 2: Goal Table Method to Retain the Design Optimization Goals

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reconstructing the Filter Design

Filter design reconstruction is fully automated by FilterSolutions. Once an optimization cycle is complete, a single mouse click on "Import Port Tuned Variable", as shown in Figure 1, will import the variables from the target simulator, adjust all resonator lengths by in accordance with the variable values, and adjust all resonator separations to retain their net gap capacitance by incorporating the tuned Yo value. The reconstructed filter is ready for its next automated port tuning export cycle, or for its final design export.

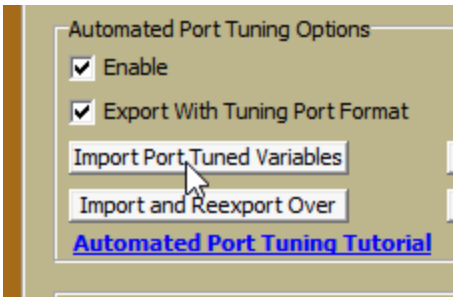


Figure 1: Reconstructing the Filter.

Reconstructing the Resonator lengths

Micro strip and strip line resonators are reconstructed by simply adding in the amount of adjustment applied by the tuned variables. In some cases, such as miniature hairpin stubs and ring resonator arms, the amount of adjustment may be scaled to match the ratio of resonator length to specific micro strip or strip line length.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

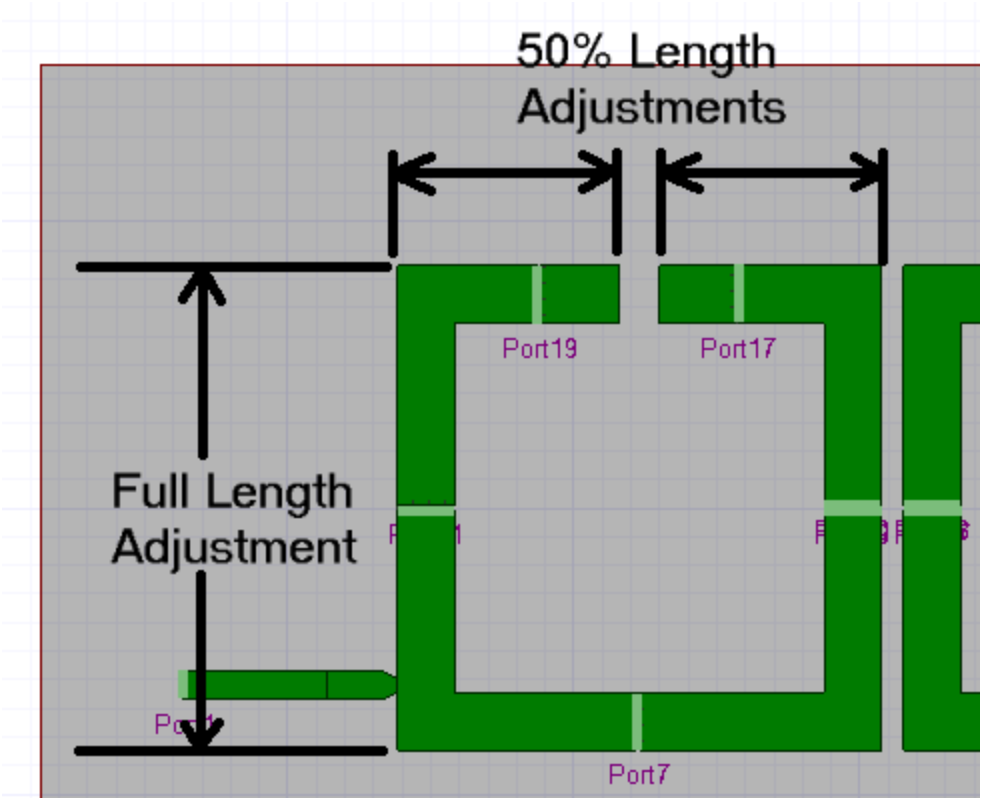


Figure 2: Resonator Length and prorating Adjustments

Adjusting the Resonator Separation

Resonator separation adjustments are accomplished by maintaining equivalent gap capacitance between the resonators, using the original gap capacitance and the tuned Yo value of the 45 deg open transmission line stubs. The process is automatic and performed upon importing the tuned variables. In the case of back coupled and cross coupled resonators, tuning ports are placed away from the virtual ground points, and compensation is applied to get a reasonably accurate estimate of the adjustment.

Example: A 5% BW, 3 pole ring resonator HFSS 3D Layout exports simulates as shown in Figure 3:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

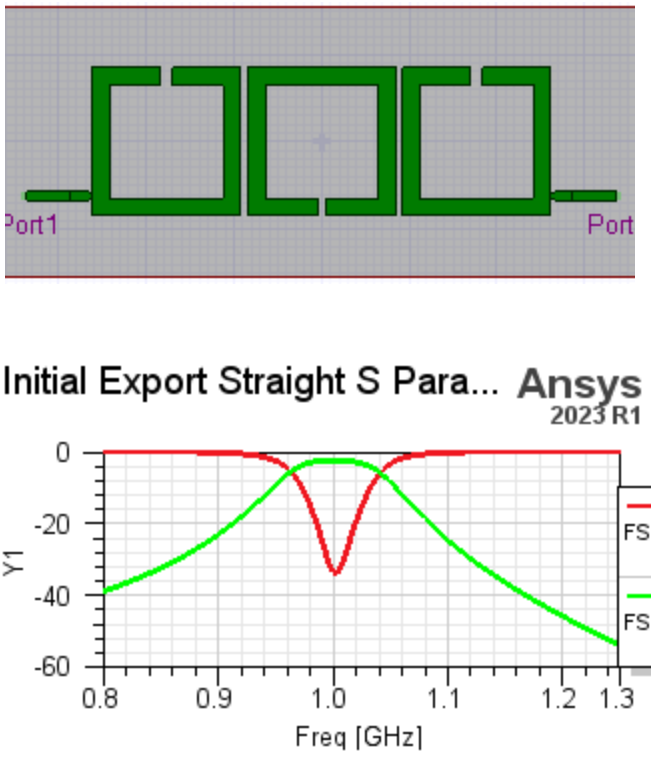


Figure 3: 3 Pole 5% BW Ring Resonator Exported to HFSS 3D Layout

Tuning the resonator separation admittance by -.001 Siemens produces a tuned schematic simulation that like Figure 4.

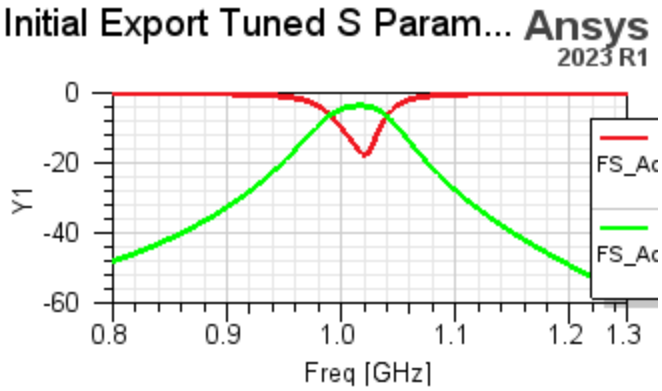


Figure 4: Tuned Schematic Admittance of -.001 Siemens

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reconstructing the filter based on equivalent resonator gap capacitance produces an approximate shape adjustment as the tuned schematic, shown in Figure 5. The slight discrepency represents the small perturbation nature of the tuning adjustments.

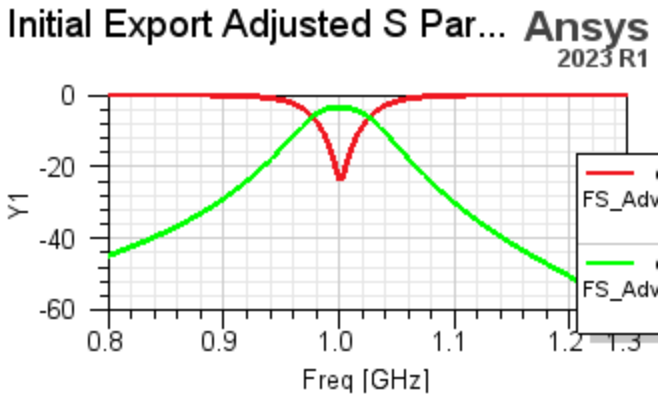
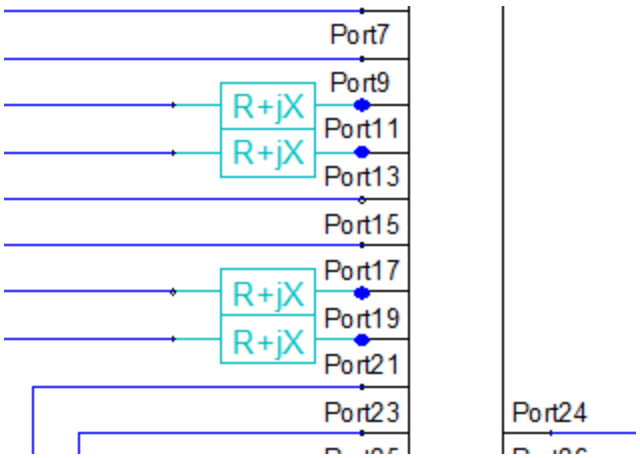


Figure 5: The reconstructed filter

Ring Resonator Gap Tuning

Ring resonators present an additional challenge in that tuning the width of the gap by adjusting the length of the micro strips or strip lines that create the gap does not actually change the gap width in the EM layout, which in turn leads to unacceptable errors in the tuning port schematic simulation and reconstruction of the filters. This problem is mitigated by the insertion of gap adjustment impedance elements, shown in Figure 1, that approximate the change in gap impedance as a function of change in gap width. If small adjustments are made to the gap width, the simulation of the impedance change element remains reasonably accurate, and maintains the acceptable accuracy of the tuned element schematic.



Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1: Ring Resonator Gap Change in Impedance Approximations

Ansys Desktop Automated Port Tuning

This chapter contains the following topics:

- HFSS Horizontal Tuning Ports
- Deembedding the HFSS Tuning Ports
- Desktop Automated Port Tuning Entries
- Circuits/Sonnet Automated Port Tuning
- Automated Port Tuning Tutorial

Horizontal Series Ports

HFSS and HFSS 3D Layout support single horizontal ports to replace dual ports for tuning ports that contain only a single micro strip or strip line elements between the ports. Tuning ports used to connect resonator separation capacitors and coupled micro strip or strip line elements continue require dual ports.

HFSS 3D Layout may employ Horizontal Tuning Ports to reduce the number of ports being used in the EM simulation and to reduce the complexity of the tuning port schematic. Horizontal ports have some additional deembedding requirements that are accounted for in the schematic design variables, namely KHdbed. KHdbed it not set up by default for tuning, but the designer may at his or her discretion adjust the value up or down to maximize deembedding accuracy, if desired or needed. Horizontal Series Ports may be deselected in the export panel, if desired and shown in Figure 1, in which case the Automated Port Tuning schematic will implement all tuning ports as the standard dual ports.

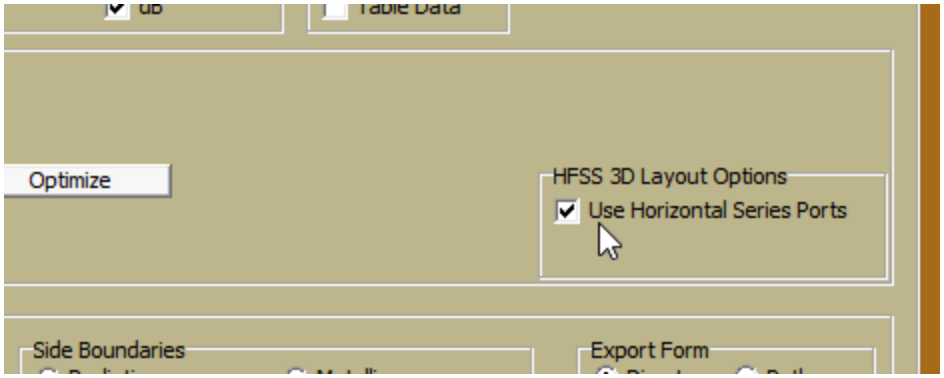


Figure 1: Selecting and Deselecting Horizontal Series Ports

The effect of the Horizontal Series Ports is to place one side of the series element on the single horizontal series ports and a ground on the other side. The horizontal port is set up in HFSS 3D

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Layout to simulate in the same manner as though a dual port pair were used to simulate the element. This may be seen in Figure 3 below.

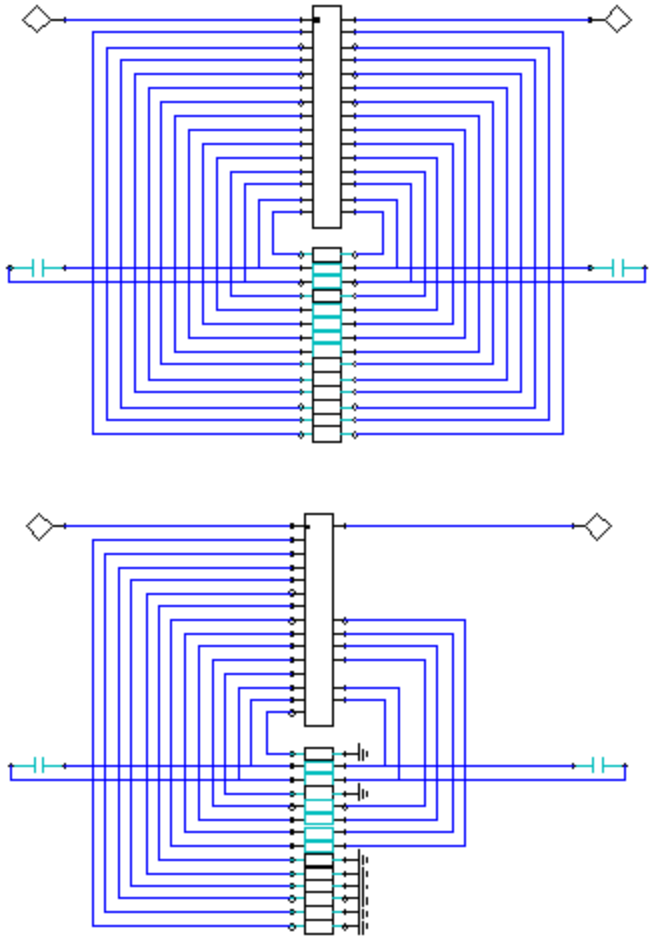


Figure 2: Dual Ports and Horizontal Ports Both Simulating the Same Design

Deembedding the Tuning Ports

HFSS and HFSS 3D Layout ports contain potential parasitic effects that have an option be removed by the port deembedding selection. This selection assumes widely separated ports, and will not produce useful results for closely spaced ports used for Automated Port Tuning. Consequently, the interactions between the closely spaced ports must be deembedded externally on the tuning port schematic. FilterSolutions provides an approximate deembedding solution of inserting a negative length micro strip or strip line in series with the port elements and of default length -0.7 times the length of the tuning port gap. Since there is no "One Size Fits All"

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

solution to the closely spaced dual port problem, a tuning variable, "Kdbed" is created in the tuning port schematic parameters that may be used to fine tune this value between a range of 0 and 2.0 to adjust the length of the deembedding micro strip or strip line for the most accurate setting.

Horizontal ports require slightly more deembedding, and are bumped up by a default factor of 1.3, which my be manually adjusted further by the designer.

Figure 1 shows the Kdbed deembedding tuning parameter in the Desktop tuning tool. If the tuning port schematic simulation is compared directly to the HFSS or HFSS 3D Layout simulation, sliding the Kdbed variable up and down will move the tuning port schemtic simulation up and down in frequency. Set Kdbed to the value that produces the best match between the two simulation traces.

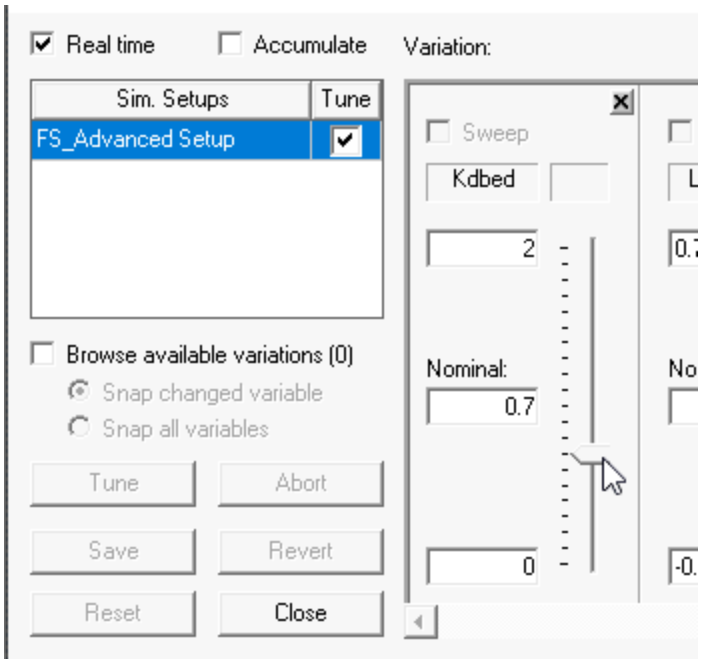


Figure 1: Tuning the Port Deembedding Constant

Desktop Automated Port Tuning Entries

The Electronic Desktop Automated Port Tuning entries are shown in Figure 1. Table 1 defines the various selections and controls available to HFSS and 3D Layout users.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

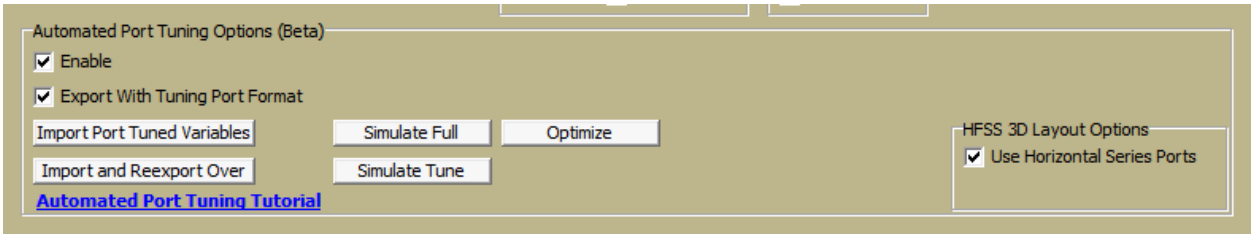


Figure 1: HFSS and 3D Layout Automated Port Tuning Selections and Controls Box

Control	Function
Enable	Check to display the Automated Port Tuning functions and controls
Export With Tuning Port Format	Install tuning ports in HFSS and 3D Layout EM layouts, and create a Circuits tuning port schematic with a sub circuit block for the HFSS or 3D Layout layout.
Import Port Tuned Variables	Import the tuned variables from the Circuits parameters, and use them to reconstruct the filter layout in FilterSolutions.
Import and Reexport Over	Perform a "Import Port Tuned Variables" virtual button push, followed by an immediate reexport in tuning port format right back to Electronic Desktop. This button used as a time saver when cycling automated port tuning operations.
Simulate Full	Press this to simulate the full Automated Port Tuning project, which includes HFSS, 3D Layout and Circuits schematics.
Simulate Tune	Simulate only the Circuits tuning port schematic. For HFSS 3D Layout, this is sufficient. For HFSS, the HFSS schematic must already be simulated.
Use Horizontal Series Ports	Check to simply the Automated Port Tuning project by using horizontal series ports, where feasible. Available with 3D Layout, only. Other options may be available for third party EM

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	when in use.
Automated Port Tuning Tutorial	View the tutorial to help get started with Automated Port Tuning.

Circuits/Sonnet Automated Port Tuning

Ansys Circuits may use Sonnet ***em***® as the EM tool for Automated Port Tuning if Sonnet is licensed for use on the designer's PC. Sonnet em is controlled by FilterSolutions as needed to simulate the planar design with Sonnet Co-Calibrated® ports, and installing the result back into Circuits as an S-parameter file in one clean sweep. The Sonnet project may be viewed with the **View Sonnet Project** control, and the final design may be simulated by Circuits when tuning ports are deselected.

To select the Sonnet as the Automated Port Tuning EM tool, launch the Ansys export panel and select **Insert Circuit Design** and then **Sonnet** as the **Select Simulator** selection, as shown in Figure 1. Port numbering in the Sonnet project is sequenced to match the Ansys Circuits S-Parameter block default pin locations.

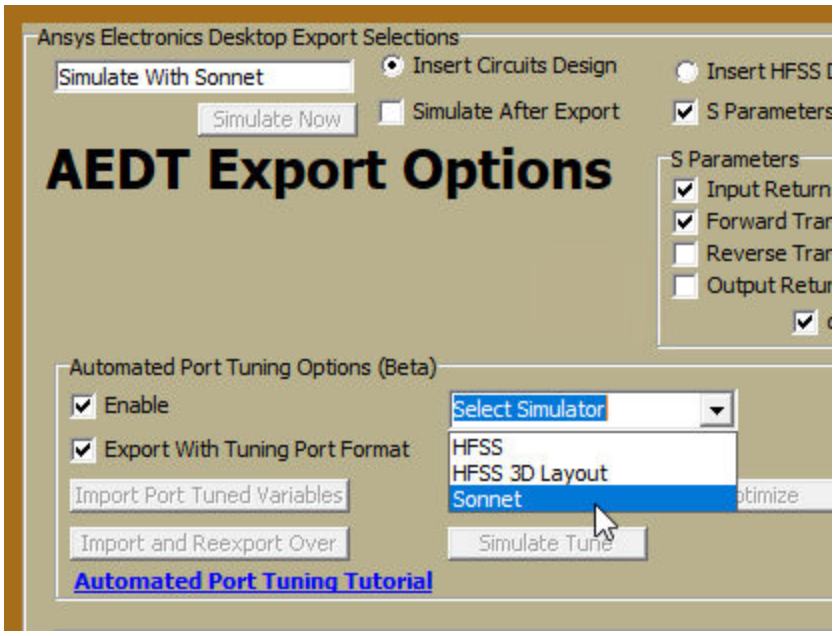


Figure 1: Selecting Sonnet ***em***® as the Automated Port Tuning EM simulator.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To view the Sonnet project with tuning ports installed, select **View Sonnet Project** after selecting the Sonnet simulator, as shown in Figure 2.

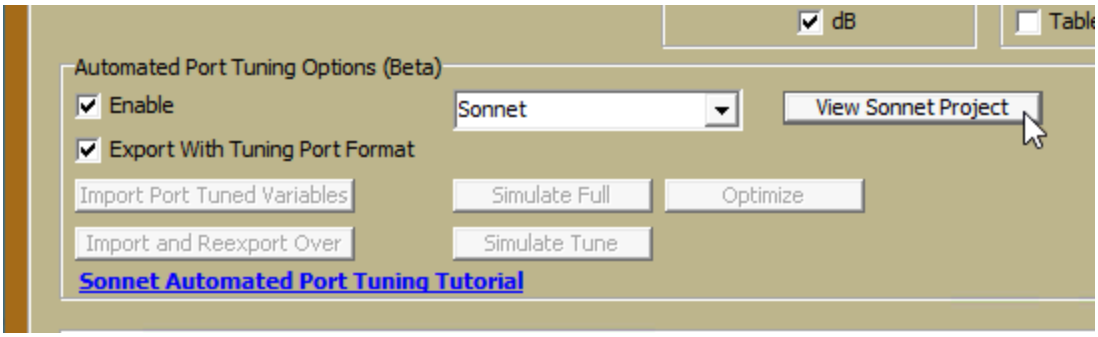


Figure 2: Viewing the Sonnet Project With Tuning Ports Installed.

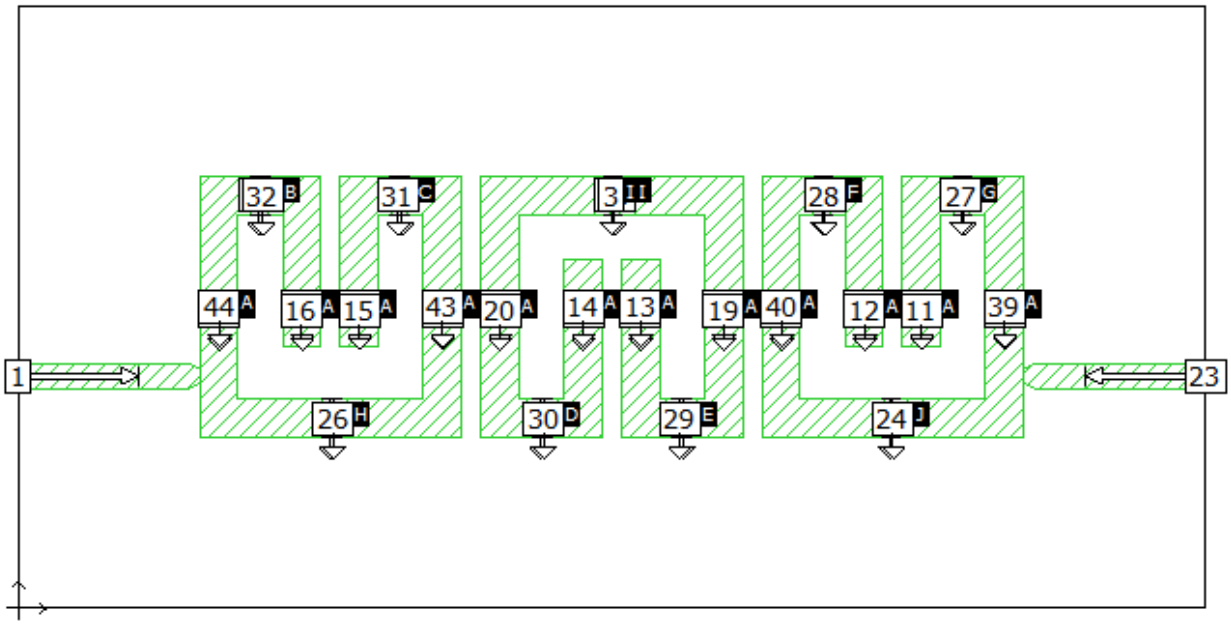


Figure 3: The Sonnet Project With Tuning Ports Installed.

Exporting Sonnet Tuning Port format to AEDT

FilterSolutions will fist invoke the Sonnet ***em*** simulator to simulate the tuning port layout, and will load the results into to an S-Parameter file containing all of the ports from the tuning port layout. The Circuits tuning port schematic will use an N-Port block in the Circuits schematic to simulate

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the S-Parameter file and thereby reproduce the results of the Sonnet ***em*** simulation. It is recommend to place the ***em*** simulation start drop down in "Auto Start" mode so that Sonnet will begin ***em*** simulations immediately upon export, shown in Figure 4.

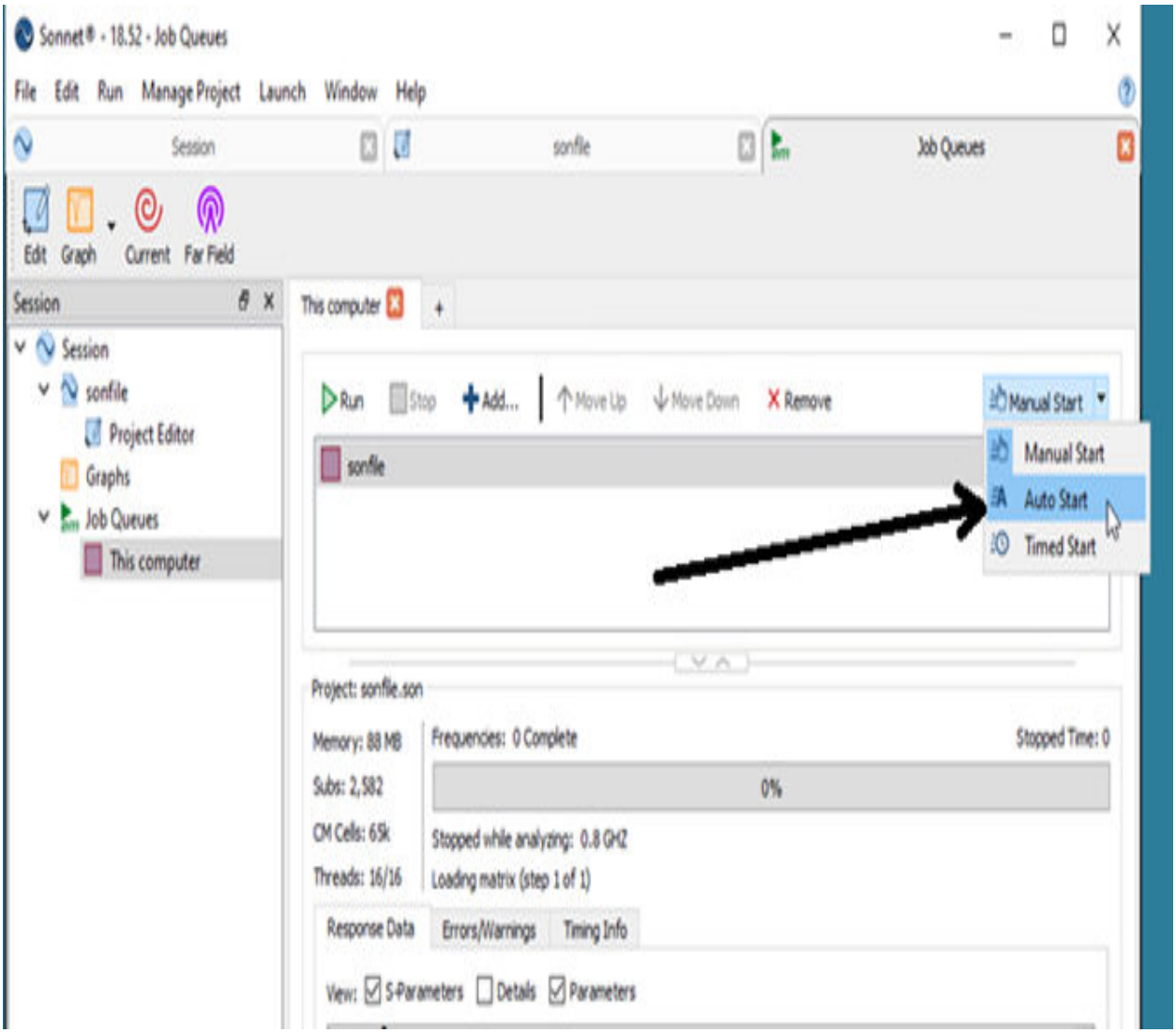


Figure 4: Selecting **Auto Start** in the Sonnet ***em*** Simulation

Automated Port Tuning Tutorial

Introduction

This tutorial walks through an example of automated port tuning with Ansys Nuhertz Filtersolutions, Circuit, HFSS, and HFSS 3D Layout using the project titled **Sample Minipins**

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

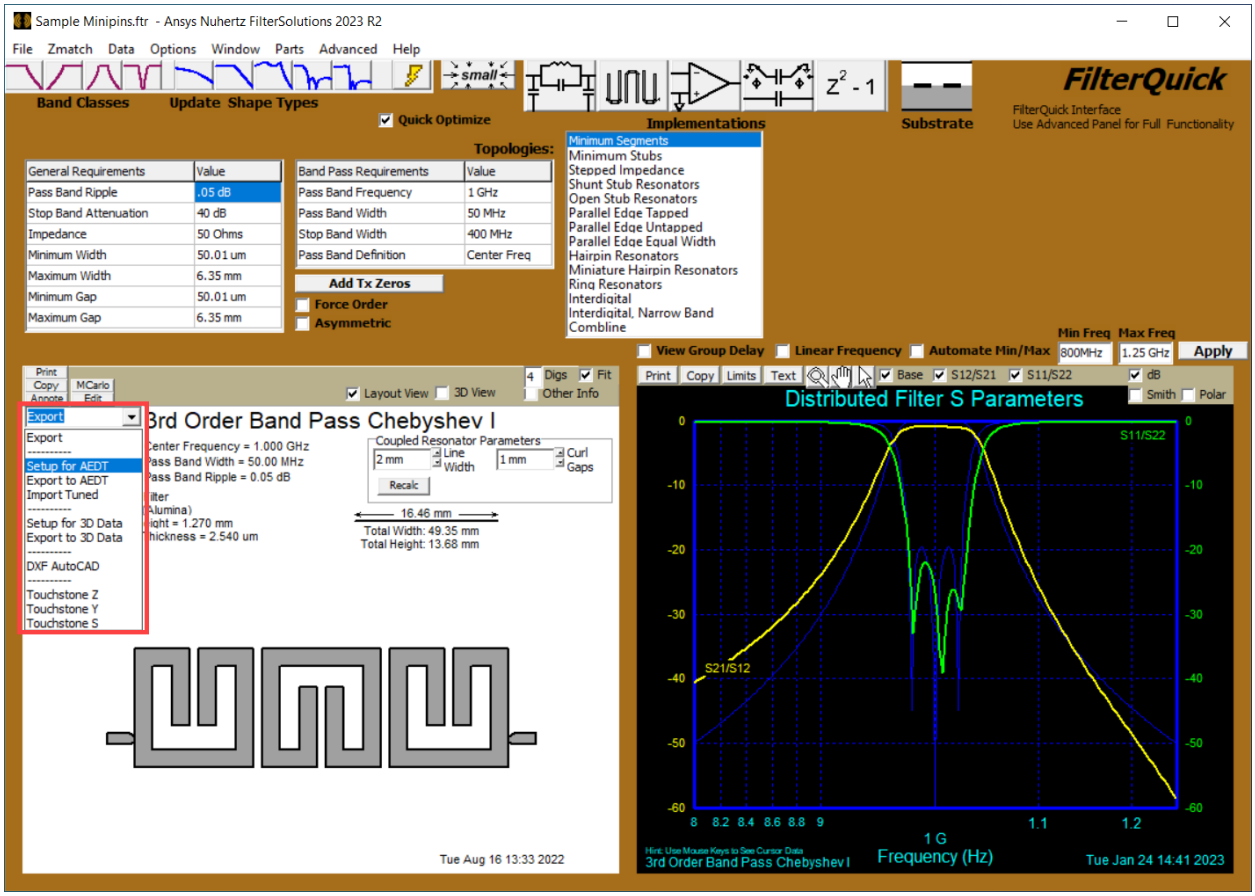
found in **C:\Program Files\AnsysEM\v###\Win64\Nuhertz\Tutorials** folder of your software download. The v### corresponds to the version number of your software download, e.g. v241 for version 24.1.

About Automated Port Tuning:

- A fast means to optimize Ansys Nuhertz FilterSolution (ANFS) planar HFSS and HFSS 3D Layout designs using Circuit cosimulation.
- Employs HFSS and HFSS 3D Layout internal tuning ports to adjust resonator length and separation.
- Generally two to four electromagnetic (EM) simulations required for completion.
- Eliminates or greatly reduces the need for traditional bulk EM optimization.

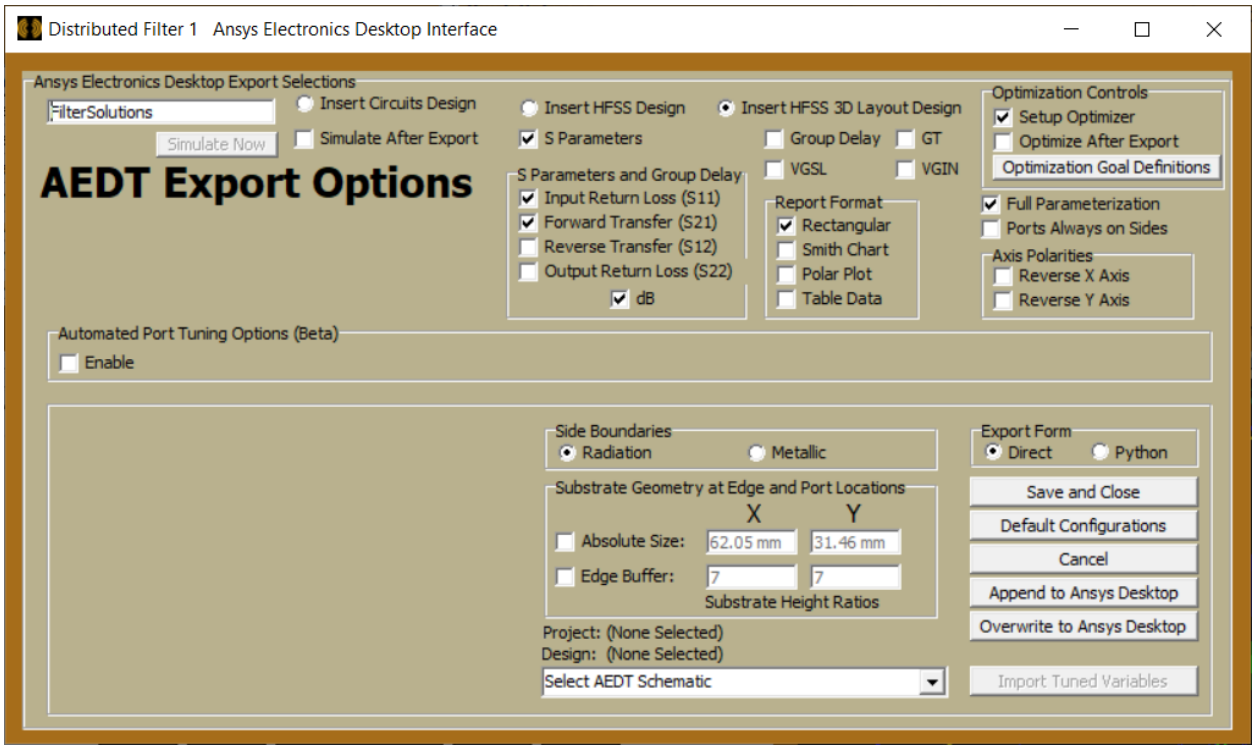
Steps:

1. Design Filter with ANFS and launch **Ansys Electronics Desktop Export** panel by selecting **Export** > **Set up for AEDT**.



The AEDT Export Panel will open:

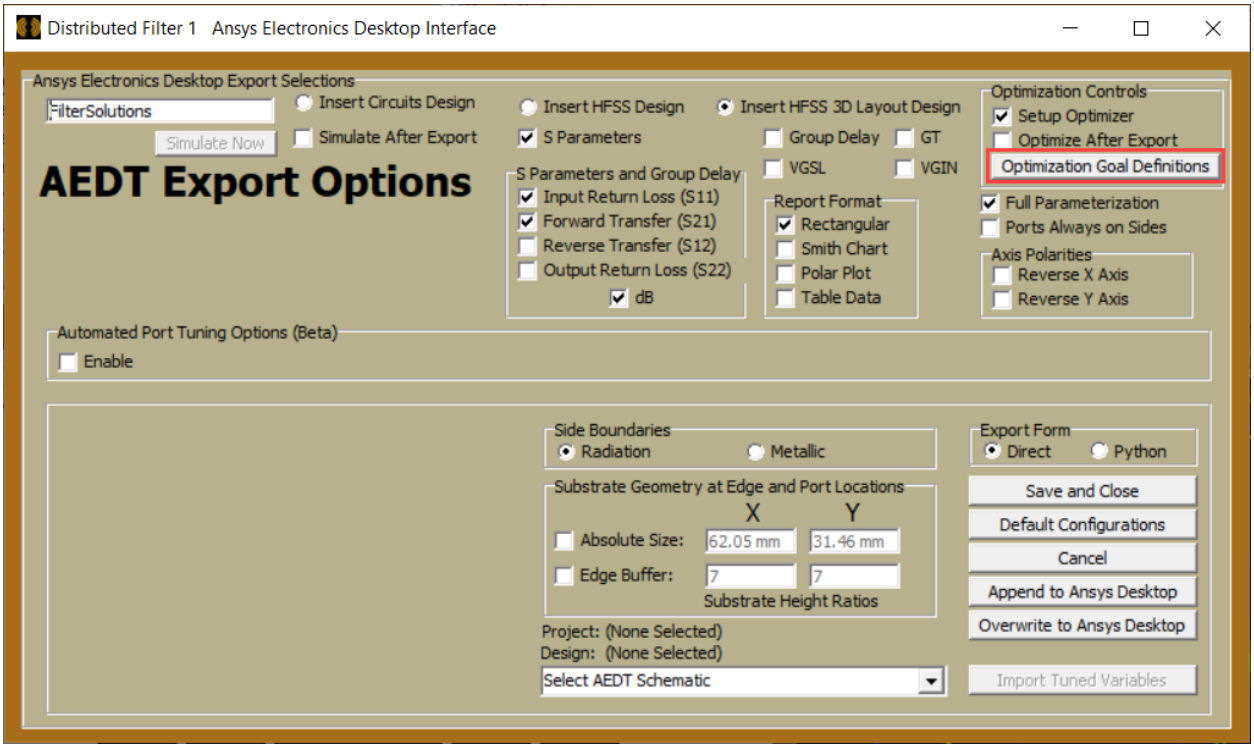
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



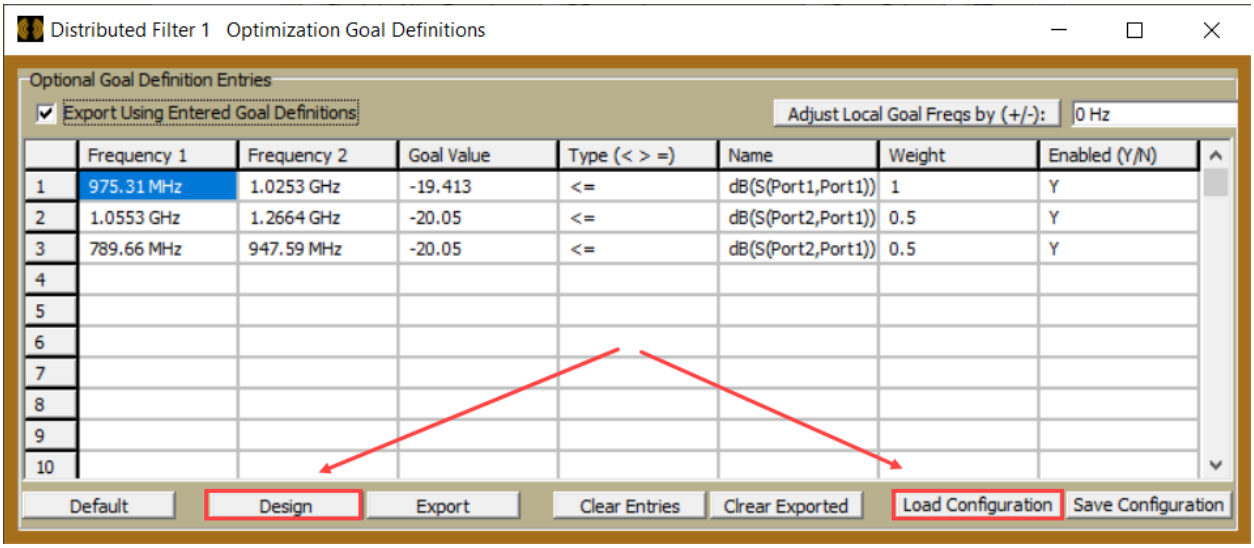
2. Optional- If it is desired to customize the goal limits:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

i. Click **Optimization Goal Definitions** in the **Optimization Controls** group box.

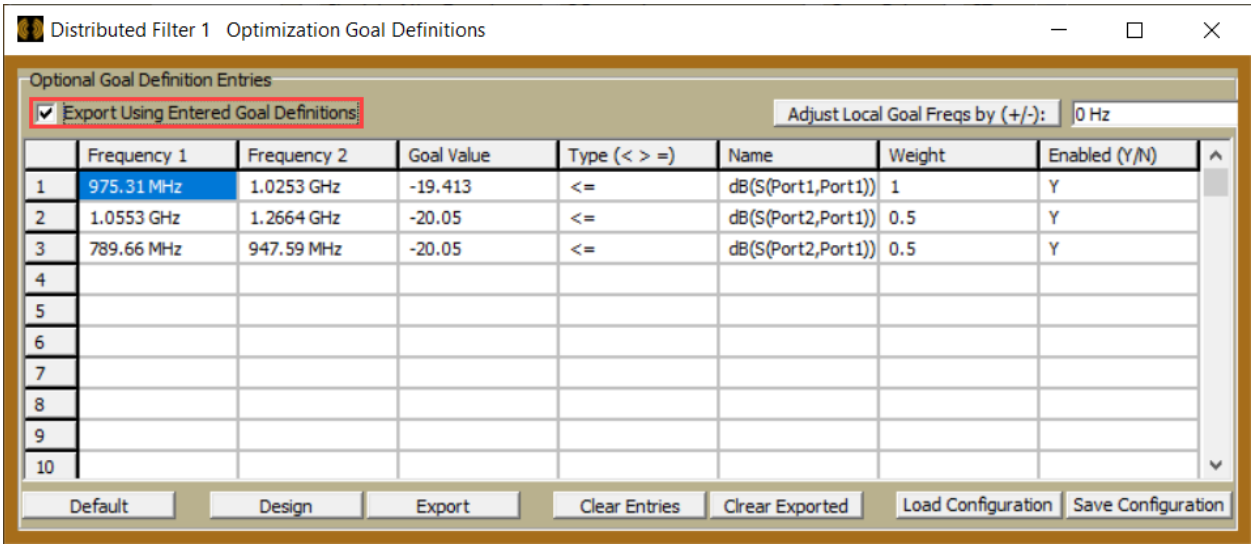


ii. Optional- Click **Design** or **Load Configuration** in the pop-up panel to fill in the existing design goals or previously saved goals, respectively.



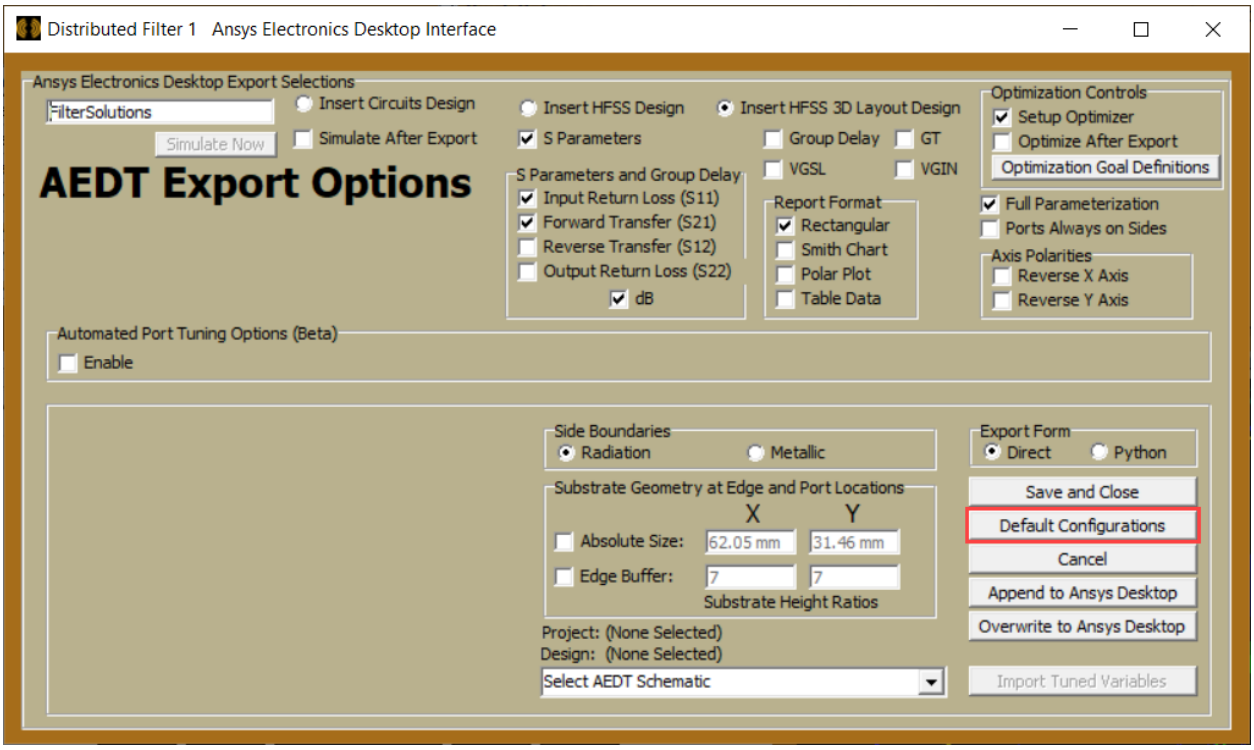
iii. Select **Export Using Goal Definitions** to use table defined goals in export.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



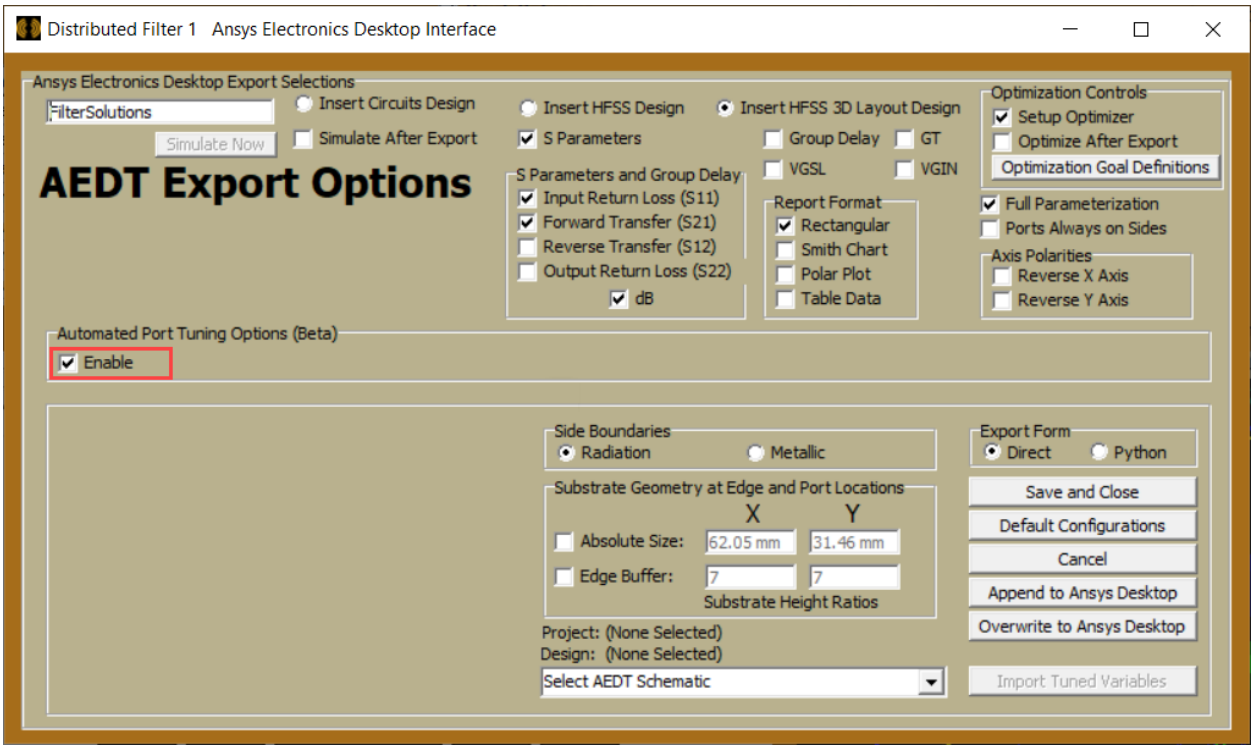
3. Set up the Export for Automated Port Tuning with HFSS or HFSS 3D Layout

   i. Select **Default Configurations** in the **AEDT Export Panel**.



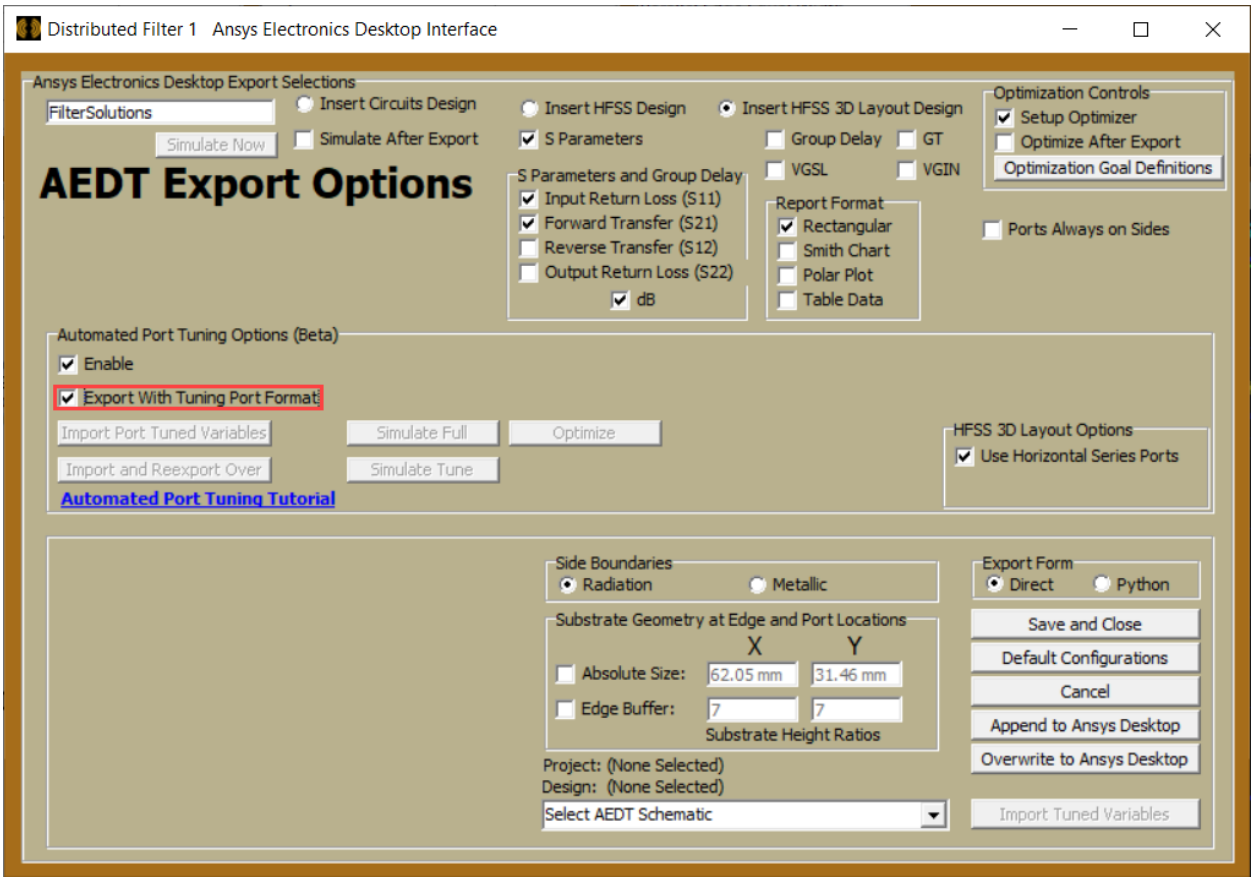
   ii. Select **Enable** in the **Automated Port Tuning Options** group box.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



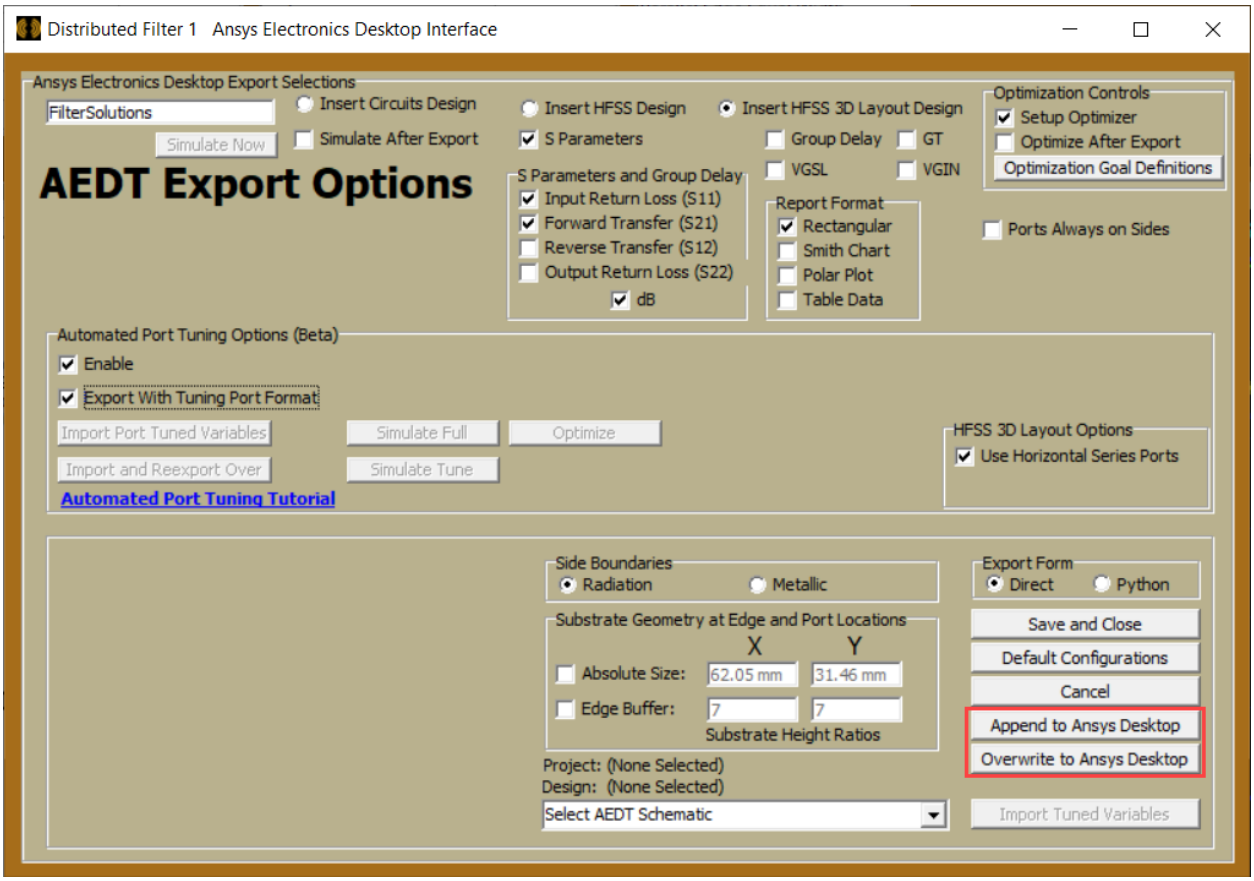
iii. Select **Export with Tuning Port Format** in the **Automated Port Tuning Options** group box

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



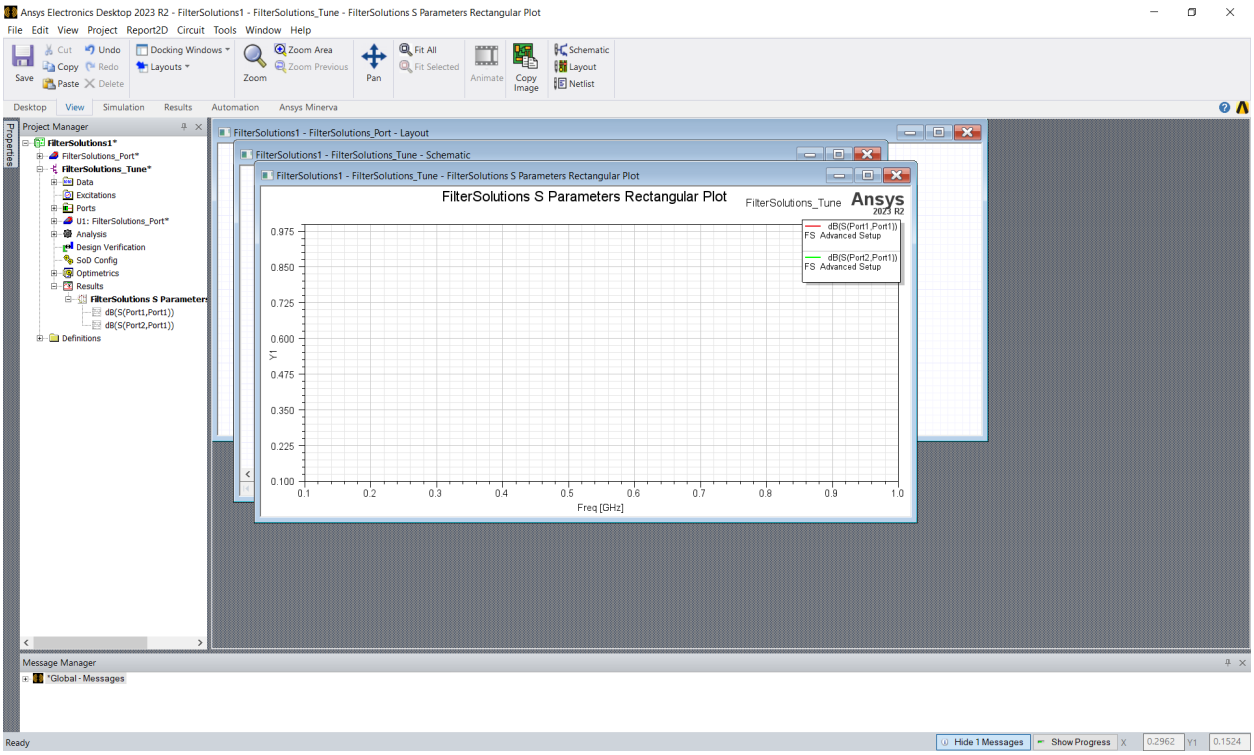
iv. Choose **Append to Ansys Desktop** or **Overwrite to Ansys Desktop**.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This step will launch AEDT with the ANFS project.

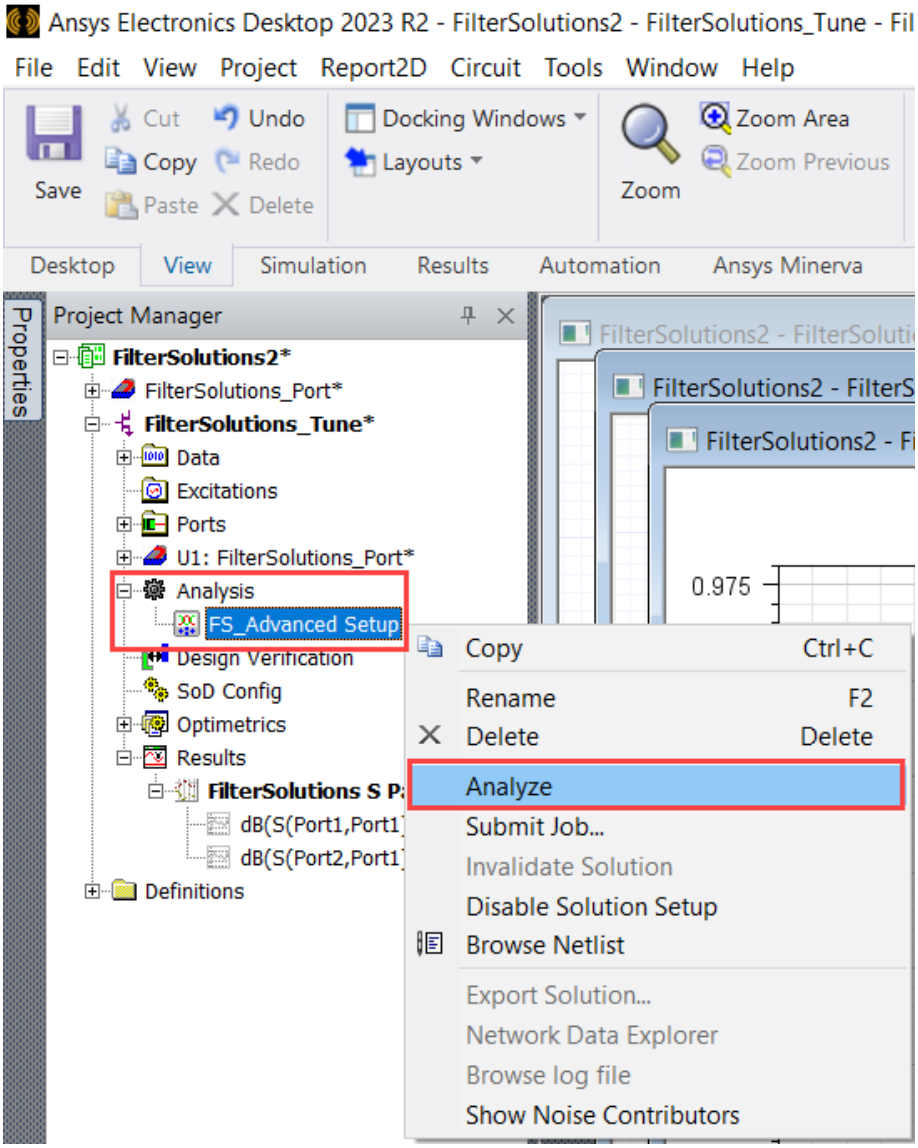
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Simulate HFSS or HFSS 3D Layout- This step can be done in Ansys Desktop (4.a) OR in the Ansys Electronics Desktop Export form from the previous steps (4.b).

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. Expand the Desktop project manager and right-click **Analysis** . Select **Analyze**. OR



b. In the Export form's **Automated Port Tuning Options** group box.

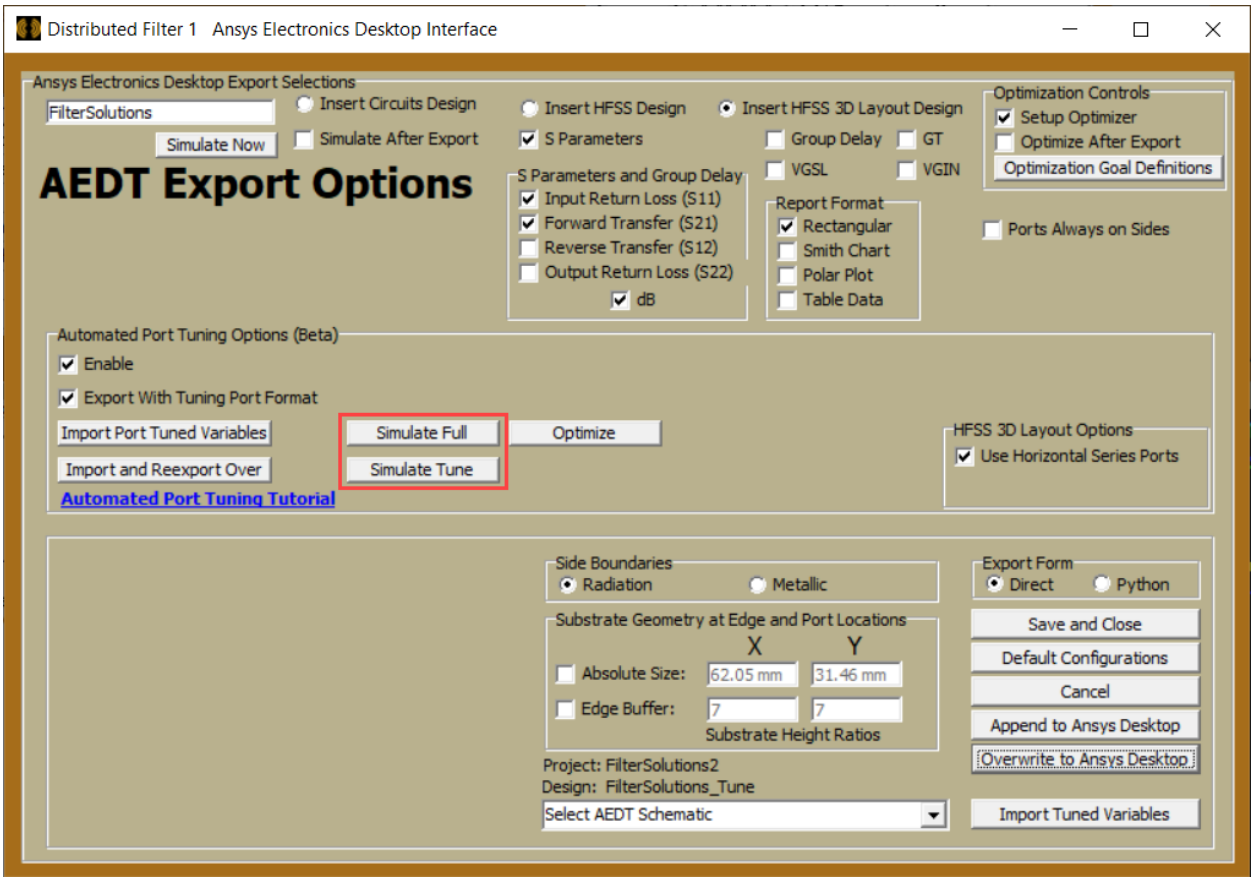
Choose **Simulate Full** to simulate both EM and Circuits schematic OR

Choose **Simulate Tune** to simulate just the circuit schematic.

Note: Desktop will automatically simulate HFSS 3D Layout if the HFSS 3D layout is not valid.

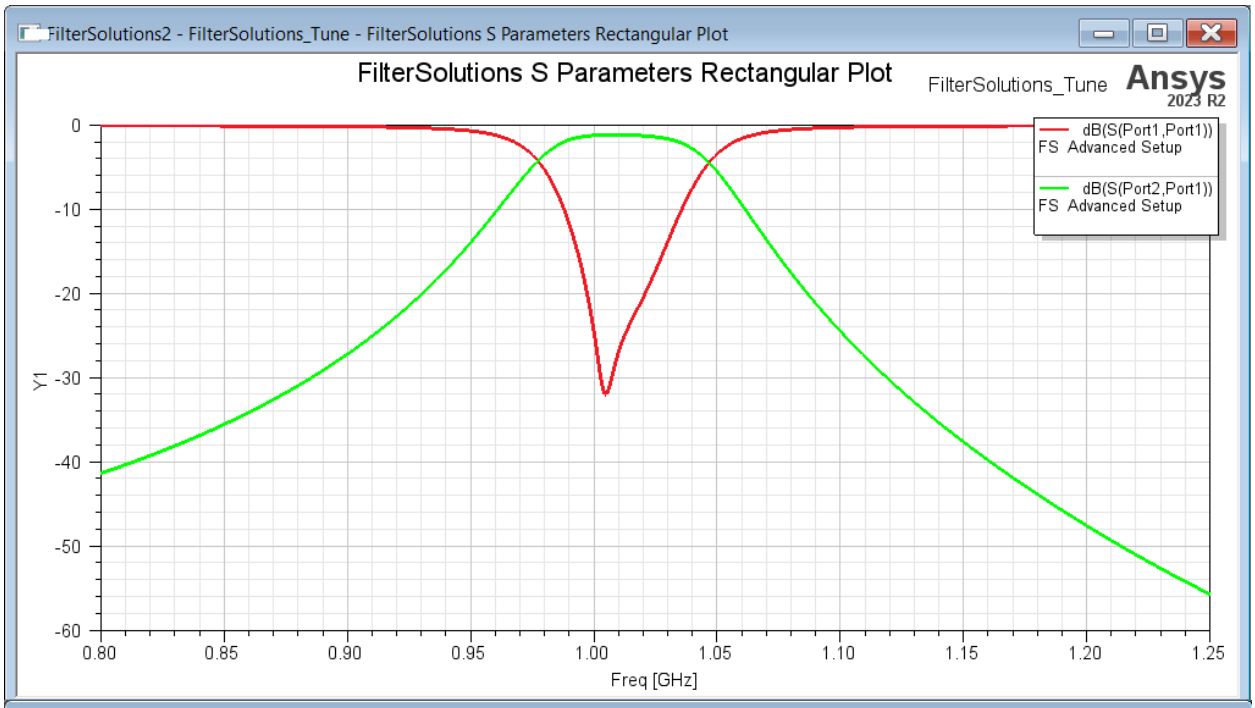
Note: Desktop will NOT simulate HFSS, regardless of whether the HFSS simulation data is valid or not.

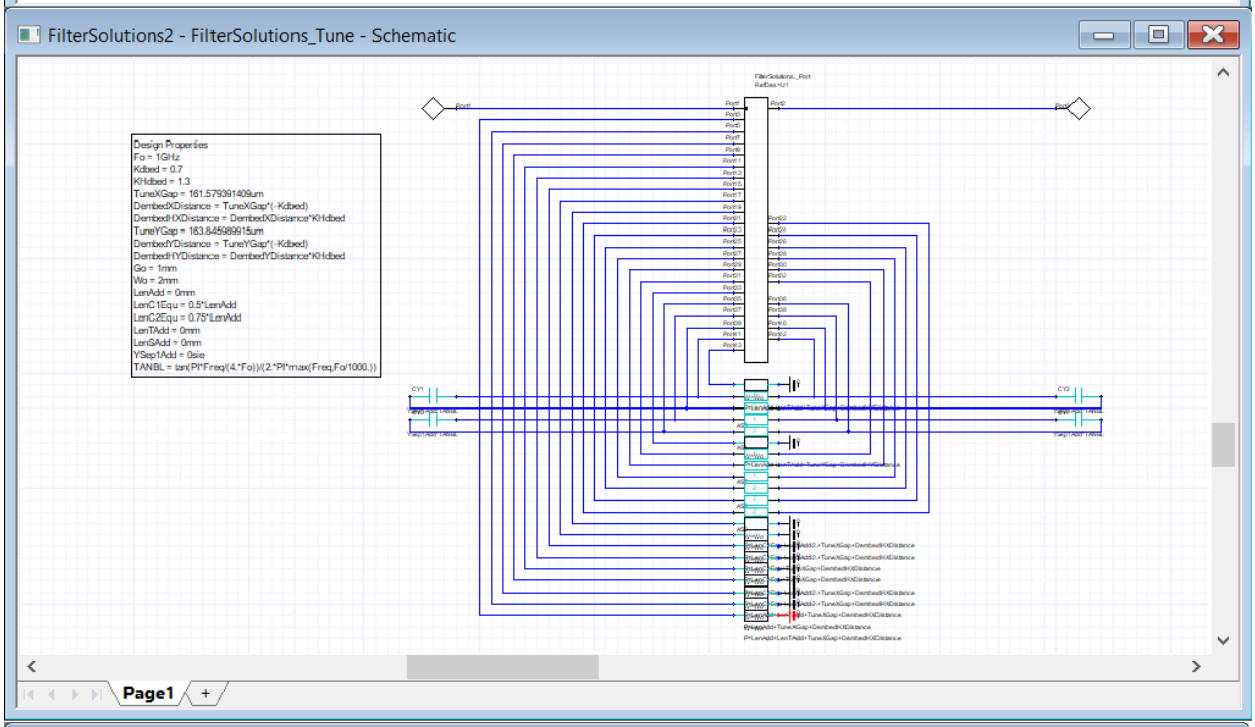
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

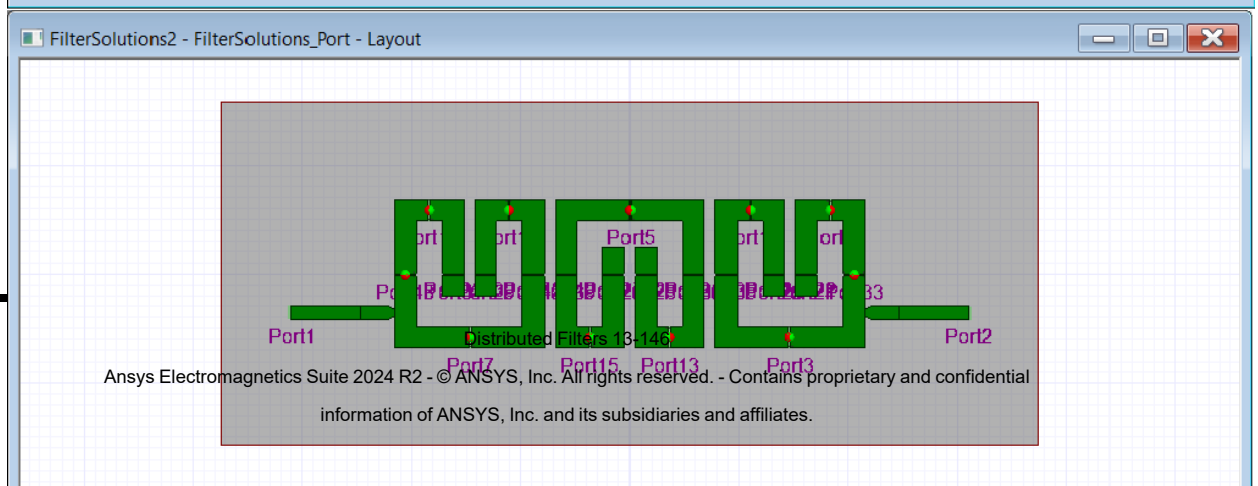


5. Confirm the HFSS 3D Layout simulation: Note the use of horizontal tuning ports for single micro strip lines in HFSS 3D Layout. Horizontal tuning ports are optional and only supported for use in HFSS 3D Layout

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



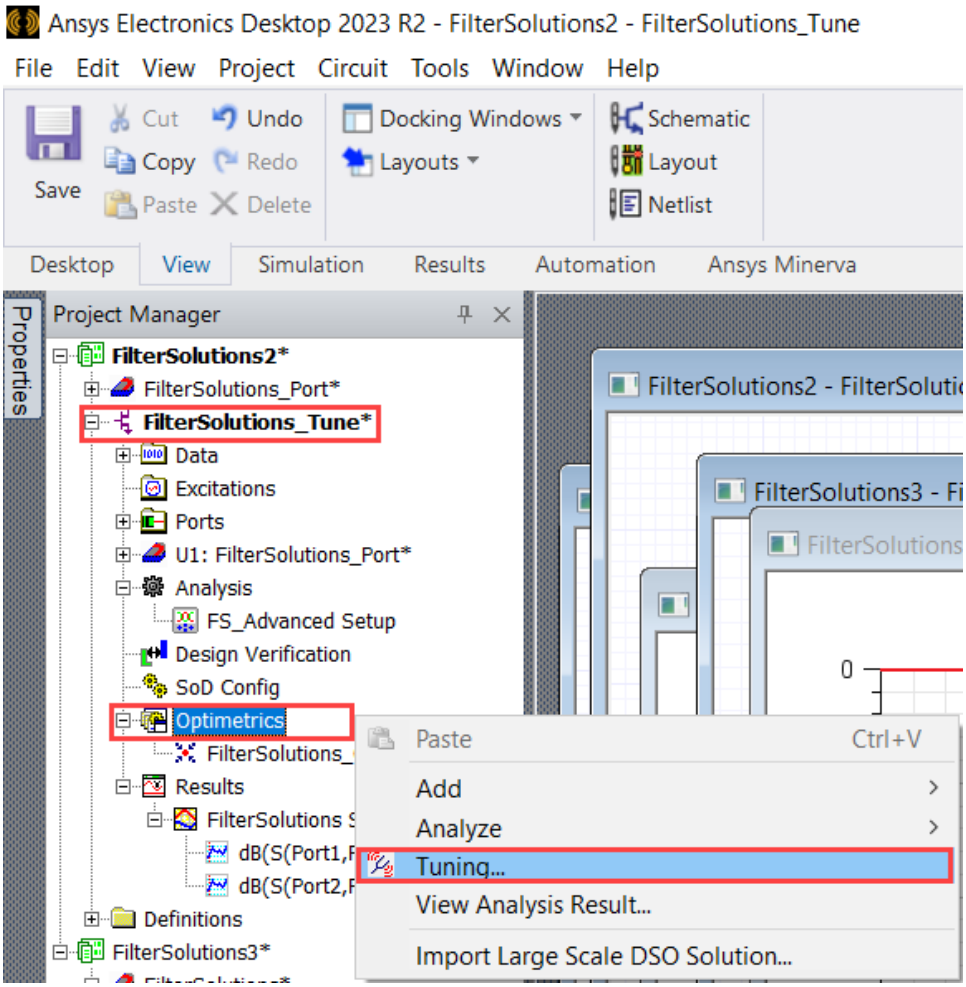




Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

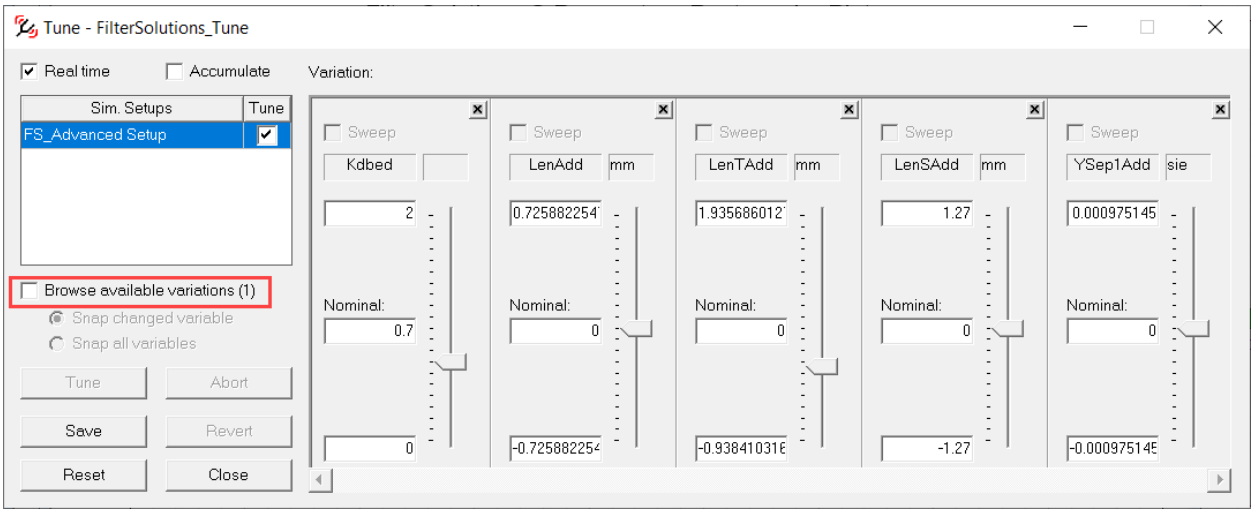
6. Optional for increased accuracy- Tune the Port Deembedding Constant:

   i. In Ansys Desktop's Project Manager, expand the FilterSolutions_Tune project, right-click **Optimetrics**, select **Tuning**.

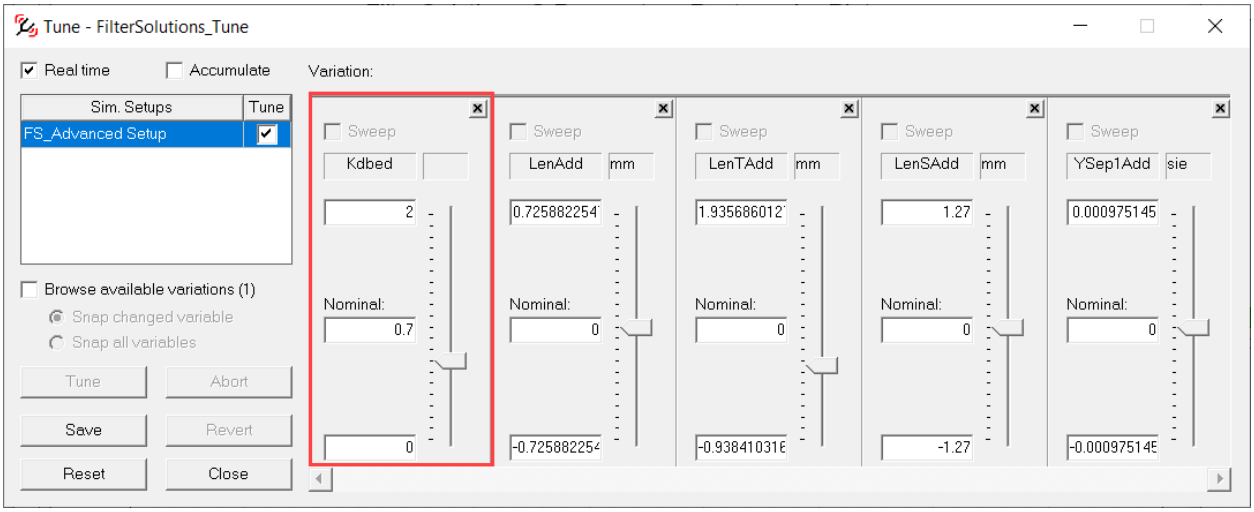


   ii. In the Tune Dialogue, deselect **Browse available variations**.

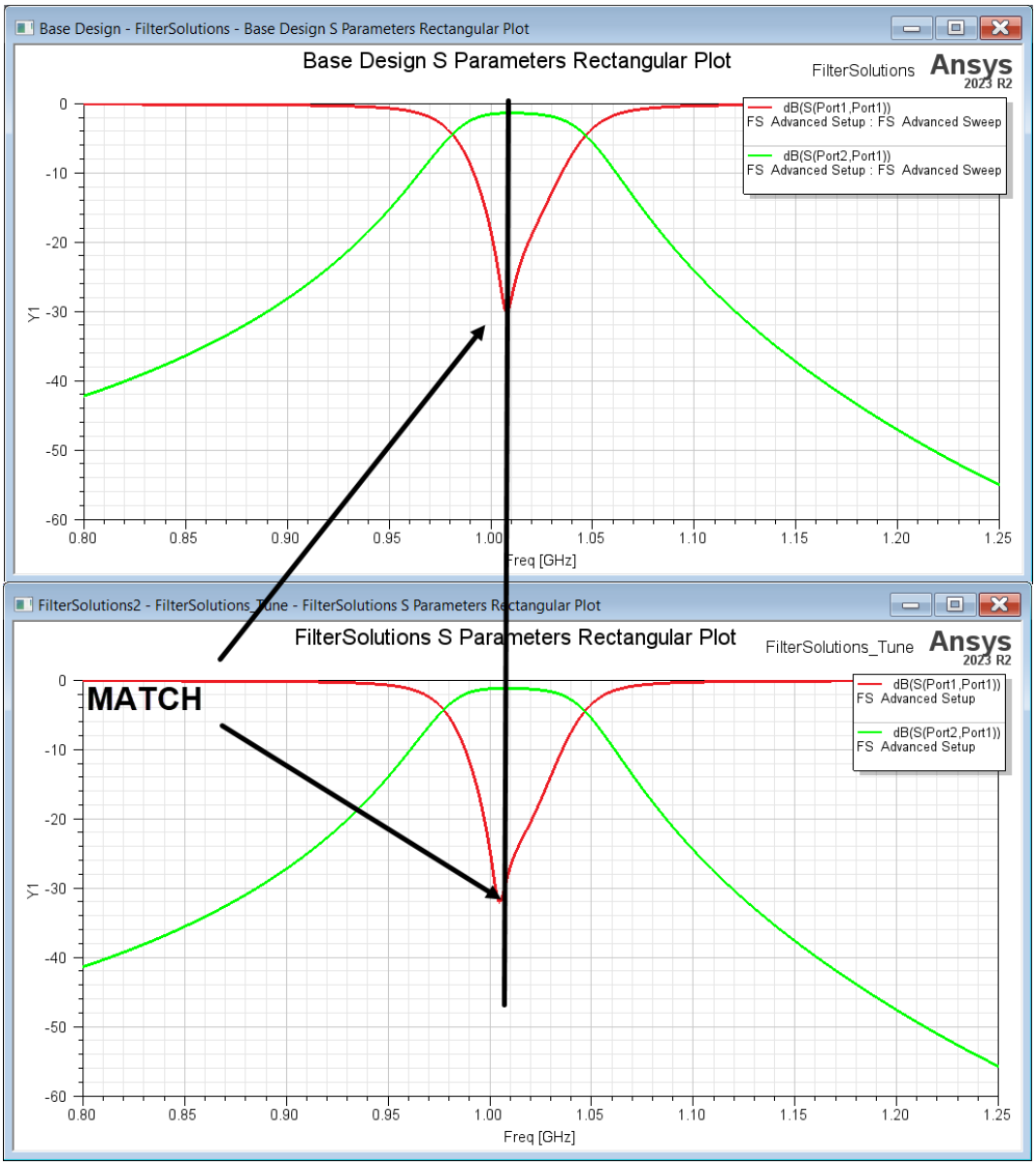
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



iii. Use the slider to tune until the center frequency of the tuning port design matches the center frequency of the design with no tuning ports.



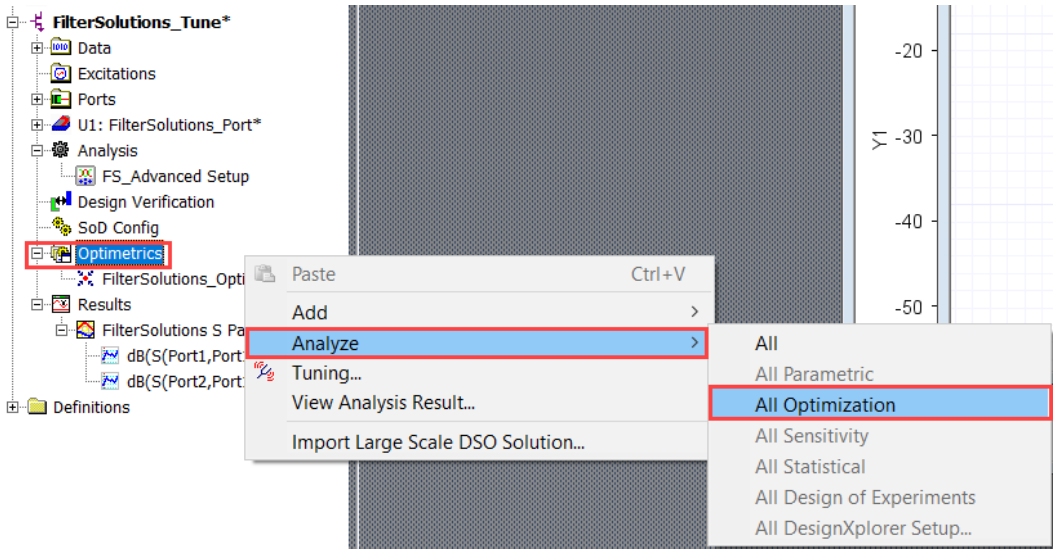
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



7. Optimize the Tuning Port Design

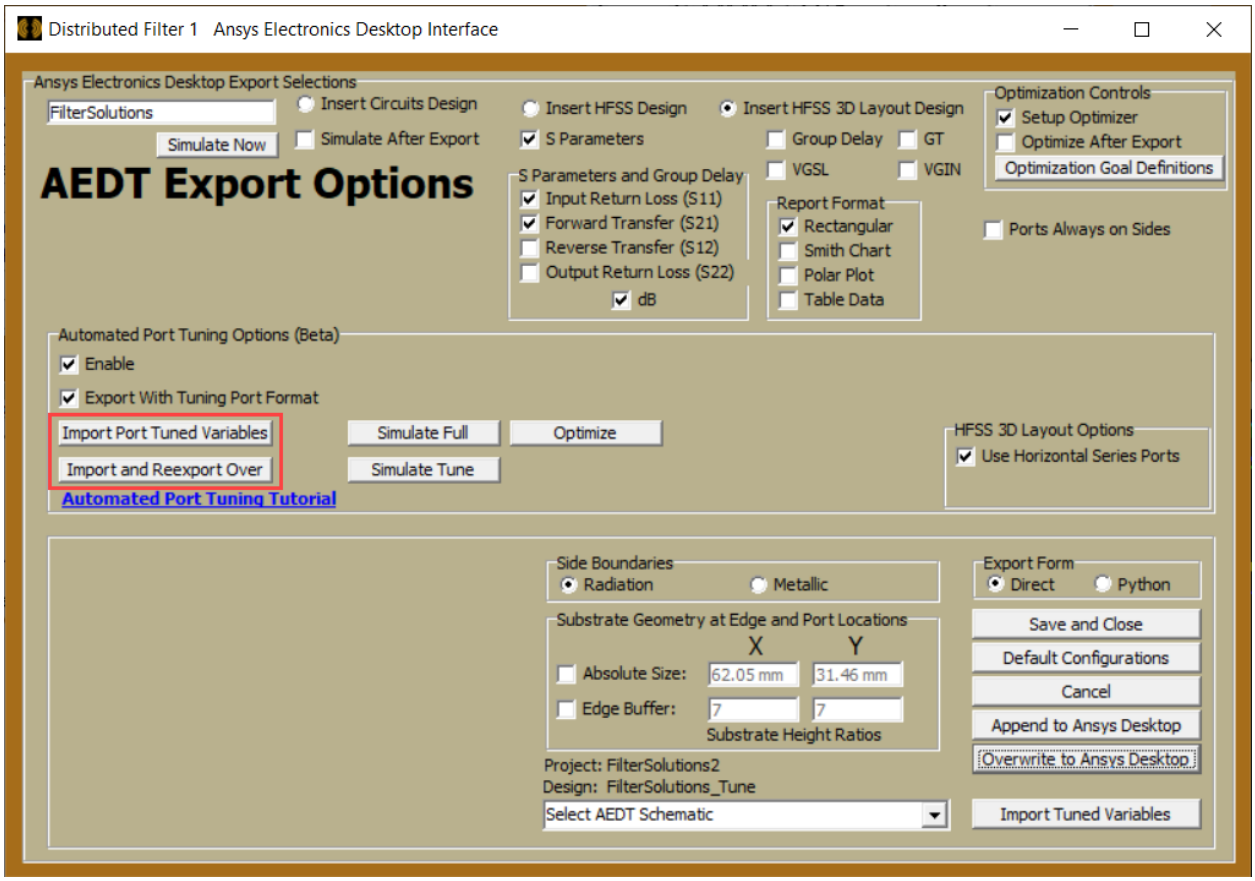
   i. Right-click **Optimetrics**, select **Analyze** > **All Optimization** OR select **Optimize** in the ANFS Export panel.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



8. Reconstruct the Optimized Design- After the optimization process completes:
   a. Select **Import Tuned Variables** to import and reconstruct the design in ANFS. OR
   b. Select **Import and Reexport Over** to send the reconstructed design immediately back to AEDT for another optimization cycle. Repeat as necessary.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



9. Export Preliminary Final Design- once the design optimization cycles produces a design of sufficient accuracy:
   i. Export the preliminary final design with no tuning ports installed to AEDT and simulate.
10. Optional- Perform Direct EM Tuning or Optimizing: If more accuracy is still needed, the Preliminary Final Design may be further tuned or optimized directly with AEDT tuners and optimizers.

    In this example, the center frequency is 0.5% too low. We can edit the side length variable (ResLen) by 0.5% down to 9.837 mm and simulate again. The final design is observed to center on 1 GHz.

Sonnet Tuning Port Projects

This chapter contains the following topic:

- Sonnet Turning Port Project
- Sonnet Automated Port Tuning Tutorial
- Sonnet Parameter Optimization Tutorial

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sonnet Tuning Port Project

Designers who posses a Sonnet license for their PC may export their Sonnet project with Co-Calibrated® tuning ports installed for use on any Sonnet supporting simulator by checking the **Include Co-Calibrated Tuning Ports** check box, as shown in Figure 1. The resulting Sonnet project will include tuning ports numbered to match the default Microwave Office sub circuit block default pin locations, as shown in Figure 2.

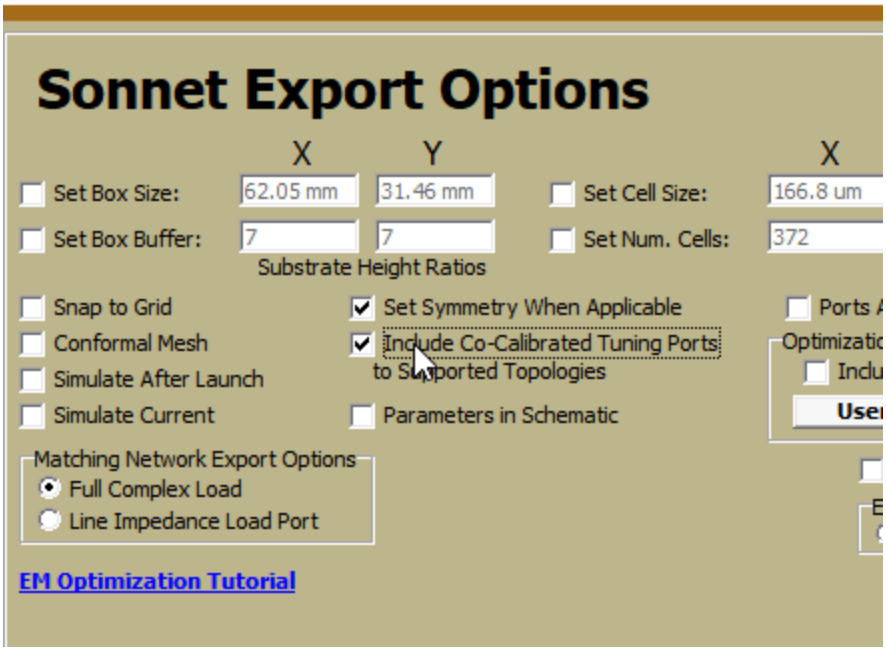


Figure 1: Exporting to a Sonnet Project With Tuning Ports

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

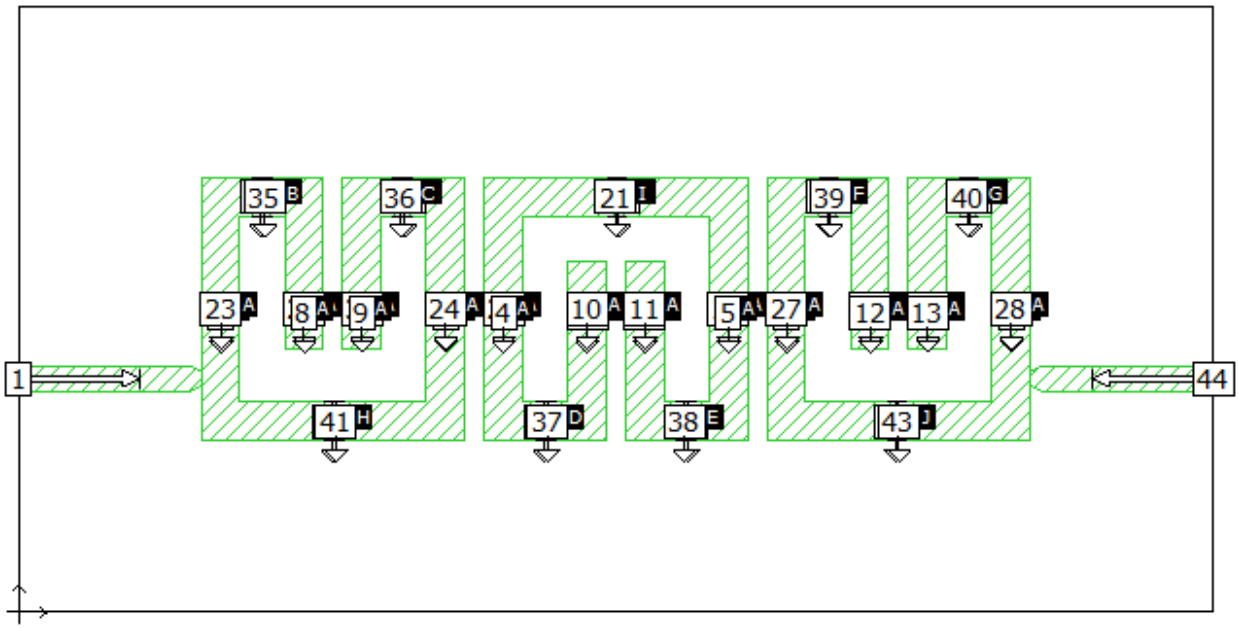


Figure 2: Exported Sonnet Project with Tuning Ports Installed

Sonnet Automated Port Tuning Tutorial

Introduction

This tutorial walks through an example of automated port tuning with Ansys Nuhertz Filtersolutions, Circuit, and Sonnet using the project titled **Sample Ringres** found in **C:\Program Files\AnsysEM\v###\Win64\Nuhertz\Tutorials** folder of your software download. The v### corresponds to the version number of your software download, e.g. v241 for version 24.1.

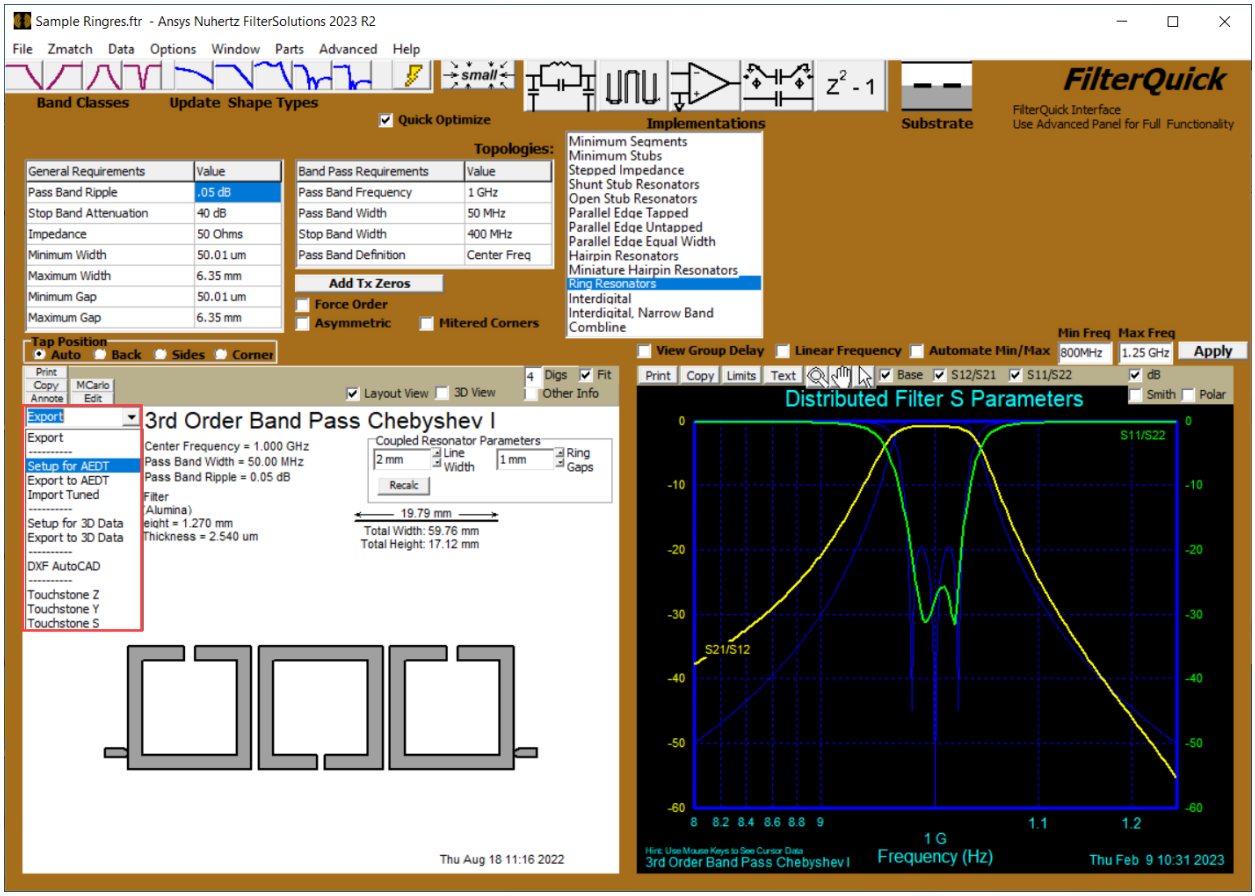
About Automated Port Tuning:

- A fast means to optimize Ansys Nuhertz FilterSolution (ANFS) 3D Layout designs using Circuit and Sonnet S-Parameter based EM cosimulation.
- Employs Sonnet layout internal Co-calibrated tuning ports to adjust resonator length and separation.
- Generally two to four electromagnetic (EM) simulations required for completion.
- Eliminates or greatly reduces the need for traditional bulk EM optimization.

Steps:

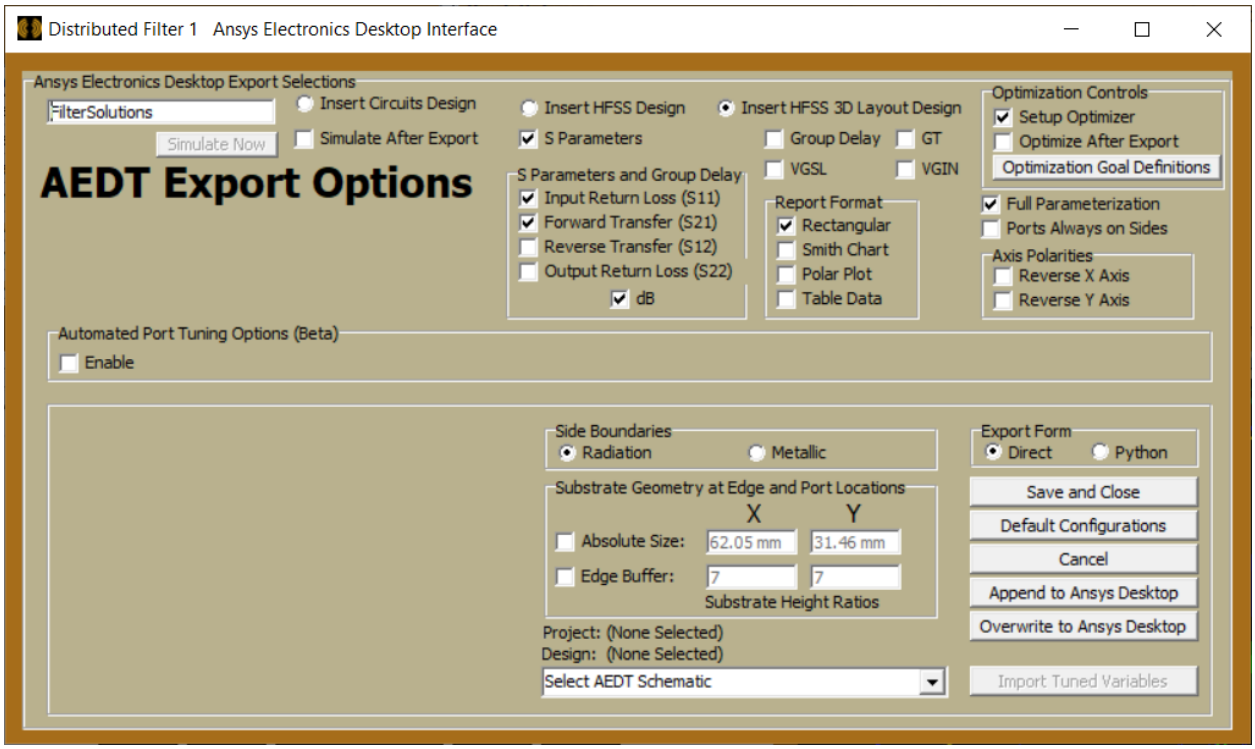
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Open the Sample Ringres project with ANFS and launch **Ansys Electronics Desktop Export** panel by selecting **Export** > **Set up for AEDT**.



The AEDT Export Panel will open:

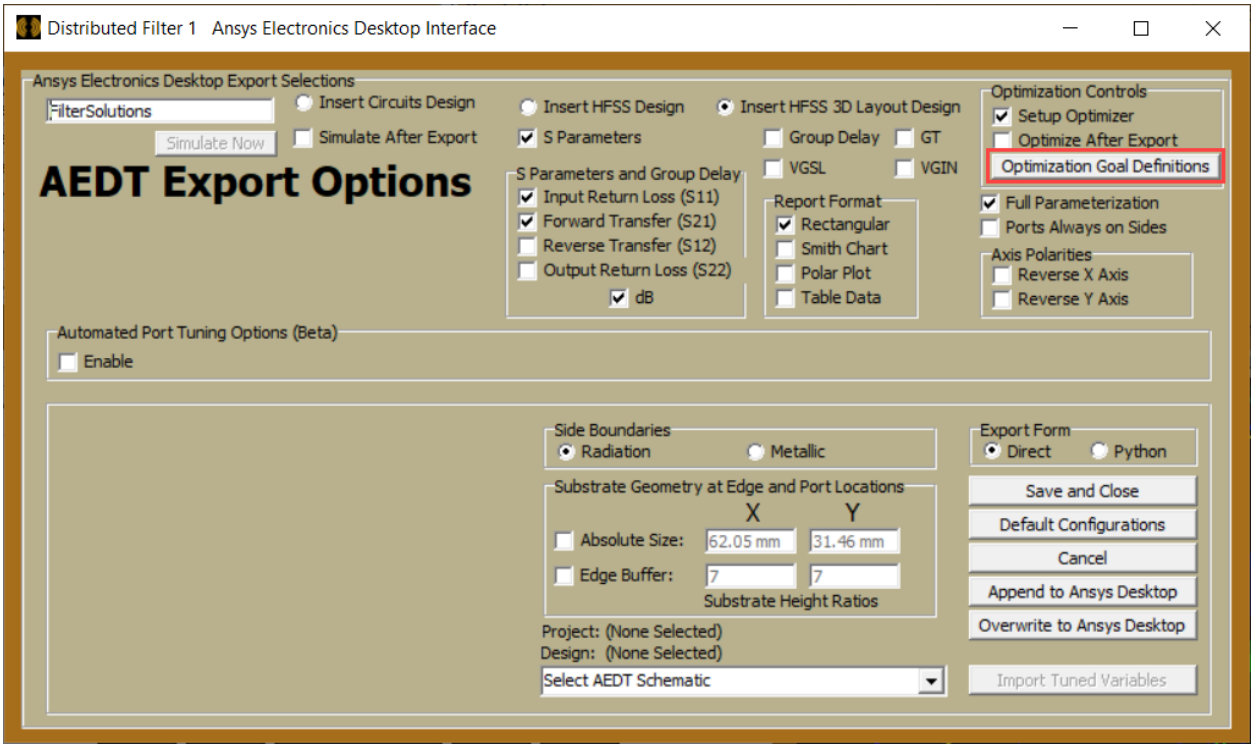
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



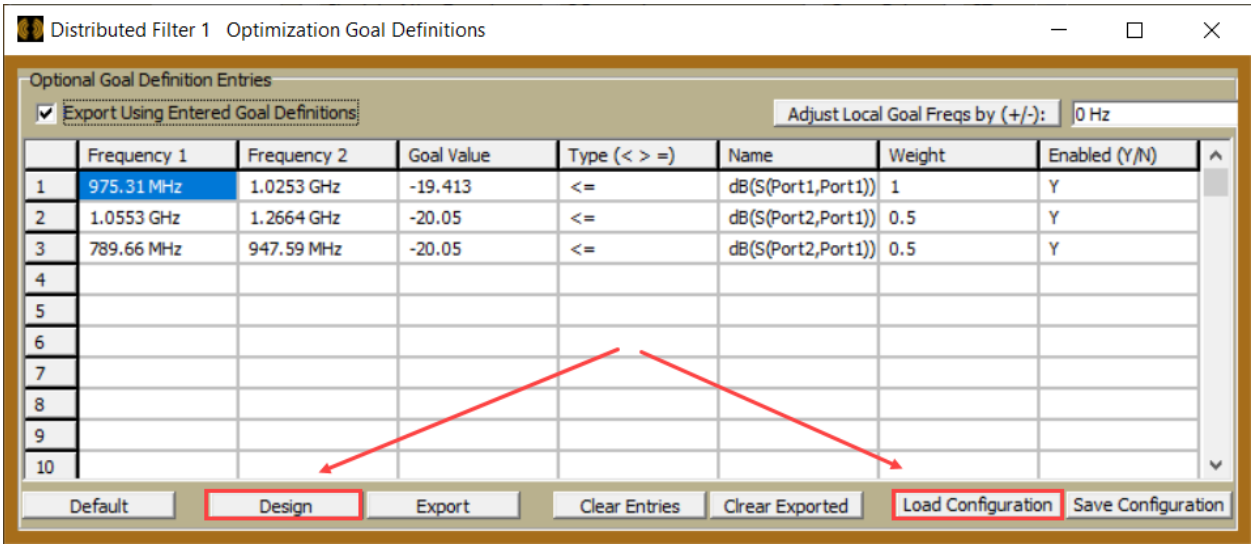
2. Optional- If it is desired to customize the goal limits:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

i. Click **Optimization Goal Definitions** in the **Optimization Controls** group box.

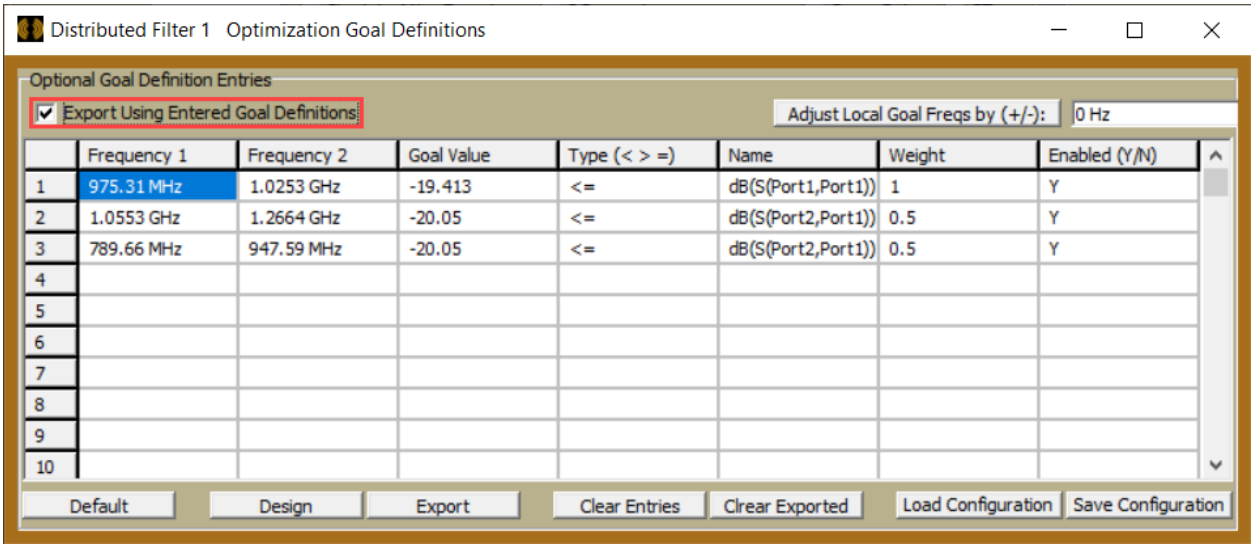


ii. Optional- Click **Design** or **Load Configuration** in the pop-up panel to fill in the existing design goals or previously saved goals, respectively.



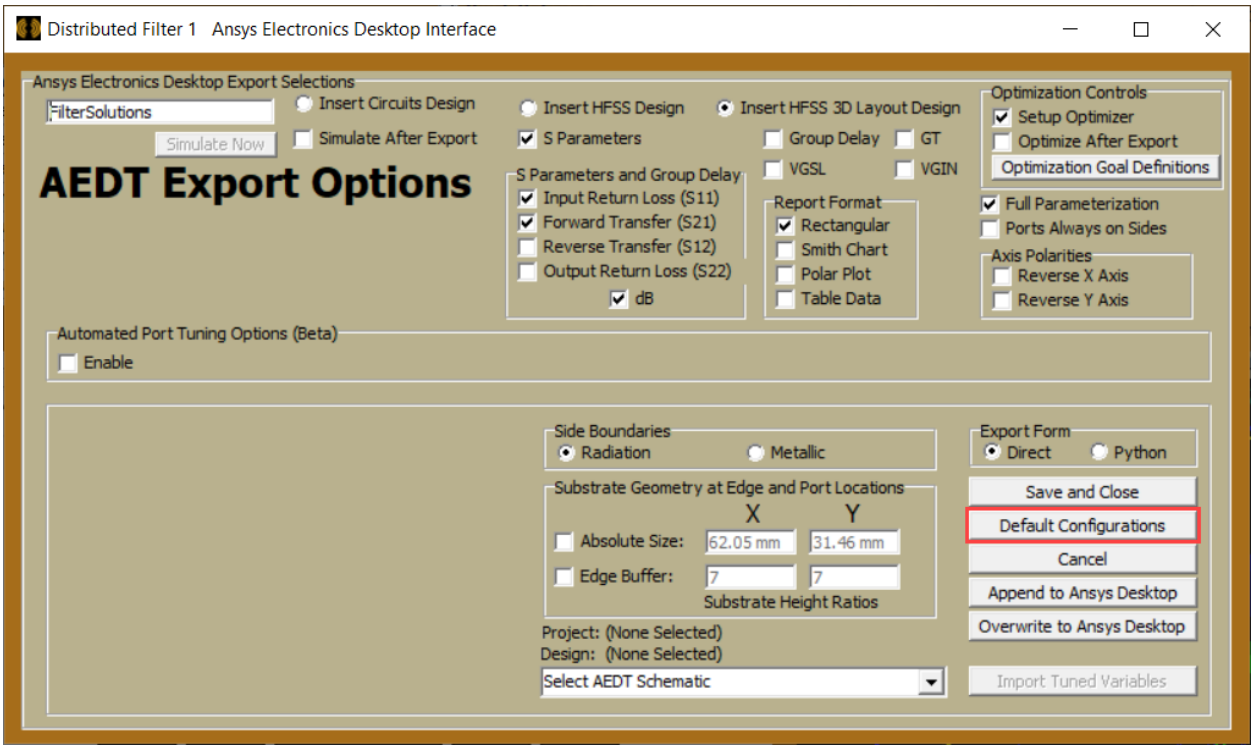
iii. Select **Export Using Goal Definitions** to use table defined goals in export.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



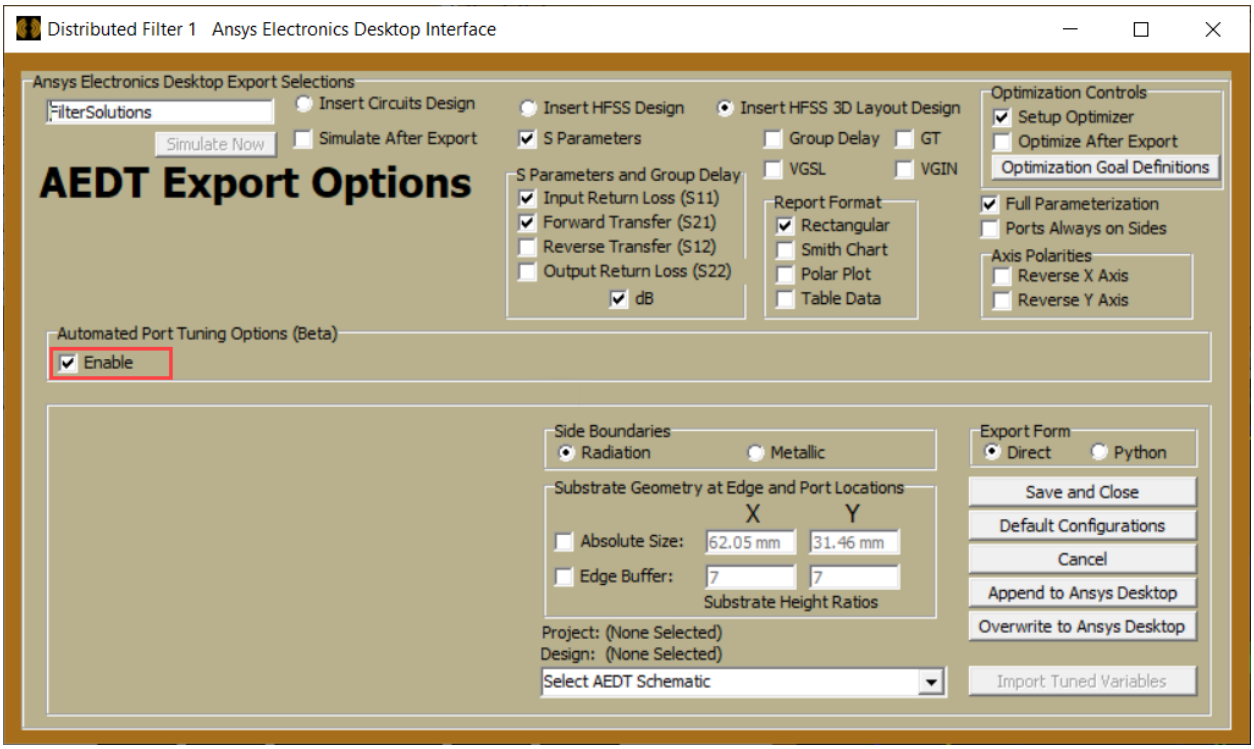
3. Set up the Export for Automated Port Tuning with **Sonnet**

   i. Select **Default Configurations** in the **AEDT Export Panel**.



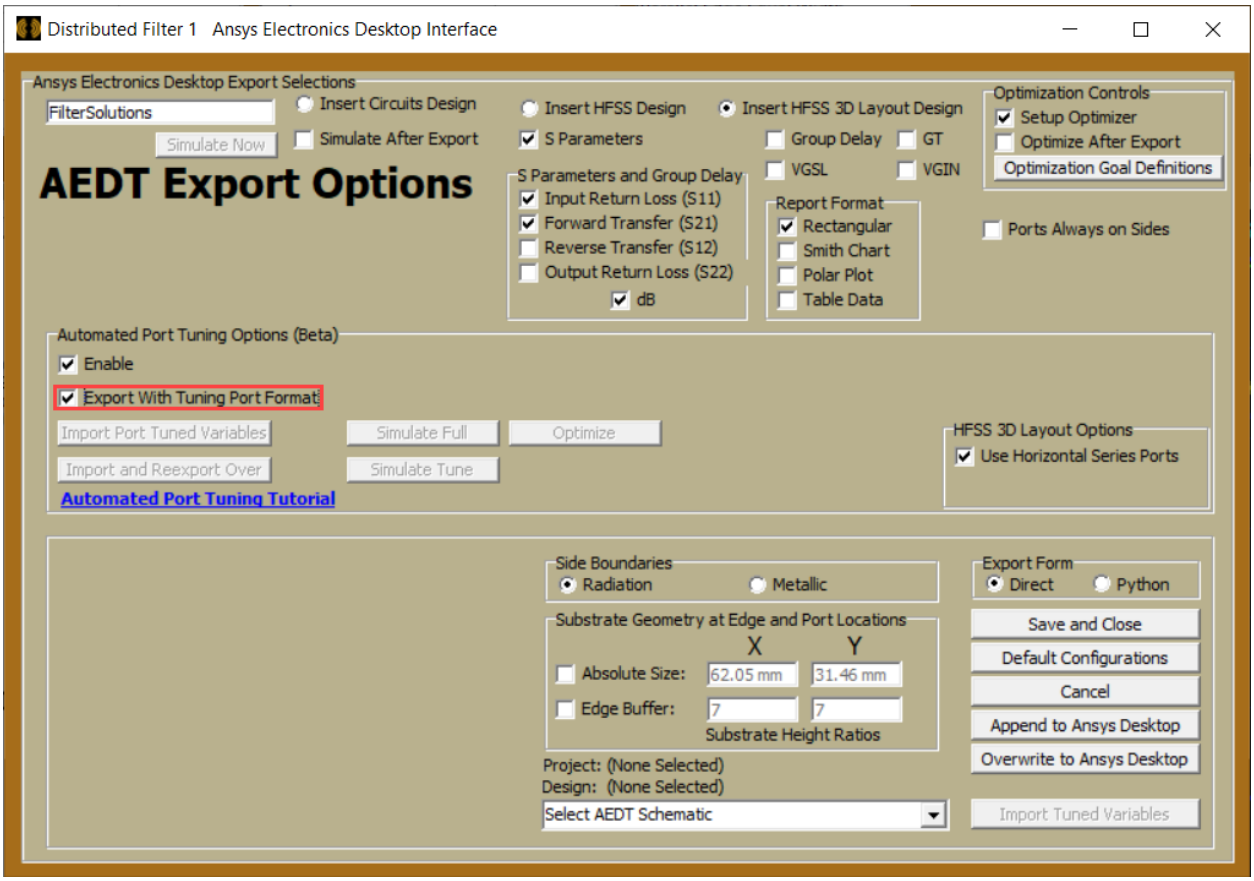
   ii. Select **Enable** in the **Automated Port Tuning Options** group box.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



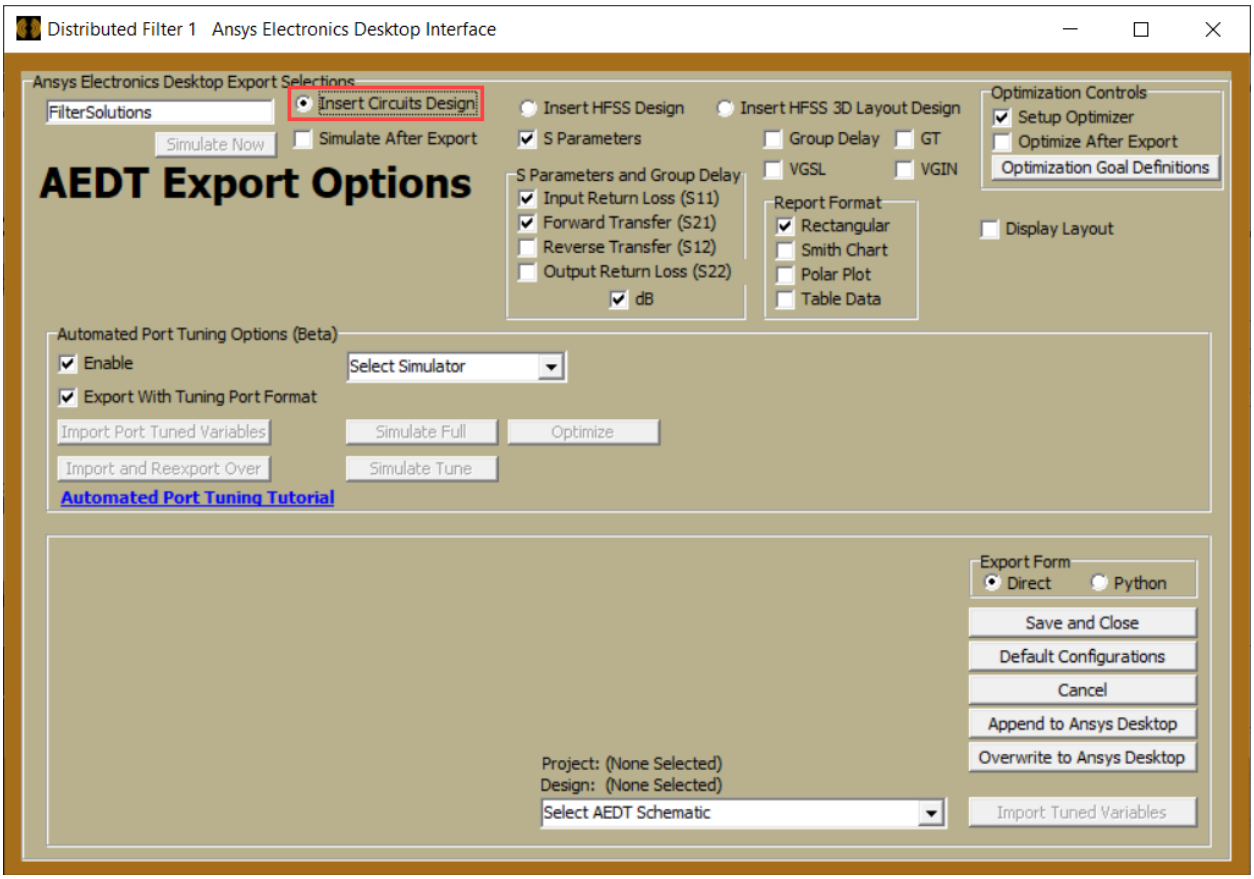
iii. Select **Export with Tuning Port Format** in the **Automated Port Tuning Options** group box

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



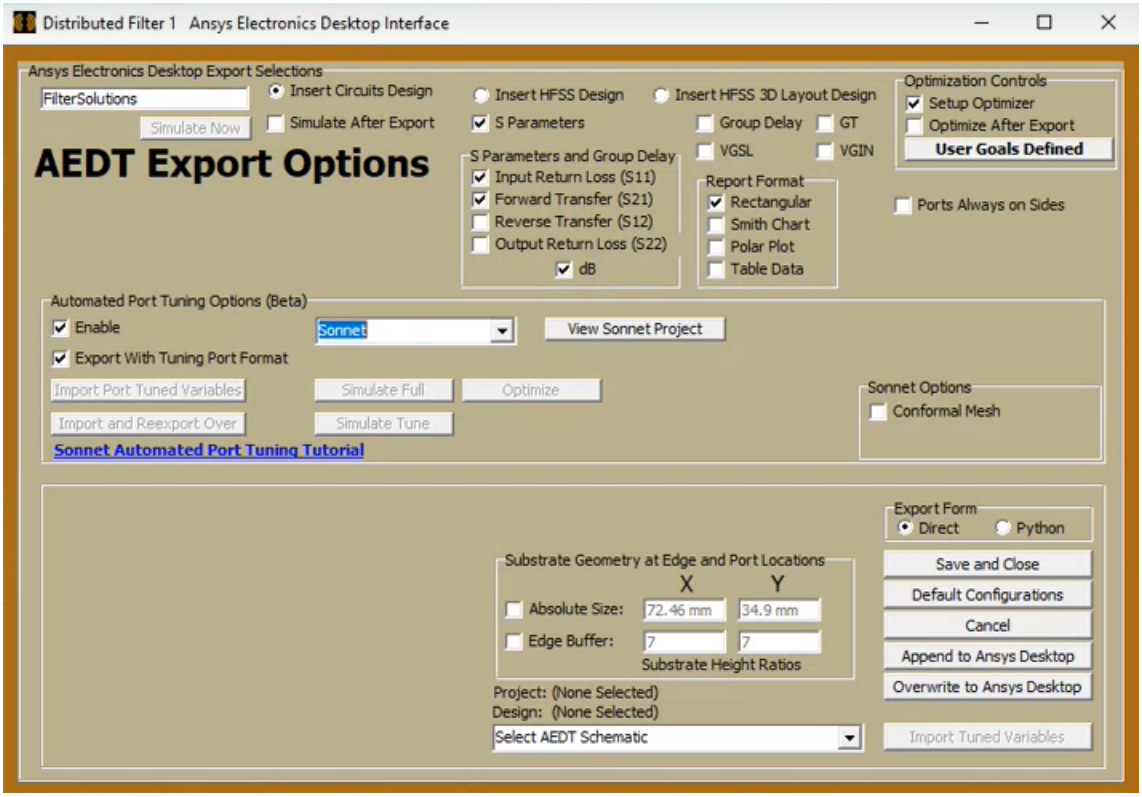
iv. Select **Insert Circuits Design**

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



v. Select **Sonnet** from the Simulator selection drop-down menu.

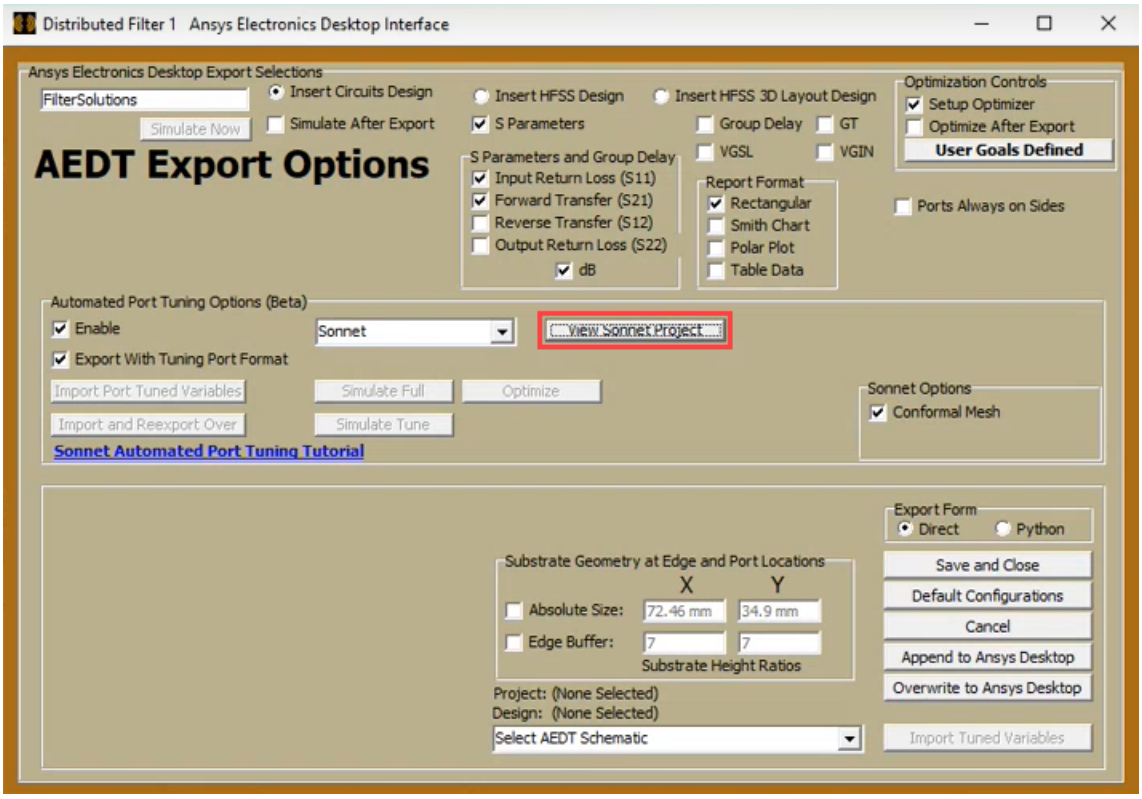
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

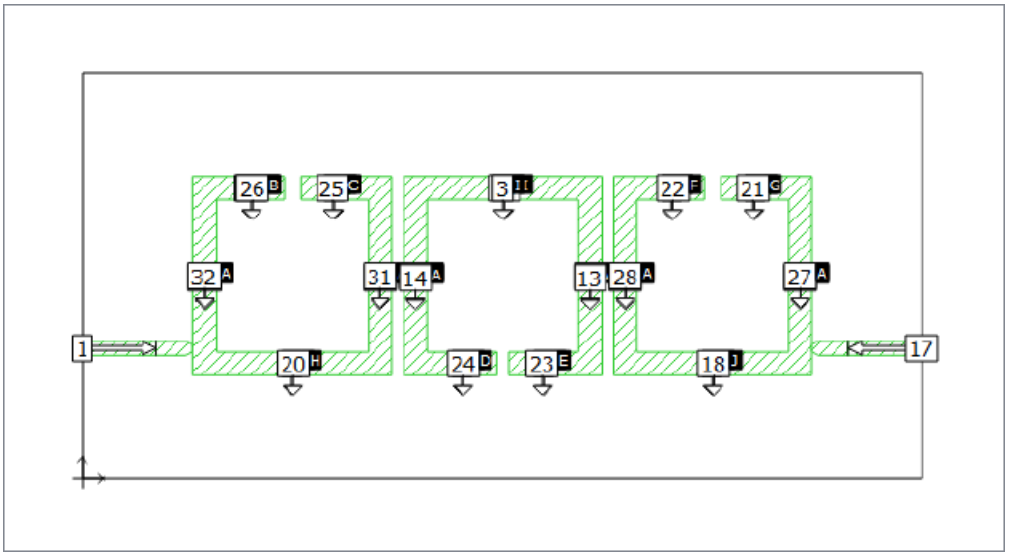


4. Optional- View the Sonnet Layout to confirm Sonnet is simulating the expected layout.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

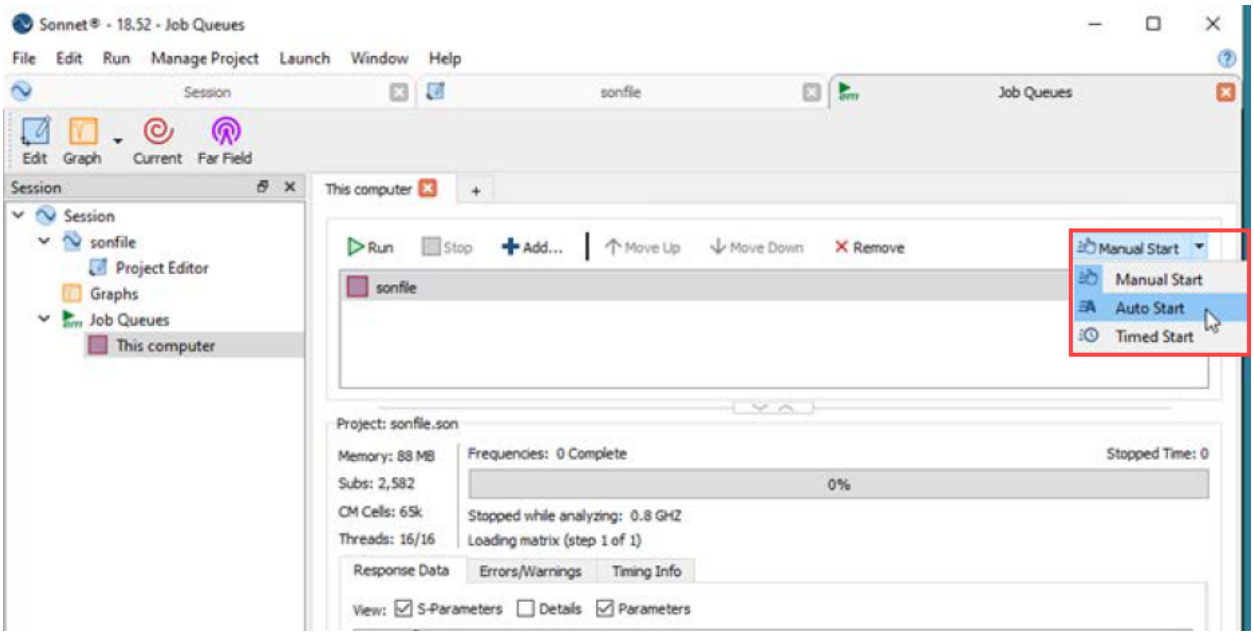
i. Select **View Sonnet Project** in the export panel.



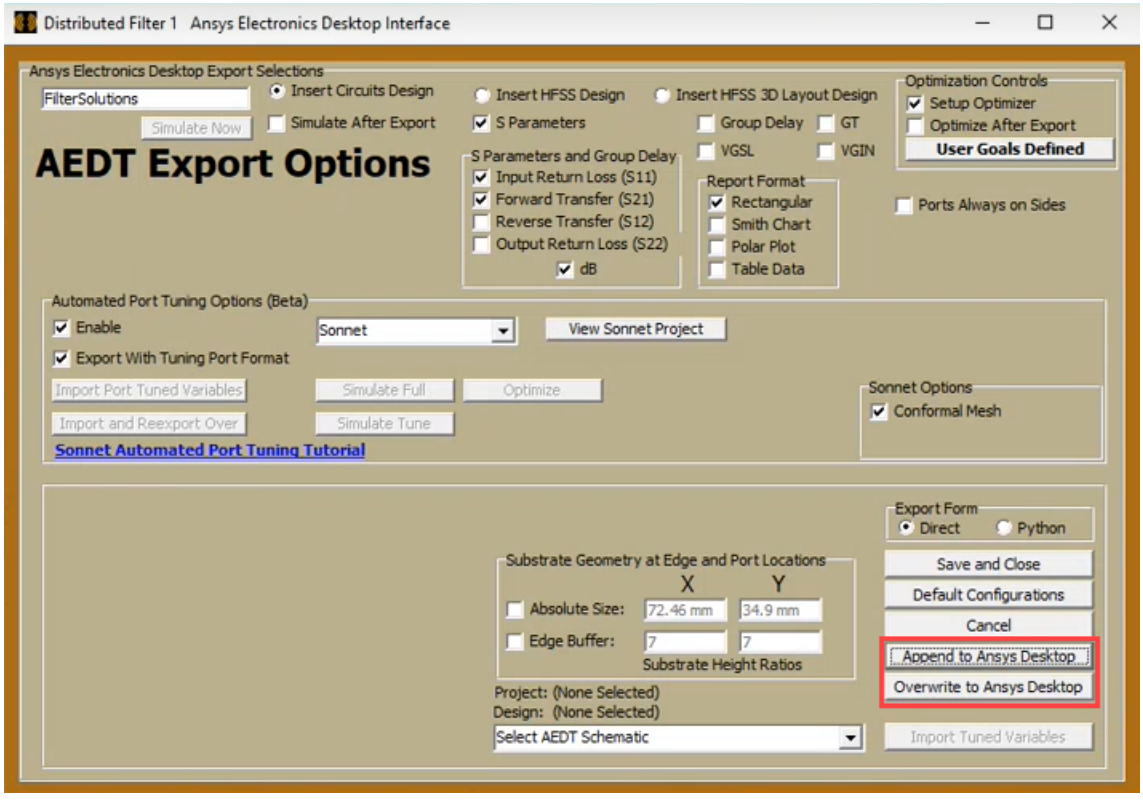


ii. **Recommended**: In the **Sonnet Job Queue**, select **Auto Start** for immediate simulation at export time(step 5).

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



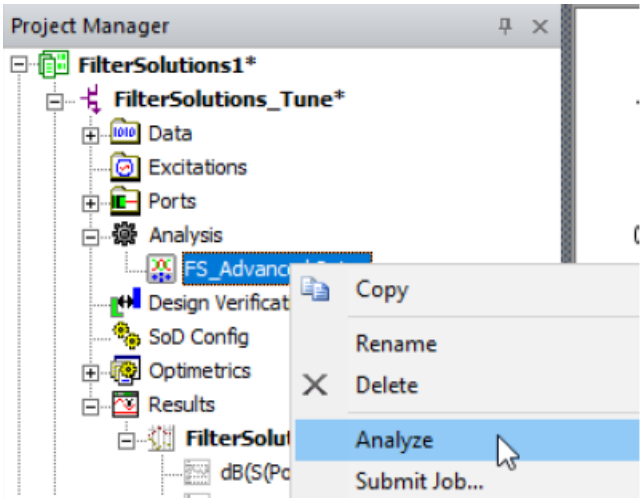
5. Export to AEDT: Choose **Append to Ansys Desktop** or **Overwrite to Ansys Desktop**.



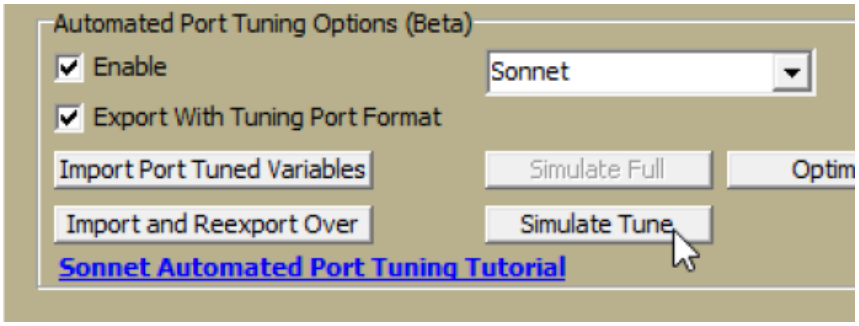
This step will launch AEDT with the ANFS project.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Simulate with Sonnet ***em***
   i. Simulate options:
      a. In Circuit's Project Manager panel, right click Analysis and select **Analyze**.

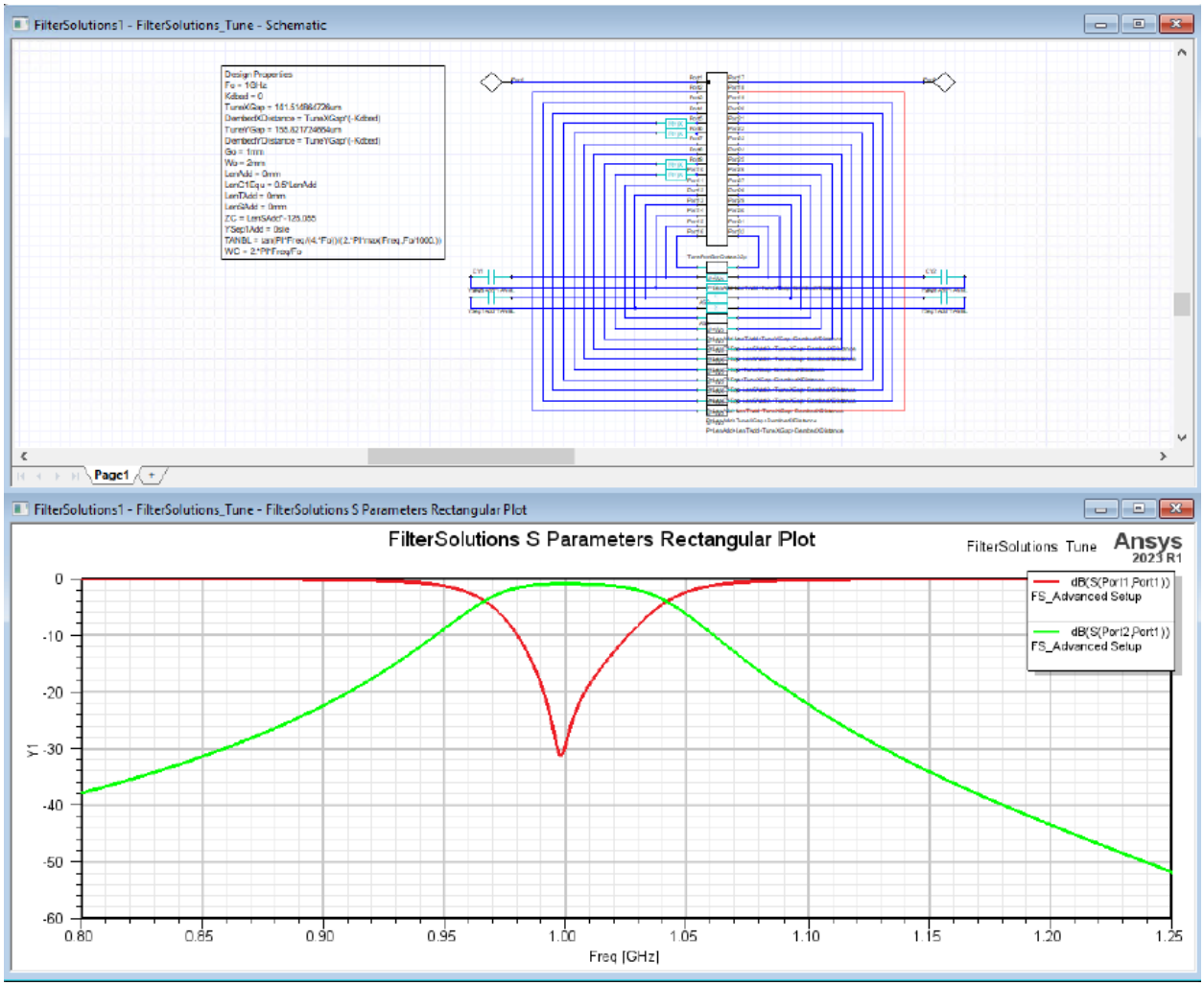


      b. In the FilterSolutions' **Export Panel**, select **Simulate Tune**.



7. Confirm the Circuits layout simulation. Note the use of an S-Parameter block to house the Sonnet simulation response.

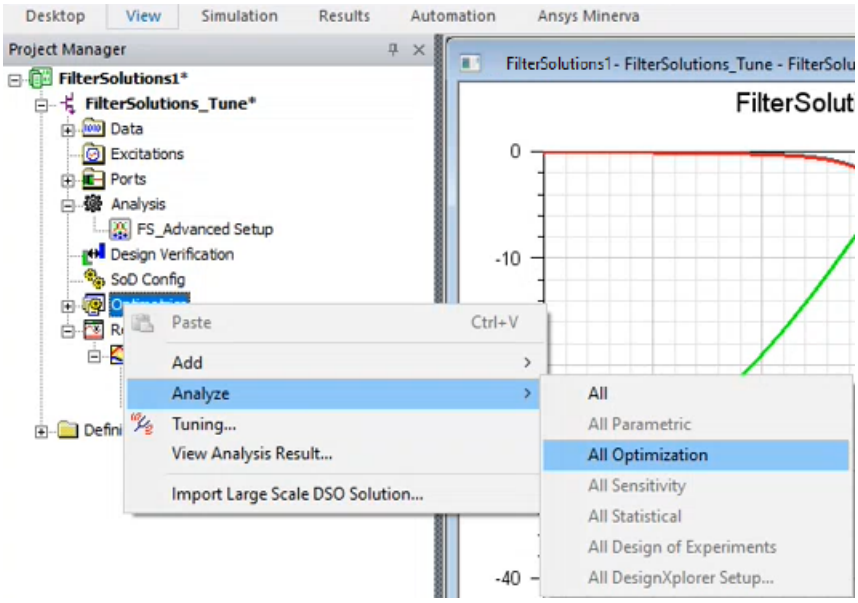
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



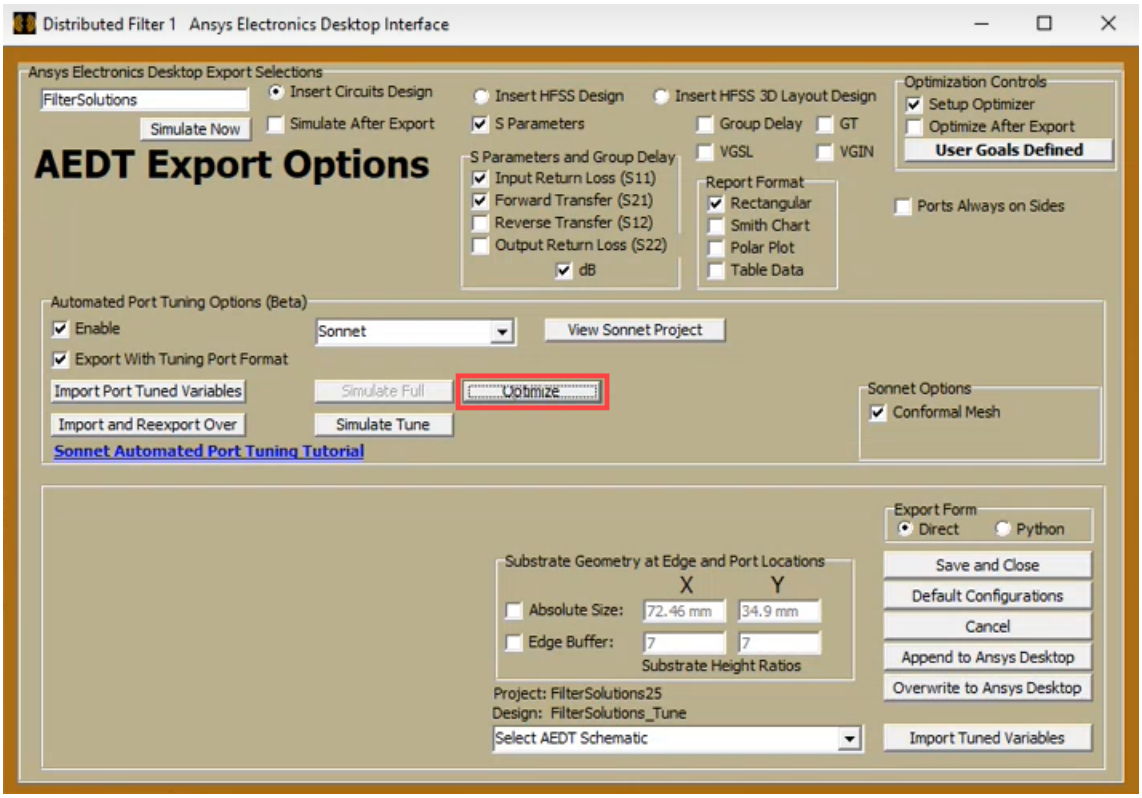
8. Optimize the Tuning Port Design in one of two ways:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. In Circuit's Project Manager, right click **Optimetrics** > **Analyze** > **All Optimizations**



b. In the ANFS export panel, select **Optimize**.

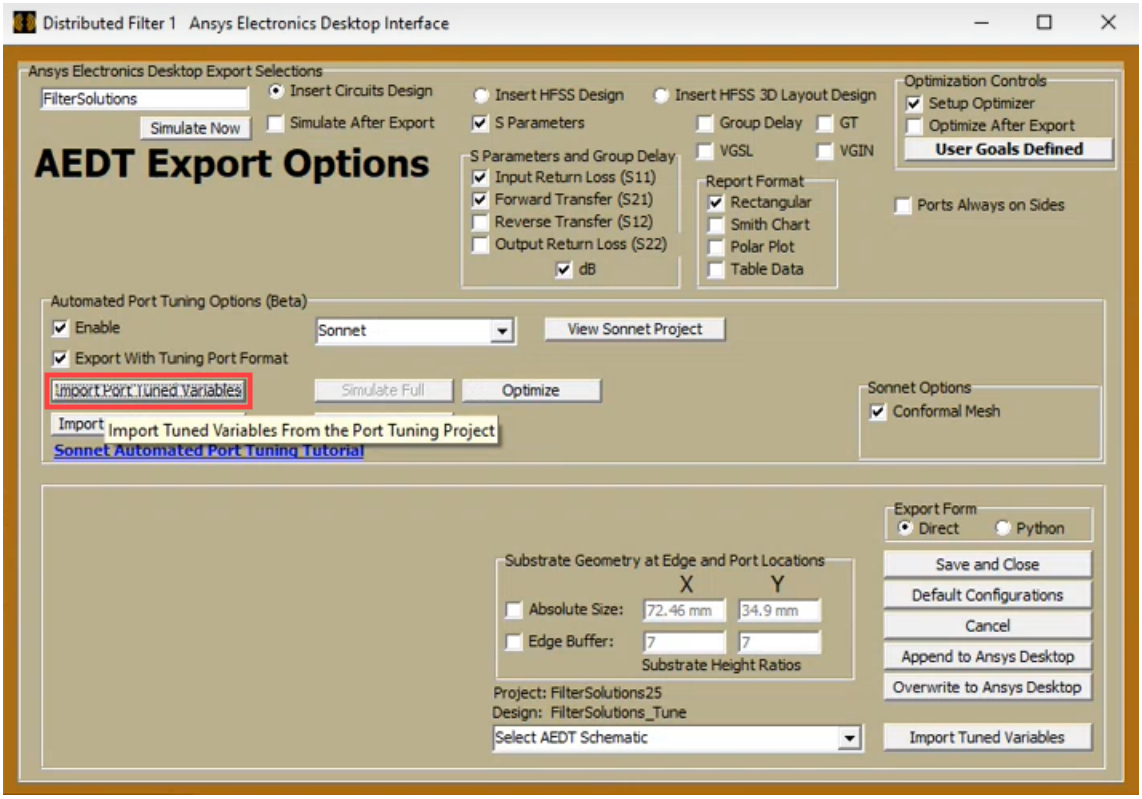


9. Reconstruct the Optimized Design.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

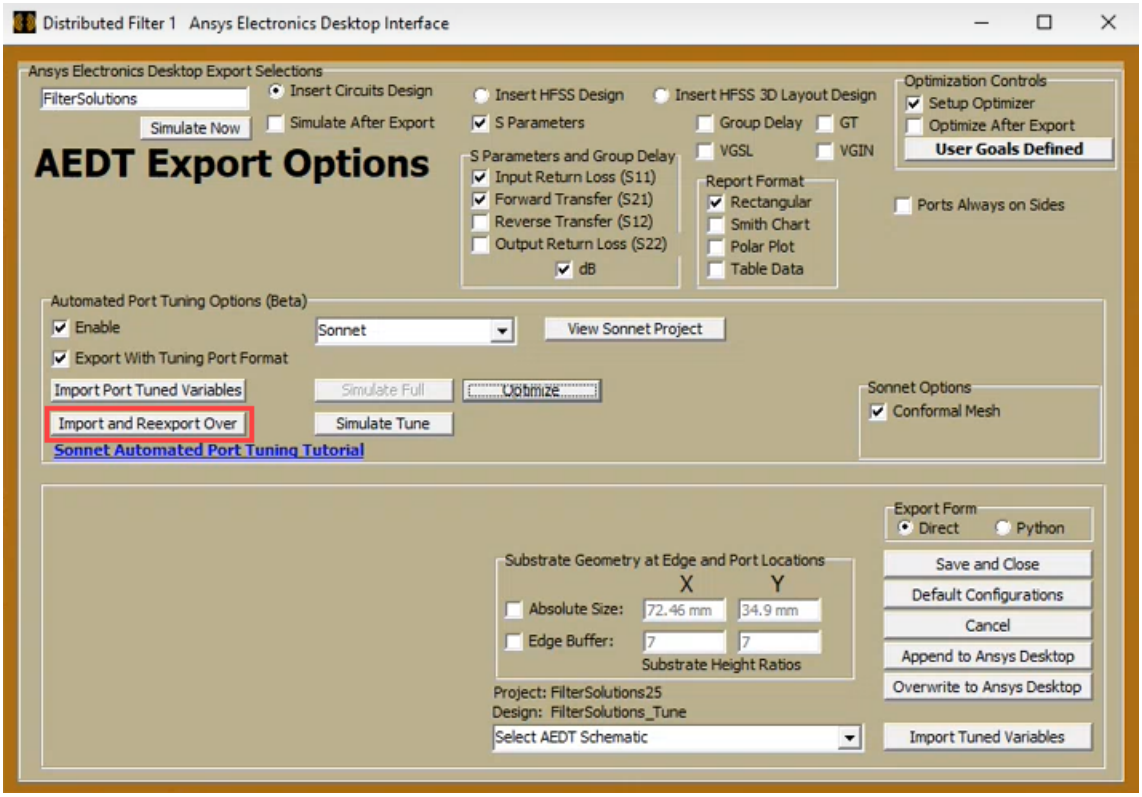
Once the optimization process completes, import and reconstruct the design to ANFS:

i. In the **ANSF Export Panel**, select **Import Tuned Variables**



ii. Optionally, to send the reconstructed design immediately back to AEDT for another port tuning optimization cycle, select **Import and Reexport Over**.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



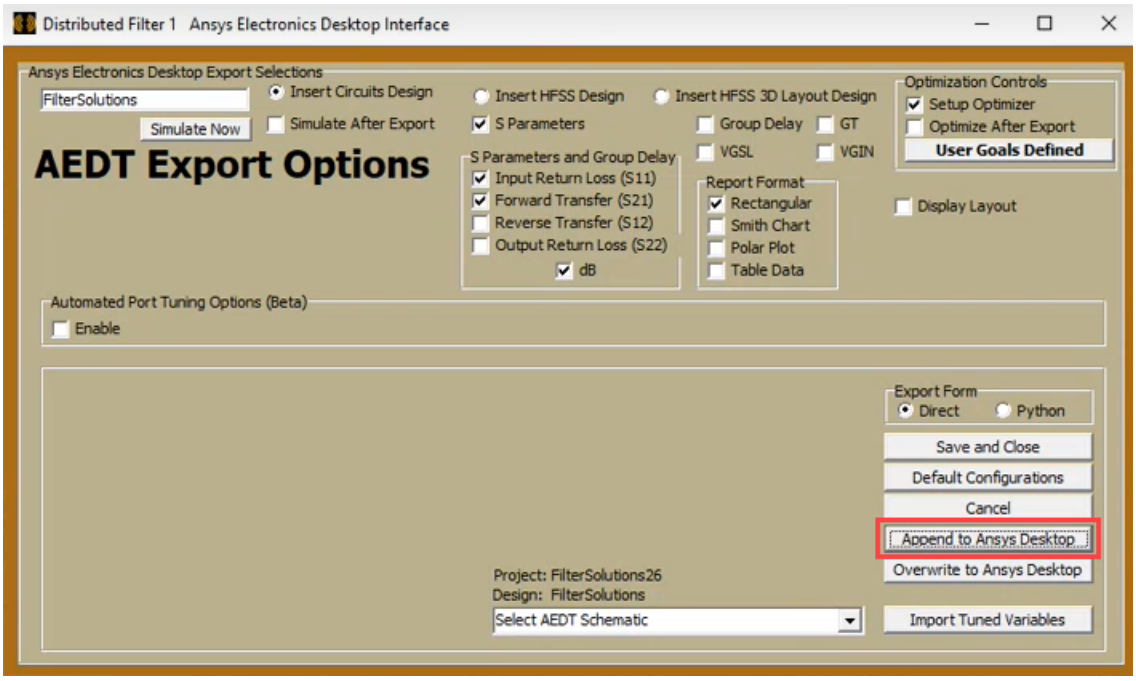
Repeat as necessary.

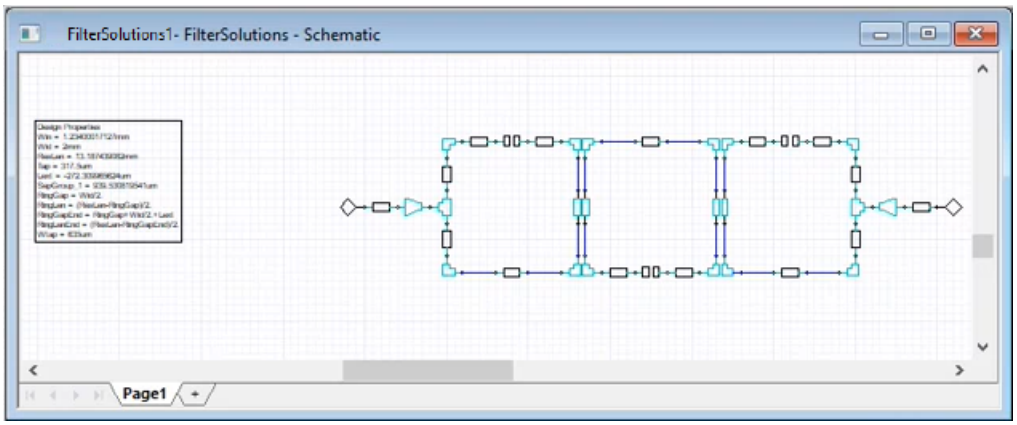
10. Export Preliminary Final Design

    Once the design optimization cycles produce a design of sufficient accuracy:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

i. Export the preliminary final design with no tuning ports installed and simulate.





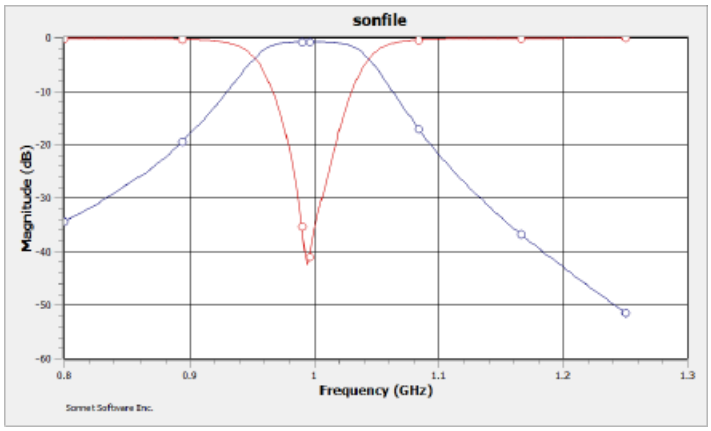
11. Optional- Perform Direct EM Tuning or Optimizing

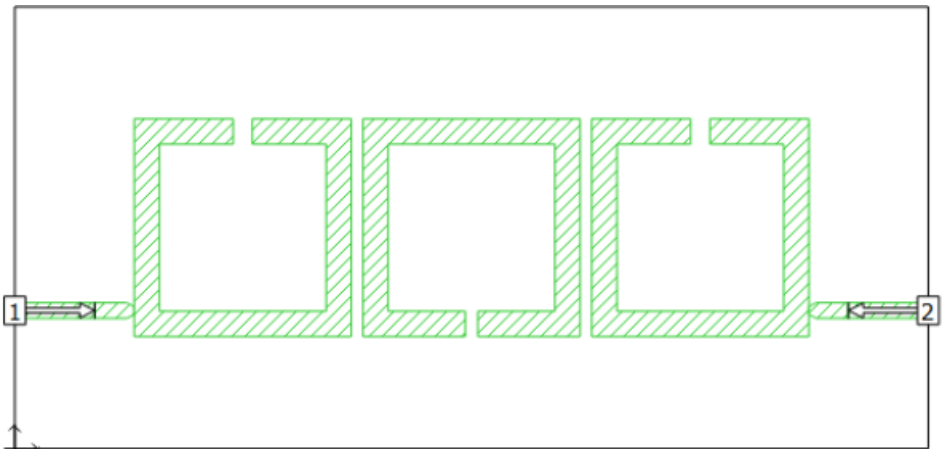
If more accuracy is still needed, the Preliminary Final Design may be further tuned or optimized directly with Sonnet tuners and optimizers.

i. In the ANFS Export Panel, Disable **Export with Tuning Port Format** and select **View Sonnet Project** to optimize directly with Sonnet.

The final design is observed to center on 1 GHz.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



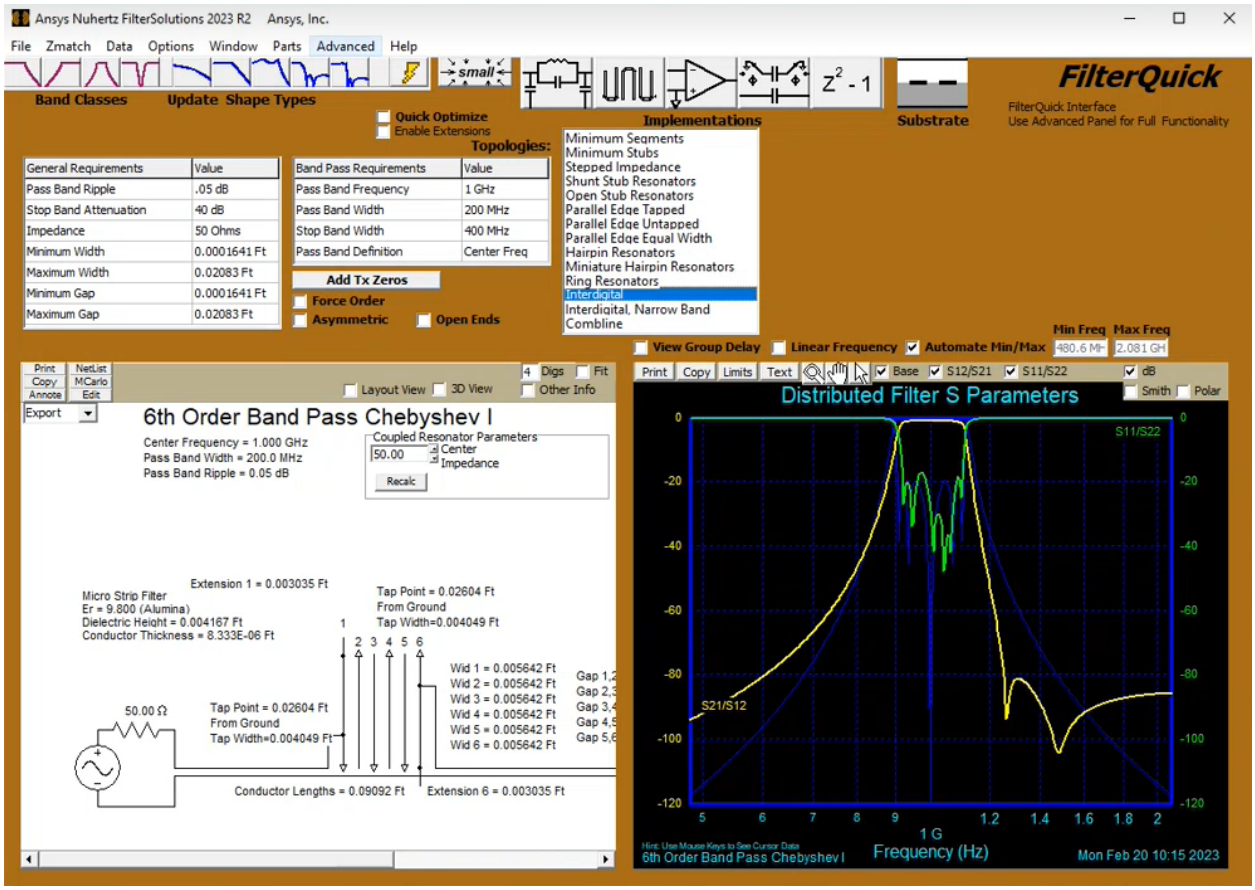


Parameter Optimization with Sonnet Tutorial

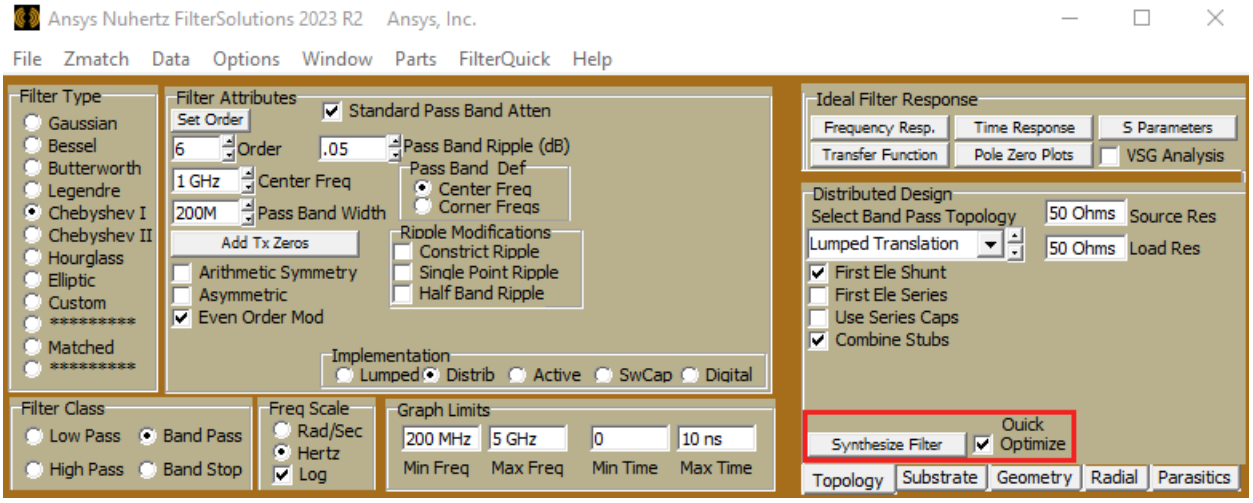
A 5 pole 10% Interdigital Example

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Launch **FilterSolutions** and design the filter.

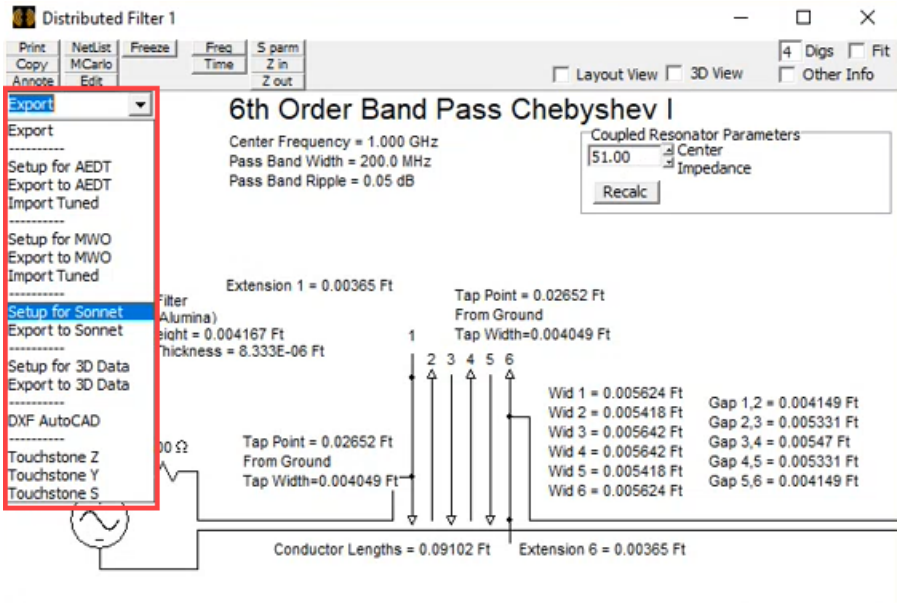


i. Check **Quick Optimize** then click **Synthesize Filter**.

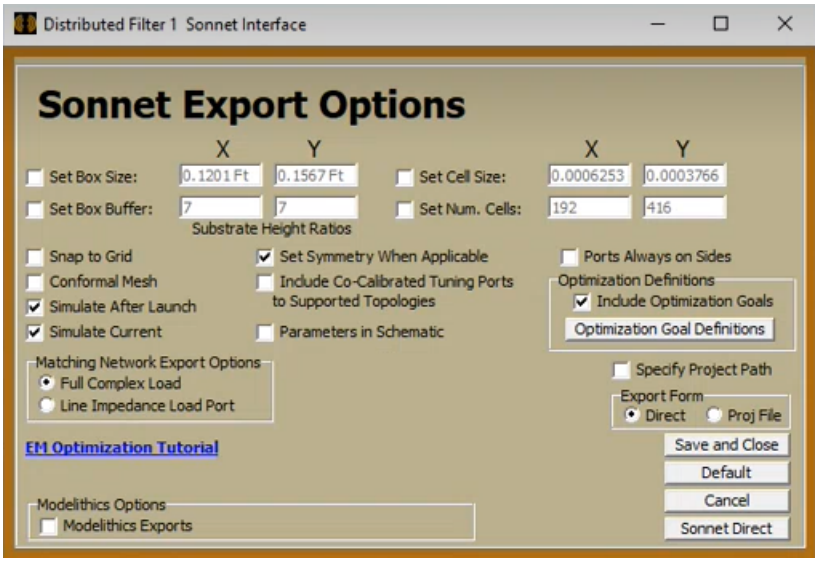


ii. From the **Export** drop down menu, select **Sonnet Setup**.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



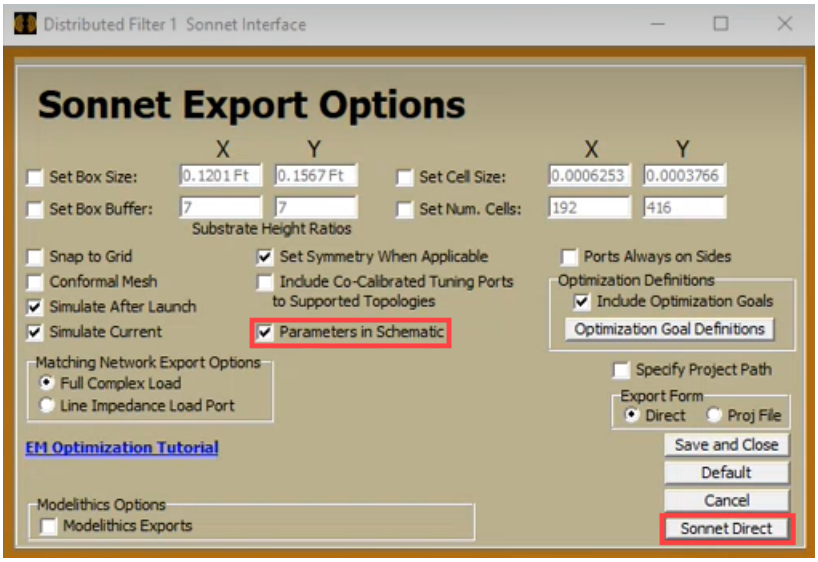
The Sonnet Export Options dialog will open:



2. Export to Sonnet
   i. Use default selections to view the frequency response prior to Sonnet optimization.
   ii. Select **Parameters in Schematic** to include parameters in geometry to be exported to Sonnet

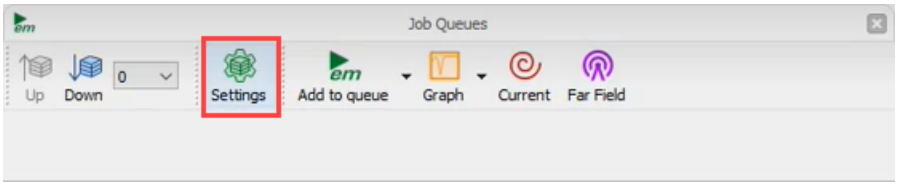
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

iii. Select **Sonnet Direct** to export geometry directly to Sonnet using the selected Sonnet export options.



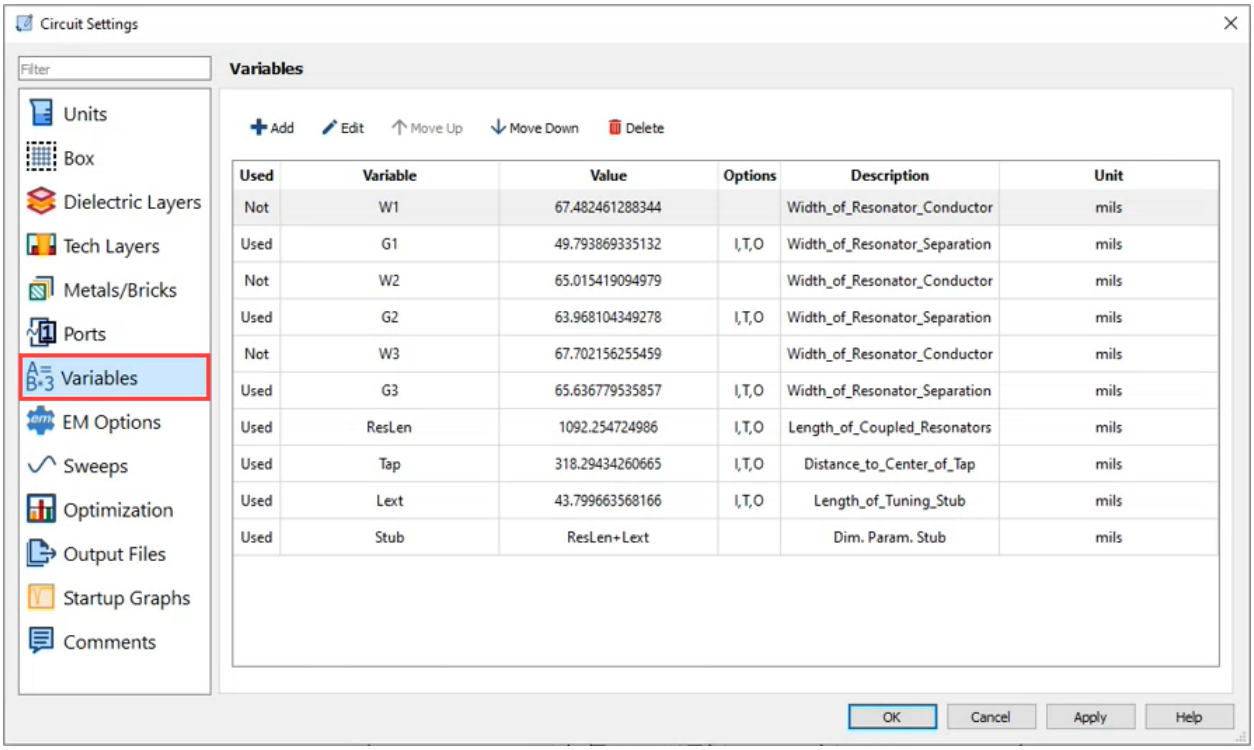
3. Parameter Tuning Setup is Automated. In Sonnet:

i. Select Settings to launch the **Circuit Settings** dialog.



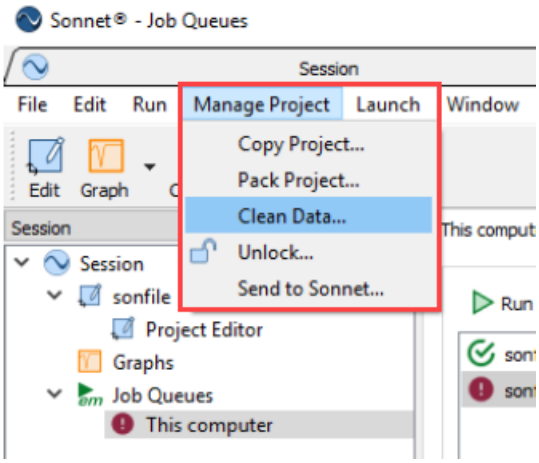
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ii. Click **Variable** to view parameters. Equations are included where needed.



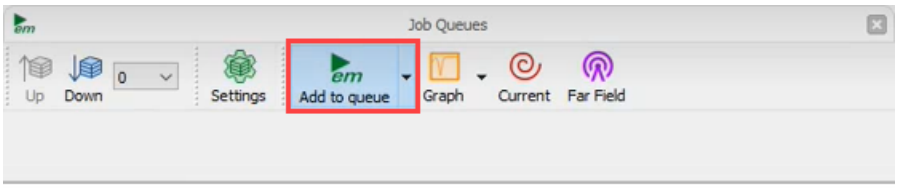
4. Simulate

i. Select **Clean Data**, if needed.

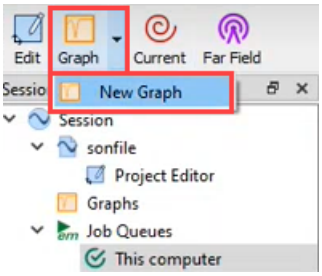


ii. Click ***em*** to simulate.

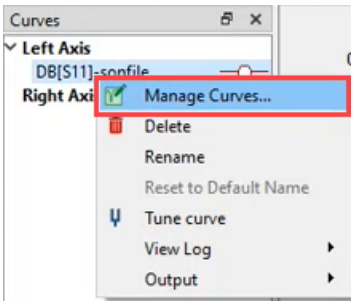
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



iii. After simulation completes: Select **Graphs** > **New Graph**.

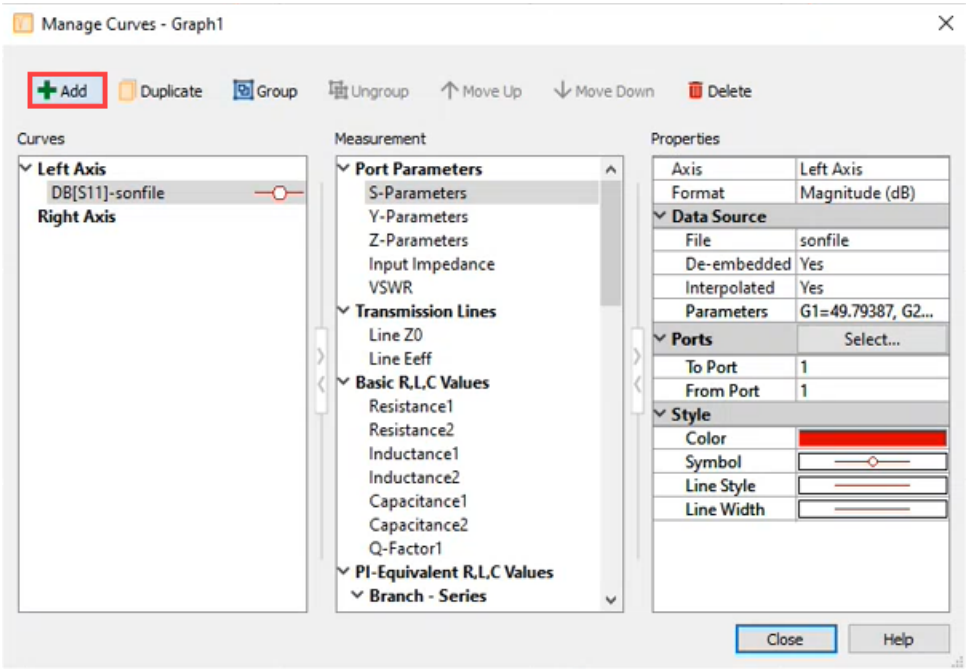


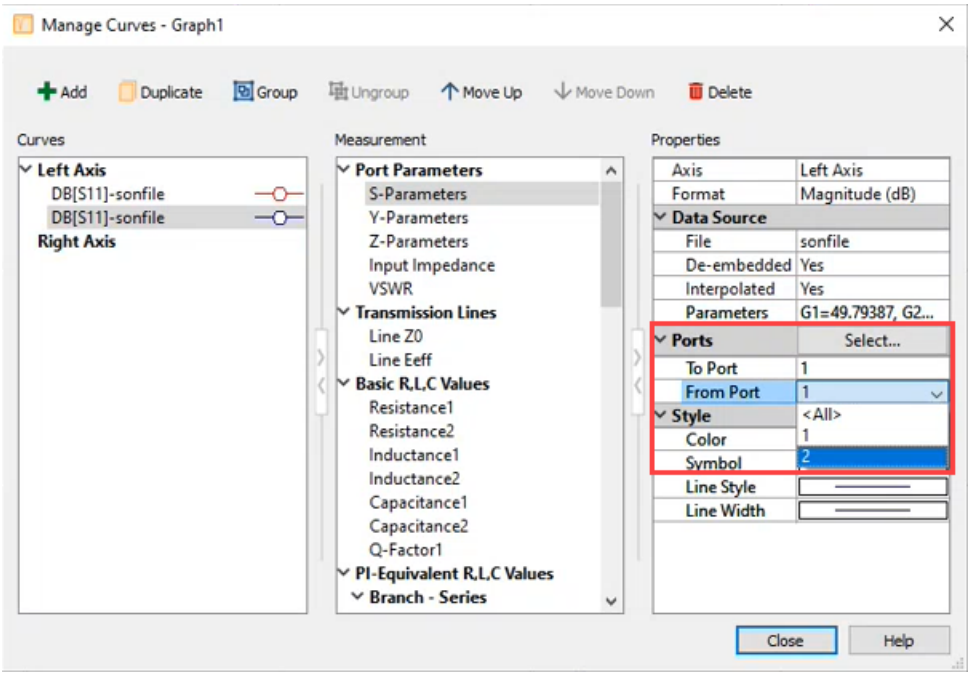
iv. Click **Left Axis** then right click **S11** to launch the **Manage Curves** window.



v. Click **+Add** and **S12**.

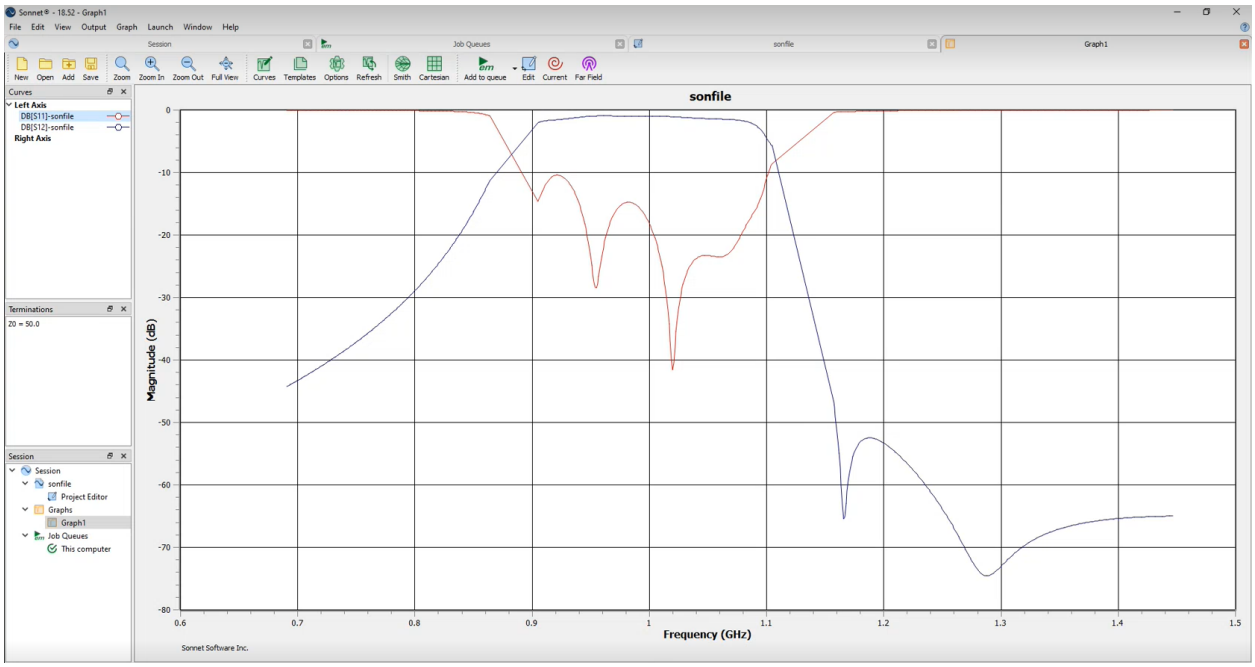
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





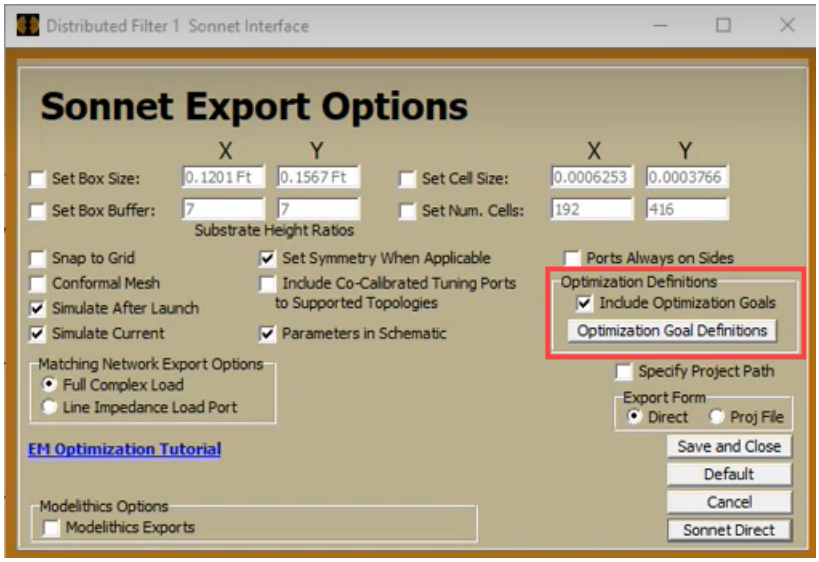
5. View the Frequency Response, determine if EM optimization is needed.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



6. Optimize

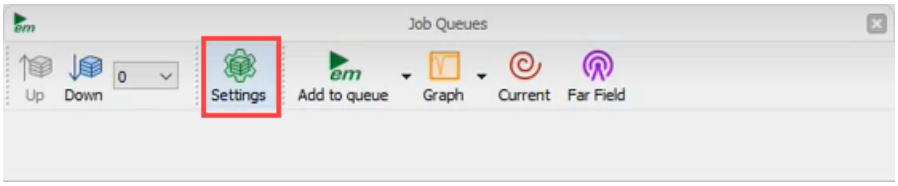
In FilterSolution's Sonnet Export Options dialog, export with **Include Optimization Goals**.

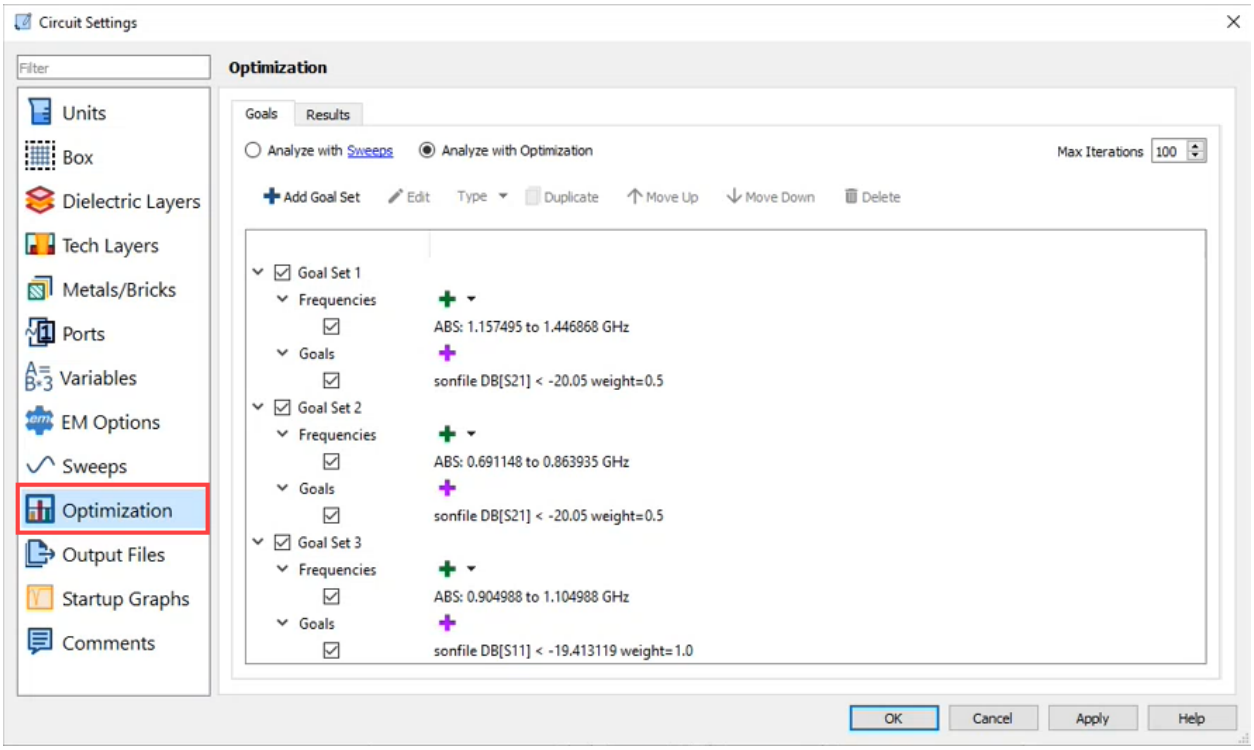


In Sonnet:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

i. Select **Settings** and **Optimization Goals** to view the optimization goals.



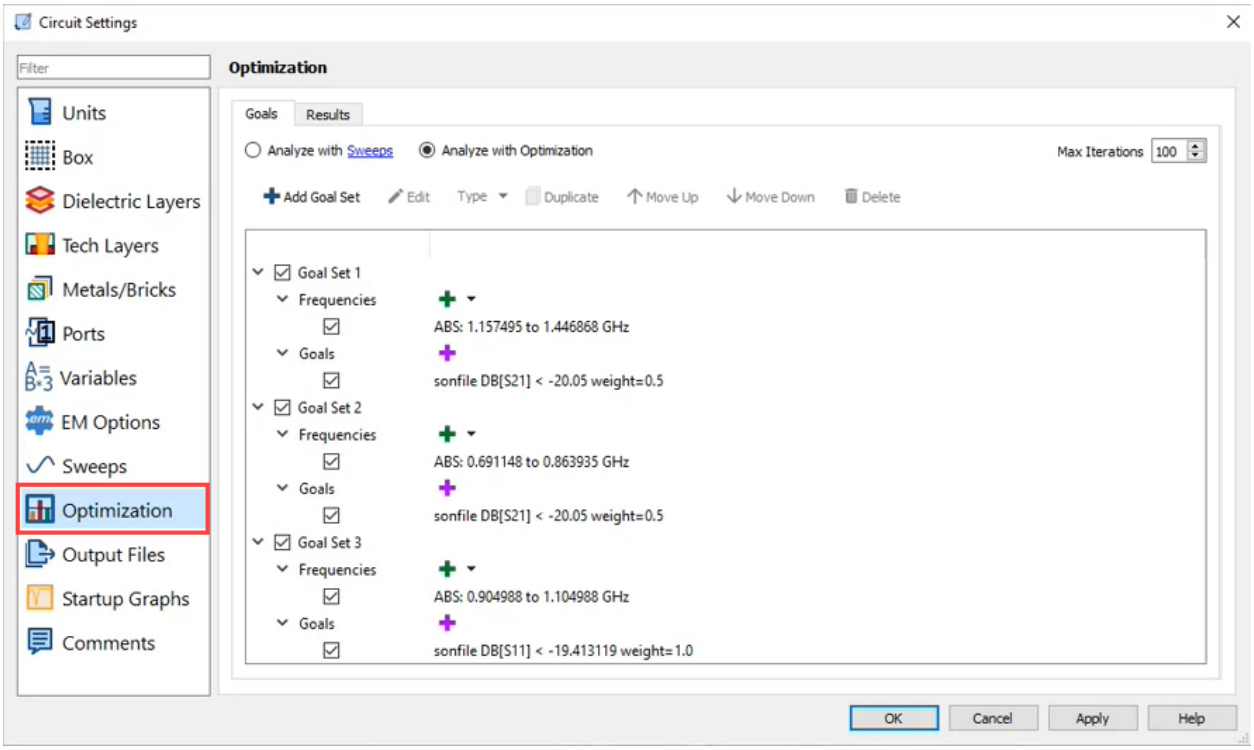


ii. Simulate the project and optimization

7. Optimize by Starting a Simulation

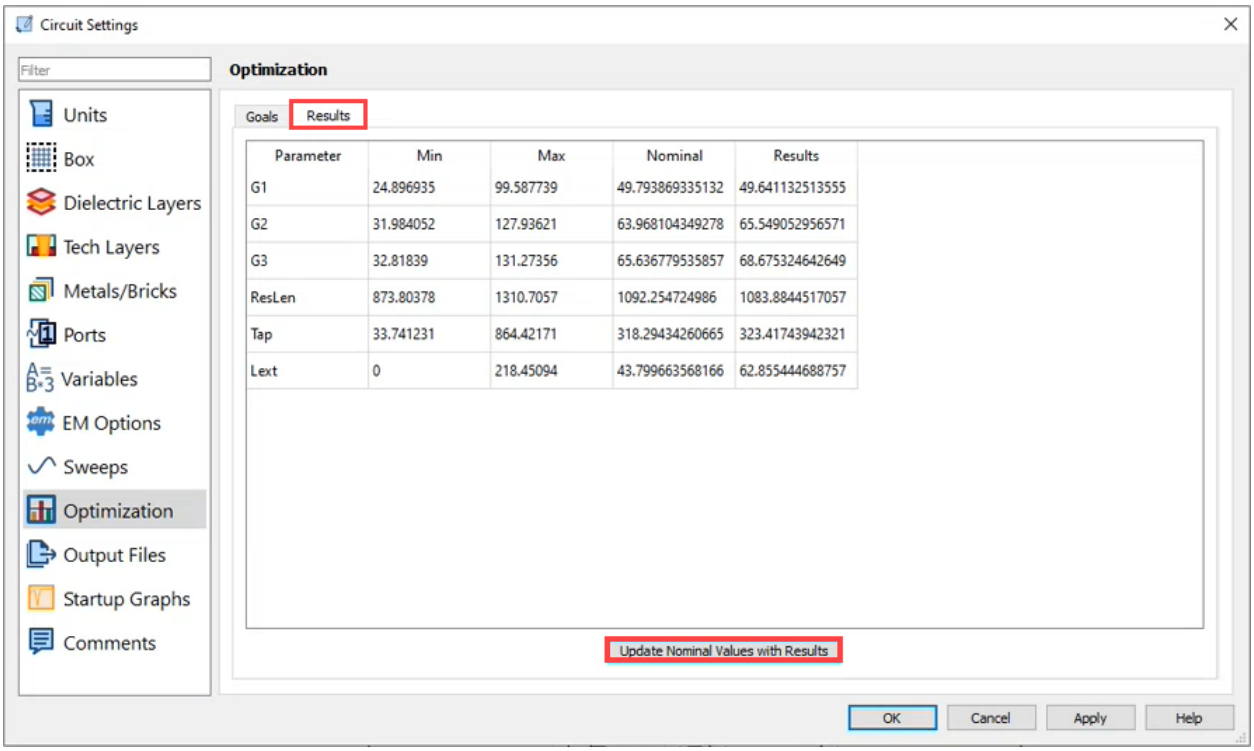
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

i. In Sonnet's **Circuit Settings**, select **Optimization**



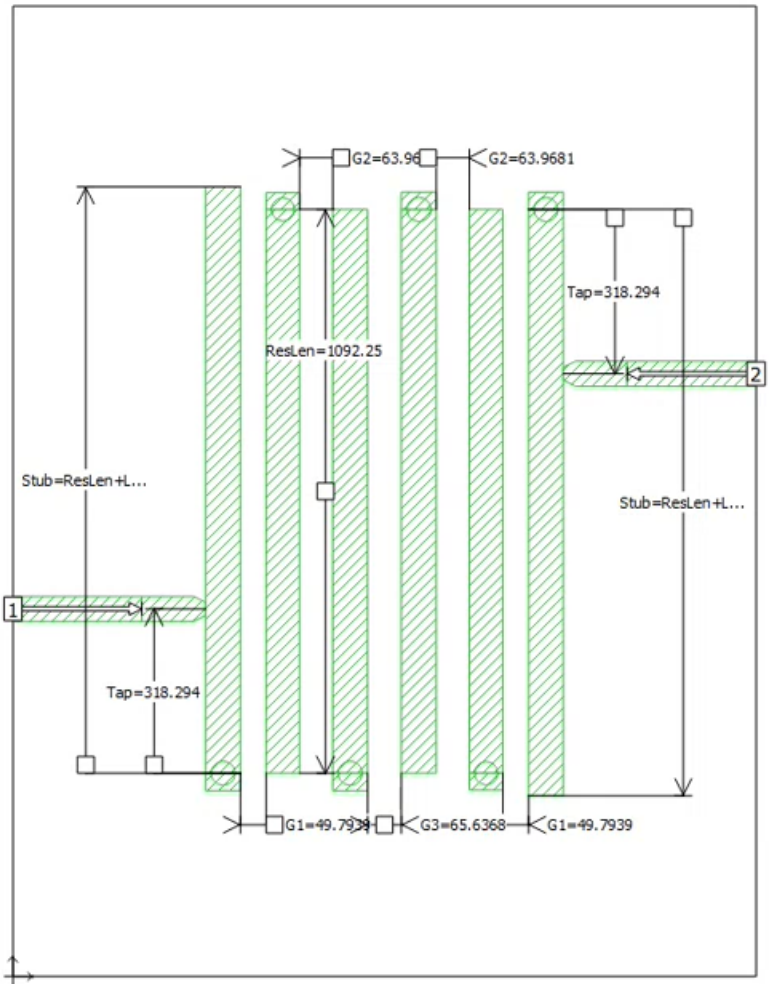
ii. Select **Results**, then select **Update nominal values with results**

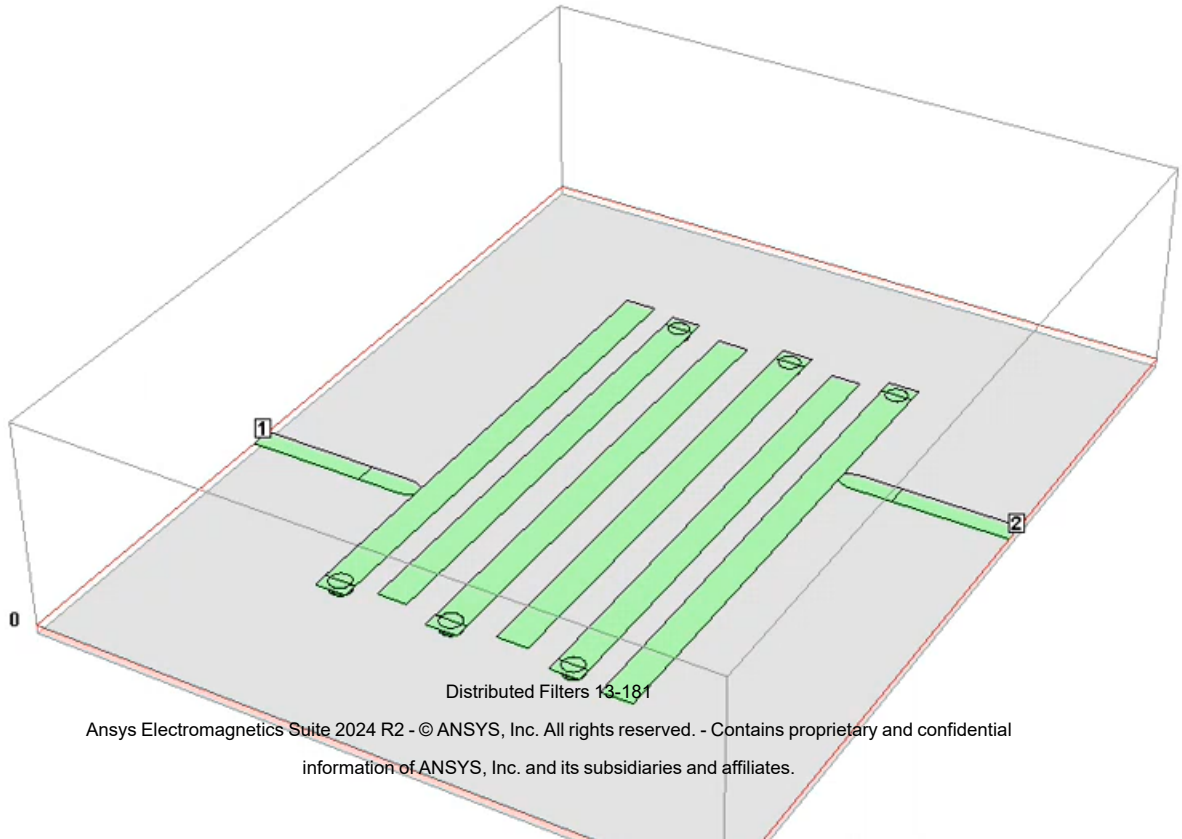
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



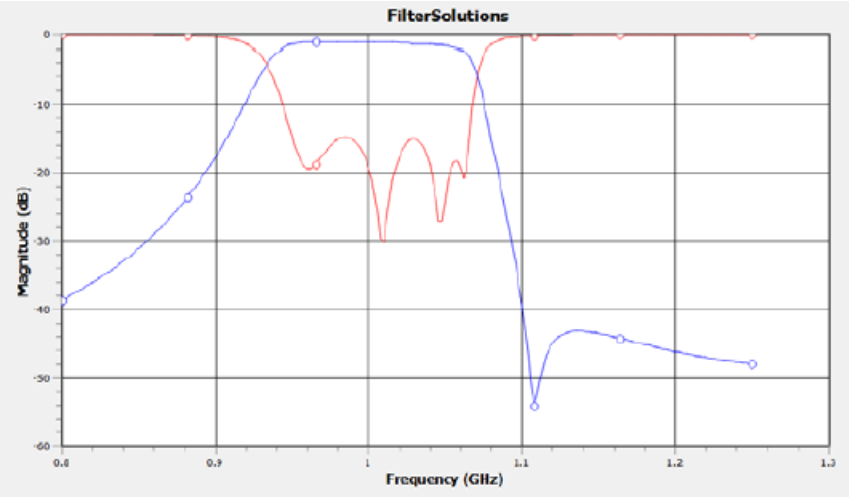
8. View the Optimized Results

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Exporting to Other Software

This chapter contains the following topics:

- Modelithics in AEDT
- 3D Data
- DXF AutoCAD
- Touchstone Files
- Sonnet
- Modelithics in Sonnet

Modelithics in FS and EDT

Lumped element Filters exported to Ansys Circuits have the option of being simulated with highly accurate Modelithics models. The filter layout, including device pad geometries, can then be generated in Circuits. FilterSolutions generated Modelithics/Circuits Schematics may be optimized by the part value and/or layout interconnect widths and lengths. The user may wish to customize the layout geometries to suit his or her specific application. Optimized or tuned Modelithics part values are confined to discrete manufacturer value through the use of Design Array variables loaded with manufacturer values supplied by Modelithics.

Loading Modelithics into Circuits

Modelithics media and licensing must be obtained from Modelithics and installed into the specific version of Electronic Desktop that it is being used on. After Modelithics is installed, it must be

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

loaded into FilterSolutions with the use of the **Load Modelithics Models** button shown in Figure 1. The loading operation only needs to be performed once following the installation of Modelithics. FilterSolutions will store the results internally for later use.

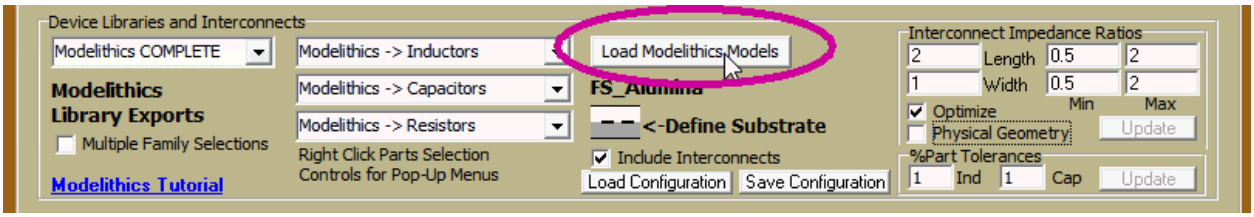


Figure 1: Loading Modelithics models into Ansys Nuhertz FilterSolutions

Selecting Modelithics Models

Modelithics inductors, capacitors, and resistors are easily selectable with the use of the right mouse click Pop up Menus, as shown below in Figure 2. Each selection shows the range of the model family for easy visual reference. Family selection may also be made directly with the use of the drop down selectors, if desired.

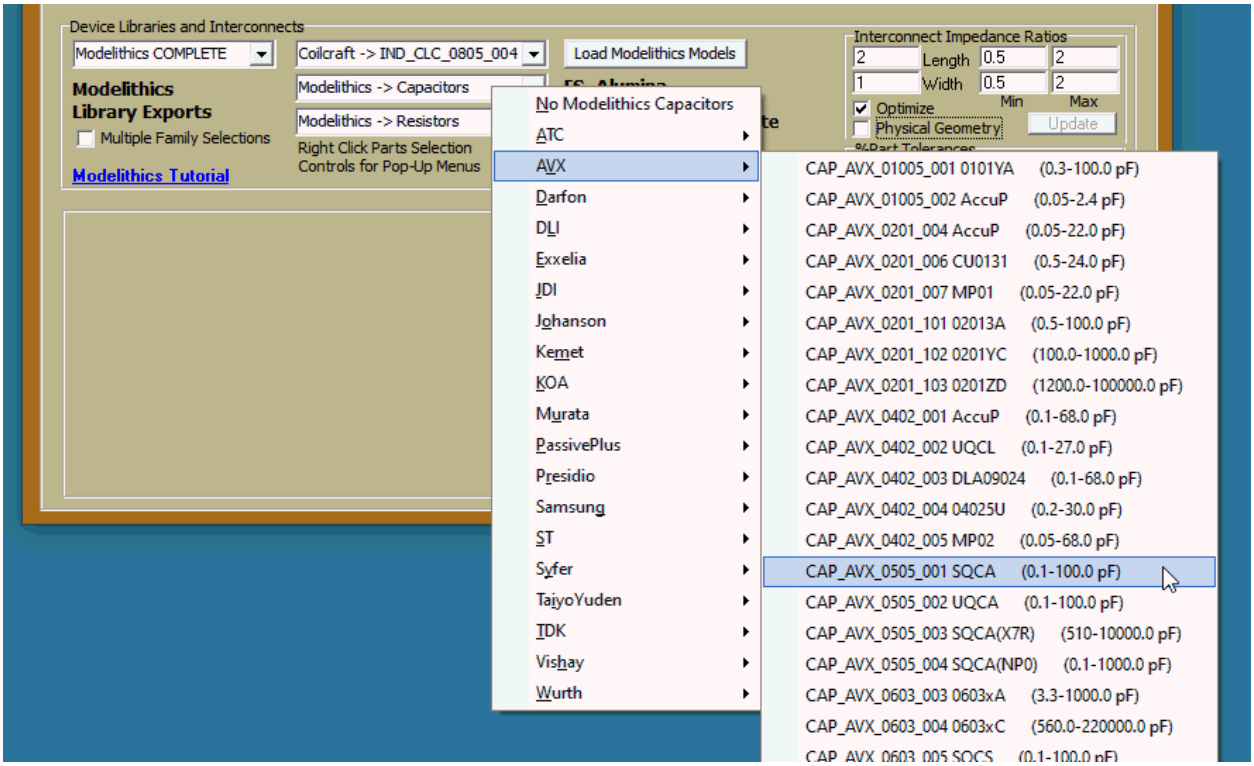


Figure 2: Selecting a Modelithics Manufacturer and Family

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selecting Multiple Families

If more than one family of parts is desired **Multiple Family Selections** may be selected, and individual families may then be selected and entered in to the multiple family tables as shown in Figure 3 either through the **Add**... button or through a drag and Drop operation. At export time, the part with the closest value to the synthesized value from all families entered will be chosen for export.

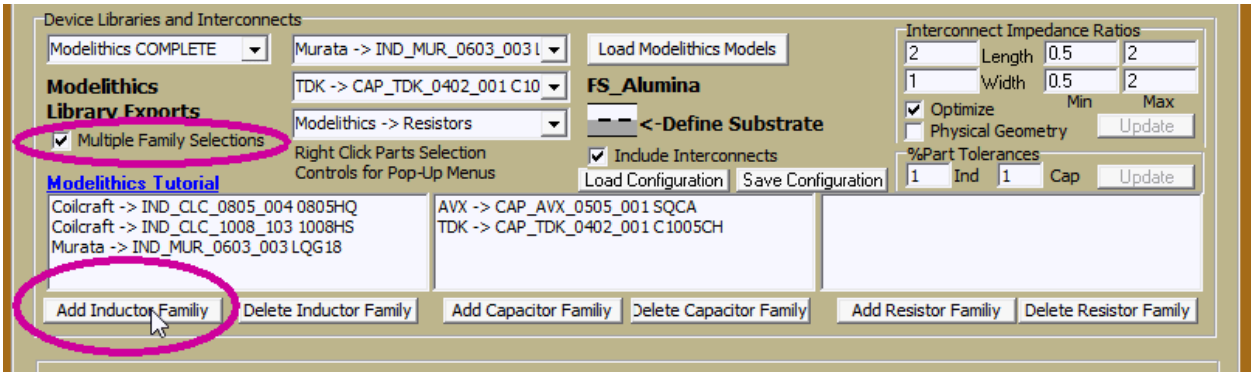
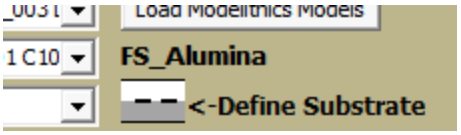


Figure 3: Selecting Multiple Part Families

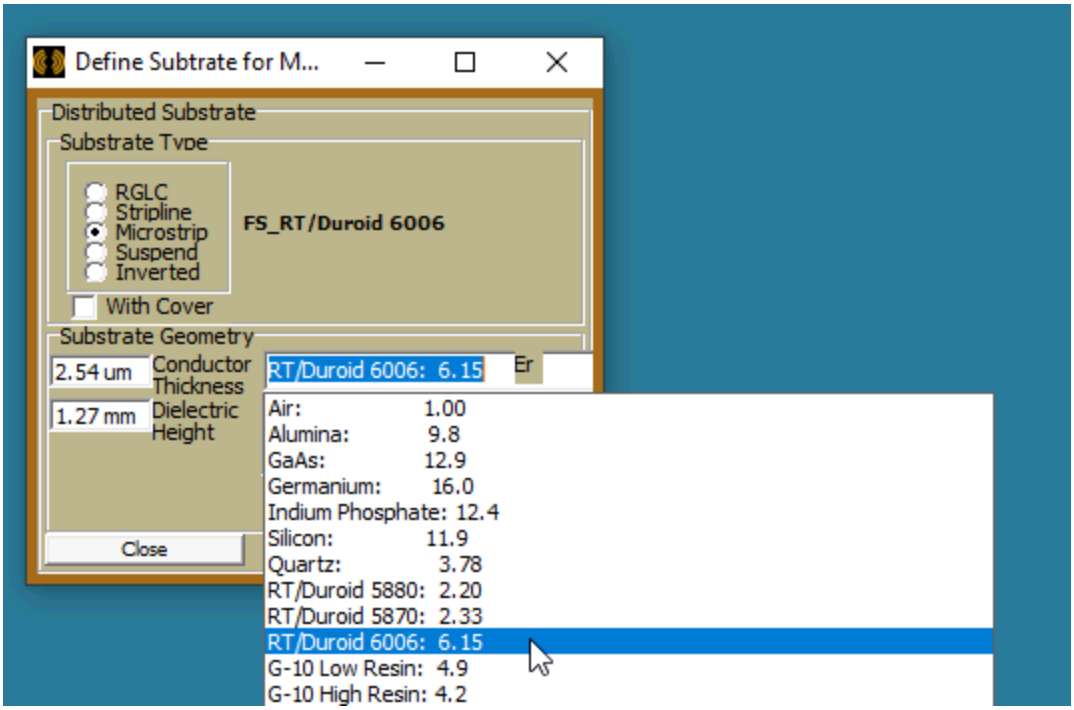
Defining the Substrate

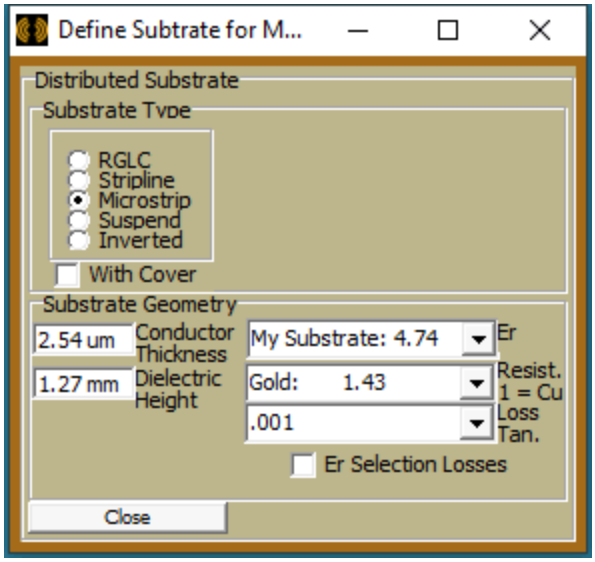
Select **Define Substrate** in the export control panel to define the substrate in the Modelithics schematic to be exported to Circuits.



The substrate definition page shown in Figure 4 below will appear. Any built-in substrate material may be selected, or a custom name and Er value may be entered by typing it in, as shown in Figure 4.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





Built-In Material Selection

Hand Entered Material Selection

Figure4: Defining the Modelithics Substrate

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Circuits Schematics and Layouts

Modelithics elements in Circuits schematics support layout cells with ports to attach to micro strip interconnect elements.

Interconnect Length Ratio and **Impedance Width Ratio** are used to set initial length and width geometry of the interconnects. Geometry may be defined in terms of substrate height ratios or physical dimensions, as desired, shown in Figure 5. Circuits/Modelithics Schematics are designed to produce valid layouts, as shown with this five pole Zigzag filter below in Figure 6, where 4 inductors and 11 capacitors may easily be seen to be included in the layout. Initial layout geometry is set to be optimized with the use of Circuits "S" and "W" equations. All Circuits equations are designed to insure a valid layout when reasonable geometry setting are entered.

The first export generally with Optimize unchecked so that only the discrete element values are optimized. Layout geometry Circuits optimization may be turned on and off remotely from the FS export panel by checking or unchecking Optimize and then selecting the **Update** button to the immediate right of Optimize.

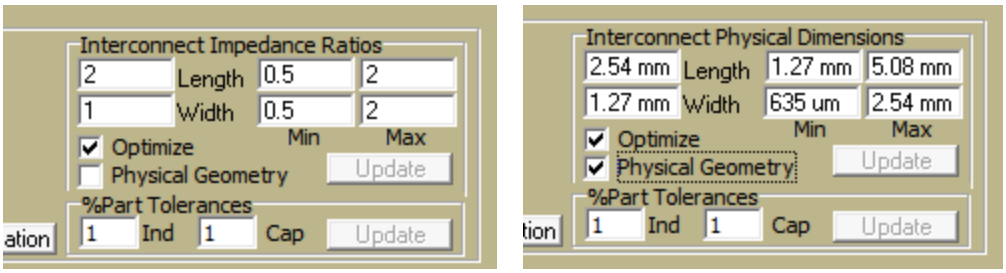


Figure 5: Interconnect geometry Defined in Terms of Substrate Height Ratios or Physical Dimensions

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

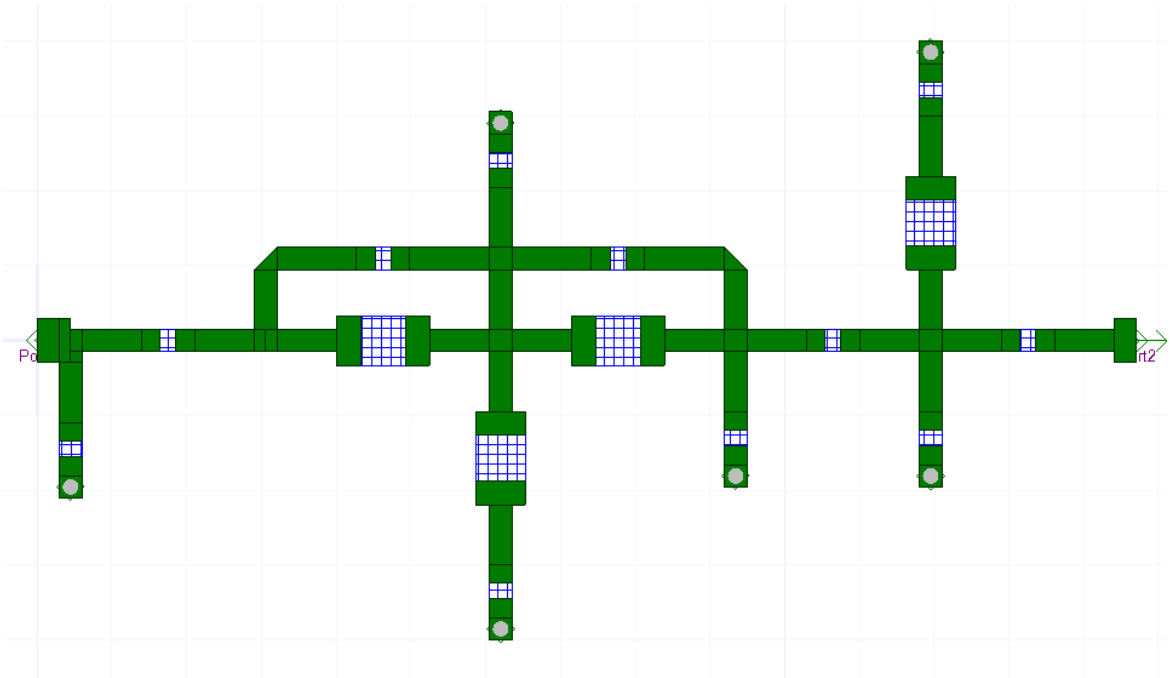


Figure 6: Valid Circuits Schematic Layout of a Modelithics Parts Filter Design

Tuned/Optimized Layouts

Modelithics layouts are designed to remain valid for reasonable width and length adjustment. All equations are embedded into the design variables and layout geometry that are needed to keep the layout valid in the presence of interconnect width and length adjustments. Figure 7 below shows the circuit layout from Figure 6 above after the interconnects have been adjusted through tuning and/or optimizing.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

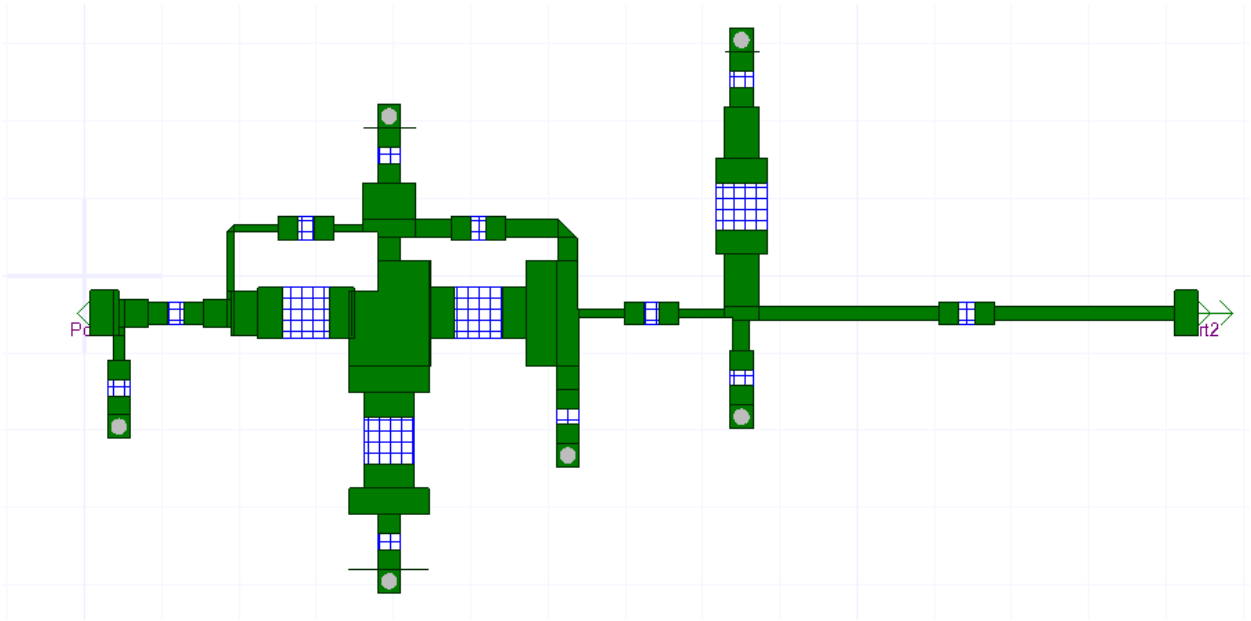


Figure 7: Valid Circuits Schematic Layout With Tuned/Optimized Interconnects

3D Data

Due to the numerous 3D applications potentially available to export FilterSolutions layouts into, FilterSolutions provides a generic file export of all the data needed to create a 3D simulation. Data may be inserted manually into 3D simulation functions, or an automated script may be written by the user to assist. Export 3D date to a text file by choosing **3D Data Setup** or **3D Data Direct** from the Export drop down menu in the upper right of the Distributed filter schematics page. **...Setup** will bring up the 3D Data export control panel **...Direct** will export directly using the existing saved setting in the 3D Data export control panel.

Please refer to the following list of data definitions:

Components	Definition
Box	This defines the box and cell geometry. Boxes and cells are used in some 3D applications. They may be ignored if not used.
Stackup	This defines the substrate layers in 3D dimensions. Each layer may be inserted into a 3D application.
Parameters	This defines the parameters used to define the filter layout. Default tunability state is provided as well as default minimum and maximum values.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Goals	Default optimization goal definitions are provided. These are usable if it is desired to export goals into a 3D simulation function.
Geometries	3D geometry definitions are provided for each layout component. A variety of geometry definitions are provided to facilitate all the potential ways a 3D application can accept data.
Elements	Lumped element definitions are included for planar filters that include lumped elements, such as many high pass filters.
Ports	Port definitions are provided to define the 3D ports.

DXF AutoCAD

Microwave printed layouts may be exported in DXF AutoCAD format for efficient importing into other planar Microwave analysis software. Strip line, micro strip and suspended substrates schematics have an "Export" selection in the tool bar. Select the "DXF AutoCAD" entry to export in a DXF formatted file.

Exported Features

DXF format files only save the layout shapes and VIAs. All other features must be manually added in the other analysis software. These features include: ideal elements, ports, substrate dimensions, conductor material definition, and substrate material definition.

DXF Shapes and VIAs

DXF defines layout shape geometry and positions in a specific layer. DXF VIAs are saved in a different layer. When importing into the other software, it may be necessary to manually define which substrate layer the shapes go in, and which layers are connected by the VIAs. See the other software Help documentation for specific DXF VIA importing instructions.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Geometry Dimension Units

> **Important:** DXF does **not** save units information. The standard units provided by FilterSolutions are meters, feet, inches, and mils. The units in which the DXF file was exported must be manually entered into the importing software.

Reference Planes and BoxBoundaries

DXF does not save the reference plane information or box boundary information. DXF cannot save the designer's reference planes, so if they are needed, they must be recreated in the importing software. If long attachments are needed, they must also be created in the importing software.

Touchstone Files

Designed filters can be exported to Y, Z, or S parameter Touchstone files that may be imported into other circuits analysis programs. Each filter schematic contains an **Export** function in the upper left that, when selected, offers Touchstone Z, Y, and S parameter file exports.

Circuits analysis programs usually include an element called "N-Port", "Ideal element", etc. that have the capability to read or import Touchstone files that contain Z, Y or S parameter data, and effectively model the designed filter when they read a FilterSolutions generated Touchstone file.

Port Definitions:

2 port filters:

Port 1 = Input.

Port 2 = output

Diplexers:

Port 1 = Input

Port 2 = Low pass output

Port 3 = High pass output

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Triplexers:

Port 1 = Input

Port 2 = Low pass output

Port 3 = High pass output

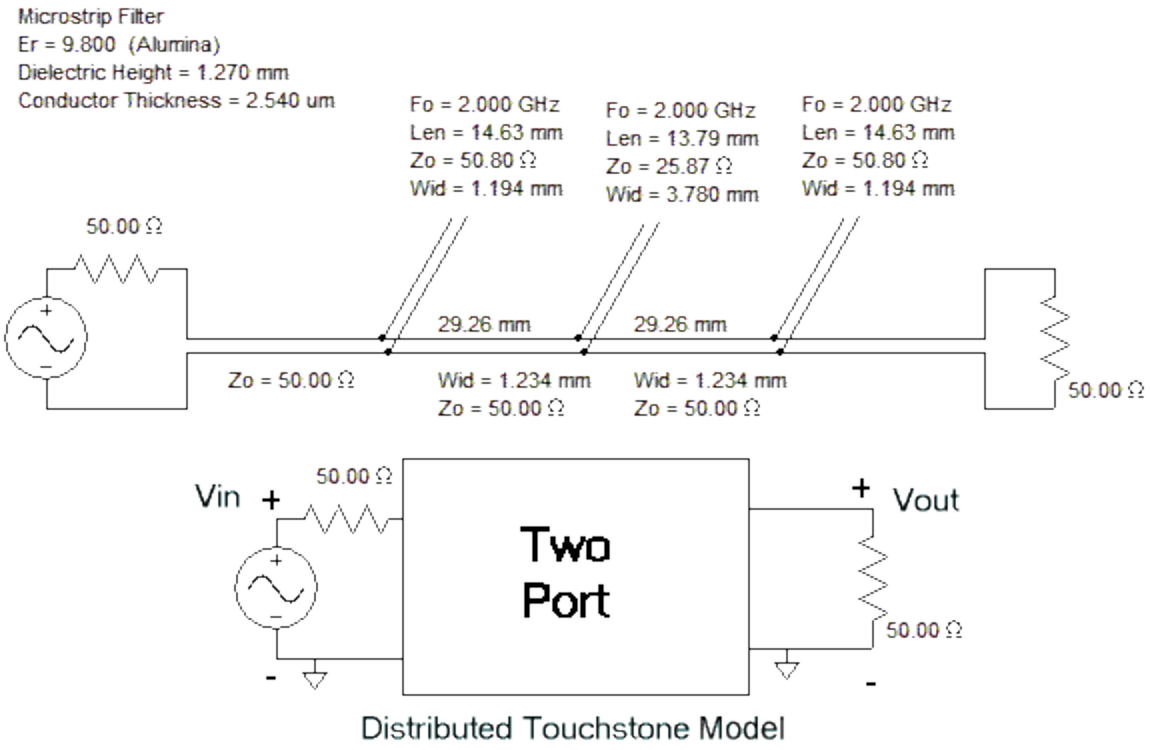
Port 4 = Band pass output

Tips:

- Z and Y parameters model only the filter network, and do not include any source or load terminations.
- S parameters Touchstone exports are available only for equally terminated filters. This is due to restrictions in standard Touchstone file data format requirements.
- beginning and ending frequencies are determined by the FilterSolutions Min Freq and Max Freq entries.
- If the Touchstone file frequency response contains noise or spikes, add a touch of loss to the reactive elements to smooth it out.

Illustrations:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

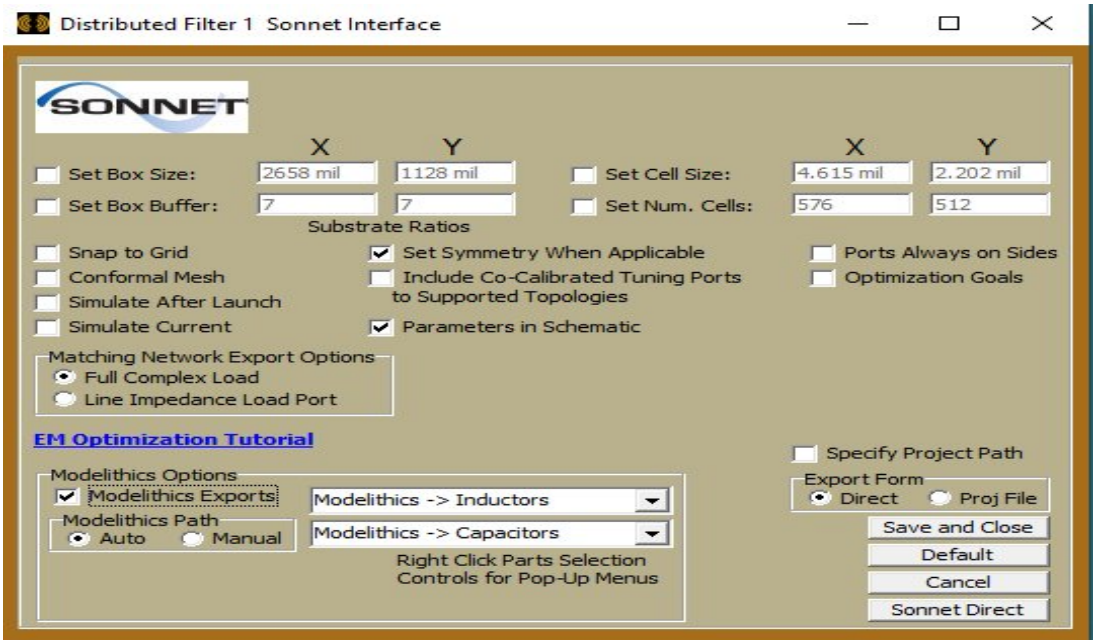


Sonnet Exports

Distributed stripline, microstrip, and suspended substrate filters may be exported to Sonnet file format for importing into Sonnet Lite or Sonnet Professional. Sonnet Software, Inc (www.sonnetsoftware.com) is dedicated completely to the development of commercial 3D Planar High-Frequency ***em***® (electromagnetic) software.

Small and medium size filters without extreme geometries may be analyzed by Sonnet Lite or Lite+, but may require that the grid resolution be adjusted manually in Sonnet. Large filters, or filters with extremely thin segments generally require Sonnet Professional.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Exporting to Sonnet

Sonnet export options exist in the "Export" selection menu in the upper left of the filter schematic. Three options exist:

1. Sonnet Setup

   Launches the Sonnet export control panel. Use this panel to set up desired Sonnet export options. Default options are uniquely calculated for each filter prior to the panel's display, and may be overwritten by selecting the appropriate checkboxes to the left of the calculated option values.

Set Box Buffer	Check to enter the distance in substrate heights between the edge of the filter and the box wall. Entries are ignored when unchecked. This selection is mutually exclusive of the "Set Box Size" selection.
Set Box Size	Check to enter the X and Y dimensions of the Sonnet box in selected length

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	units. Entries are ignored when unchecked. Mutually exclusive with "Set Box Buffer". This selection is mutually exclusive of the "Set Box Buffer" selection.
Set Cell Size	Check to enter the X and size of cells to be used in selected length units. Entries are ignored when unchecked. This selection is mutually exclusive of the "Set Num Cells" selection.
Set Num Cells	Check to enter the number of cells to be used in the box X and Y directions. Entries are ignored when unchecked. This selection is mutually exclusive of the "Set Cell Size" selection.
Snap to Grid	Check to snap the Sonnet export to the grid.
Conformal Mesh	Check to use conformal mesh to reduce memory requirements and computation time. Sonnet recommend reading the chapter "Conformal Mesh" in the Sonnet Users Guide before using conformal mesh.
Simulate After Launch	Causes the Sonnet project to simulate immediately after exporting.
Set Symmetry When Applicable	Check to force upper and lower box symmetry for filter layouts that qualify. Useful for speeding up the simulation while maintaining accuracy.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Include Co-Calibrated Ports...	Check to include Co-Calibrated tuning ports. Useful for port tuning on platforms not directly supported by Nuhertz.
Simulate Current	Check to include current density in the exported Sonnet project.
Matching Network Export Option	Used in Zmatch. Options: a. User Selectable Prompt at export time which of the option below you wish to use. b. Full Complex Load Model the complex load in a Touchstone format file to go with the exported Sonnet project. This is useful to determine the exact behavior of the Sonnet matching network when terminated by the complex load. c. Line Impedance Load Substitute the line impedance, usually 50 ohms, for the complex load. This is useful for Sonnet to generate S port parameters of the designed matching network.

2. Sonnet Direct

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Export directly to Sonnet using the export panel entries defined in Sonnet Setup.

3. Sonnet Save

   Save the filter design in Sonnet format using the export panel entries defined in Sonnet Setup.

To export to Sonnet, design a distributed stripline, microstrip, or suspended substrate filter with finite source and load resistance. Select "Export" in the tool bar, and then, "Sonnet Save" to export to a Sonnet format file, or "Sonnet Direct" to export directly to Sonnet. For the file option, select a path and file name, and the filter will be exported into a format that is readable by Sonnet.

Tips:

- Microstrips and suspended substrates with thick air between an upper cover and the conductor should use free space on top instead of a lossless cover. This will reduce high frequency EM simulation glitches as well as EM simulation time.
- Resonator corners and parallel edge coupled angle are selected in the in the Options, accessible from the main menu, and used for Sonnet exports definitions.

Restrictions:

- Load and Source resistance must both be finite.
- Parallel resonators composed of capacitors in parallel with a metal segment are not supported.
- Parallel resonators composed of lumped inductors and capacitors are not supported

Modelithics® in Sonnet

Planar filters exported to Sonnet that include lumped elements (frequently found in high pass filters) have the option of being simulated with highly accurate Modelithics models.

Modelithics Options Panel

To export Sonnet lumped elements to Modelithics, it may be required to first set up the correct path to the Modelithics installation if multiple Modelithics versions are installed or if the Modelithics installation is not in the default directory. If only one version is installed, it is generally OK to select the **Auto** Modelithics path. Select the **Manual** Modelithics path if it is desired (or required) to select a specific Modelithics installation. Select the specific Modelithics

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

installation by selecting **Get Path** and browsing for the "library.xml" file that resides in the desired Modelithics installation.

Once the path is set properly, select the desired inductor and/or capacitor model family from the pop-up menus as shown in Figure 1, and export. The nearest Modelithics model value will be used and the Modelithics model will be inserted into the Sonnet lumped element.

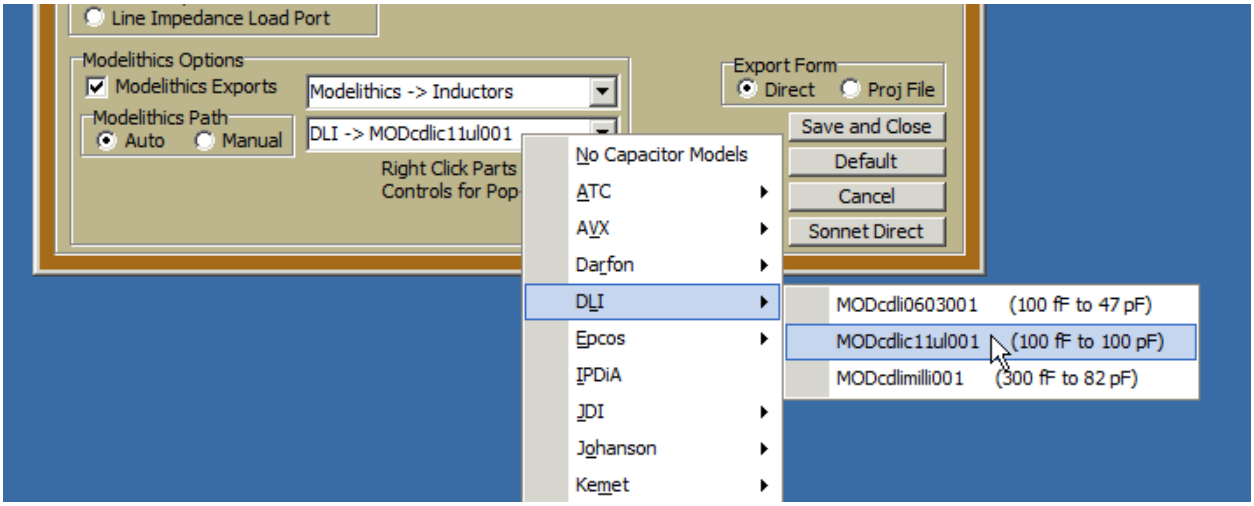


Figure 1: Selecting Modelithics model families from FilterSolutions Sonnet Export Page

Editing Modelithics in Sonnet

Double click on a exported Modelithics part that is desired to edit. The Components dialog will appear. Make any edits as desired in the components dialog. Geometries and values may be edited under the "Parameters" button..

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

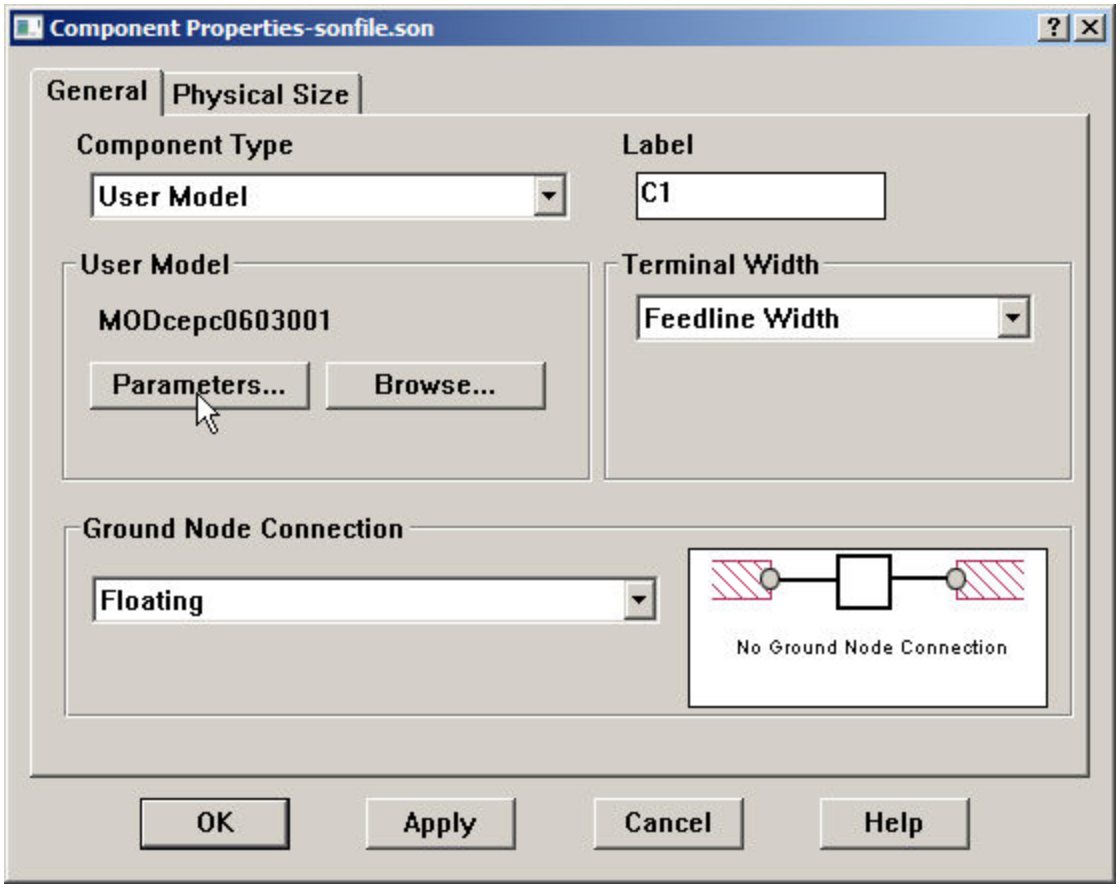


Figure 2: Editing a Modelithics Parts With the Components Dialog

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14 - Active Filters

This chapter contains the following topics:

- Thomas Biquads
- Akerberg-Mossberg Biquads
- Sallen & Key
- Multiple Feedback (MFB)
- Leap Frog Active Filters
- Parallel Active Filters
- GIC Stages
- Active Stages
- Band Pass Architectures
- Opamp and Element Parasitic Effects
- Sensitivity
- Modify Active Filters
- Net Lists
- Exporting to Other Software

Thomas Biquads

This chapter contains the following topics:

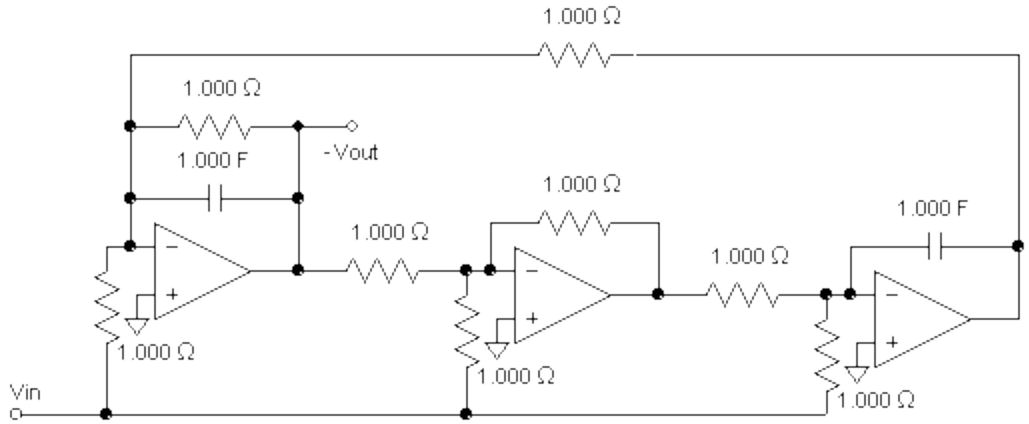
- Thomas Biquad Description
- Design Equations
- Third Order Thomas Biquads

The Thomas biquad is a three amplifier biquad that is capable of realizing virtually any biquad of any given Q, Wo, and Gain. Its main advantage is that it is highly resistant to opamp imperfections, provides very high quality notches, and is easy to design and tune, although high speed computers have lessened the need for easy design and tuning capability. Thomas biquads are composed of three inverting single stages and a feedback loop. Each stage is either an integrator, a gain, or a first order lag as determined by the feedback elements. Each stage accepts a summation from the input through a resistor or a capacitor.

FilterSolutions provides two Thomas biquad implementations which will be referenced here as Thomas 1 and Thomas 2. The Thomas 1 biquad is easier to design and requires fewer capacitors. The Thomas 2 biquad provides higher quality notches in the presence of real elements.

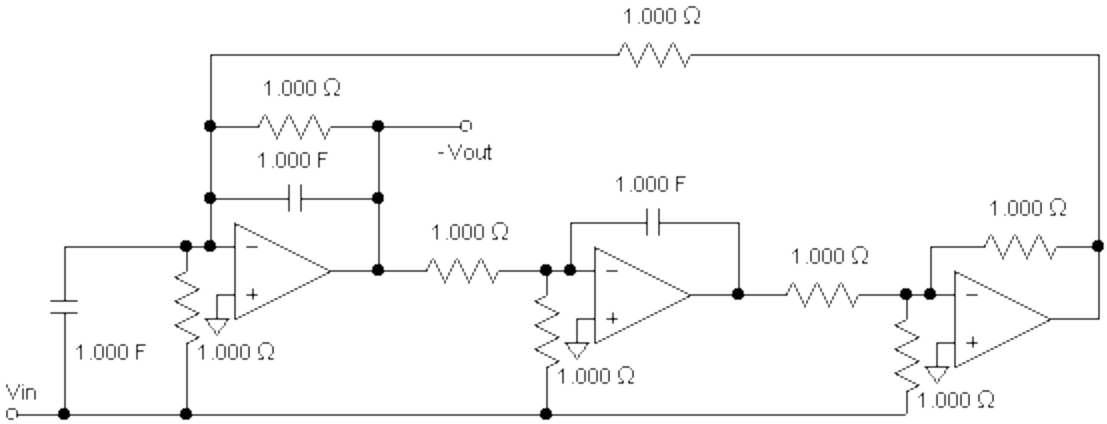
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Thomas 1 biquad active filter circuit with all elements set to unity is shown below. Note that there are only two capacitors. Also note that if the middle summation resistor on the input to the second opamp is not needed, then the output of the first opamp is the inverse of the second opamp. This provides some sign flexibility for this biquad.



Thomas 1 Biquad

The Thomas 2 biquad active filter with all elements is shown below. Note the capacitor on the summation input to the first opamp.



Thomas 2 Biquad

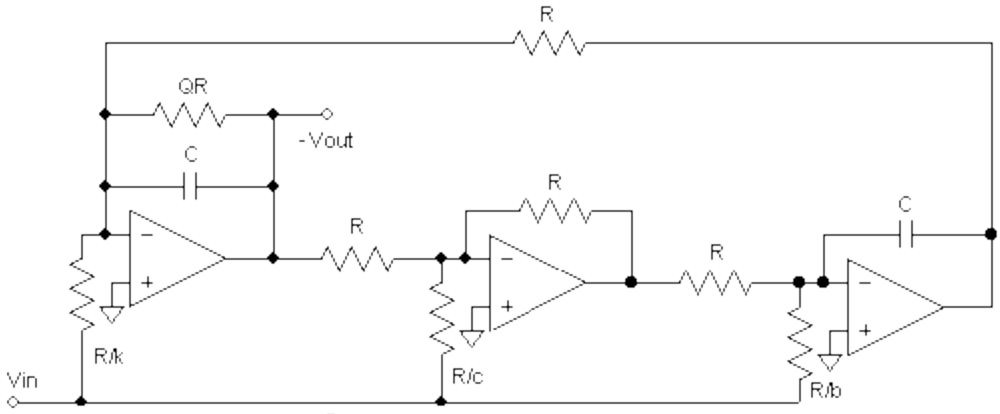
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Advantages and Disadvantages

Biquad	Advantages	Disadvantages
Thomas 1	• No more than two capacitors are required. • Sign change capability when center input resistor is not installed.	• Less notch quality when element value errors are introduced
Thomas 2	• Better notch quality when real elements errors are introduced. • Sign change capability for band pass stages.	• The sign is always negative for all stages other than band pass stages. • Three capacitors required for high pass, notch, and all pass stages.

Thomas Biquad Design Equations

The design equation for both the Thomas 1 and Thomas 2 biquads are shown below. From the Thomas 1 biquad equations, it is seen that the notch always requires two elements to subtract. When real elements are installed into the filter, the error in the element values degrades the quality of the notch. It may also be seen from the equations that all biquad realizations may be made without the need for a third capacitor.

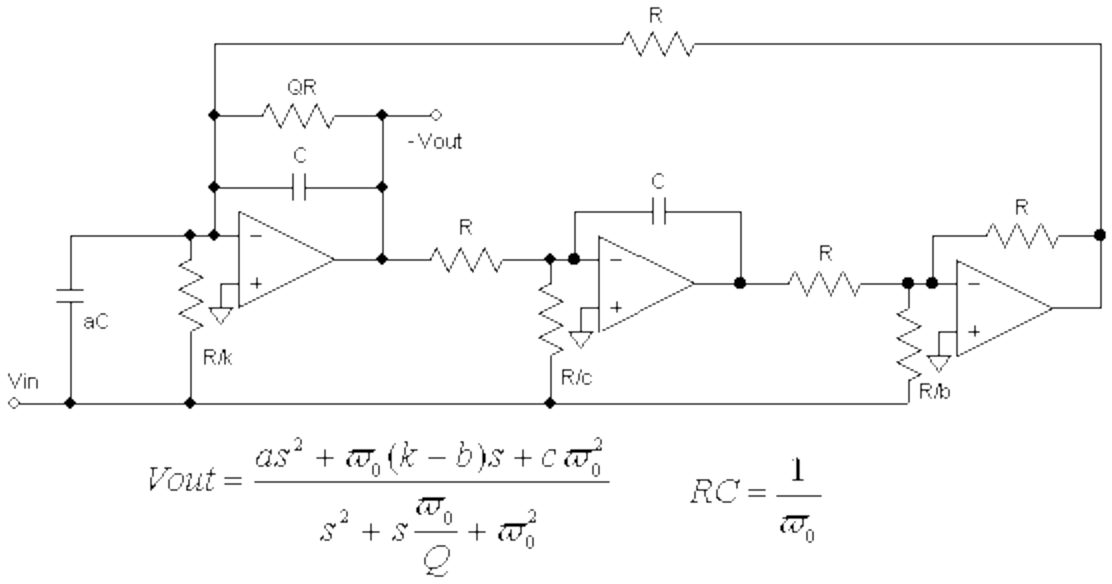


$$Vout = \frac{cs^2 + \omega_0\left(\frac{C}{Q} - k\right)s + b\,\omega_0^2}{s^2 + s\frac{\omega_0}{Q} + \omega_0^2} \qquad RC = \frac{1}{\omega_0}$$

Thomas 1 Design Equations

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Thomas 2 design equation below show that by removing the k and b resistors, a notch is created that is immune to element value errors. However, a third capacitor is required to realize all high pass and notch filters.



$$Vout = \frac{as^2 + \varpi_0(k-b)s + c\,\varpi_0^2}{s^2 + s\frac{\varpi_0}{Q} + \varpi_0^2} \qquad RC = \frac{1}{\varpi_0}$$

Thomas 2 Design Equations

Thomas Biquad With Third Additional Pole

When **Absorb 1st Orders** is selected in the Active Filter control panel, isolated single poles may be absorbed into adjacent second order biquads with the use of an additional resistor and capacitor. This is useful when building odd order filters because it negates the need for an additional single amplifier stage. Any Thomas Biquad may absorb an isolated pole.

Akerberg-Mossberg Description

This chapter contains the following topics:

- Akerberg-Mossberg Description
- Design Equations
- Third Order Akerberg-Mossberg Biquads

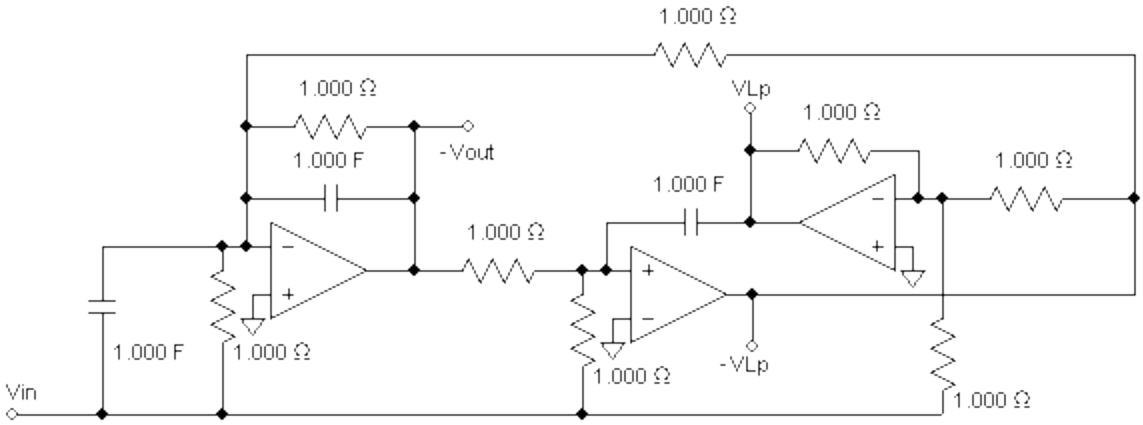
Akerberg-Mossberg Biquad

The Akerberg-Mossberg biquad exceeds the performance of the Thomas biquads for opamp imperfections and matches the Thomas 2 biquad notch performance in the presence of element

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

value errors. This increased performance is obtained by replacing the positive integrator in the Thomas 2 biquad second and third opamps with a Miller integrator. The Miller integrator uses two matched opamps in a configuration that tends to cancel errors due to opamp imperfections.

The output of the Akerberg-Mossberg biquad may be from any of the three opamps.



Akerberg-Mossberg Biquad

Miller Integrator

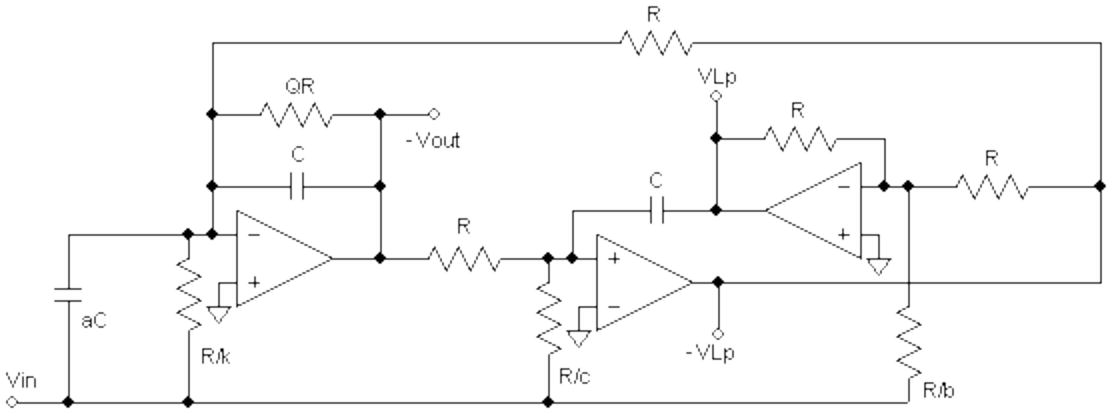
The opposing second and third opamps in the Akerberg-Mossberg biquad shown above constitute the Miller integrator. The outputs of the two opamps are equal and opposite. If matched opamps are used in the Miller integrator, this opposing arrangement tends to cause errors due to opamp imperfections to cancel, and thereby create a high quality output signal for the biquad as a whole.

Biquad	Advantages	Disadvantages
Akerberg-Mossberg	• Highest tolerance to opamp imperfections of all biquads available in FilterSolutions. • Notch quality is immune to element value error just as in the Thomas 2 biquad. • Low pass sign change capability.	• Sometimes three capacitors are required as opposed to only two for the Thomas 1 biquad. • Three opamps are required as opposed to the one and two opamp biquad designs. • Sign is fixed for high pass, band pass, notch, and all pass stages.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Akerberg-Mossberg Design Equations

The design equations for the Akerberg-Mossberg are shown below. It is easily seen that be setting both k and b to zero will generate a notch that is immune to degradations due to element imperfections. It is also seen that the sign of band pass filters is changed by setting either k or b to zero. Since it has been observed that the b input to the Akerberg-Mossberg filter degrades the performance of the Miller integrator, FilterSolutions creates all Akerberg-Mossberg band pass filters using the k input. If a sign reversal is desired, the Thomas 2 biquad may be used instead.



$$Vout = \frac{as^2 + \varpi_0(k-b)s + c\varpi_0^2}{s^2 + s\frac{\varpi_0}{Q} + \varpi_0^2} \qquad VLp = \frac{k\varpi_0^2}{s^2 + s\frac{\varpi_0}{Q} + \varpi_0^2} \qquad RC = \frac{1}{\varpi_0}$$

Akerberg-Mossberg Design Equations

Akerberg-Mossberg With Third Additional Pole

When **Absorb First Orders** is selected in the Active Filter control panel, isolated single poles may be absorbed into adjacent second order biquads with the use of an additional resistor and capacitor. This is useful when building odd order filters because it negates the need for an additional single amplifier stage. The Akerberg-Mossberg biquad may absorb an isolated pole.

Sallen & Key

This chapter contains the following topics:

- Sallen & Key Single Amplifier Biquads
- Positive Gain Equal Element Stages
- Twin T Notch Configuration
- Fourth Order Stages

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sallen & Key Single Amplifier Biquads

Sallen & Key (also known as VCVS and Positive Gain SAB) single amplifier biquads (SAB's) are positive gain for all but even notch stages implemented with an injector resistor. The inclusion of injector resistor, R7, is used to create notch stages. When R7 is used in conjunction with a positive gain band pass stage, the sign of the stage is reversed from positive to negative and a negative gain notch stage is created. A Twin T notch configuration may be used for any notch stage and is discussed under the Twin T section of Help.

The Main Advantages of Sallen & Key to MFB are:

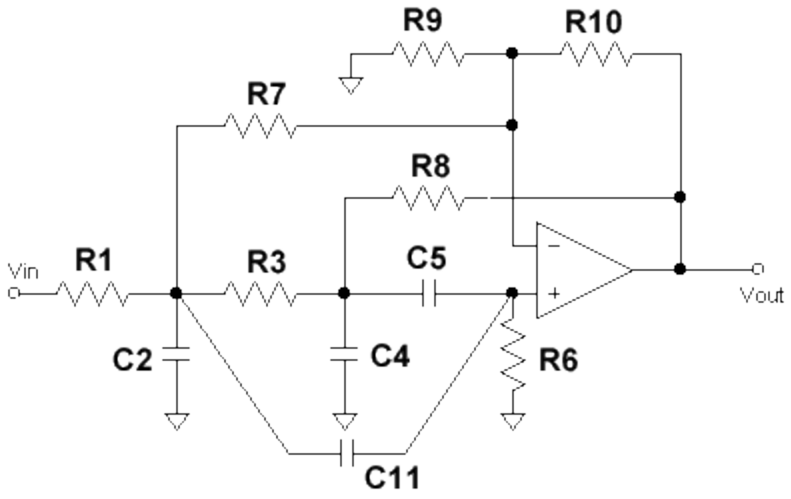
1. Generally lower opamp error sensitivity
2. Unlimited gain selection for notch stages.

The Main Disadvantages of positive SA's to negative SA's are:

1. Generally high element error sensitivity.
2. A Higher element count is sometimes needed.

The element count and sensitivities to element value and opamp error vary from configuration to configuration and should be checked prior accepting a stage topology.

The generic low pass form for the positive SA is shown below.



Sallen & Key Single Stage Amplifier

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Not all of the resistors and capacitors are needed at the same time. The table below states the functions of the individual elements.

Element	Function
R1, C2	These elements are used to create third order filters. High pass filters interchange the resistor and capacitor. If a second order stage is specified, these elements are not installed. Third order stages are not available for all pass stages.
R3, C5, R8, R6	These elements comprise the basic Sallen & Key filter. They are ALWAYS installed for second and third order filters. C5 and R6 are present in first order filters. Resistors and capacitors are interchangeable depending on the type of filter being designed. (high pass, band pass, etc.).
C4	This element is used to control gains of less than unity for second and third order low pass and high pass filters. It is also installed for band pass, notch and all pass filters. This element may also be a resistor
R9, R10	These elements are used to control gains of greater than unity for all low pass and high pass filters and for high gains of band pass filters. High and low pass filters with gains of unity or less set R10 to zero and omit R9. These elements are also installed for notch and all pass filters. These elements are always resistors.
R7	This element is known as the injection resistor. When inserted into a band pass filter, it converts the band pass stage to an even notch stage, and changes the gain from positive to negative. It is installed on all band stop, notch, and all pass filters. R7 is also used in conjunction with C11 to create low pass and high pass notch stages. This element is always a resistor.
C11	This element is used in conjunction with R7 to convert a second order low pass or high pass stage to a second order high pass notch or low pass notch stages. The sign of the gain is maintained when C11 is used. This element may be either a resistor or capacitor.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sallen & Key Equal Element Stages

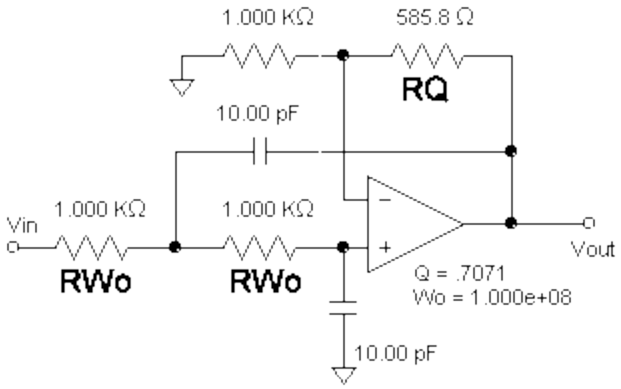
When a high or low pass positive gain SAB is created with equal resistor and capacitor values, the resulting filter is tunable in both Wo and Q. When the **Equal Caps Gain** or **Equal Resis Gain** option is selected in the **Active Control Panel**, all high and low pass positive SAB stages are created with a gain that sets all capacitors and resistors to equal values, except for the gain resistors.

For the following filter stages below, Wo and Q are adjusted according to the following statements:

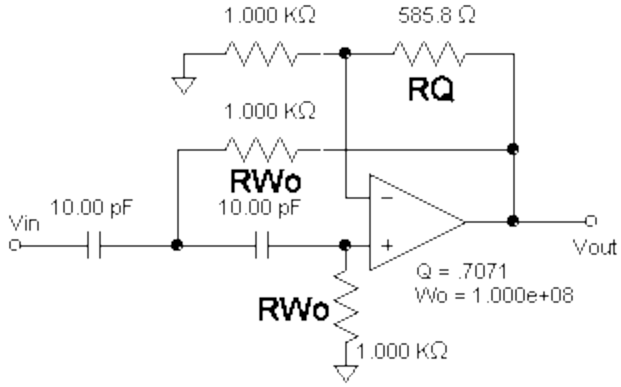
RQ changes the Q of the stage without altering Wo.

RWo changes the Wo of the stage without altering Q.

Low Pass Stage:



High Pass Stage:



Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

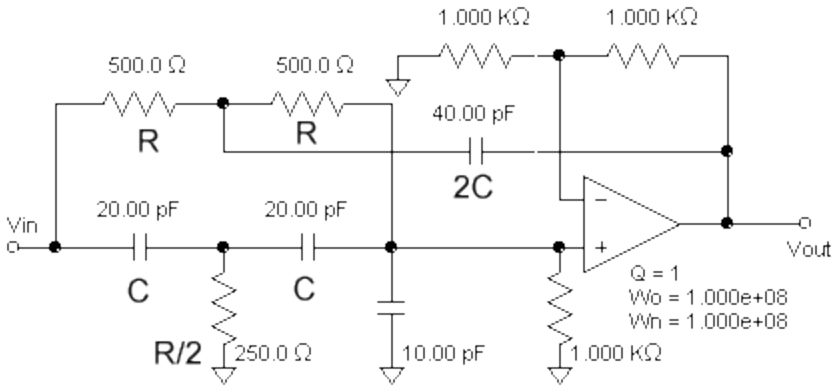
Twin T Notch Configuration

FilterSolutions supports an alternative positive gain notch configuration known as the Twin T. Twin T notch stages require more elements than the standard configuration and has limited gain flexibility, is less immune to element value errors and opamp error than the standard configuration. Nevertheless, it is supported for those who wish to use it and/or have a specific need for it.

Even gain notch sections may be created with either an injector resistor or a twin T. The default is to use an injector resistor. Checking the **Twin T Even Notch** box in the Active Filter Control Panel will cause even notch stages to be created with a twin T. All pass stages are not realizable with a twin T, and therefore are always created with the injector resistor.

Twin T Stage

The following is an example of a Twin T notch. The arithmetic for the elements values with the R and C labels is always the same for second order Twin T notch stages. To change these ratios will cause the stage to produce a third order results. The values for R, C, and all other elements are calculated by FilterSolutions.



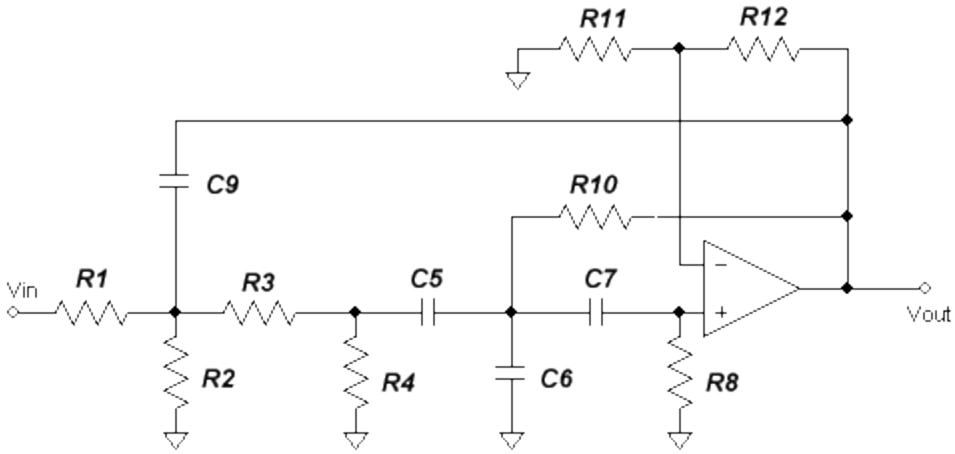
Sallen & Key Fourth Order Stages

Sallen & Key 4th order single amplifier all pole stages are possible by selecting **Sallen & Key** Active Implementation selection for a low pass, high pass or band pass all pole filter and checking the **Absorb 2ndOrders** checkbox. An additional feedback paths are created that will produce the desired fourth order response. For all but very low Q stages, it is necessary to add gain resistors to insure that all resistors and capacitors are positive and the impedance spread is reasonable.

In the fourth order active stage below, R3 and R4 are the two elements added to convert a second order stage into a fourth order stage. **R1, R2, and C9** are added to convert the third

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

order stage into a fourth order stage. (**R2** may be omitted if **R4** is included.) **R11 and R12** are gain resistors that are necessary for all but very low Q stages.



Fourth Order Single Amplifier Sallen & Key All Pole Active Stage.

Any fourth order all pole stage is realizable with the above architecture by selecting the appropriate capacitor and resistor locations and values. This includes low pass, high pass, band pass, and asymmetrical band pass stages. With the exception of R4, the above resistor and capacitor locations depict a band pass arrangement.

Disadvantages

Single amplifier fourth order stages have the disadvantage in that they are more sensitive to element value error than second and third order single amplifier stages, especially for high Q stages. Monte Carlos should be conducted to insure that you filter design requirements may be met with available resistors and capacitors.

Multiple Feedback (MFB)

This chapter contains the following topics:

- MFB Single Amplifier Biquads
- Positive Feedback Enhancement
- Alternate Notch Configuration
- Fourth Order Stages

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MFB Single Amplifier Biquads

MFB (also known as Bridged T and Negative Gain SAB) single amplifier biquads (SAB's) are negative gain for all but notch stages implemented with an injector resistor. The inclusion of injector resistor, R8, reverses the sign of the gain, making the gain of the MFB SA positive. This network can realize any filter (low pass, high pass, band pass, band stop, high notch, low notch, and all pass) with positive element values. The inclusion of injector resistor R8 reverses the sign of the gain.

The Main Advantage of negative SA's to positive SA's are:

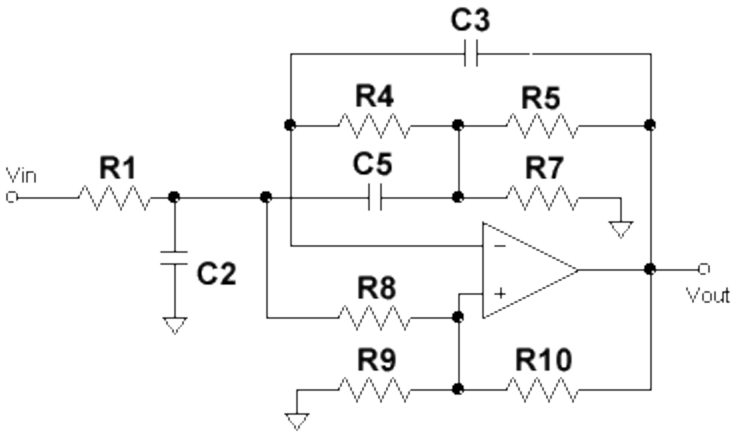
1. Generally lower element error sensitivity and sometimes a lower element count.
2. A lower element count is sometimes needed.

The Main Disadvantage of negative SA's to positive SA's are:

1. Generally higher opamp error sensitivity, although this error may be minimized for high Q stages with the use of positive feedback enhancement (Resistors R9 and R10).
2. Fixed gain selection for notch stages.

The element count and sensitivities to element value and opamp error vary from configuration to configuration and should be checked prior accepting a stage topology.

The generic low pass form for the negative SA is shown below.



MFB Single Stage Amplifier

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Not all of the resistors and capacitors are needed at the same time. The table below states the functions of the individual elements.

Element	Function
R1, C2	These elements are used to create third order filters. High pass filters interchange the resistor and capacitor. If a second order stage is specified, these elements are not installed. Third order stages are not available for all pass stages.
C3, R4, R5, C6	These elements comprise the basic **MFB** filter. They are ALWAYS installed for second and third order filters. C3 and R4 are present in first order filters. Resistors and capacitors are interchangeable depending on the type of filter being designed (high pass, band pass, etc.).
R7	This element is used to control gains for second and third order filters. It is usually, but not always, installed. It may also be a capacitor.
R9, R10	These are the positive feedback enhancement resistors. Their presence is optional and they have the effect of reducing opamp sensitivity to high Q (Q>=10) filters. These elements are always resistors.
R8	This element is known as the injection resistor. Its presence converts the gain from negative to positive. It is installed on band stop, notch, and all pass filters. The presence of this resistor also fixes the gain of the filter in second order stages.

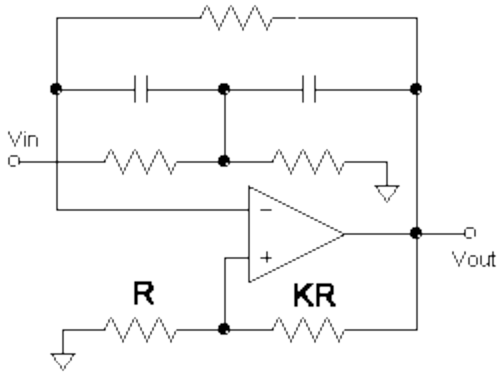
Positive Feedback Enhancement

> **Important:** Positive Feedback Enhancement is a feature applicable to second and third order MFB's only

When operating near the limitations of the op amp with a high Q band pass or notch filter, the degradation of the filter performance may be minimized with the use of positive feedback enhancement elements. This done by adding a positive feedback path to the filter with the resisters labeled R and KR in the diagram below. The amount of feedback is determined by the enhancement ratio K. K is the value entered into the Active Control Panel Opamp section. Each

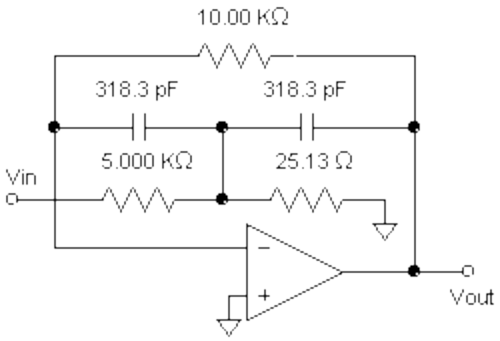
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

biquad stage in the filter may have the enhancement ratio set independently by clicking on the op amp.

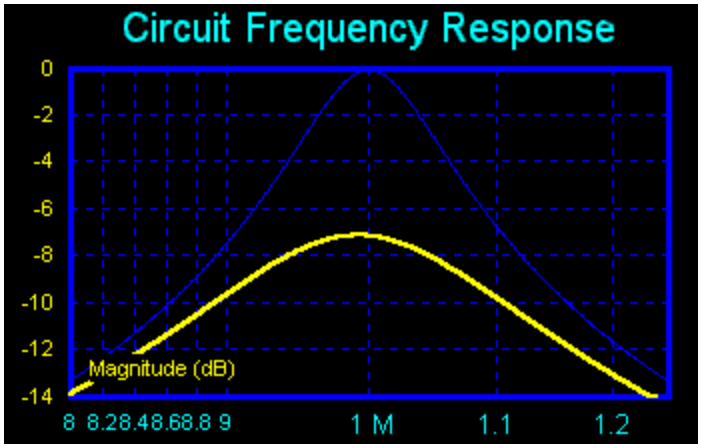


Positive Feedback Enhancement

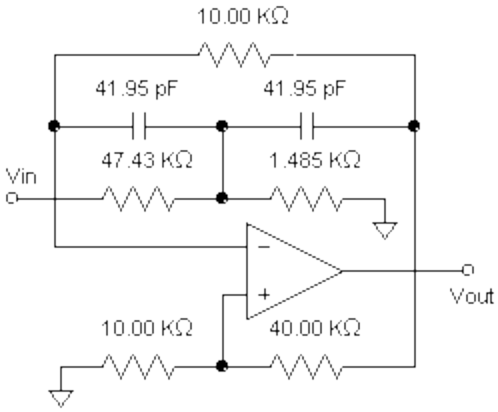
The unenhanced and enhanced band pass filters with Qs of 10 and finite gain opamps are shown below to illustrate. The unenhanced filter suffers a degradation of 7 dB at the center frequency. An enhancement ratio of 4 reduces the degradation to only 2 dB. The ideal filter trace is shown in dark blue.

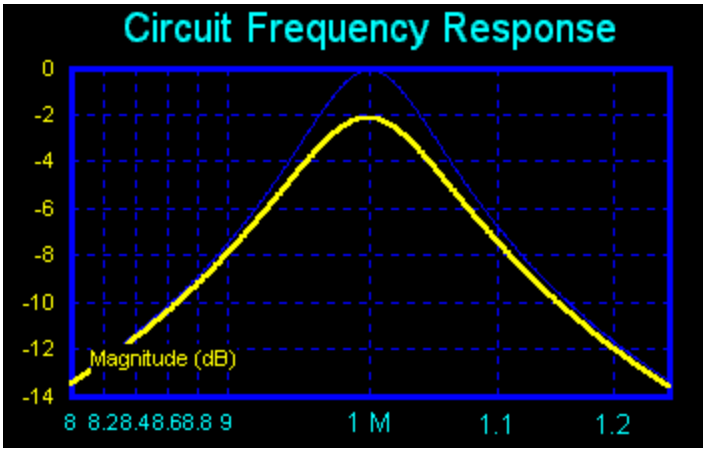


Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Negative SAB With No Positive Feedback Enhancement





Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

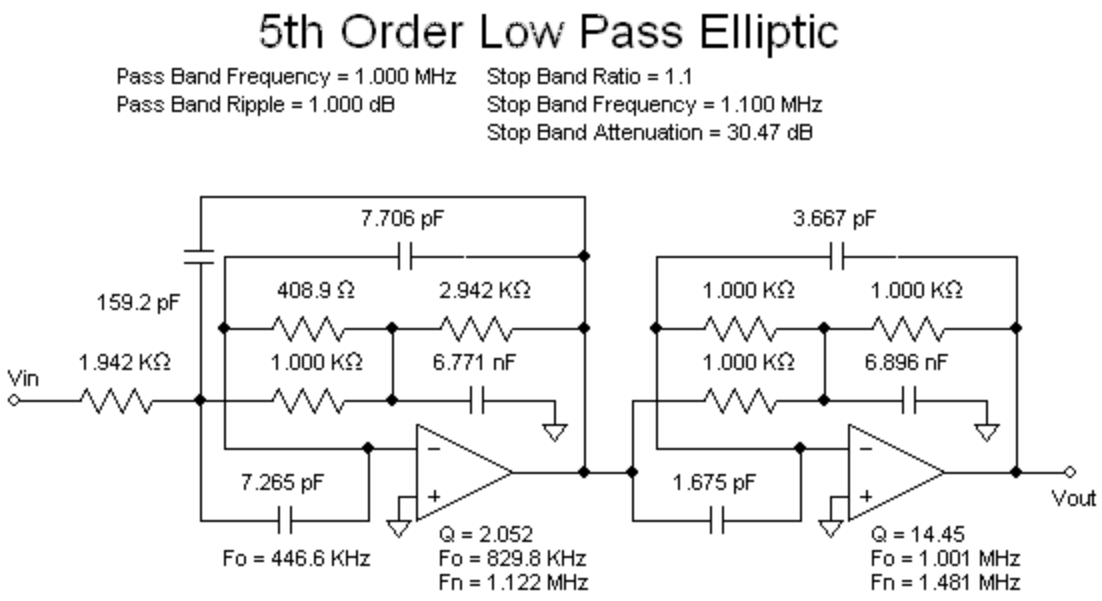
Negative SAB With Enhancement Ratio of Four

MFB Alternate Notch Configuration

MFB notch section with different high pass and low pass gains may be implemented with an alternate configuration that allows the gain to be adjustable and not inverted. The positive node of the opamp is tied directly to ground in this configuration. The injection element is moved from the positive input to the negative input of the opamp.

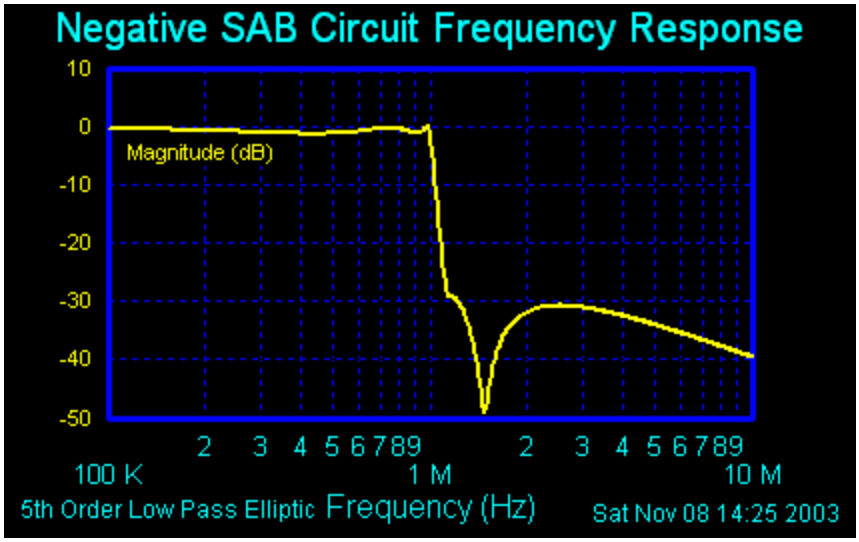
The drawback to the alternate notch configuration is that the notch quality is very poor and positive feedback enhancement is unavailable. However, in applications where notch quality is not important and opamp parasitics are not a factor, the alternate notch configuration produces a simpler filter.

An example of a fifth order Elliptic filter alternate notch configuration frequency response:



As may be seen in the following frequency response, the basic structure of the Elliptic response is preserved, and the notch quality is poor:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

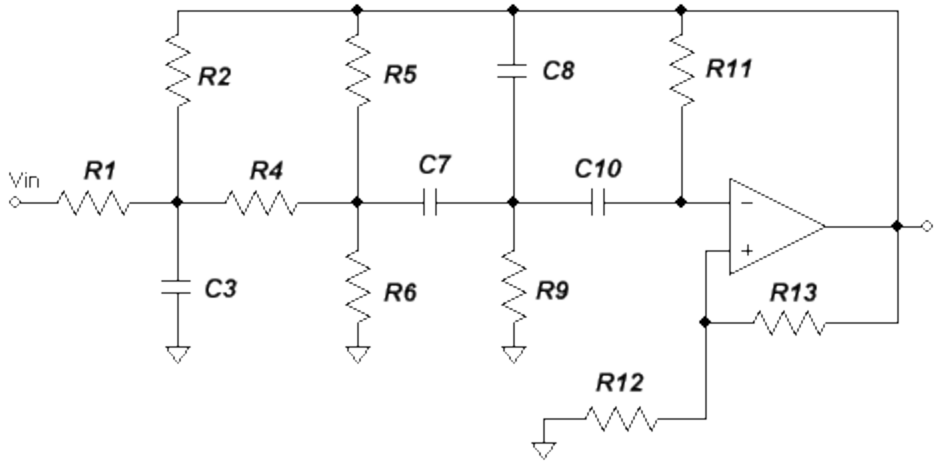


MFB Fourth Order Stages

MFB 4th order single amplifier all pole stages are possible by selecting **"MFB"** Active Implementation selection for a low pass, high pass or band pass all pole filter and checking the "**Absorb 2ndOrders"** checkbox. An additional feedback paths are created that will produce the desired fourth order response. Positive feedback enhancement, which is optional for second and third order filters, is required for fourth order filters. You may set the positive feedback yourself, or FilterSolutions will set it for you to a value that attempts to minimize the impedance spread of the elements..

In the fourth order active stage below, R3 and R4 are the two elements added to convert a second order stage into a fourth order stage. **R1, R2, and C3** are added to convert the third order stage into a fourth order stage. (**R2** may be omitted if **R4** is included.) **R12 and R13** are positive feedback enhancement resistors that are necessary for all negative gain fourth order stages.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Fourth Order Single Amplifier MFB All Pole Active Stage.

Any fourth order all pole stage is realizable with the above architecture by selecting the appropriate capacitor and resistor locations and values. This includes low pass, high pass, band pass, and asymmetrical band pass stages. With the exception of R5 and R6, The above resistor and capacitor locations depict a band pass arrangement.

Disadvantages

Single amplifier fourth order stages have the disadvantage in that they are more sensitive to element value error than second and third order single amplifier stages, especially for high Q stages. Monte Carlos should be conducted to insure that you filter design requirements may be met with available resistors and capacitors.

Leap Frog Active Filters

This chapter contains the following topics:

- Leap Frog Active Filters
- Leap Frog Transmission Zeros

Leap Frog Active Filters

Leap frog active filters are active simulations of passive LC filters. The advantage in doing this is that equally terminated passive LC filters distribute the element value error over the entire filter, which in turn has the effect of minimizing adverse effect of element value errors on the frequency

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

response. If such a filter is simulated in an active filter, then the synthesized active filter is expected to exhibit similar robust qualities.

Leap frog filters are relatively simple to create for all pole low pass and band pass all pole filters. The design complexity is greater for other forms of filters. Currently, band pass Leap Frogs are limited to all-pole designs. Low pass and high pass Leap Frogs support all-pole, as well as imaginary transmission zeros, such as in Elliptic as well as Butterworth and Chebyshev with manual imaginary zeros.

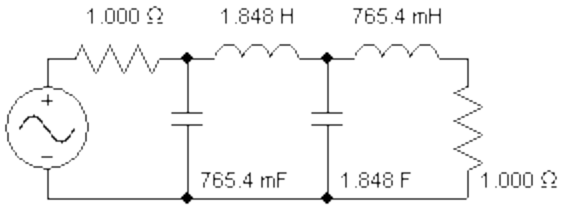
Alternate Topology Checkbox (available for low pass and high pass only) reverses the gain sign of each section in the filter. This results in an overall gain change for odd order filters, and greatly effects the tx zero topologies for filters with transmission zeros.

Synthesis of Filters

FilterSolutions uses the synthesis algorithm from *Filter Theory and Design, Active and Passive by Sedra & Bracket, pages 714 to 721, ISBN 0-916469-14-2* for low pass and band pass all-pole designs.

Essentially, a string of integrators alternating between positive unity gain and negative unity gain. Positive gain stages are implemented with a Miller integrator to maximize performance with real opamps. Each integrator output has a feedback and a feed forward resistor. The end integrators have additional resistors to account for the terminations. In the case of even order Chebyshev filters that are not modified for even order passive implementation, the value of the source resistor is calculated to be the maximum obtainable value that is less than the load resistor. The values of the feedback and feed forward resistors are calculated from the prototype LC filter and resistive constant entered by the user. The capacitor values of the leapfrog filter are calculated from the cutoff frequency of the filter and resistive constant entered by the user.

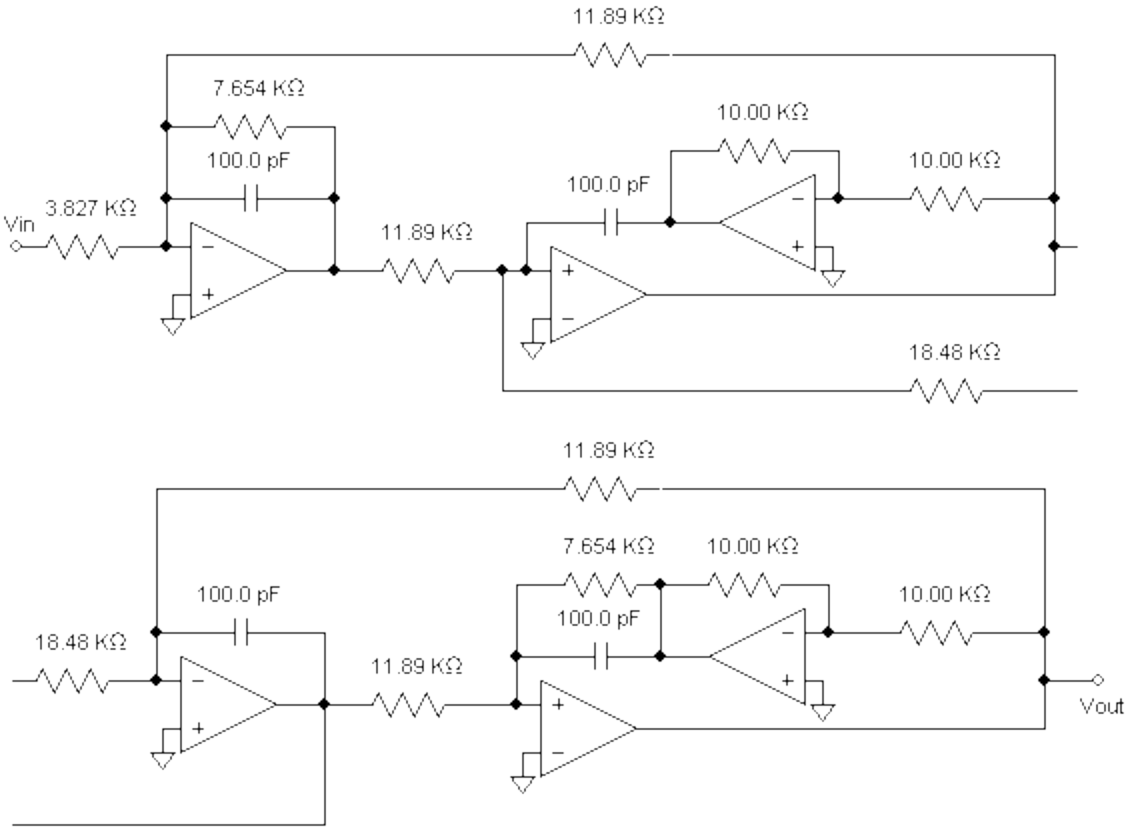
The following is a 4th order low pass LC passive Butterworth filter prototype.



Passive LC Ladder

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The element values, cutoff frequency, and user entered resistive constant are used to calculate the feedback, feed forward, and end piece resistors of the following Leap Frog filter:



Equivalent Leap Frog Active Filter

High pass designs use differentiaters instead of integrators. Transmission zeros utilize both a differentiater and an integrator.

Leap Frog Transmission Zeros

Three methods are supported for the detailed implementation of transmission zeros.

1. Parallel Miller integrator and differentiator (4 opamps, Figure A1 Below)
2. Traditional negative gain integrator and differentiator summed together into a negative gain summer. (3 Opamps, Figure B1 Below)
3. Parallel traditional negative gain integrator and differentiator. (2 Opamps, Figure C1 Below)

Low pass and high pass Leap Frog filters support all transmission zeros options. Band pass Leap Frog filters support option C) only.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Negative gain stages require option C), and is the most efficient and most robust option.

Positive gain stages require A) or B), which are less efficient and less robust. A) tends to be more robust in the pass band, B) tends to be more robust in the stop band. Option C) may be switched in and out for tx zeros spaced when spaced about by multiples of two (as in Elliptics) with the use of the **Alternate Topology** checkbox. Switching **Alternate Topology** effective changes the sign of each stage in the Leap Frog filter.

Low pass and high pass Elliptic filters may be set to always use option C) by selecting, **Optimal Setting** checkbox. Band pass filters only support option C), so the checkbox is not displayed.

Low pass and high pass leap frog transmission zeros:

Min Opamps checkbox selects between option A) and C).

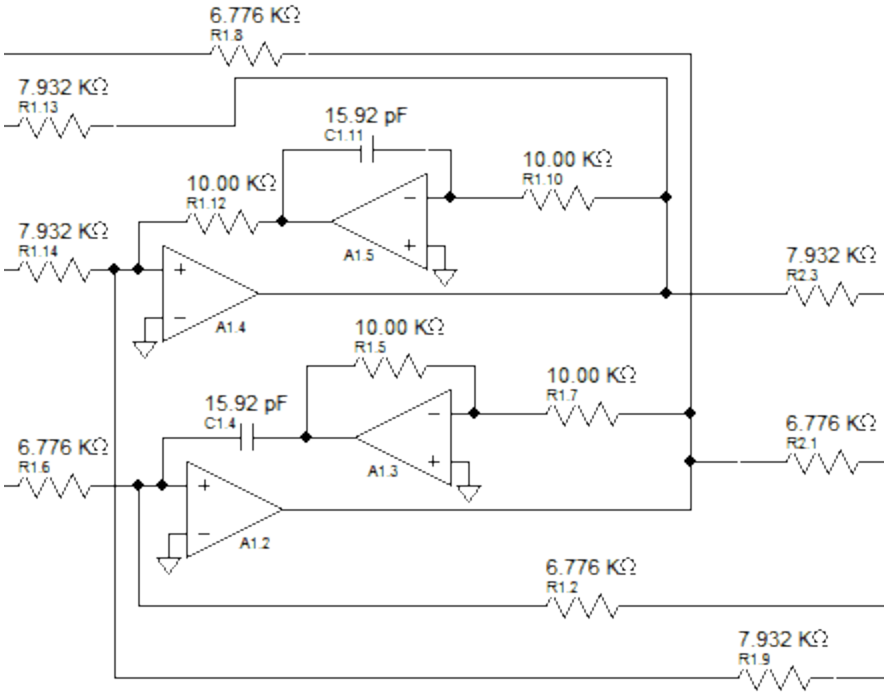


Figure A1:Leap Frog Transmission Zero Negative Gain Stage With Miller Integrator and Differentiator

Option (A above

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

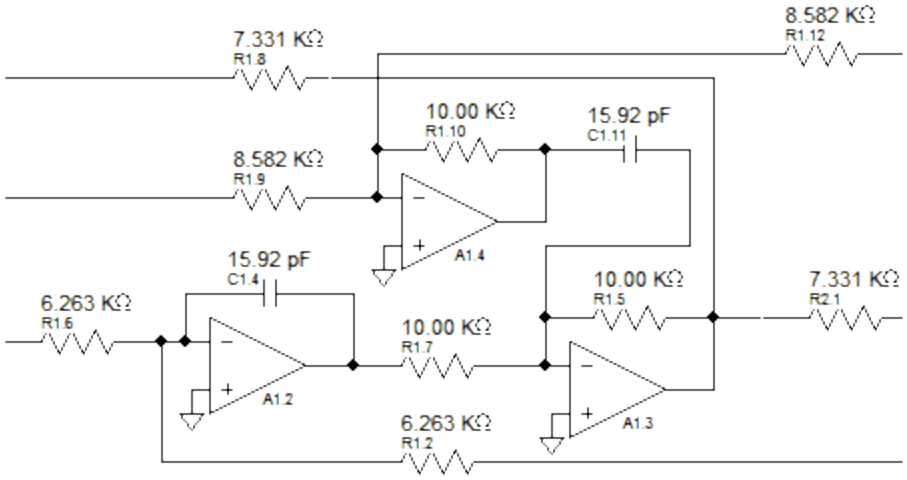


Figure B1:Leap Frog Transmission Zero Negative Gain Stage With Summed Integrator and Differentiator

Option (B above)

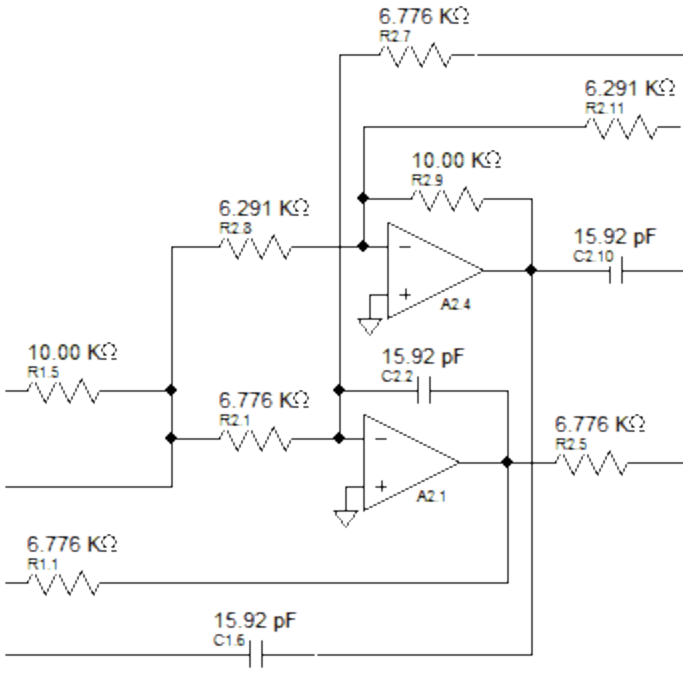


Figure C1:Leap Frog Transmission Zero Positive Gain Stage With Miller Integrator and Differentiator

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Option (C above)

Band Pass Transmission Zeros:

Band pass transmission zero pairs utilize cascaded integrator/differentiator summers as shown below in Figure 2.

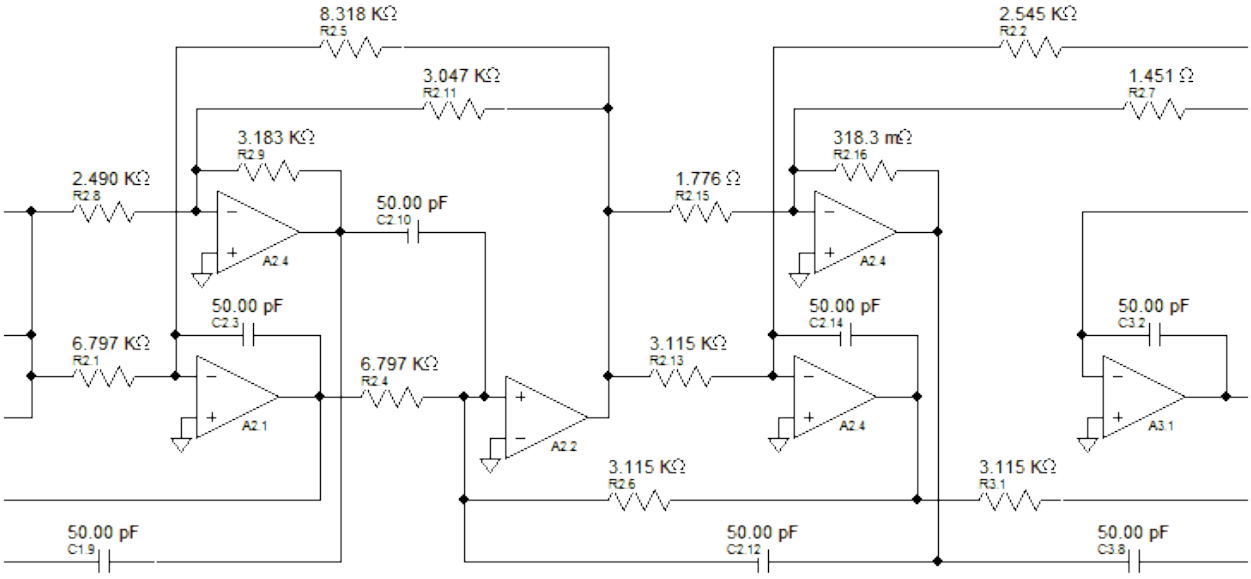


Figure 2: Leap Frog Band Pass Transmission Zero Pair

Parallel Active Filters

This chapter contains the following topic:

- Parallel Active Filters

Parallel Active Filters

When large filters are needed, they may be generated in either cascade form or parallel form. The other filter type selections, Thomas biquads, Sallen & Key, and MFB, use the cascade transfer function to derive the filter stages, and the stages are multiplied together. With the Parallel selection, the Parallel transfer function is used, and the stages are summed together. The advantage of a Parallel active filter is that errors due to discrete element values and real opamps are not amplified in successive stages. The disadvantage is that the parallel filter may

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

be much larger than the cascade, and attenuated areas such as notches tend to be of poor quality due to the multiple additions.

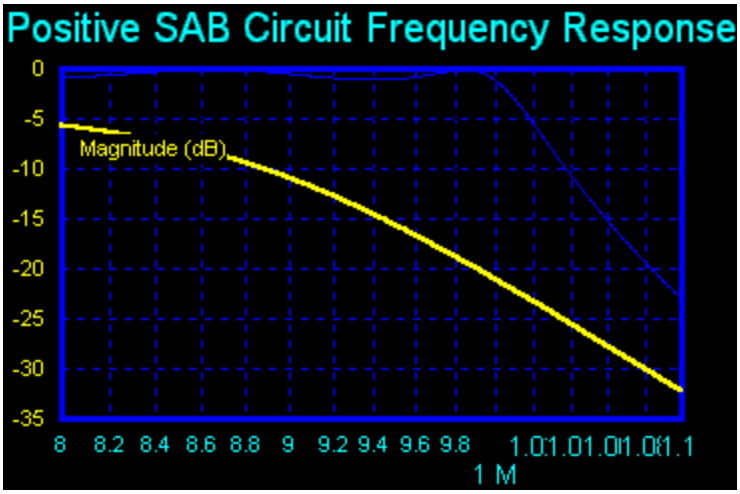
Sallen & Key and MFB first and second order single amplifier stages are used in the construction the Parallel filter. The final stage is an optional summation circuit used to add the other stages together. The summation circuit is included when "Final Gain Stage" is checked in the filter control panel. The advantage of the summation circuit is that the gain of the filter may be easily set. The disadvantage is that the additional opamp may degrade the filter performance. If the final summation circuit is not selected, the outputs of the individual stages are passively summed together.

Parallel Stage Gain Settings

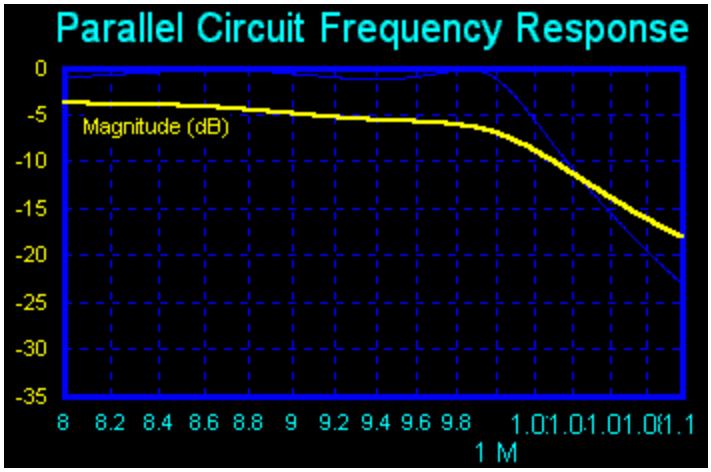
Each stage of the parallel filter is initially set to a gain that minimizes the number of elements needed in the stage. This gain may be undesirable for the application. If this is the case, you may change the gain of the stage by left clicking on the opamp. The associated summation resistor will change automatically to maintain the overall filter integrity. If "Final Gain Stage" is selected, the output summation amplifier will be updated automatically to maintain desired filter gain.

Example

A ninth order Chebyshev low pass filter with 1MHz corner frequency and a finite gain opamp is shown for the Positive SAB and the Parallel filters. Notice attenuation at the corner frequency of the positive SAB compared to the Parallel filter. Also notice the attenuation of the parallel filter is poorer than the positive SAB.



Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



GIC Stages

This chapter contains the following topics:

- GIC Biquads
- Third Order GIC's
- GIC Ladders
- Bridged GIC Ladders

GIC Biquads

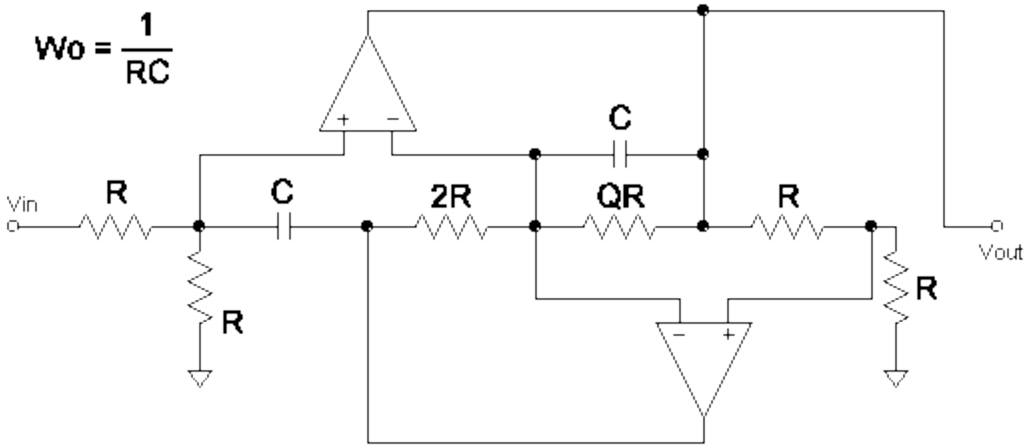
GIC Biquads are two opamp biquads with good high frequency performance. As expected, the high Q performance is generally better than a single opamp stage, but less that of a Thomas or Akerberg stage. All but the even notch stages are tunable. The high pass, low pass and band pass stages are gain adjustable. The notch and all pass stages have a fixed gain of unity. All GIC stages have equal capacitor values, unless a capacitor is required to adjust the gain. Like the Thomas 2 and Akerberg biquads, the GIC notch stages do not rely on element values for notch quality, and are thus immune from degradations in notch quality due to element value errors.

Care should be taken to watch the voltage level out of the lower opamp. It may be significantly greater than the upper. Even though the lower opamp does not connect to the following stage, it is possible for it to saturate. Click on an opamp to bring up the stage control panel and select **Alternate Opamp** to select the output of the lower opamp.

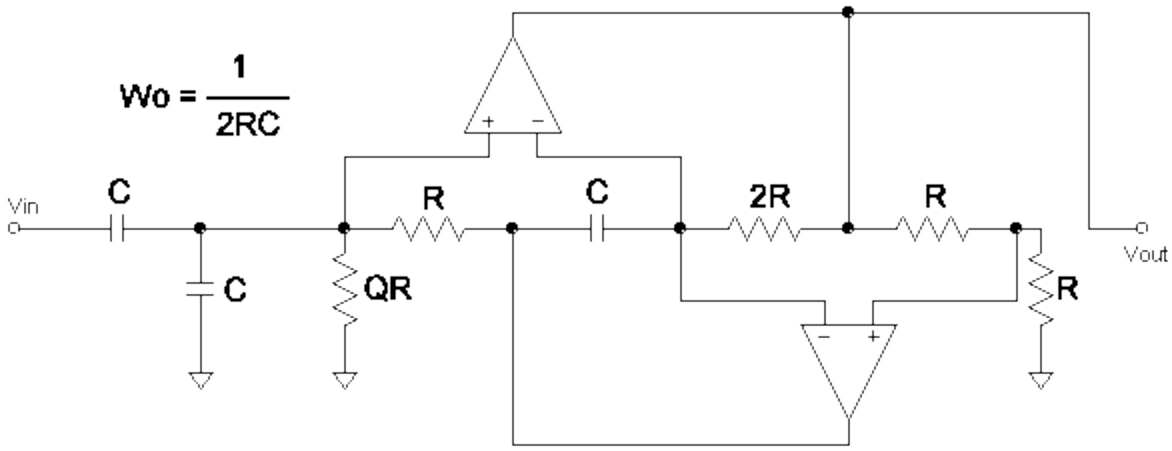
Each stage is described below with tuning characteristics:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Low Pass Unity Gain:

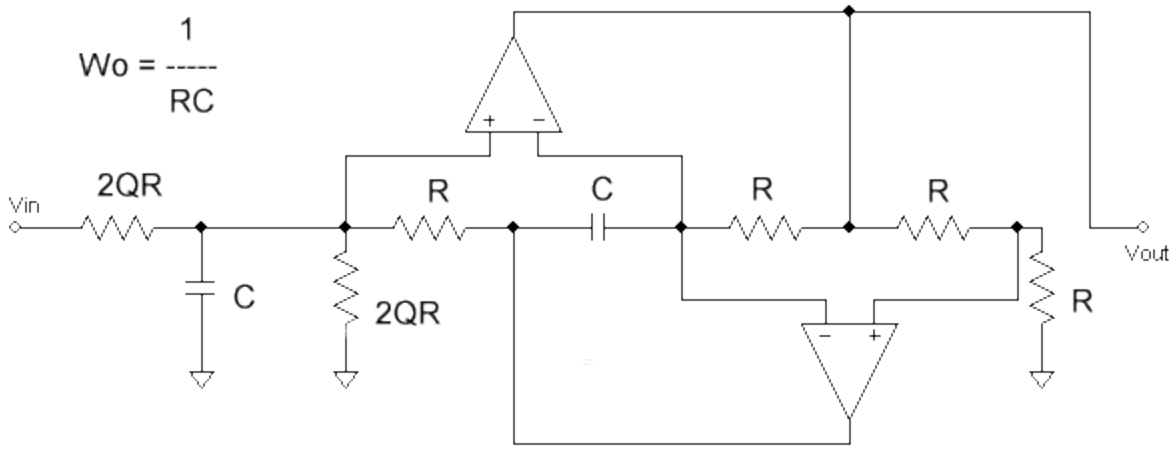


High Pass Unity Gain:

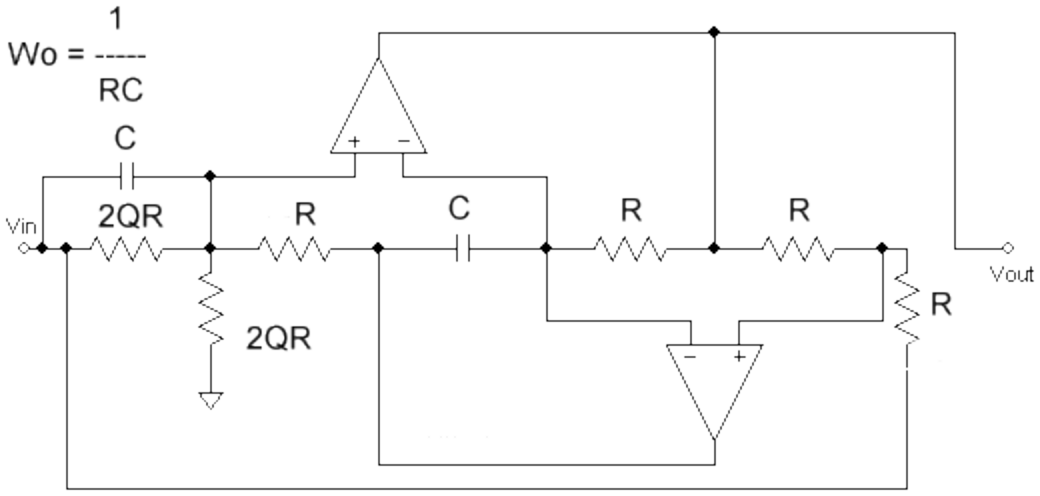


Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Band Pass Unity Gain:



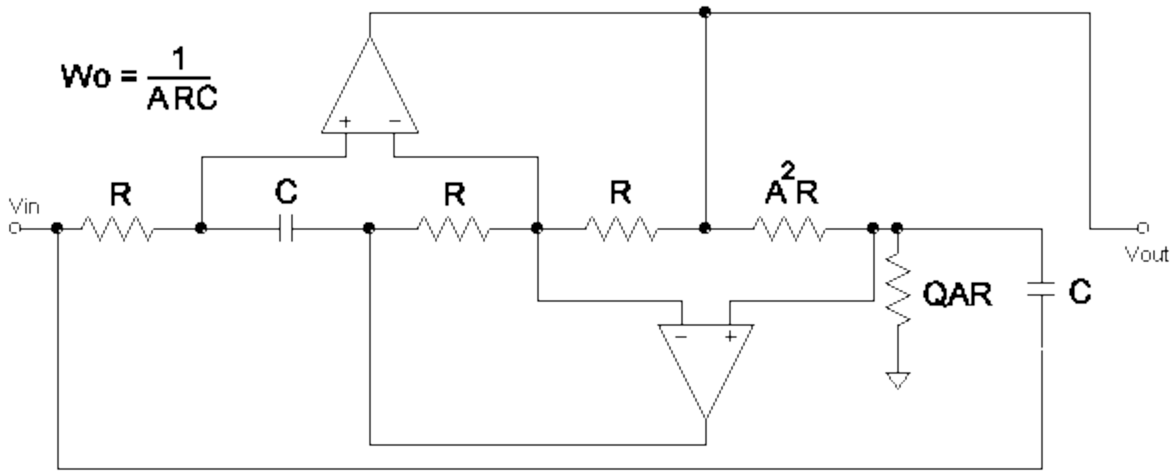
Even Gain Notch:



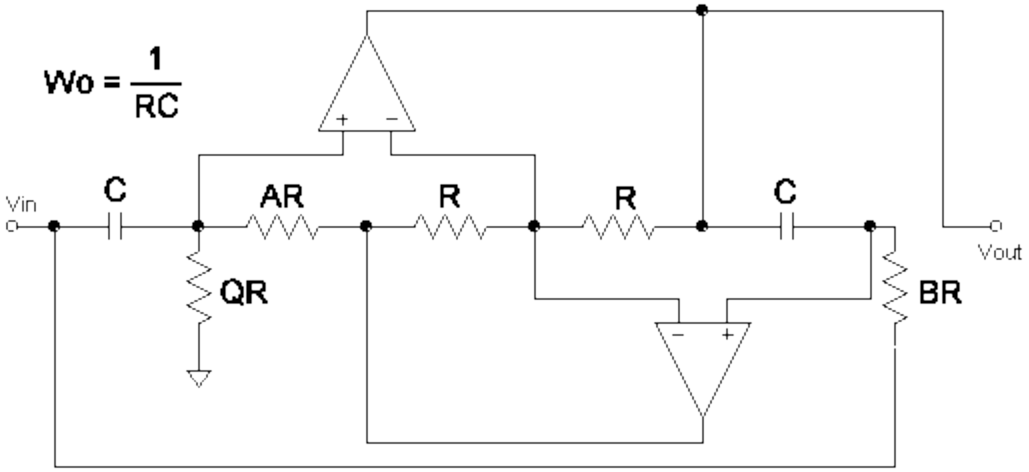
Low Pass Notch:

Adjust Wo by changing R, not A.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

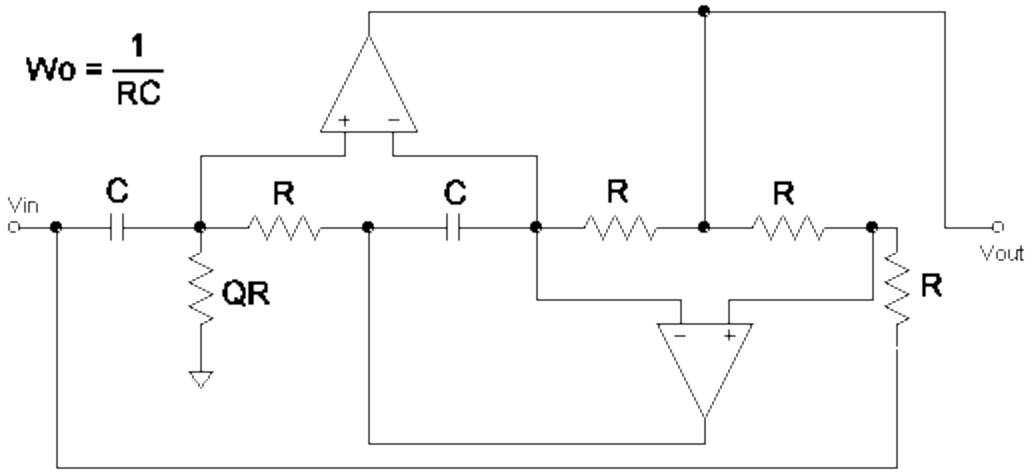


High Pass Notch:



Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

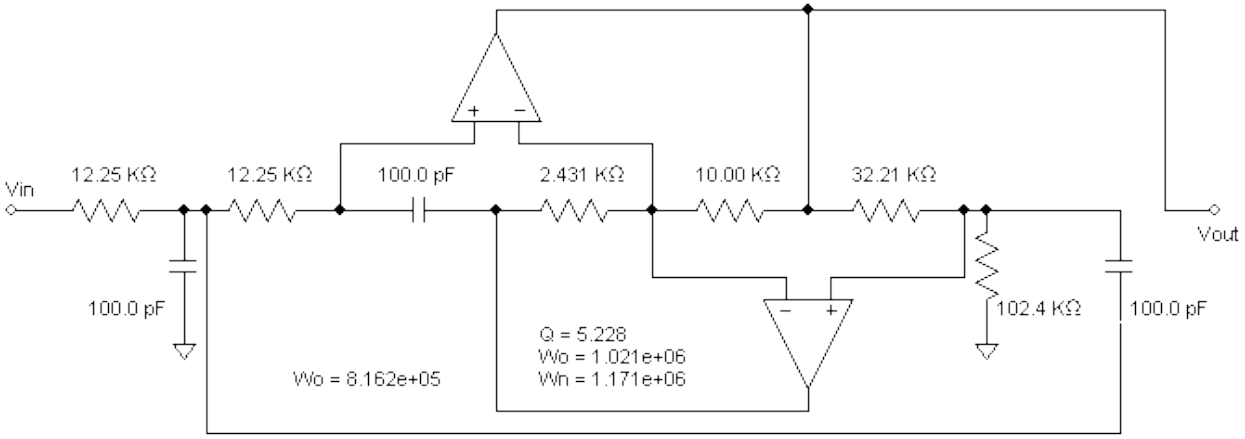
All Pass:



Third Order GICs

Like the other biquads, low pass and high pass all pole and notch GIC biquads may have an additional RC pair added at the front end to produce a third order GIC stage. This addition destroys the tuning capability of the stage, but is useful for conserving opamps. The identical capacitor value characteristic of GIC stages is, preserved, however. Single poles are automatically incorporated into the first second order stage whenever "Absorb Odd Orders" is selected in the Active control panel.

The addition below of the initial 12.25K resistor and the shunt 100pF capacitor provide for the addition of a single low pass pole:



Third Order Low Pass Notch GIC Stage

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

GIC Ladder Filters

GIC Ladder filters are active simulations of passive LC filters constructed with the classic "Ladder" topology. Any low pass, and high pass filter is realizable, and narrow band all pole band pass filters are realizable. GIC Ladder filters use impedance simulations of inductors (Z = LS), D Devices (Z = 1/(DS^2)), and M devices (Z = MS^2). "D" and "M" terminology arise from the next letter in the alphabet following "C" for capacitor and "L" for inductor respectively.

Options

Low pass and high pass filters have on option to synthesize the filter without upper series capacitors ("**Top Caps"**) When this option is selected, FilterSolutions will do whatever impedance adjustment of end resistance adjustment necessary to eliminate these capacitors. This option is useful to minimize or eliminate the number of capacitors that cannot be adjusted without distorting the filter frequency response. Capacitors in the GIC generally may be set to anything desired by adjusting the resistors.

Band pass GIC Ladder filters only approximate the desired band pass response. Therefore, the option exists to approximate with a low pass heavy response or a high pass heavy response. Low pass approximations generally require fewer capacitors than high pass approximations.

Limitations

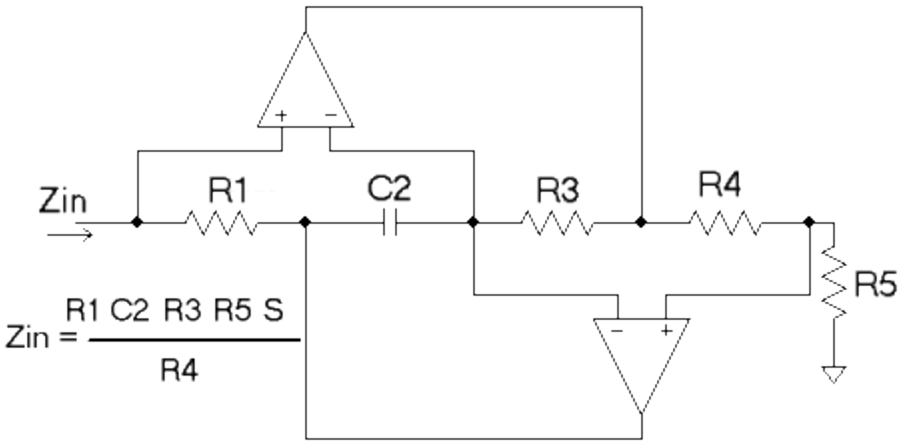
Band pass filters only realize narrow band responses. If the band width is too wide, the user is prompted to narrow it, or to select a different active implementation.

Impedance scaling

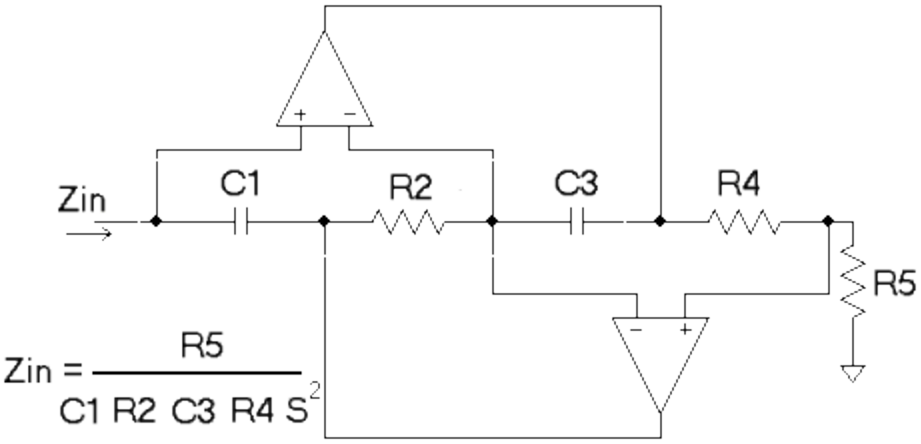
High pass all pole filters only need to simulate inductors to create an RC active filter. Other filters may need to be scaled by S or 1/S to remove all inductors from the filter. These filters usually end up with D and M devices, and use Frequency Dependant Negative Resistors (FDNRs) to simulate these devices. FDNR s owe their name to their real negative impedance values that are a function of frequency. (R[D,w] = -1/(Dw^2) and R[M.,w] = -Mw^2.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Inductor Simulation:

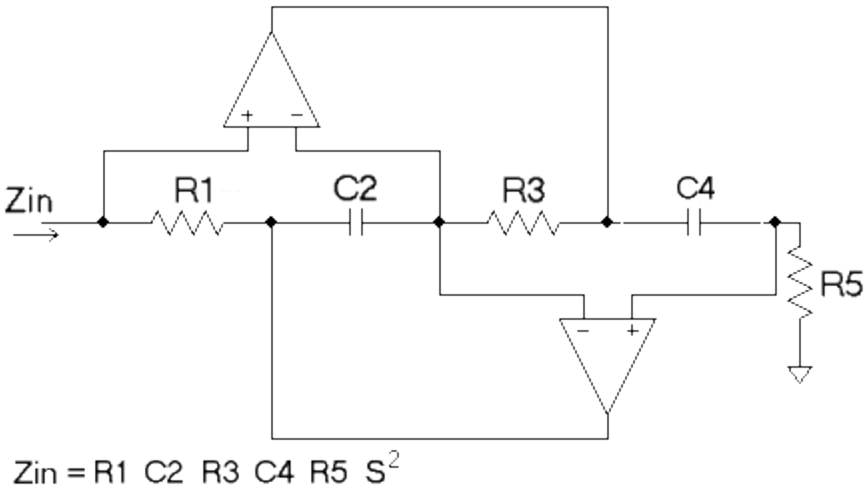


D Device Simulation:



Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

M Device Simulation:



Bridged GIC Ladder Filters

Bridged GIC Ladder filters may be used to synthesize delay equalized Chebyshev I filters without the need for all pass stages or coupled inductors. More information may be found under "Delay Equalizers" -> "Off Axis Real and Quadruplet Zeros" section of the Help, or the short Help video, "Bridged Element Delay Equalization".

Bridged GIC Ladder filters consist of cascaded shunt GIC stages with a series of one or more bridge resistors connecting the GIC stages on opposite sides of the filter, and are used to formulate off axis zeros, as shown below in Figure 1.

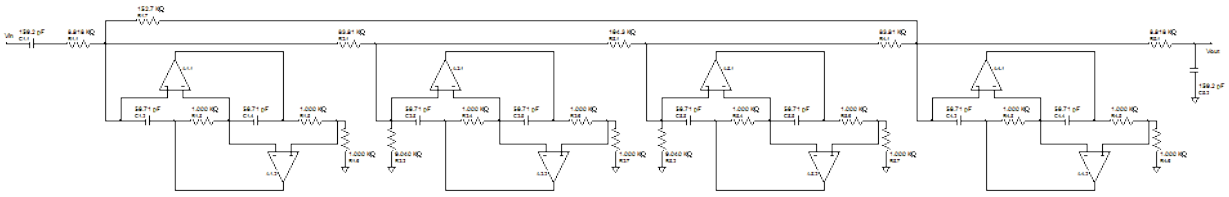


Figure 1. Bridged GIC Ladder Topology Example

Important: To create band pass GIC Ladder filters, **band pass, Chebyshev Type I MUST be selected**.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Bridged tubular filters may be created from Chebyshev I filters with stop band zeros and off axis real and quadruplet zeros.. **The passive circuit schematics will have a check box in the tool bar for "Tubular"**. Check this box, and a bridged tubular resonator band pass filter will be displayed if the filter design requirements meet all of the all the bridged tubular design criteria If the selected pass band is too wide or other criteria is not met, you will receive an error message informing you that you need to reduce the width of the pass band or instructed to meet other criteria.

Required Criteria:

In addition to requiring Chebyshev I or Elliptic selection, bridged tubular resonator filters must also meet the following:

i. Band width less than center frequency. To obtain accurate results, the band width should generally be much less than the center frequency.

Active Stages

This chapter contains the following topics:

- Stage Parameters
- All Pass Stages
- Differential Stages

Stage Parameters

Each active filter has one or more of the following parameters displayed: Wo, Fo, Q, Wn, Fn. Below are definitions and descriptions of these parameters.

Wo and Fo

For second order stages, these are defined as the geometric mean of the two poles' distance from the origin. For first order stages, these are defined as the distance of the single pole from the origin. For third order stages, the two complex poles are used to calculate and display Wo and Fo. If all poles are real, the two smaller ones are used. Wo and Fo provide an approximate measure of the corner frequency or center frequency location. Wo is in units of R/S. Fo is in units of Hz.

Q

Q is only defined for second and third order stages. For second order stages, Q is defined as Wo divided by the sum of the real part of the pole locations. For third order stages, the two complex

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

poles are used to calculate and display Q. If all poles are real, the two smaller poles are used. Q provides a measure of sharpness to the stage frequency response. High Q implies a very narrow sharp frequency response. Low Q provides a wide shallow frequency response. The minimum value for complex poles is 0.5. Q is dimensionless.

Stages are sequenced such that low pass and low Q stages are physically position prior to high Q and hi pass stages. Low pass takes precedence over low Q. Stages with equal Q are sequence by increasing Wo. This minimizes the possibility of opamp slew rate limiting, and internal signal ringing. Stages may be physically sequenced by the user by clicking on any opamp of any stage and selecting the stage number of the desired stage to switch places with.

Wn and Fn

Wn and Fn are used only in stages with zeros. For second order stages, these are defined as the geometric mean of the two zeros distance from the origin. When used in notch filters stages, Wn and Fn provide an exact measure of the null frequency location. Wn is in units of R/S. Fn is in units of Hz.

Wn's are carefully paired with Wo's and Q's to minimize gain differences between the low frequency and high frequencies of all notch stages. This minimizes internal noise amplification of the filter. Stage Wn's may be physically sequenced by the user by clicking on any opamp of any stage and selecting the stage number of the desired stage Wn to switch places with.

Gains

Active filter stage gains are set such that the maximum output at each stage is equal, whenever possible. This minimizes the possibility of opamp saturation. Some active stage gains are fixed and cannot be adjusted. In this case, the stages with gains that are adjustable are all set to the same maximum output level. Stages that do not have fixed gains are individually adjustable by the user by clicking on any opamp of the stage, and manually entering the desired stage gain.

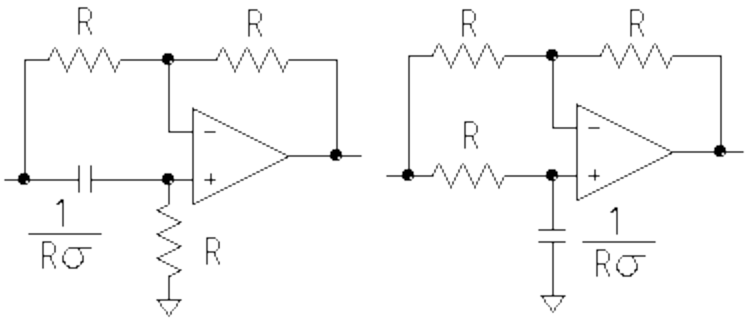
Active All Pass Stages

All pass stages may be added to the active filter with the continuous pole/zero plot function. Select where you want a pole to go, depress and hold the left mouse key, then click the right key. The all pass stage will then appear in the filter the next time you click "Circuits". **If "RTU" is checked** on top of the pole/zero plot, the all pass stage will be added automatically. You may also watch the filter design update in real time by sliding around the poles and zeros with the cursor when **RTU** is checked.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

First Order Stages

First order all pass stages use the following topologies:



1st Order All Pass Stages

The required sign of the stage determines which one to use.

Second Order Stages

Second order all pass stages may be created from either Thomas Biquads, or from Positive or Negative SAB's. The positive and Negative SAB's require an injection resistor. See the Help sections on these biquads for more information.

Differential Stages

FilterSolutions supports differential stages for MFB all pole topologies. To select a differential topology, check the **Balance** check box in the menu bar above the schematic.

Nondifferential stages are converted to differential stages by copying all series elements to the bottom half of the filter, removing the grounds on the shunt elements and cutting the impedances of shunt elements in half.

In the example below, a fifth order two stage nondifferential Chebyshev filter is converted to a a full differential filter.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

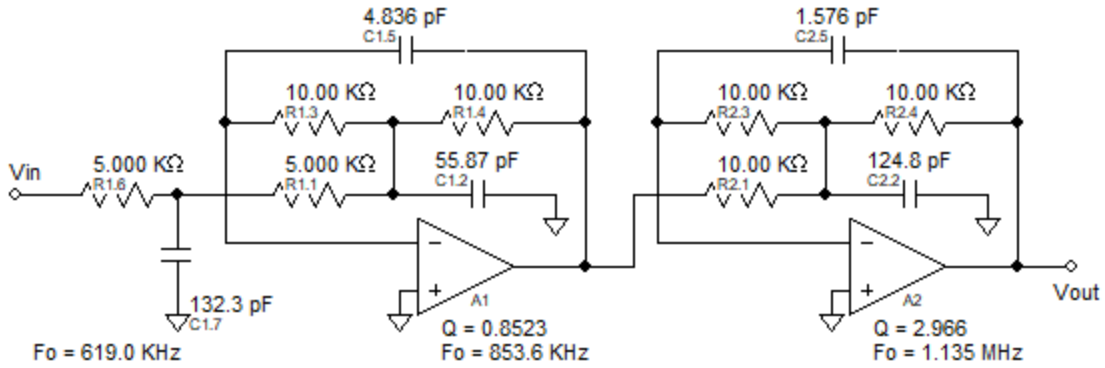


Figure 1: Fifth Order Chebyshev With Stage

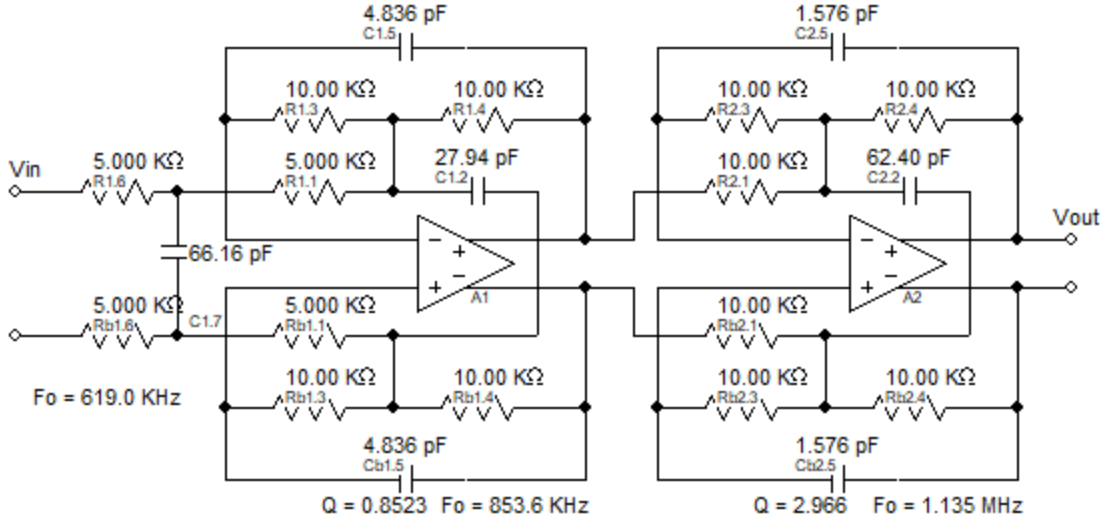


Figure 2: Fifth Order Chebyshev With Differential Stage

Differential Op Amps

Differential op amps are implemented as two standard op amps with the positive nodes tied together. Positive feedback compensation elements, if any, are included internally in the differential op amp model. The differential op amp model can be seen in the net list. Figure 3 depicts the differential op amp models used by Nuhertz.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

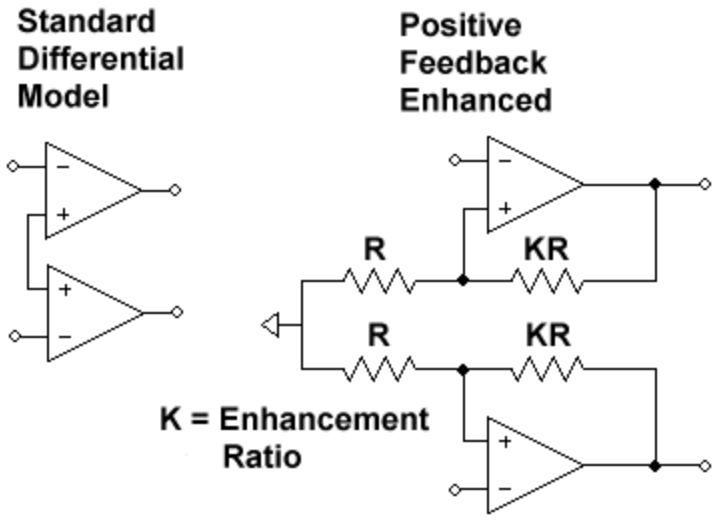


Figure 3: Op Amp Differential Models Used in Nuhertz Simulations and Net Lists

Band Pass Architectures

This chapter contains the following topic:

- Active Band Pass Architectures

Active Band Pass Architectures

FilterSolutions offers two ways to architect the active band pass filter: integrated band pass architecture and high pass/low pass architecture. The differences produce different element values and stage gains for the band pass stages. In the integrated architecture, all stages in the filter are 2nd order band pass stages. The high pass/low pass architecture uses high pass and low pass stages. Odd order high pass/low pass architectures also use one band pass stage. One significant advantage of the high pass/ low pass architecture is if poles exist on the real axis, they may be absorbed into other biquads resulting in a savings of one opamp.

The two different architectures are produced by rearranging the stage transfer function numerators. The integrated architecture uses a single zero in each stage of the filter. The high pass/low pass architecture places two zeros in half the stages, and no zeros in half the stages. The total number of zeros is the same for both architectures.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Which Architecture to Use:

In general, the **integrated band pass architecture works better for narrow band filters**, and the **high/low pass architecture works better for wide band filters**. This is due to potentially huge, undesirable gains that may saturate opamps if the wrong architecture is used.

Illustration:

To build a 3rd order filter with corner frequencies of 1000 and 5000 Hz, the following cascade transfer function may be used:

Both transfer functions are identical in performance. The only difference is the distribution of zeros among the stages.

$$\frac{8.000}{(S^2 + 1.681*S + 5.275)} \quad \frac{S}{(S^2 + .3187*S + .1896)} \quad \frac{S}{(S^2 + 2.000*S + 1)}$$

$W_0 = 2.297$	$W_0 = .4354$	$W_0 = 1$
$Q = 1.366$	$Q = 1.366$	$Q = .5000$

Integrated Band Pass Architecture

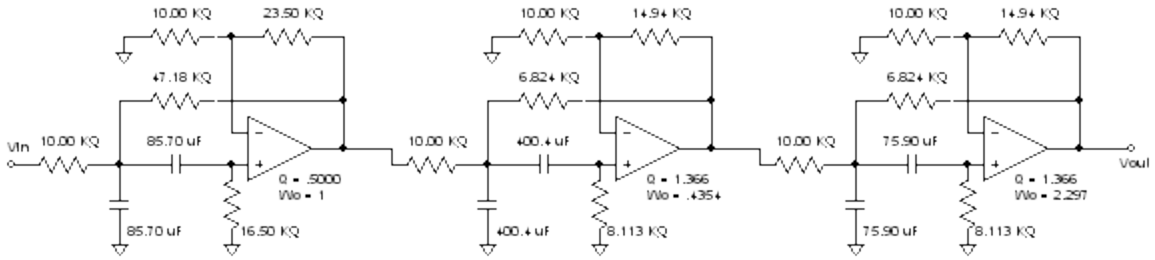
$$\frac{8.000}{(S^2 + 1.681*S + 5.275)} \quad \frac{S^2}{(S^2 + .3187*S + .1896)} \quad \frac{S}{(S^2 + 2.000*S + 1)}$$

$W_0 = 2.297$	$W_0 = .4354$	$W_0 = 1$
$Q = 1.366$	$Q = 1.366$	$Q = .5000$

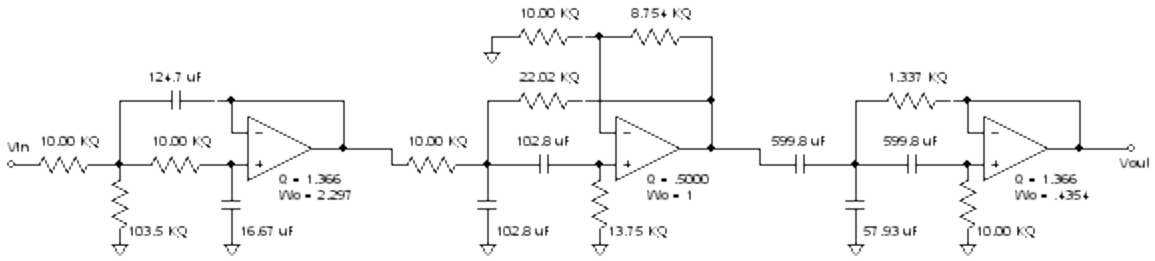
High Pass/Low Pass Band Pass Architecture

The transfer functions shown above each produce unique active circuit architectures shown below

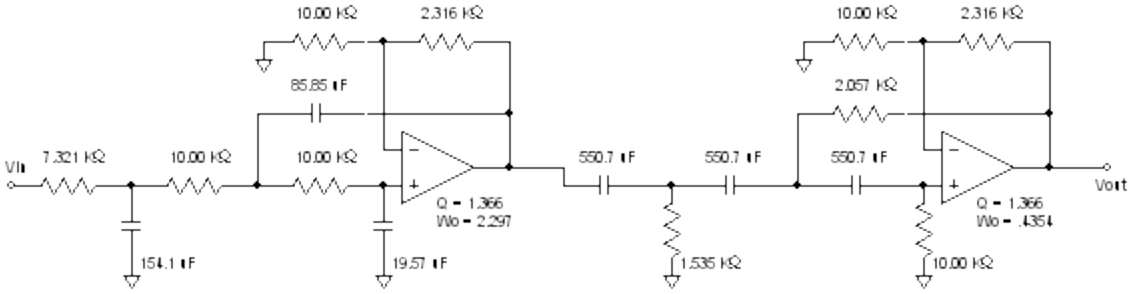
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Integrated Band Pass Architecture



High Pass/Low Pass Band Pass Architecture



High Pass/Low Pass Architecture With Real Poles Absorbed

Opamp and Element Parasitic Effects

This chapter contains the following topics:

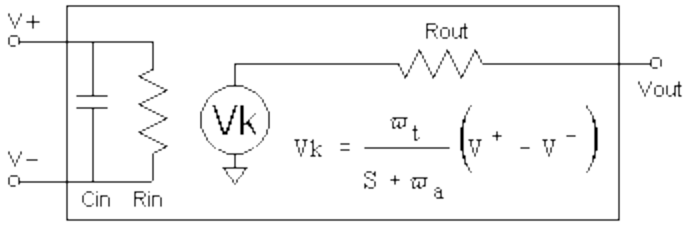
- Real Opamps
- Resistor Parasitic Shunt Node Capacitance
- Capacitor Losses

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Real Opamps

FilterSolutions provides real opamp models for more fine tuned analysis. It is possible then to for you to more closely estimate the performance of the filter when it will be constructed.

FilterSolutions uses the following opamp model for real opamp simulations:



Real Opamp Model

In addition, physical opamp Spice models may be inserted into netlists. See "Exporting to Other Software".

User adjustable variables include:

Acronym	Description	Default Value
Cin	Input Capacitance	Zero
Rin	Input Resistance	Infinite
wt	Gain BandWidth Product	Infinite
wa	Bandwidth	Infinite
Rout	Output Resistance	Zero

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

It is advisable to test run the filter with the real opamp setting and all opamp parameters set to their ideal default states first. The ideal opamp filter trace will visible in dark blue behind the real opamp trace, and you may easily determine the accuracy of the real opamp simulation. Large filters with high Q and high gain stages are particularly difficult to simulate and may take noticeable time to execute.

The default values of wt and wa are infinite. Under this condition, internal simulations simply use a large fixed gain for the opamp. When setting wt and wa to finite values, both must be set to a finite value, not just one.

Resistor Parasitic Capacitance

Active filters in integrated circuit design that use real physical resistors must contend with parasitic shunt capacitance associated with each resistor. Since the shunt capacitance tends to be distributed across the resistor surface, an acceptable model is a unit length RC transmission line. FilterSolutions permits the selection of this shunt capacitance for analysis purposes by selecting, "Real Parameters", deselecting "Ideal Resistors", and entering the desired default shunt capacitance in the text entry to the right of the checkbox.

Updates to Parasitic Shunt Capacitance

Since the parasitic shunt capacitance may not be the same for each resistor, parasitic shunt node capacitance may be updated by left clicking on a resistor and entering the desired shunt node capacitance for the selected resistor in the pop-up Change Control Panel.

Display of Parasitic Shunt Capacitance

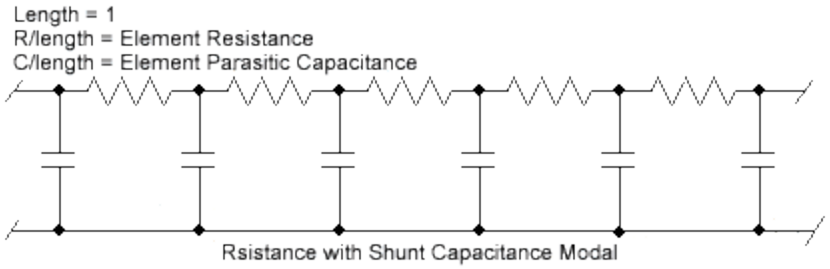
The active circuits display shows the parasitic shunt capacitance value near each resistor that has a nonzero parasitic shunt capacitance value assigned. The active filter netlist also includes the parasitic shunt node capacitance values for external analysis, modeled as a unit length RC transmission line.

Parasitic Shunt Capacitance model

Parasitic shunt capacitance models unit length transmission lines. The length of the line is one, and the R/length and C/length are set to the actual resistance and parasitic values, so that the net R and C on the line exactly match the actual resistance and parasitic capacitance. L/length and G/length are set to zero.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Since L is zero, the speed of the line, 1/sqrt(LC), is infinite, so the RC transmission line may be set to any length desired, so long as the net R and C equate to the actual element resistance and parasitic capacitance. A unitless line length of one is convenient.



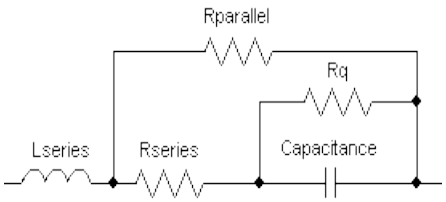
Note: Versions 12.1.0 and prior modeled parasitic shunt capacitance as a lumped capacitor at each node of the resistor. When filter files from Versions 12.1.0 and before are read in, the shunt node capacitance is doubled to maintain equivalent total parasitic capacitance.

Active Capacitor Losses

Finite Q, series resistance (ESR), and parallel resistance parasitics are modeled. Select, "Real Parameters" in the main Active control panel to enter default values. Use the left mouse key to edit values. Series capacitor resistance is sometimes referred to as equivalent series resistance (ESR).

Standard parts lists may also include parasitic values by adding the terms, "Q= ", "Rs= ", and "Rp= " to a capacitor line.

The following model is used for analysis purposes when series, parallel, and/or finite Q effects are entered.



Capacitor model with Parasitic Effects

Rq is computed as a function of frequency: Rq = Q/(C*W)

where C is capacitance in Farads, and W is frequency in R/S

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sensitivity

This chapter contains the following topics:

- Monte Carlo Analysis
- Element Sensitivity Tables
- Element Sensitivity Plots

Monte Carlo Analysis

Active filters may be statistically analyzed through Monte Carlo analysis. The percent tolerance of each resistor and capacitor may be set individually, or a default value for all resistors and all capacitors may be used.

To execute a Monte Carlo on a passive LC circuit, open the desired graph to be analysis, such as frequency response, time response, reflection response, etc., use the left mouse key to select any resistor or capacitor to open the change control panel, select "Random" in the change control panel, and fill out the remainder or the information at the bottom of the Change Control panel. If "Maintain Old Traces" is selected, the graphical display will accumulate all the traces for easy visual interpretation. **Only schematic analysis graphs are updated with Monte Carlo analysis traces.** Ideal analysis graphs are not updated.

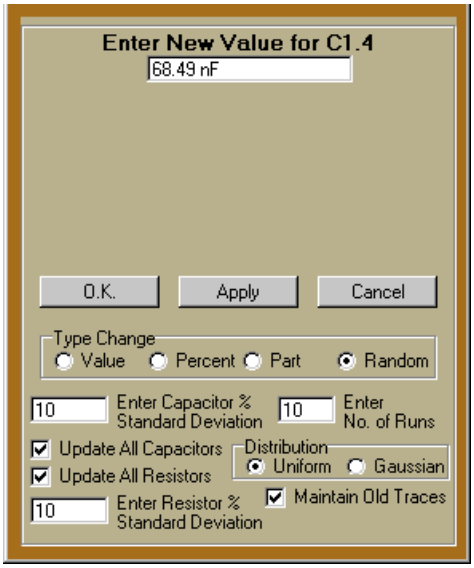
Selecting element tolerance

The change control panel permits a global selection of tolerance values for all resistors and all capacitors. **The global value is only applied to resistors and capacitors that do not have a tolerance assigned.** To assign a specific tolerance value to a specific resistor or capacitor, use the left mouse button to select the resistor or capacitor, check the **Set Tolerance (%)** and enter the desired tolerance in the text entry to the right of the checkbox. The percent tolerance value will appear in the schematic near the element. If a specific percent tolerance is not assigned to an element, the global value in the change control panel is used for Monte Carlo analysis.

Example

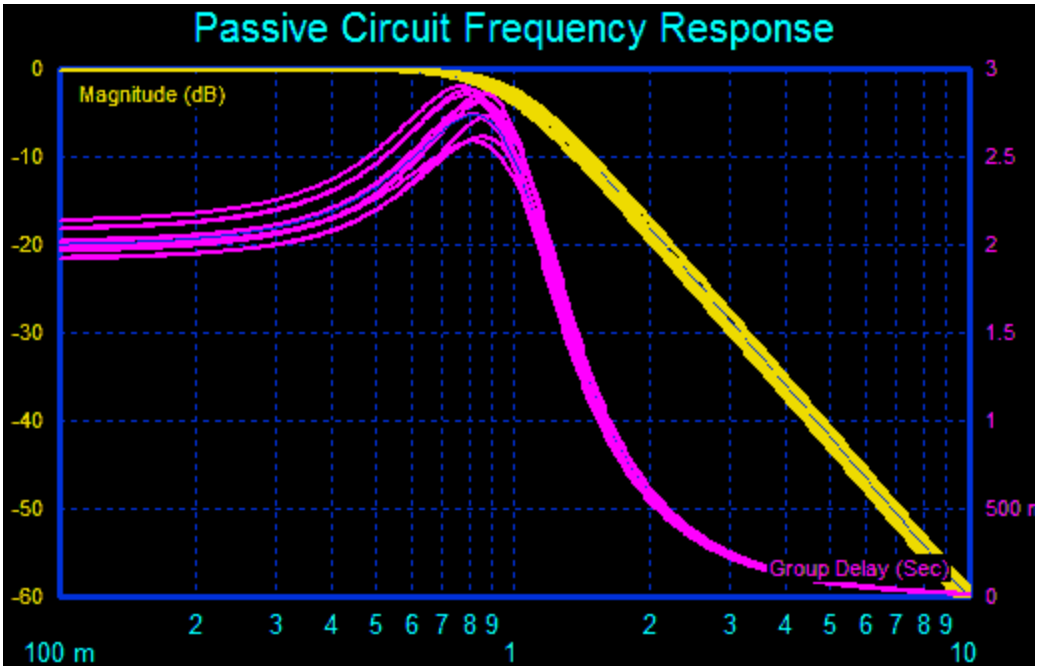
A third order Butterworth filter is analysis with a Monte Carlo analysis. All resistors and capacitors are set to 10% tolerance. 10 runs are selected to be generated, and old traces are selected to be remembered, as seen in the example below.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Monte Carlo Setup

When the Change Control Panel above is filled out, select **Apply** or **OK** to generate the Monte Carlo analysis graphs on all graphs currently open on the selected filter schematic. The frequency response should look like the graph below.



Monte Carlo Graph

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Element Sensitivity Tables

The effect of element value errors on magnitude, phase, and group delay filter performance may be analyzed for all elements simultaneously by selecting the **Sens.** button on top of the passive filter schematic. The sensitivity table showing numerical sensitivity values at a specific frequency appears on the schematic. A quick visual scan of the table may indicate which elements need higher precision, and which ones may be low cost loose precision parts.

Sensitivity Definitions

To obtain meaningful sensitivity numbers with reasonable values for all filters, FilterSolutions has adapted the following definitions to measure sensitivity of parts with respect to frequency response aspects:

Magnitude in dB	Arithmetic change in dB per % change in element value
Magnitude in Arithmetic Volts	% change in magnitude per % change in element value
Phase Angle in Degrees	Arithmetic change in degrees per % change in element value
Phase Angle in Radians	Arithmetic change in radians per % change in element value
Group Delay	% change in group delay per % change in element value

Frequency Selection

Low pass and high pass filters default to the cut off frequency, otherwise known as pass band frequency, to perform the sensitivity analysis. Band pass and band stop filters default to the center frequency. Band pass and band stop filters permit the selection of the lower and upper cutoff frequencies, and all filter permit the selection of a user defined frequency.

Specific element sensitivity analysis

Click any element in the sensitivity table to see more information of the effect of filter performance on that specific element. More information on this subject is available in the Help under Element Sensitivity Plots.

Element Sensitivity Plots

Left click any element in the element sensitivity tables to generate an element sensitivity frequency plot. (Select the **Sens.** button in the passive filter control bar to generate element

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

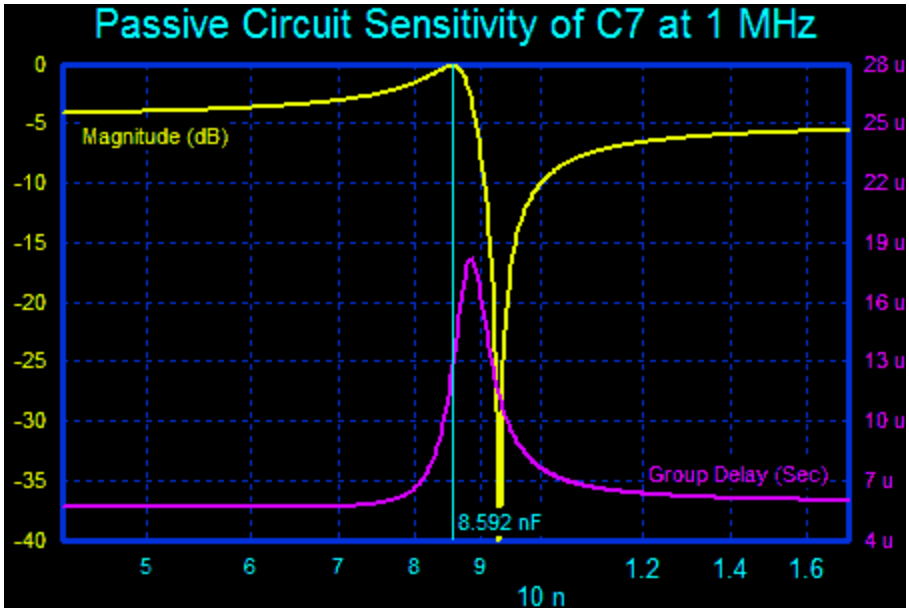
sensitivity tables.)

Element sensitivity tables only provide a sensitivity information about the element value. Since sensitivity is not linear, this information has its limits in usefulness. Element sensitivity plots provide magnitude, phase, and group delay frequency information about a range of element values. FilterSolutions provides provide frequency information as a function of element value in the range 1/2 to 2 times the actual value for semi-log plots, and 1/2 to 1.5 times the actual value for arithmetic plots. The "Log" switch setting in the Main control panel determines which type, semi-log or arithmetic, of plot to display.

Elements defined with S-Parameter files will continue to use the nominal value for sensitivity plots.

Example:

The example below shows a case where the sensitivity table would show what might be considered a good low magnitude sensitivity response at the cutoff frequency of 1MHz, whereas the component sensitivity plot shows a potentially significant degradation of performance as the component value increases.



Element Sensitivity Graph

Modify Active Filters

This chapter contains the following topics:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Capacitor and Resistor Selections
- Modify Stages
- S-Parameter and Spice Model Imports
- S-Parameter and Spice Model Exports
- Stage Gains
- User Entered Capacitors
- Modify Elements
- Standard Parts

Capacitor and Resistor Selections

The active filter schematics have controls designed to select parts for all the active filter stages. Capacitors and resistors are selected from the user parts list or industrial standard parts (See topic for standard parts) . Resistors may be calculated around capacitor values in the parts list, or capacitors and resistors may be substituted for analysis purposes.

Cap Compute

Select the user or industrial part list for the capacitors, and calculate the resistors. Some capacitors may also be calculated, and sometimes no solution exists for one or more stages, and an error message is displayed. If an error message is displayed, it will be necessary to click on the opamps of the stage(s) that failed to compute and manually select capacitors that succeed in computing resistors.

Caps Select, Resis Select

Select the desired user parts list or industrial parts value selection, and all capacitors and/or resistors will be updated to the nearest value. No new synthesis calculations are performed, but all subsequent analysis will use the new parts. Parts are substituted in the same manner as edits with the mouse keys. New parts are color coded and may be removed with the use of the "Restore" button.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

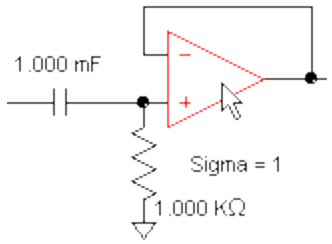
Recommended Design Sequence

It is recommended to update resistors with "Cap Compute" first, then update remaining capacitors and all resistors with "Caps Select" and "Resis Select". This insures a design that maximizes use of available capacitors and minimizes design error resulting from modified element values. All parts selection are remembered for future filter designs, and the sequence of selection is the previously stated recommended sequence.

Modify Active Filter Stages

The filter that FilterSolutions provides may not necessarily be in a form that best fits the designer's needs. Therefore, options are provided to change some of the features of each stage. For example, the gain of one of the stages may be too low or too high. You may change the gains of that stage.

To modify a filter stage, pass the cursor of the opamp of that stage. It will turn red as shown:

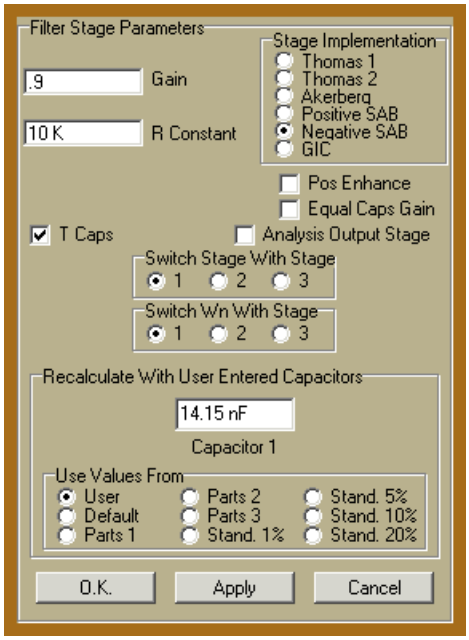


When you left click the mouse key, the Stage Control Panel will appear with several change options to choose from. The change options are a function of the parameters in the stage you selected to change. The default values in the panel will match the selected stage parameter values. You may modify any or all of the displayed parameters, select **OK**, and the associated filter stage will assume the new parameter value. **Best results are obtained when changing only one parameter at a time.**

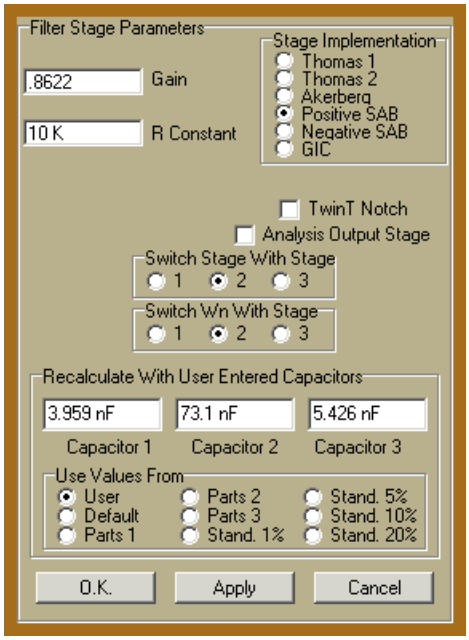
Negative SAB's with an injector resistor (needed for high quality notch and all pass functions) have a fixed gain. The gain will be displayed in the Gain control of the panel, but you will not be able to modify it.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modify Stage Control Panel



Stage Control Panel



Stage Control Panel

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Stage Control Panel Entry Definitions

Alternate Notch	Check to use the alternate notch configuration for negative SAB notches with unequal low and high gains.
Enhancement Ratio	Enter the new positive feedback enhancement ratio for negative SAB's when "Pos Enhance" is checked.
Equal Caps Gain	Check to set the stage gain to a value that will produce equal capacitors for Positive SAB 2nd order low pass stages.
Equal Resis Gain	Check to set the stage gain to a value that will produce equal capacitors for Positive SAB 2nd order low pass stages.
Gain	Recalculate the stage with a new gain.
OK	Closes the control panel and updates the stage.
Pos Enhance	Check to apply positive feedback enhancement to 2nd and 3rd order negative SAB's
R Constant	Rescale the stage resistors and capacitors with a new resistive constant.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Recalculate With User Entered Capacitors	Recalculate the resistor values of the stage using user-entered capacitor values. (Cap Computer control on the schematic also does this for all stages.)
Switch Stage With Stage	Select another stage to change places with
Switch Wn With Stage	Select a stage to swap Wn with.
Stage Implementation	Recalculate the stage with a different implementation.
T Caps	Use capacitors rather than resistors to create the "T" elements in negative SAB all pass and even gain notch stages.
TwinT Even	Selects a TwinT notch for positive SAB second order notch filters. An injection resister is used otherwise.
Use Parts From	Select the source of the new capacitor values used to recalculate the stage.
Analysis Output Stage	Moves the filter frequency and time analysis to the output of the selected stage.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Alternate Opamp	Displayed for GIC biquad stages only. Check to view the output of the alternate opamp, that is, the opamp that is not used to out the stage voltage.

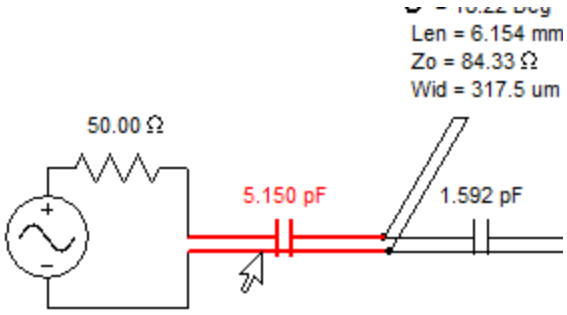
S-Parameter and Spice Model Imports

FilterSolutions permits you to import S-Parameter from Touchstone format files and Spice model files in standard Spice format, both by individual file selection and by automatic nearest match. When you do this, the original element impedance is replaced with the S Parameter or Spice model defined impedance for each frequency. Interpolation is employed between the frequencies for S Parameter files. Any passive linear element S-Parameter or Spice model file is acceptable. Active element S-Parameters or Spice models, such as a two pin amplifier, will not be accepted for importation. Elements defined with S-Parameter or Spice model files will continue to use the nominal value for sensitivity analysis and Monte Carlo analysis.

S-Parameter Files and Spice parameters for Coilcraft inductor downloads may be found at: www.coilcraft.com/models.cfm

Once an S Parameter or Spice model is imported into an element, all analysis performed on the associated filter will use the imported S-Parameters or Spice models instead of the element impedance. S-Parameter or Spice model files that are electrically asymmetrical will post pin numbers on the schematic.

To import a specific S Parameter or Spice models files, left click on the desired element, select, **File** in the popup Change Control Panel, select the desired file, and **OK**. The name of the S Parameter file then appears above the element, as shown below in Figure 1.



Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

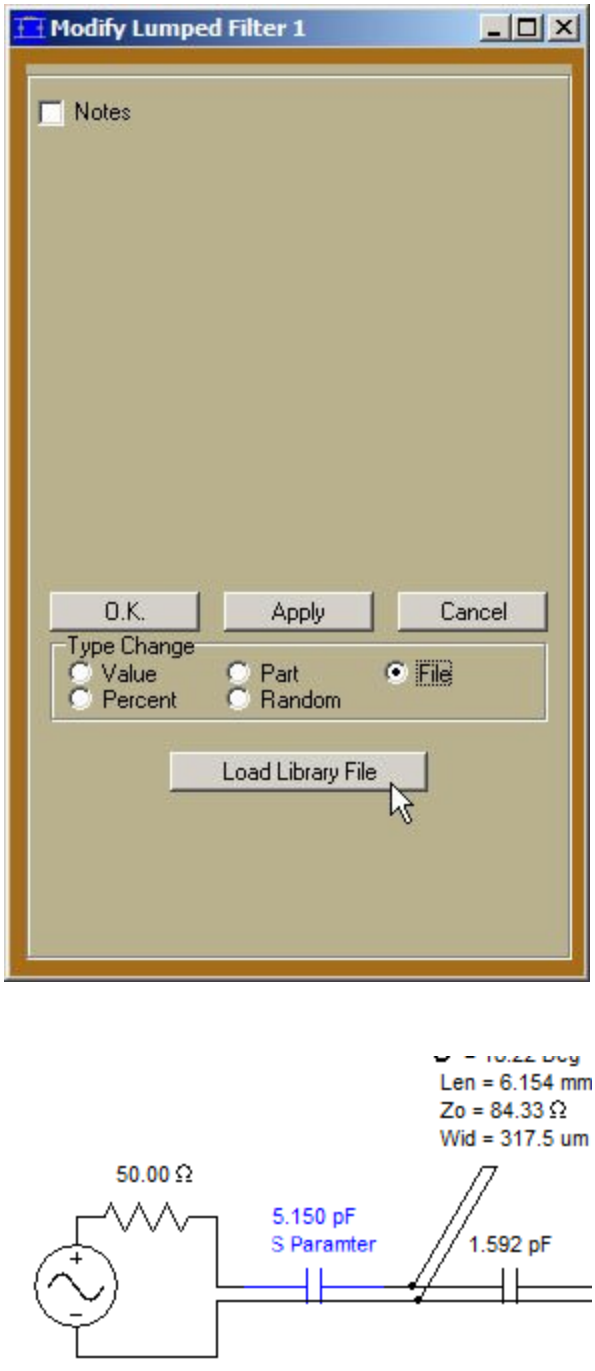


Figure 1: Selecting a Desired S-Parameter or Spice Model File

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Update to Nearest Matching S-Parameter or Spice Model File

To update to the nearest matching S-Parameter or Spice model file, select **Part** and **File**, as shown below, then select, **Browse** to select the desired directory containing the S-Parameter or Spice model files, as shown below in Figure 2. Select, **Menu** to bring up the manufacturer and family pop-up menu. Select any file in the Browse directory to select and define the desired directory of S-Parameter files. Once selected, FilterSolutions will remember the directory until another directory is selected by the user. Check, **All Inductors** to update all the inductors in the filter. Care should be taken to insure that all S-Parameter and Spice model files are from components valid for the frequency and power range of the designed application.

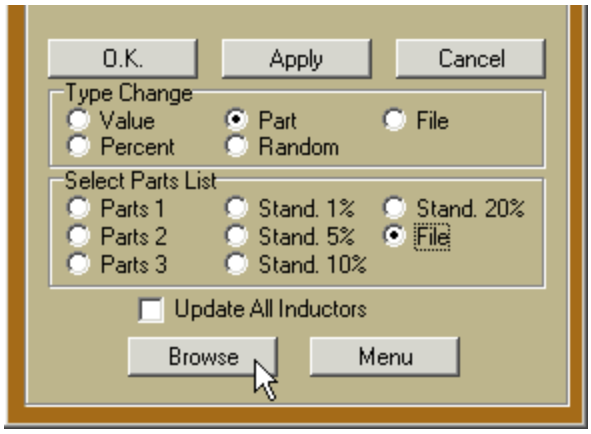


Figure 2: Update to the Nearest Matching S-Parameter or Spice Model File

S-Parameter and Spice Model Exports

S-Parameter or Spice model files that have previously been imported can be exported. Automatic S-Parameter and Spice model selection may make it difficult to determine which S-Parameter or Spice model file is used in any particular element. Left clicking an element that is loaded with a S-Parameter or Spice model file pulls up the element Change Control Panel which may be used to export the specific S-Parameter or Spice model file that is loaded into the selected element. Select **File** to bring up the Touchstone features, then save the element S-Parameter or copy it into the Windows clipboard for pasting elsewhere, as shown in Figure 1.

Additionally, S-Parameter and Spice model files may be saved or viewed by selecting the appropriate button.

In all cases, the original S-Parameter or Spice model file will be saved or copied, including comments, header, and numerical resolution.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

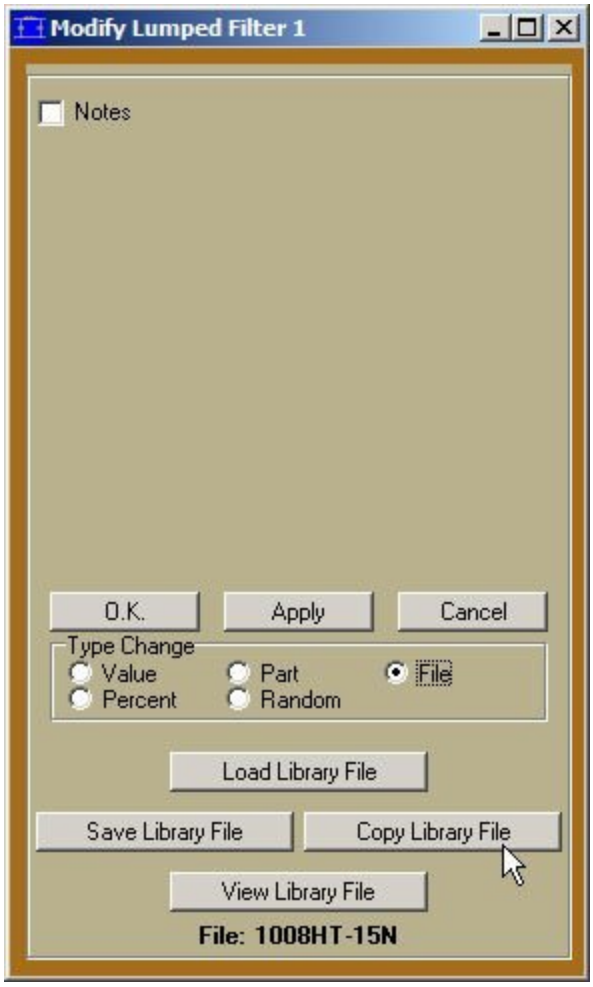


Figure 1. Saving and Copying an Element S-Parameter or Spice Model file

Active Stage Gains

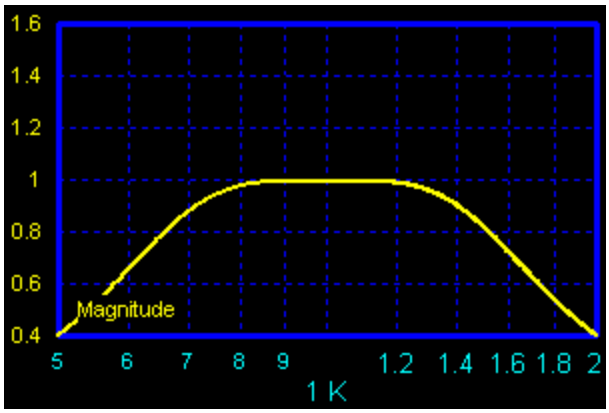
FilterSolutions allows one to view and modify the gain of each stage of an active filter, as shown in the **Modify Stages** section of the Help contents. It is important to remember that the product of the stage gains is not necessarily equal to the overall filter gain as seen in the Active Control Panel. The reasons for this are summarized below.

Band Pass Filters

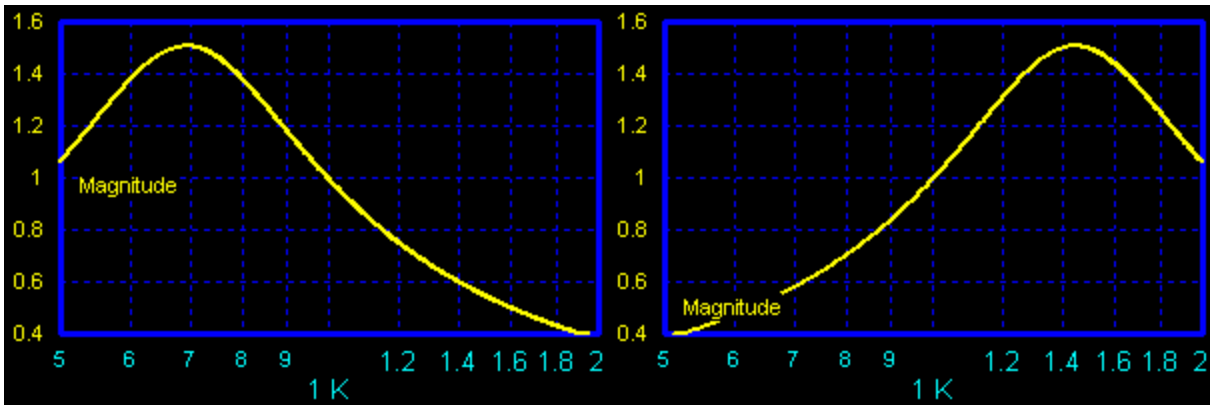
The gain of the filter as shown in the Active Control Panel is taken at the center frequency of the filter. The gain of the individual stages is taken at the center frequency of the respective stage,

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

which will not be the same of the center frequency of the filter. To illustrate, the frequency response of a second order Butterworth band pass filter is shown below along with the frequency response of both cascaded stages.



2nd Order Band Pass Filter Gain of 1.000

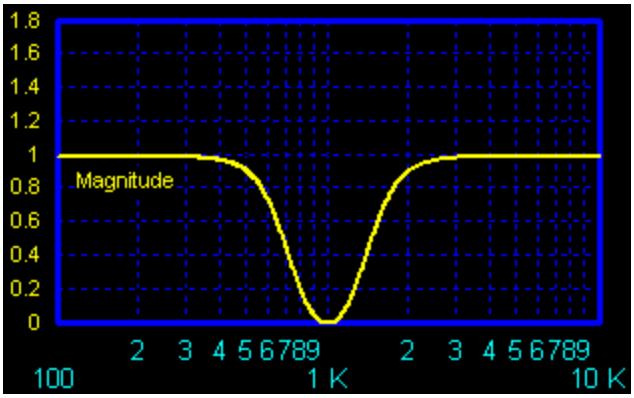


Band Pass Stage Gains of 1.510

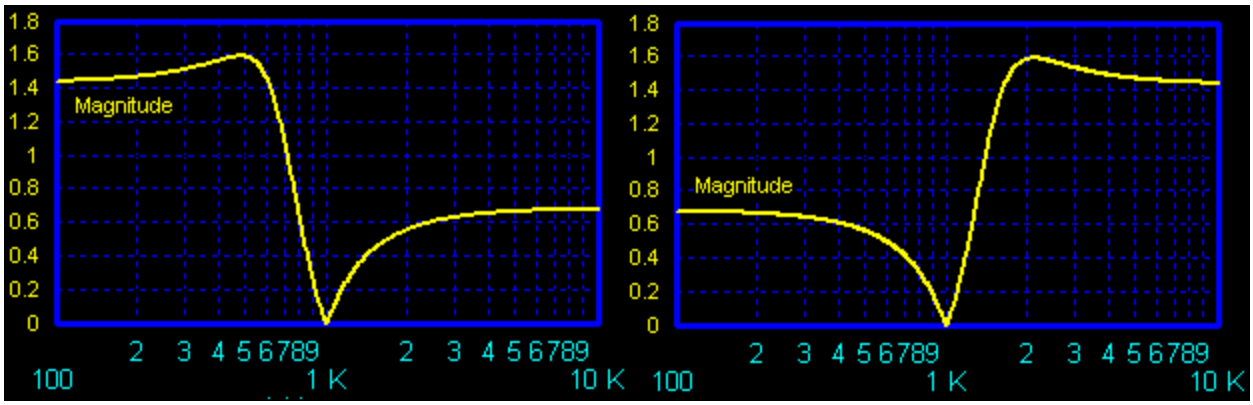
Band Stop Filters

The gain of the band stop filter as shown in the Active Control Panel is taken at the zero and infinite frequencies of the filter. The gain of the individual stages is different at zero and infinity. The stage gain in FilterSolutions is taken at zero or infinite frequency, depending on which has the higher gain, of the respective stage, which will not be the same of the zero and infinite frequencies of the filter. To illustrate, the frequency response of a second order Butterworth band stop filter is shown below along with the frequency response of both cascaded stages.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2nd Order Band Stop Gain of 1.000



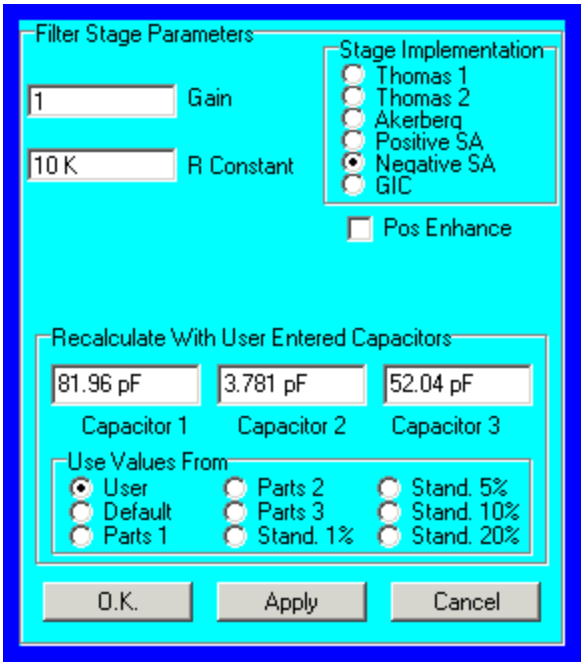
Band Stop Stage Gains of 1.443

User Entered Capacitors

FilterSolutions allows values to be entered for the desired capacitor in an active filter, and will then calculate necessary resistor values around those capacitors. If FilterSolutions is unable to create the active filter, you will be prompted to input different capacitor values.

To change the desired capacitor values, **right click on the opamp** in the same stage that the capacitor you want to change is in. The Modify Control Panel will appear with current capacitor values entered in the capacitor change entries, as shown below.

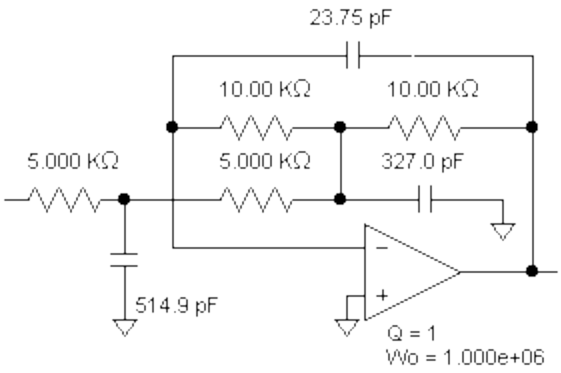
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



User Entered Capacitors

The default capacitor values seen in the panel generally are from the filter stage. You may enter a value, select the calculated default value, select the nearest capacitor from the standard parts lists, or select the nearest capacitor from the 1%, 5%, 10%, and 20% standard industrial values.

The sequence of capacitors is generally from left to right, as you can see below.



Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To update the the filter with the desired capacitors, enter the capacitor value in the appropriate text entry slot in the panel, and select **OK** The resistors will be recalculated to create a filter with the capacitors you specified.

1%, 5%, 10%, and 20% Standard Parts

The values for 1%, 5%, and 10% selections are standard industrial values. These values are listed below:

20% Parts	1.00, 1.5, 2.2, 3.3, 4.7, 6.8, 10.0, 15.0...
10% Parts	1.00, 1.20, 1.50, 1.80, 2.20, 2.70, 3.30, 3.90, 4.70, 5.60, 6.80, 8.20, 10.00, 12.00...
5% Parts	1.00, 1.10, 1.20, 1.30, 1.50, 1.60, 1.80, 2.00, 2.20, 2.40, 2.70, 3.00, 3.30, 3.60, 3.90, 4.30, 4.70, 5.10, 5.60, 6.20, 6.80, 7.50, 8.20, 9.10, 10.00, 11.00...
1% parts	1.00, 1.02, 1.05, 1.07, 1.10, 1.13, 1.15, 1.18, 1.21, 1.24, 1.27, 1.30, 1.33, 1.37, 1.40, 1.43, 1.47, 1.50, 1.54, 1.58, 1.62, 1.65, 1.69, 1.74, 1.78, 1.82, 1.87, 1.91, 1.96, 2.00, 2.05, 2.10, 2.15, 2.21, 2.26, 2.32, 2.37, 2.43, 2.49, 2.55, 2.61, 2.67, 2.74, 2.80, 2.87, 2.94, 3.01, 3.09, 3.16, 3.24, 3.32, 3.40, 3.48, 3.57, 3.65, 3.74, 3.83, 3.92, 4.02, 4.12, 4.22, 4.32, 4.42, 4.53, 4.64, 4.75, 4.87, 4.99, 5.11, 5.23, 5.36, 5.49, 5.62, 5.76, 5.90, 6.04, 6.19, 6.34, 6.49, 6.65, 6.81, 6.98, 7.15, 7.32, 7.50, 7.68, 7.87, 8.06, 8.25, 8.45, 8.66, 8.87, 9.09, 9.31, 9.53, 9.76, 10.0, 10.20...

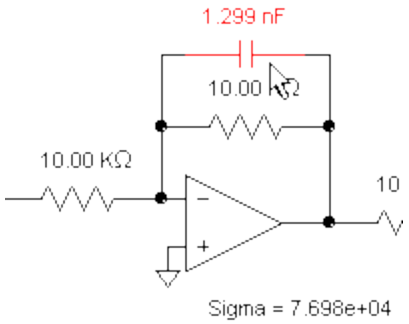
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modify Active Circuit Elements

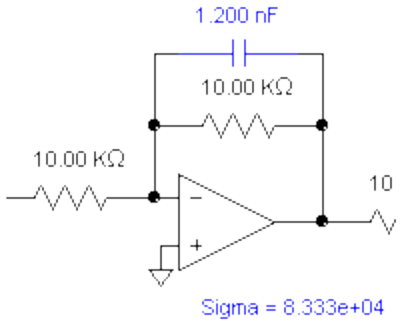
Active Circuit elements may be modified, and the effect displayed in real time. Element values may be updated to selected values, by percent change, to the nearest standard part, to the nearest standard industrial value, or random value. The random values feature may be used to perform Monte Carlo analysis.

To modify an active circuit element:

Pass the cursor over the element you desire to modify. It will turn red. Left click the mouse to change the element value.

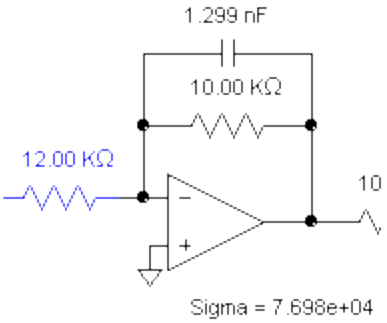


Elements are displayed in blue after they are modified. If the frequency information changes, it is also displayed in blue.



Unaltered frequencies continue to be displayed in black.

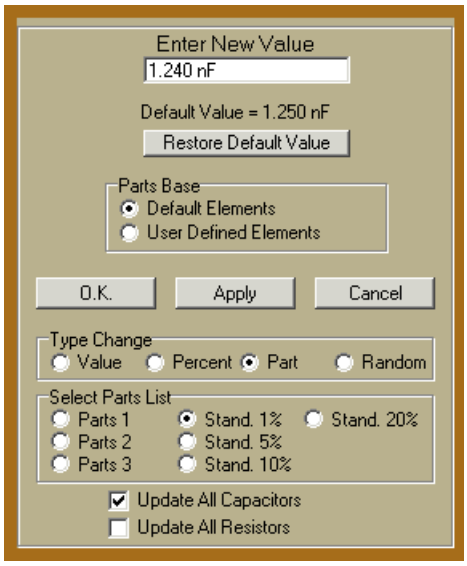
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The colors of red and blue may be changed with the Color Control Panel.

Change Control Panel

Click the left mouse key to bring up the change control panel



The Change control panel entries are defined in the following table:

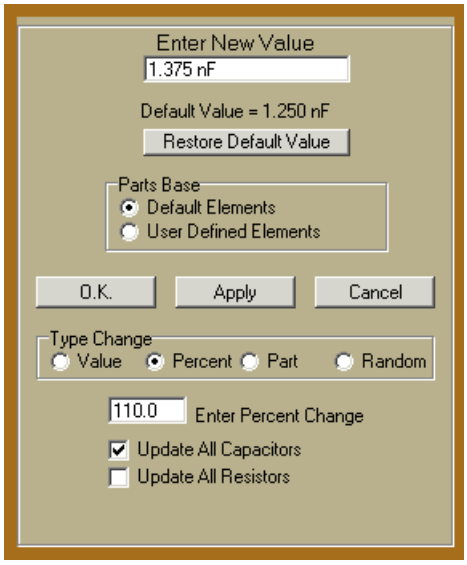
Enter New Value	Enter the desired new value of the component. This value will be calculated if "Percent" or "Part" is selected in the Type Change selection.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Restore Default Value	Restores the element value to the original unmodified value.
Parts Base	When "Percent" or "Part" is selected, the "Parts Base" selection determines whether to apply the part value computation to the default part value or the current value.
OK	Closes the control panel and updates the element value in the circuit.
Apply	Update circuit and graphical displays in real time without closing the Change Control Panel.
Cancel	Closes the control panel and does not update the element value.
Type Change	Specify to update element to a fixed value, percent change in value, or update to the nearest standard part.
Select Parts List	Determines which part list to use in the standard parts selection. Part Lists are maintained on the Main Control Panel. The 1%, 5%, 10%, 20% selections update the part to the nearest standard 1%, 5%, and 10% standard value.
Update All...	Check to update all like parts in the filter.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When a **Percent** is selected in the **Type Change** selection, the Change panel looks like this.



Enter the desired percentage change. All other selections work the same as described above.

1%, 5%, 10% and 20%Standard Parts

The values for 1%, 5%, 10%, and 20% selections are standard industrial values. These values are listed below:

20% Parts	1.00, 1.5, 2.2, 3.3, 4.7, 6.8, 10.0, 15.0...
10% Parts	1.00, 1.20, 1.50, 1.80, 2.20, 2.70, 3.30, 3.90, 4.70, 5.60, 6.80, 8.20, 10.00, 12.00...
5% Parts	1.00, 1.10, 1.20, 1.30, 1.50, 1.60, 1.80, 2.00, 2.20, 2.40, 2.70, 3.00, 3.30, 3.60, 3.90, 4.30, 4.70, 5.10, 5.60, 6.20, 6.80, 7.50, 8.20, 9.10, 10.00, 11.00...
1% parts	1.00, 1.02, 1.05, 1.07, 1.10, 1.13, 1.15, 1.18, 1.21, 1.24, 1.27, 1.30, 1.33, 1.37, 1.40, 1.43, 1.47, 1.50,

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	1.54, 1.58, 1.62, 1.65, 1.69, 1.74, 1.78, 1.82, 1.87, 1.91, 1.96, 2.00, 2.05, 2.10, 2.15, 2.21, 2.26, 2.32, 2.37, 2.43, 2.49, 2.55, 2.61, 2.67, 2.74, 2.80, 2.87, 2.94, 3.01, 3.09, 3.16, 3.24, 3.32, 3.40, 3.48, 3.57, 3.65, 3.74, 3.83, 3.92, 4.02, 4.12, 4.22, 4.32, 4.42, 4.53, 4.64, 4.75, 4.87, 4.99, 5.11, 5.23, 5.36, 5.49, 5.62, 5.76, 5.90, 6.04, 6.19, 6.34, 6.49, 6.65, 6.81, 6.98, 7.15, 7.32, 7.50, 7.68, 7.87, 8.06, 8.25, 8.45, 8.66, 8.87, 9.09, 9.31, 9.53, 9.76, 10.0, 10.20...

Monte Carlo Analysis

A Monte Carlo sensitivity analysis may be performed with the use of the "Random" selection. Enter the desired standard deviation in percent, and FilterSolutions will automatically update the selected element, or all like elements, by a random number from a Uniform or Gaussian (Normal) distribution. The mean of the distribution is 0%, and the maximum tolerance of the Uniform distribution or the standard deviation of the Gaussian distribution is entered by the user.

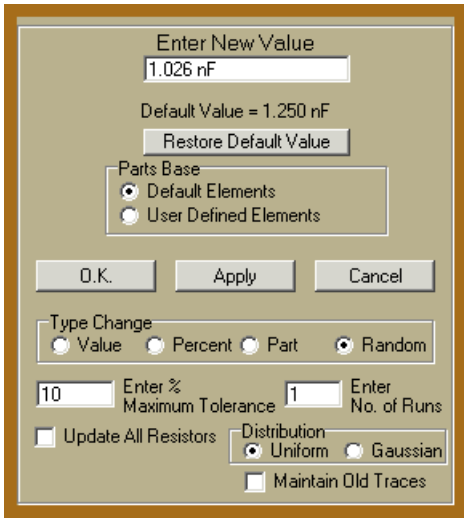
Manufactured parts are usually specified as a maximum not-to-exceed percentage error without any specifications for the shape of the error distribution between the upper and lower limits, so neither Uniform nor Gaussian are totally accurate models of the true error, but can be used to produce reasonably accurate results.

All graphical analysis traces are automatically updated each time the "Applied" button is depressed. Check the "Maintain Old Traces" checkbox to prevent old graphical traces from being overwritten. Enter the number of desired simulation runs per button click on the "No. of Runs" box.

Monte Carlo analysis may be performed on the original ideal parts, or the user selected part values.

See more on Monte Carlo analysis in the Help section.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Random updates are particularly useful in investigating the robustness of a filter to element errors.

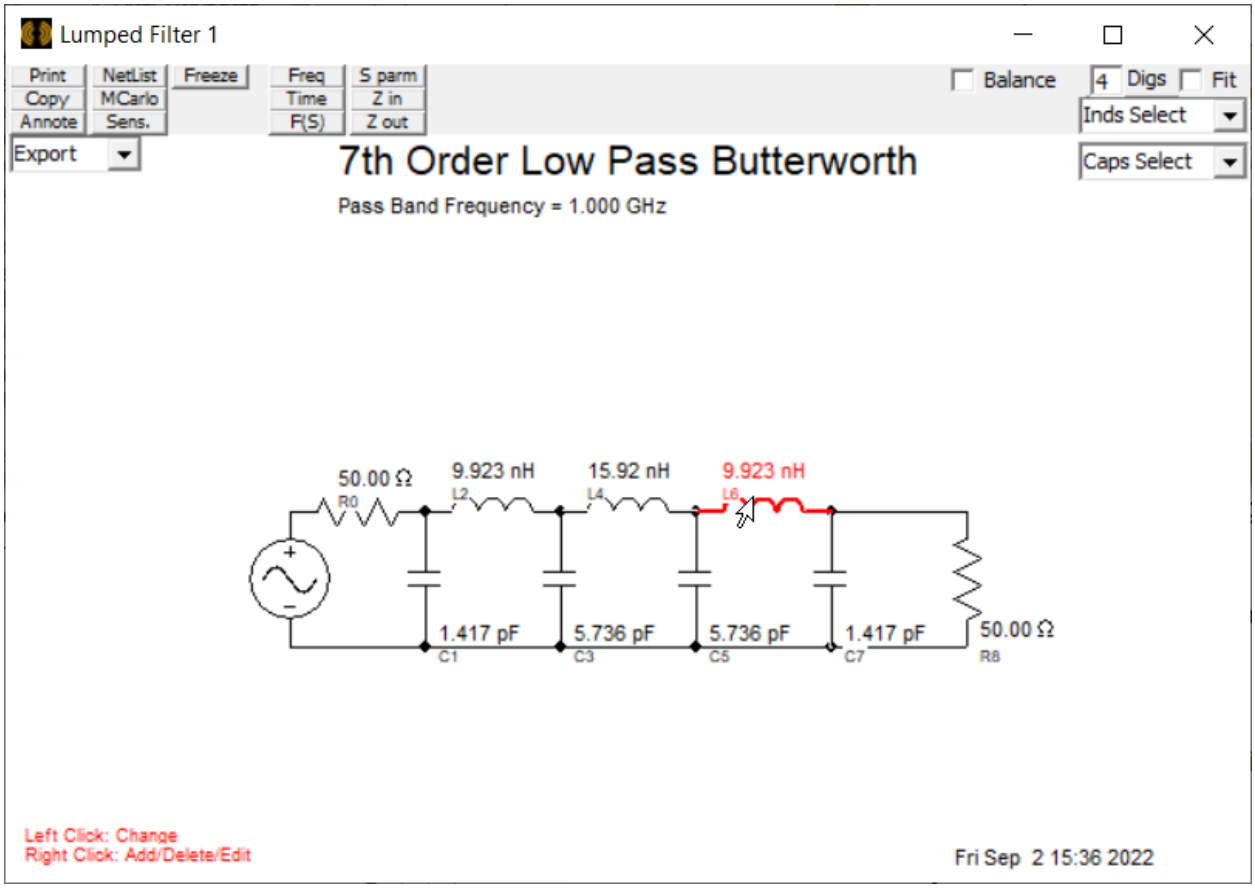
Standard Parts

FilterSolutions provides a flexible means to update the lumped or active filter elements values with the nearest value from the available standard parts, or the nearest 1%, 5%, and 10% standard part value. Up to three standard parts databases are maintained by **FilterSolutions**. If you need more than three, they may easily be maintained in another text or word processing document and copied or pasted in and out of FilterSolutions.

To update filter parts to the nearest value in the parts list:

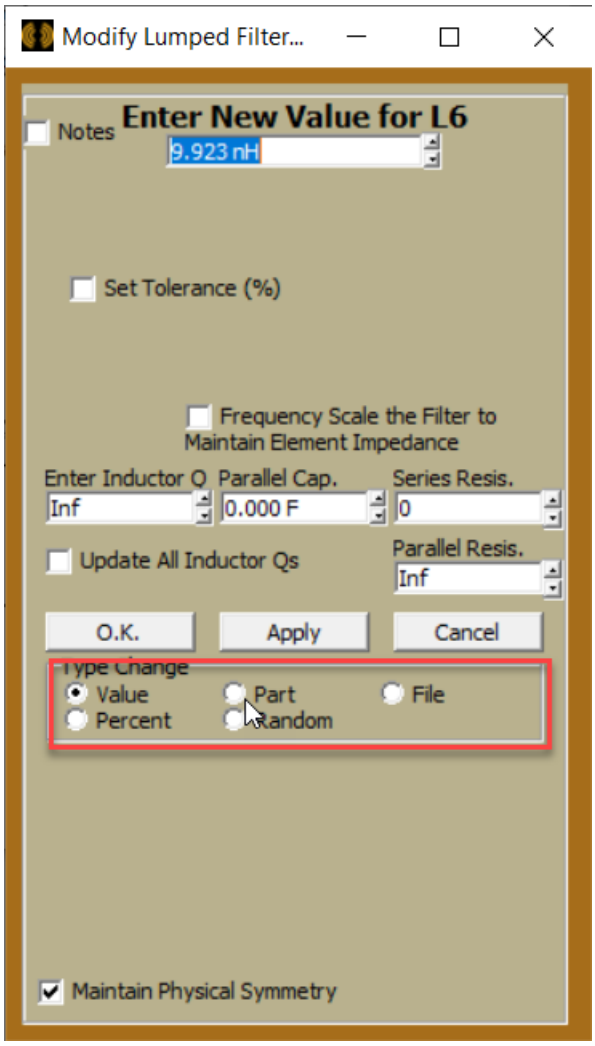
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select an element in the filter display.



2. Select **Parts** in the **Type Change** group box.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Select **OK**

Standard part updates are also available in the upper right corner of passive and active schematics.

Passive and active filters may have individual element or all like elements set to the nearest standard value from the selected database. The checkbox at the bottom of the Change control panel determines if one or all elements are updated.

You may select the default element value or the currently assigned element value as the base for the nearest part. A selection box in the change control panel determines the choice.

The upper part of the FilterSolutions standard parts window is shown below with some sample standard parts.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting the Q (Lumped filters only)

To associate a specific part value with a specific Q, add "Q=50", where 50 is an example Q value to the part line. For example:

Setting Series and Parallel Parasitic Resistance (Lumped filters only)

To associate a specific part value with a specific resistance, add "Rs=5" and "Rp=525", where 5 is an example series resistance value, and 525 is an example parallel resistance. The standard parts form is shown in Figure 1 below.

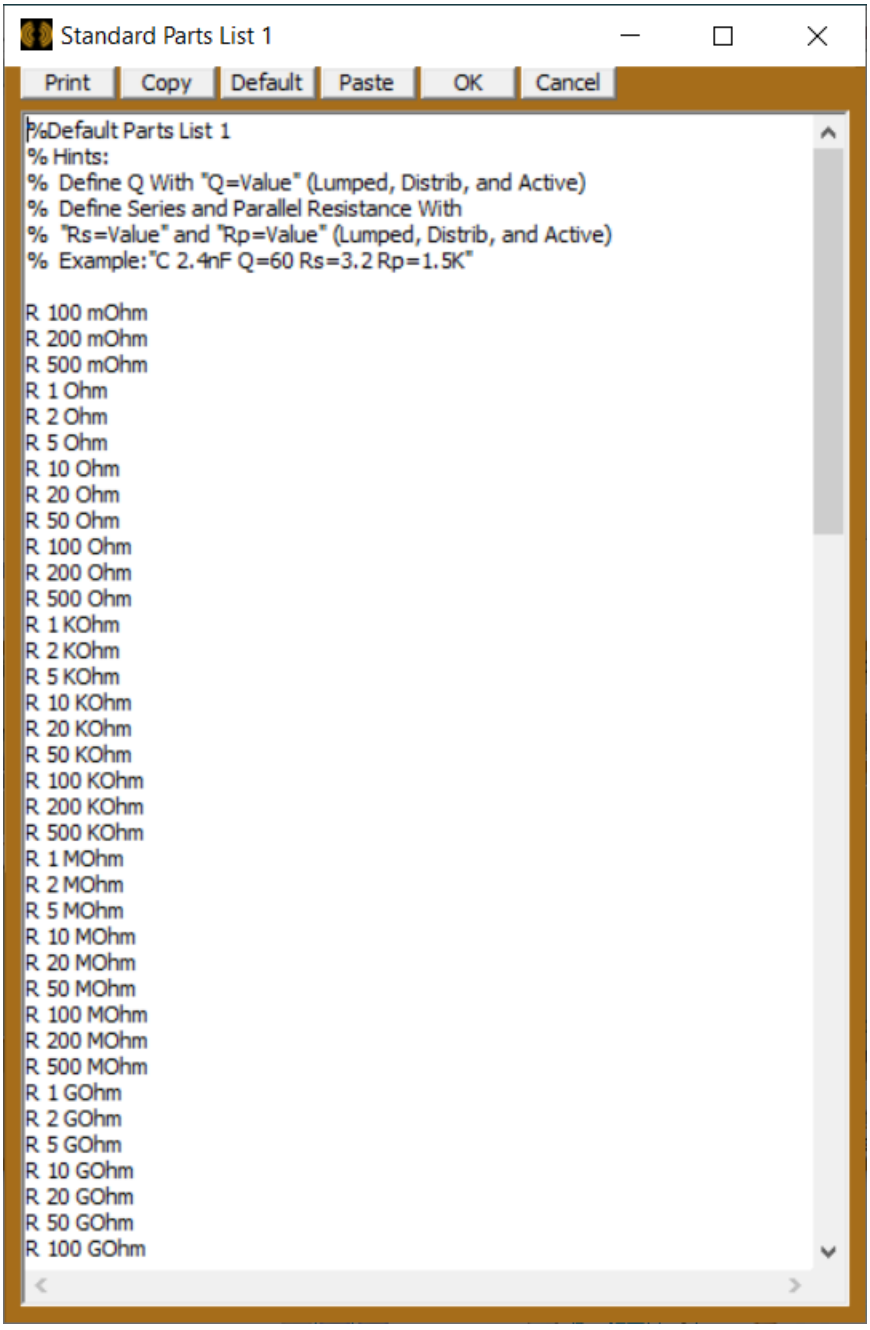
For Example:

"L 32uH Q=40 Rs = 3.5 Rp=7.25K" will have the effect of setting all inductors whose nearest part value is 32 uH to a Q of 40, a series resistance of 3.5 Ohms, and a parallel resistance of 7.25K Ohms.

Saving in an External File

FilterSolutions saves the user defined parts list in its own file. However, if it is desired to save the parts list in an external file, press Ctrl+C to copy the parts list and Ctrl+P to paste in the external file.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

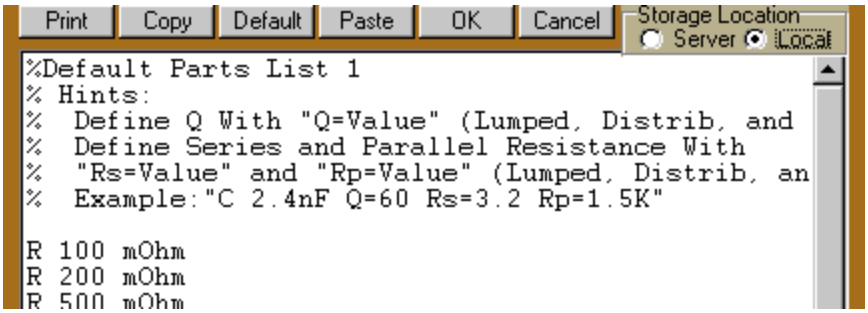


Saving on a common network server (Floating licenses only)

Network users of FilterSolutions may save standard parts on the network manager with the use of an Administrator's password. Network users may retrieve this common standard parts lists for use in their projects. This helps insure that all users on a project are using the same standard parts list. Users and administrators may select in the upper right where their part lists come

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

from, as shown in Figure 2 below. Server parts are read-only for users, and read/write for administrators. The license key issued by Nuhertz determines user/administrator status.



Parts Files

If administrators do not wish to use the administrative passwords to upload standard parts to Nuhertz Server, parts files may be manually copied into their respective locations instead. Client FilterSolutions parts files are located in the FilterSolutions_ini directory Default: (C:\Program Files (x86)\Nuhertz Technologies\FilterSolutions_ini\) under the Nuhertz Technologies program directory. They are names, parts0.dat, parts1.dat and parts2.dat for standard parts lists 1, 2, and 3 respectively. These files may be manually copied into the server Nuhertz Server program directory (Default: C:\Program Files (x86)\Nuhertz Technologies\Nuhertz Server\) as well, removing the necessity of providing an administrators password to the individual(s) charged with maintaining the parts list.

The buttons across the top perform the following functions:

Print	Print the displayed standard parts
Copy	Copy the displayed parts to the windows clipboard. Select the portion to be copied, or select all or nothing to copy all text in the text box.
Default	Sets the standard parts to the default parts list.
Paste	Paste text from the windows clipboard into the standard parts text box.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

OK	Save you standard parts list in the FilterSolutions database and exit.
Cancel	Exit without saving the parts list.
Storage Location	Determines where the standard parts lists is read from. Administrators may write to the server parts lists. Users may only read from it.

Parts List Formats:

Comments	Precede the comment line with a "%" character. Up to 10 comments are allowed.
Numerical Entries	Integer, floating, and engineering notation are all acceptable. Six significant digits are stored in the database. Example: 12, 0.00047, 470n, 50pF are all acceptable numerical entries. All numerical entries are sorted in ascending order in the database. Up to 500 numerical entries are allowed per element type.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Capacitors	Precede the capacitance numerical entry with a "C" or "c" character.
Inductors	Precede the inductance numerical entry with a "L" or "l" character.
Resistor	Precede the capacitance numerical entry with a "R" or "r" character.

1%, 5%, 10% and 20% Standard Parts

The Change Control Panel has selections for 1%, 5%, 10%, and 20%. When these options are selected, the nearest part value is selected from a list of standard industrial values. These values are listed below:

20% Parts	1.00, 1.5, 2.2, 3.3, 4.7, 6.8, 10.0, 15.0...
10% Parts	1.00, 1.20, 1.50, 1.80, 2.20, 2.70, 3.30, 3.90, 4.70, 5.60, 6.80, 8.20, 10.00, 12.0...
5% Parts	1.00, 1.10, 1.20, 1.30, 1.50, 1.60, 1.80, 2.00, 2.20, 2.40, 2.70, 3.00, 3.30, 3.60, 3.90, 4.30, 4.70, 5.10, 5.60, 6.20, 6.80, 7.50, 8.20, 9.10, 10.00, 11.00...
1% Parts	1.00, 1.02, 1.05, 1.07, 1.10, 1.13, 1.15, 1.18, 1.21, 1.24, 1.27, 1.30, 1.33, 1.37, 1.40, 1.43, 1.47, 1.50, 1.54, 1.58, 1.62, 1.65, 1.69, 1.74, 1.78, 1.82, 1.87, 1.91, 1.96, 2.00, 2.05, 2.10, 2.15, 2.21, 2.26, 2.32, 2.37, 2.43, 2.49, 2.55, 2.61, 2.67, 2.74, 2.80, 2.87, 2.94, 3.01, 3.09, 3.16, 3.24, 3.32, 3.40, 3.48, 3.57, 3.65, 3.74, 3.83, 3.92, 4.02, 4.12, 4.22, 4.32, 4.42, 4.53, 4.64, 4.75, 4.87, 4.99, 5.11, 5.23, 5.36, 5.49, 5.62, 5.76, 5.90, 6.04, 6.19, 6.34, 6.49, 6.65, 6.81, 6.98, 7.15, 7.32, 7.50, 7.68, 7.87, 8.06, 8.25, 8.45,

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	8.66, 8.87, 9.09, 9.31, 9.53, 9.76, 10.0, 10.2...

Net Lists

This chapter contains the following topic:

- Net List

Active Netlists

To help retrieve the designed active circuits, FilterSolutions will output you filter in a netlist. From the control bar of any displayed active circuit, select **Netlist**, and the Netlist box will be displayed on the left side of the filter window. The number of significant digits used on the netlist will match the number in the "Digs" box on the right side of the controls bar. The netlist contains entries for AC and Transient analysis, including graph scaling. An AC source is used for the AC analysis. A Pulse source is installed, but commented out for the transient analysis. To run a transient analysis it is necessary to comment out the AC source, and remove the comment from the Pulse source.

Component Indexing

netlist component indexes use three or four digits. The first two digits represent the filter stage the component is in, the third and fourth digit represents the index within that stage. This simplifies the task of finding a specific component within a filter netlist. Component values may be copied directly to the window clipboard from the netlist.

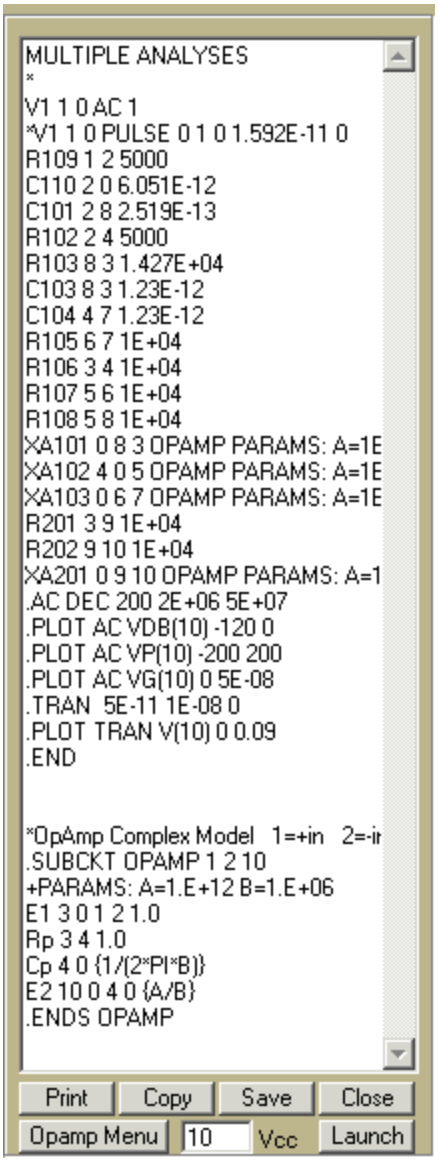
Op Amps

FilterSolutions models ideal opamps with very high gain netlist opamps. Real opamp models use an RC low pass filter to simulate the bandwidth and gain bandwidth product. Input and output impedances are simulated with resistors and capacitors. Voltage controlled sources are used to buffer the RC filter from other impedances.

Suitable pp amp Spice models from Analog Devices may be found at:
https://www.analog.com/en/design-center/simulation-models/spice-models.html

The following is an example of a FilterSolutions active netlist:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note that the bottom parts of the netlist are hidden below the text window. The scroll bar may be used to scroll up and down the netlist to observe these hidden parts.

Print	Sends the netlist to a printer.
Copy	Copies all or part of the netlist to the windows clipboard. Partial text may be copied by highlighting the desired text. All text may be copied by selecting all the text or none of the text.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Save	Saves the netlist to a text file.
Close	Closes the netlist window.

Opamp Menu	Select an opamp spice file from the internal or customize parts library. Refer to "Parts Library" in the Table of Contents for more on this. Note: if more oapmp Spice files are added by the user, some minor editing of the interface may be needed if the new Spice files do not conform to the existing format.
Vcc	Enter the Voltage desired to power the physical opamp in the netlist, if a physical opamp is selected for use with the "Opamp Menu".
Launch	Only visible is an application is installed in the system that uses documents with a .CKT suffix, such as Microcap. "Launch" will launch that application and load it with the displayed netlist.

Exporting to Other Software

This chapter contains the following topics:

- CKT Spice Files
- Touchstone Files

CKT Spice Files

Active filter netlists may be automatically loaded into .CKT simulations applications, such as Microcap. If an application is installed on the user's system that reads .CKT suffix files, a "Launch" button appears below the netlist that will launch the application and import the netlist, as shown in Figure 1. This is useful to validate the Nuhertz simulation results, or to simulate with selected physical opamp Spice models.

Note: if more oapmp Spice files are added by the user, some minor editing of the interface may be needed if the new Spice files do not conform to the existing format.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

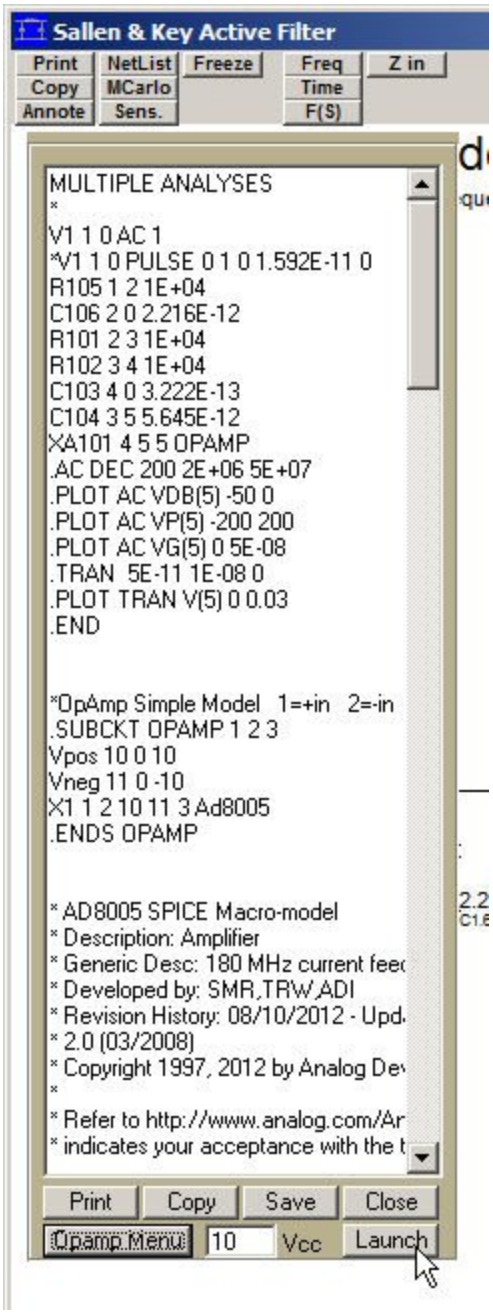


Figure 1: Launching a .CKT Application With The Imported netlist

Touchstone Files

You may easily export designed filters to Y or Z parameter Touchstone files that me be imported into other circuits analysis programs. Each filter schematic contains an "Export" function in the

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

upper left that, when selected, offers Touchstone Z and Y parameter file exports.

Circuits analysis programs usually include an element called "N-Port", "Ideal element", etc. that have the capability to read or import Touchstone files that contain Z or Y parameter data, and effectively model the designed filter when they read a FilterSolutions generated Touchstone file.

Port Definitions:

2 port filters:

Port 1 = Input.

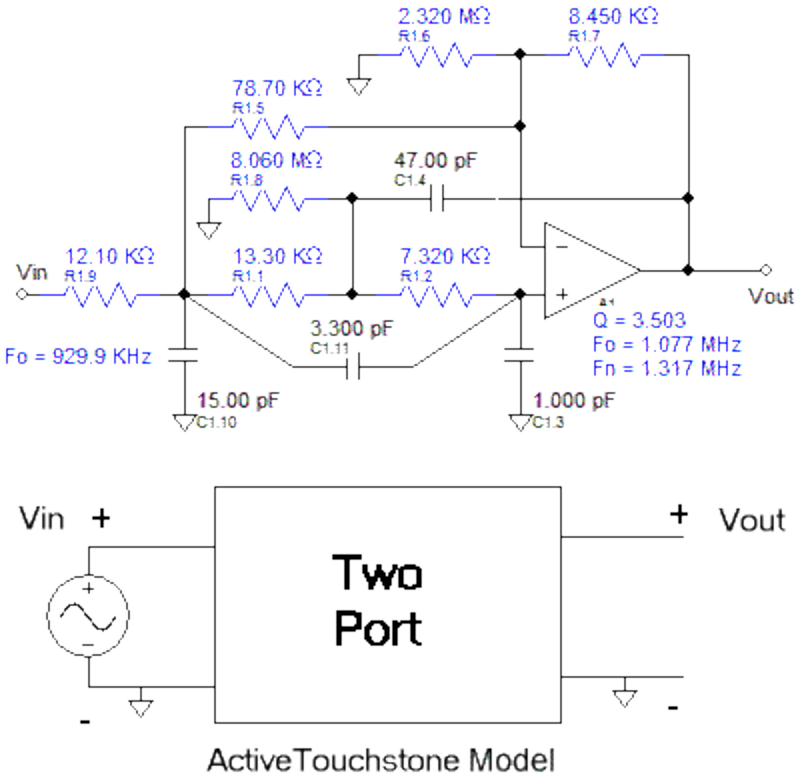
Port 2 = output

Tips:

- Z and Y parameters model only the filter network, and do not include any source or load terminations.
- Beginning and ending frequencies are determined by the FilterSolutions Min Freq and Max Freq entries.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Illustrations:



Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15 - Switched Capacitor Filters

This chapter contains the following topics:

- Purpose
- Component Definitions
- Z Transforms
- Biquad Architectures
- Band Pass Architectures
- Op Amps
- Modify Switched Capacitor Filters
- Net Lists
- Exporting to other Software

Purpose

This chapter contains the following topic:

- Purpose

Purpose and Definitions

The purpose of building switched capacitor filters is to allow filter circuits to be economically constructed in integrated circuits. In integrated circuits, resistors are relatively expensive and difficult to control, and their numbers should be minimized to maximize the quality and efficiency of the design. Capacitors, amplifiers, and switches, on the other hand, are relatively inexpensive and more easily controlled. Switched capacitor filters use switched capacitors to simulate resistors in active filters and thereby minimize the cost and maximize the quality of the filter.

Switched Capacitors behave like digital filters rather than analog filters. The clock switching frequency is analogous to a digital filter update rate, and, just like in digital filters, affects the alias frequency of the filter. Since a switched capacitor filter behaves like a digital filter, the components are calculated with the objective of meeting Z transform requirements, rather than S transform requirements as in analog filters.

ANY digital filter may be implemented as a switched capacitor filter, including FIR filters. The required size of an FIR filter may make it impractical to construct using switched capacitor filters, but they can, in theory, be built. FilterSolutions supports switched capacitor IIR filters and FIR filters.

Component Definitions

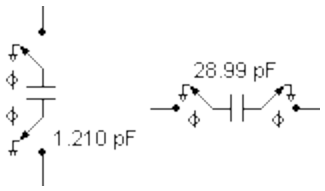
This chapter contains the following topics:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Positive Switched Capacitors
- Negative Switched Capacitors
- Unswitched Capacitors

Positive Switched Capacitors

The graphical symbols used by FilterSolutions to designate a positive switched capacitor are:



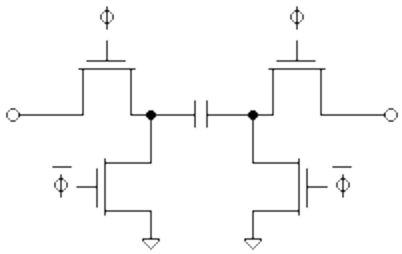
Positive Switched Capacitor Symbol

The switched are synchronized such that the capacitor is alternately in line and grounded. Current impulses alternately charge and discharge the capacitor. **When averaged over time**, the current impulses make the positive switched capacitor look like a **resistor of value T/C** to the rest of the circuit, where T is the switching period, and C is the capacitance.

The switch polarity may be read as "Ground on True" or "Ground on False" provided all switches in the switched capacitor filter use the same polarity definition.

Switch Models

The electronic switches may be constructed from Mosfets or other electronic devices that provide good quality electronic open/short switching capability. It is essential in the switch design that the control and inverted control signals **DO NOT OVERLAP.** A common inverter device by itself is generally inadequate for this requirement.



Positive Switched Capacitor Model

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Z Transform

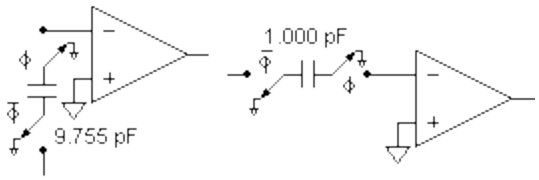
When used in a switched capacitor filter, the Z Transform of the admittance of the positive switched capacitor is Y*T where Y is the admittance, which reduces to simply C. H(Z) = C is used when calculating switched capacitor Z transforms.

Impedance

When calculating the input impedance to a filter, FilterSolutions uses the average impedance calculation of T/C.

Negative Switched Capacitors

The graphical symbols used by FilterSolutions to designate a positive switched capacitor are:



Negative Switched Capacitor model with Opamps

The switched are asynchronous such that one side of the capacitor is always grounded, and the other is always in line. Current impulses alternately charge and discharge the capacitor, however, the current on the positive side of the capacitor is the opposite that of the positive switched case, and the time of the current impulse is delayed by a single clock cycle. This negative current flow into the positively switched terminal makes the negative switched capacitor look like a negative resistor from the perspective of the rest of the circuit when the positively switched side is connected to a zero volt node, as seen in the diagrams above. The current impulses seen from the negatively switched side, however, are not reversed, so the input impedance is still positive as seen from the negatively switched side. **When averaged over time**, the current impulses make the positive switched capacitor look like a **resistor of value T/C** to the rest of the circuit, where T is the switching period, and C is the capacitance. The Z in the denominator represents the single frame delay of the Z Transform.

Negative switched capacitors **may not float**. Although it is possible to arrange the negative switched capacitor in a floating configuration, the proper impedance and transfer function computations require that each side of the negative switched capacitor be attached to a fixed voltage reference, usually and input voltage and a zero volt opamp input node.

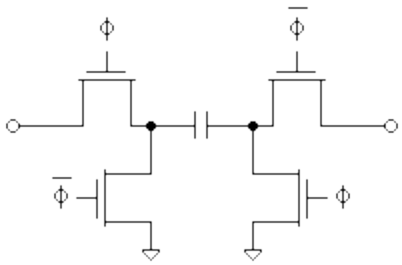
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

It is essential in the switch design that the control and inverted control signals **DO NOT OVERLAP.** A common inverter device by itself is generally inadequate for this requirement. Note that the negative switched capacitor is identical to the positive switched capacitors with the exception that the polarity of one side of the device is reversed.

The switch polarity may be read as "Ground on True" or "Ground on False" provided all switches in the switched capacitor filter use the same polarity definition.

Switch Models

The electronic switches may be constructed from Mosfets or other electronic devices that provide good quality electronic open/short switching capability. It is essential in the switch design that the control and inverted control signals **DO NOT OVERLAP.** A common inverter device by itself is generally inadequate for this requirement. Note that the negative switched capacitor is identical to the positive switched capacitors with the exception that the polarity of one side of the device is reversed.



Negative Switched Capacitor Model

Z Transform

When used in a switched capacitor filter, the Z Transform of the admittance of the positive switched capacitor is -Y*T/Z where Y is the admittance, which reduces to simply -C/Z. H(Z) = C/Z is used when calculating switched capacitor Z transforms.

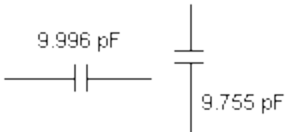
Impedance

When calculating the input impedance to a filter, FilterSolutions uses the average impedance calculation of T/C.

Unswitched Capacitors

The graphical symbols used by FilterSolutions to designate a positive switched capacitor are:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The unswitched capacitors, as seen above, are just ordinary capacitors, with standard impedance computations

Z Transform

When used in a switched capacitor filter, the Z Transform of the admittance of the unswitched capacitor is Y*T where Y is the admittance, which may be computed as CS/T. If S is replaced by the Z Transform, S = (Z-1)/(TZ), the equation reduces to **H(Z) = C(Z-1)/Z**

Impedance

When calculating the input impedance to a filter, FilterSolutions uses the standard L1/(CS).

Z Transforms

This chapter contains the following topics:

- IIR Z Transforms
- FIR Z Transforms
- User Entered Z Transforms
- Which Z Transform to Use

IIR Z Transform

IIR(Infinite Impulse response)

Switched capacitor filters may generally be designed to meet any desired IIR Z transform. A brief description of available Z transform implementation is shown below. **More information is available in the Digital IIR sections of the Help file**.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Bilinear

The Bilinear uses trapezoidal integration to implement the digital Z transform. It is usually the most accurate digital implementation for filtering a continuous variable. However, significant warping of the frequency spectrum occurs.

Bilinear with Prewarping

Prewarping the frequency spectrum allow for exact frequency performance matches at one or more frequencies. Selecting prewarping for low pass and high pass frequencies has the effect of eliminating error at the cutoff frequency. Selecting prewarping for band pass and band stop frequencies has the effect of eliminating error at both corner frequencies.

Matched Z

The Matched Z transform employs a basic simple method of translating analog poles and zeros to digital poles and zeros on the cascaded continuous transfer function..

Impulse Invariant

The Impulse Invariant transformation retains the exact impulse response of the analog system. It is desirable in cases where the impulse time response of the filter is important to be retained. The transformation is properly performed on the parallel transfer function.

Step Invariant

The Step Invariant transformation retains the exact impulse response of the analog system. It is desirable in cases where the step time response of the filter is important to be retained. The transformation is properly performed on the parallel transfer function.

Modified Impulse Invariant

The Modified Impulse Invariant transformation performs an Impulse Invariant computation on the cascaded transfer function biquads instead of the parallel. This introduces slight errors in the impulse response, but has the benefit of sometimes resulting in a smaller capacitor spread.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modified Step Invariant

The Modified Step Invariant transformation performs an Step Invariant computation on the cascaded transfer function biquads instead of the parallel. This introduces slight errors in the impulse response, but has the benefit of sometimes resulting in a smaller capacitor spread.

FIR Z Transform

FIR (Finite Impulse Response)

Switched capacitor filters may generally be designed to meet any desired FIR Z transform. **More information is available in the Digital FIR sections of the Help file**.

Finite impulse response filters have an advantage over the above IIR filters in that the group delay may be held constant. FIR filters have a disadvantage is that the pass band is less controlled and they may be excessively large to be of practical use in switched capacitor filters. FilterSolutions supports all FIR filters that are supported with digital filters.

User Entered Z Transform

The Custom Filter feature may be used to enter any IIR or FIR Z Transform of the designer's choice. Digital poles and zeros may be manually added, deleted, and relocated using the pole/zero plots and mouse keys. More information is available under the Custom Filter and Pole/Zero Plot entries of the Help file.

Which Z Transform to Use

The Z transform best to use is a combination of accuracy requirements and capacitor spread requirements. The accuracy and capacitor spread of each transform varies from filter to filter. If capacitor spread is the overriding quality requirement, select the "Min Cap Spread" check box, and FilterSolutions will select the Z transform with the best capacitor spread. If frequency response of the most important quality factor, then the Bilinear Transformation with prewarping is usually the best to use, even though the capacitor spread is not the best.

Biquad Architectures

This chapter contains the following topics:

- Double Op Amp Biquads
- Negative Single Amplifier Stages
- Positive Single Op Amp Stages
- Cascade Biquads
- Parallel Biquads

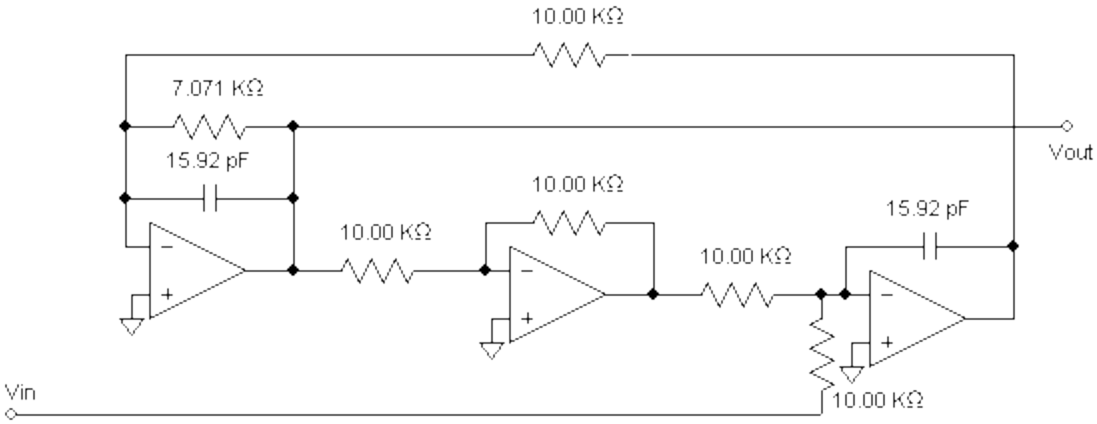
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Double Op Amp Biquads

All first and second order Z transforms may be realized with the double opamp biquads **except when the numerator and denominator are of equal order and their leading coefficients are of opposite sign.**

Switched Capacitor double opamp stages come in low Q and High Q varieties. In general, the low Q biquads tend to have less capacitor spread that high Q biquads for Q less than unity. **FilterSolutions checks that actual capacitor spread of both biquads for both positive and negative gain sign, and uses the biquad with the minimal spread regardless of the Q or sign of the gain.**

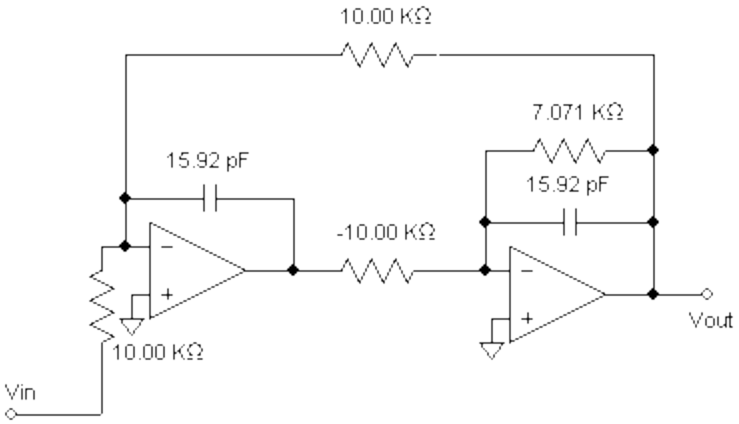
Switched capacitor double opamp stages are baselined from the active filter Thomas Biquads, one of which is shown below. Whereas the Thomas Biquads have inverting gain amplifiers, the switched capacitor double opamp biquads instead incorporate a negative resister between the two opamps. The inverting side is attached to the output of the first opamps, and the non inverting side is attached to the negative terminal of the second. Since the negative terminal is a fixed zero volts, the negative resistor may be simulated with a negative switched capacitor.



Thomas 1 Biquad

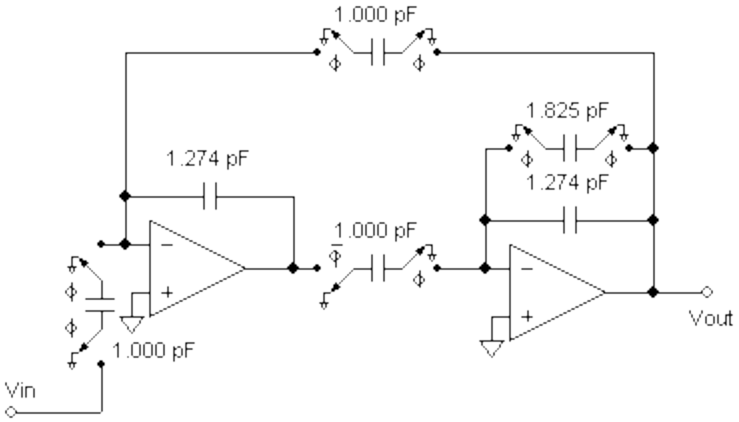
An equivalent analog active filter shown below uses a negative resister in lieu of the inverting center opamp. It is functionally identical to the Thomas 1 Biquad.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Thomas 1 Biquad With Negative Resistor

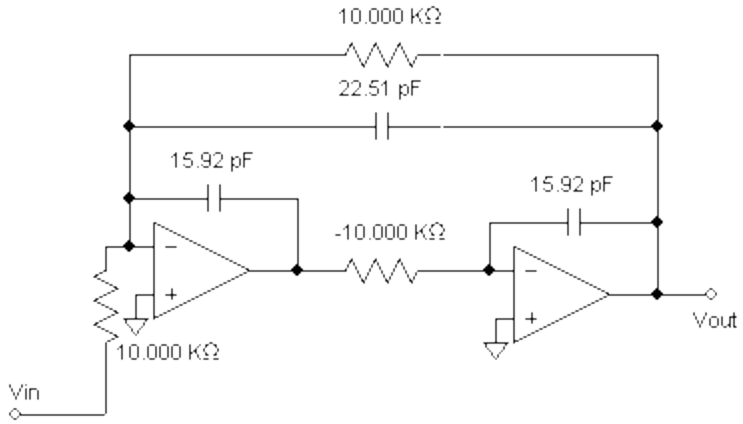
Switched capacitors may be used to simulate both positive and negative resistances, as shown below in the switched capacitor low Q biquad. Note that the capacitors are scaled such that the smallest capacitor is 1pF. If 1pF is the minimal allowable capacitor size, this scaling is optimal to minimize the size of all capacitors. Capacitor values are calculated from the Z Transform of the desired filter to be designed.



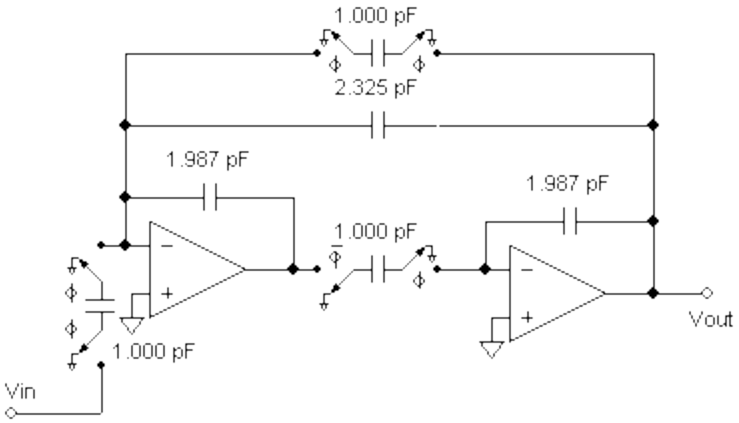
Switched Capacitor Low Q Biquad

Switched capacitor high Q biquads are just a minor variation from the Thomas 1 Biquad in that a feedback capacitor is takes the place of the second opamp feedback resistor to set the stage Q.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Switched Capacitor High Q Biquad Analog Equivalent



Switched Capacitor High Q Biquad

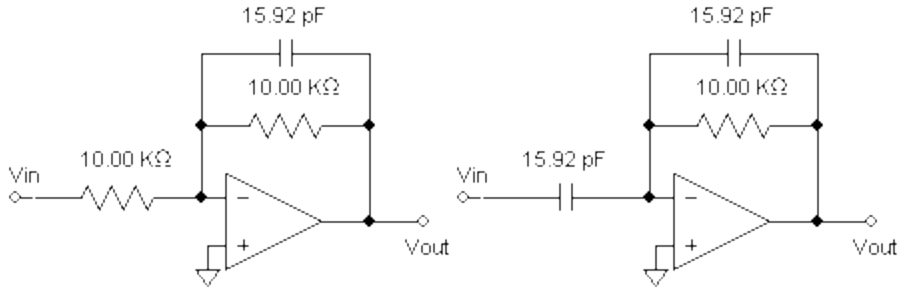
Negative Single Opamp Stages

First order Z transforms may be realized with the negative single op stages for all negatively signed first order Z Transform numerators and positively signed zero order Z Transform numerators.

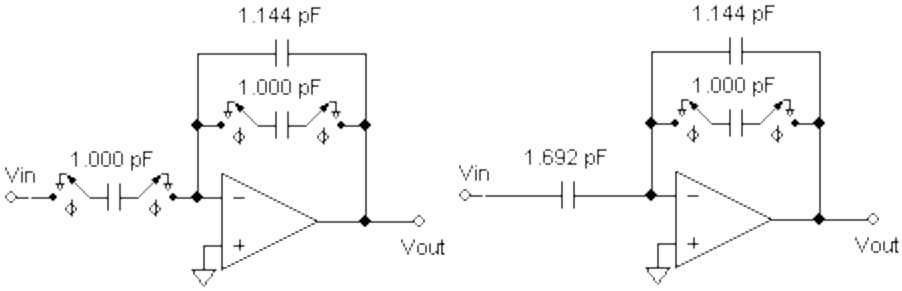
Switched capacitor negative single op stages are baselined from the active filter first order Negative SAB's, which are shown below. Whereas the analog Negative SAB's are always negative, only the negative single op switched capacitor stages used to implement first order Z Transform numerators are negative. Zero order Z Transform numerator implementations are positive.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FilterSolutions prioritizes minimum capacitor spread over desired sign of the gain when choosing negative single stages over positive single stages. The user may manually change the stage type by left clicking on an opamp.



Analog Negative Single Order Stages



Switched Capacitor Negative First Order Stages

Positive Single Opamp Stages

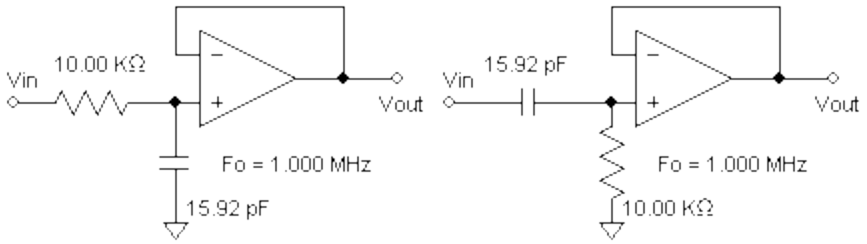
First order Z transforms may be realized with the positive single op stages for all positively signed first order Z Transform numerators and negatively signed zero order Z Transform numerators.

Switched capacitor positive single op stages are baselined from the active filter first order Positive SAB's, which are shown below. Whereas the analog Positive SAB's are always positive, only the positive single op switched capacitor stages used to implement first order Z Transform numerators are positive. Zero order Z Transform numerator implementations are negative.

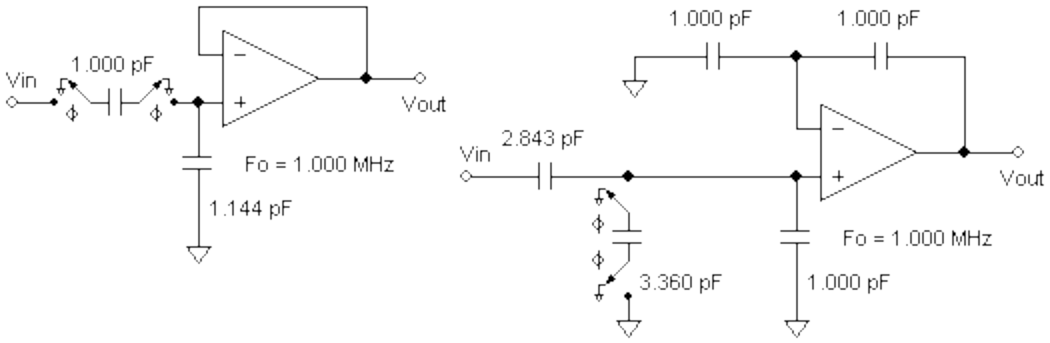
FilterSolutions prioritizes minimum capacitor spread over desired sign of the gain when choosing positive single stages over negative single stages. The user may manually change the stage type by left clicking on an opamp.

Positive single order switched capacitor stages are restricted in that they may not utilize negative

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Analog Positive Single Order Stages



Switched Capacitor Positive First Order Stages

Cascade Switched Capacitor Biquads

The most straightforward method to design switched capacitor filters is by breaking down large complex filters into biquads, just like active filters. Unlike active filters, switched capacitor filters are designed to meet Z transform requirements rather than S transform requirements. The Z Transform is broken down into cascaded first and second order terms. Each term becomes a filter stage. Switched capacitor values are then calculated for each stage to meet the Z Transform requirements of each term.

First Order Cascaded Terms

First order stages may be composed of positive single opamp stages or negative. FilterSolutions calculates both, and uses the one yielding the smallest capacitor spread. If the spread is identical, FilterSolutions uses the one that produces the desired gain sign. Positive and negative single op stages may be manually selected by left clicking the opamp in the stage of interest.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Second Order Cascaded Terms

Second offer stages may be composed of low Q stages or High Q stages. Generally, low Q stages yield better capacitor spreads for Q's below one, and High Q stages yield better spreads for Q's above one. FilterSolutions computes both, and uses the one with the best spread. Low Q and High Q stages may be manually selected by left clicking the opamp in the stage of interest. FilterSolutions also computes both the positive and negative gain stages when possible, and uses the gain sign that yield the minimum capacitor spread.

Cascade vs. Parallel Biquads

In general, cascade biquads yield better performance for Bilinear, Matched Z, Modified Impulse Invariant, and Modified Step Invariant filters. However, the higher performing filters may not necessarily produce the best capacitor spread. The decision of to use cascaded vs. parallel switched capacitor filters is a function of the accuracy and capacitor spread design requirements

Parallel Switched Capacitor Biquads

Large complex switched capacitor filters may be broken down into a sum of parallel biquads, just like active filters. Unlike active filters, switched capacitor filters are designed to meet Z transform requirements rather than S transform requirements. The Z Transform is broken down into parallel first and second order terms. Each term becomes a filter stage. Switched capacitor values are then calculated for each stage to meet the Z Transform requirements of each term. When the Z Transform to be implemented does not have constant offset term, FilterSolutions calculates both the positive sign and negative sign filters, and then uses the one with the smallest capacitor spread.

First Order Cascaded Terms

First order stages may be composed of positive single opamp stages or negative single opamp stages. The sign of the first order Z Transform term determines which one to use.

Second Order Cascaded Terms

Second offer stages may be composed of low Q stages or High Q stages. Generally, low Q stages yield better capacitor spreads for Q's below one, and High Q stages yield better spreads for Q's above one. FilterSolutions computes both, and uses the one with the best spread. Low Q and High Q stages may be manually selected by left clicking the opamp in the stage of interest. The sign of the parallel switched capacitor stage is determined by the sign of the associated parallel Z Transform term.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Final Stage

Parallel switched capacitor filters, like active filters, may or may not have a final summing gain stage. When **Final gain stage** is selected, FilterSolutions computes a positive gain or negative gain summation stage. The final gain is determined by the user gain entry.

Parallel vs. Cascade Biquads

In general, cascade biquads yield better performance for the (unmodified) Impulse Invariant and Step Invariant filters. However, the higher performing filters may not necessarily produce the best capacitor spread. The decision of to use parallel vs. cascaded switched capacitor filters is a function of the accuracy and capacitor spread design requirements

Band Pass Architectures

This chapter contains the following topic:

- Switched Cap Band Pass Architectures

Switched Capacitor Band Pass Architectures

Two varieties of cascaded band pass filters are offered by FilterSolutions, integrated and high/low pass. Leaving the "Hi/Lo Pass Stages" box unchecked produces the integrated band pass filters. Checking this box produces the high/low pass stage band pass filters. The difference affects the capacitor value selections, including spread, and internal voltage movements. In general, **wide band pass filters are better suited for high pass and low pass stags, and narrow bands are better suited for the integrated**. Attempting to build a wide band filter with integrated band pass stages, or a narrow band filter with high pass and low pass stages may produce undesirable internal voltages, which may saturate opamps.

Integrated Band Pass Filters

In integrated band pass filters, each stage of the filter is a band pass stage. The gain of these stages may become huge for wide band filters, but are generally acceptable for narrow band. Each term is constructed from a band pass Z domain transfer function, which in turn is constructed from an S domain transfer function.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

High/Low Stage Band Pass Filters

In high/low stage band pass filters, each stage of the filter is a high pass or low pass stage. The gain of these stages may become huge for narrow band filters, but is generally acceptable for wide band. Each term is constructed from a high pass or low pass Z domain transfer function, which in turn is constructed from an S domain transfer function.

More detailed technical information is available in the Active Filters Help section.

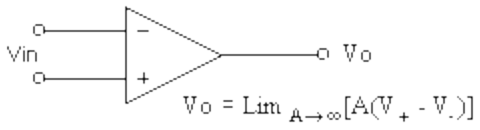
Op Amps

This chapter contains the following topic:

- Ideal Op Amps

Ideal Opamps

Opamps (operational amplifiers) are simply devices used to amplify a differential input voltage which be modeled as shown:



Ideal opamps have other normally finite features idealized. Input resistance is infinite; output resistance is zero; input differential voltage is zero; and output slew rate is infinite. All FilterSolutions switched capacitor analysis is performed assuming that all op amps are ideal.

Modify Switched Capacitor Filters

This chapter contains the following topics:

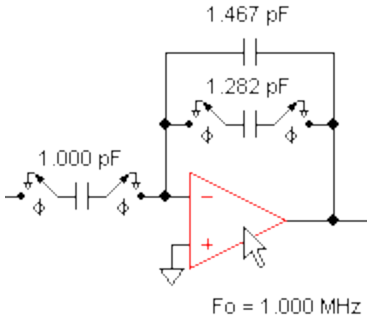
- Modify Stages
- Modify Elements

Modify Switched Capacitor Stages

The filter that FilterSolutions provides may not necessarily be in a form that best fits the designer's needs. Therefore, options are provided to change some of the features of each stage. For example, the gain of one of the stages may be too low or too high. You may change the gains of that stage.

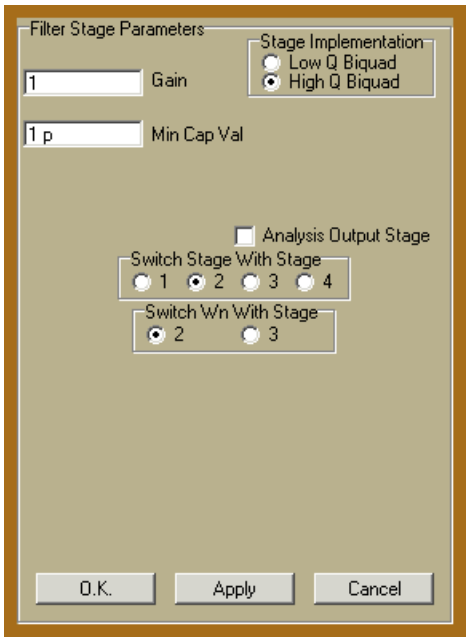
To modify a filter stage, pass the cursor of the opamp of that stage. It will turn red as shown:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



When you left click the mouse key, the Stage Control Panel will appear with several change options to choose from. The change options are a function of the parameters in the stage you selected to change. The default values in the panel will match the selected stage parameter values. You may modify any or all of the displayed parameters, select "OK", and the associated filter stage will assume the new parameter value. **Best results are obtained when changing only one parameter at a time.**

Modify Stage Control Panel



Stage Control Panel

Stage Control Panel Entry Definitions

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Analysis Output Stage	Moves the filter frequency and time analysis to the output of the selected stage.
Apply	Apply the selected changes without closing the window.
C Constant	Select a new minimum capacitance for the selected stage.
Cancel	Cancel the selected changes
Gain	Recalculate the stage with a new gain.
OK	Closes the control panel and updates the stage.
Switch Stage With Stage	Select another stage to change places with
Switch Wn With Stage	Select a stage to swap Wn with.
Stage Implementation	Recalculate the stage with a different implementation.

Modify Switched Capacitor Elements

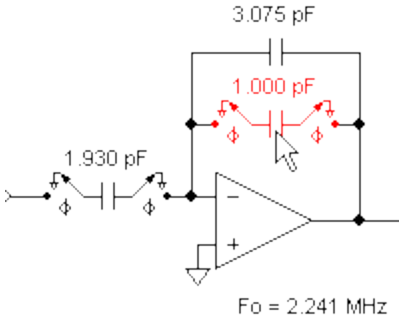
Active Circuit elements may be modified, and the effect displayed in real time. Element values may be updated to selected values, by percent change, to the nearest standard part, to the

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

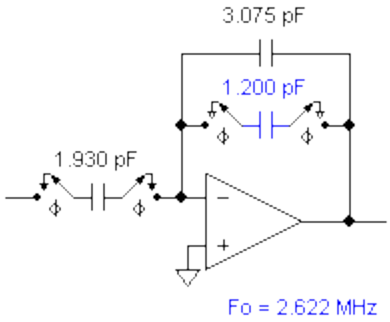
nearest standard industrial value, or random value. The random values feature may be used to perform Monte Carlo analysis.

To modify a switched capacitor circuit element:

Pass the cursor over the element you desire to modify. It will turn red. Left click the mouse to change the element value.

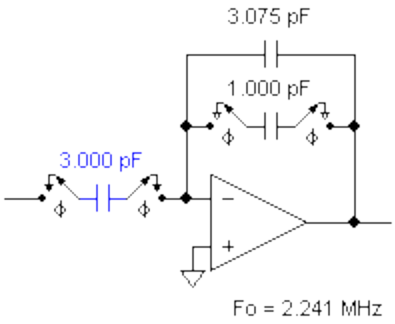


Elements are displayed in blue after they are modified. If the frequency information changes, it is also displayed in blue.



Unaltered frequencies continue to be displayed in black.

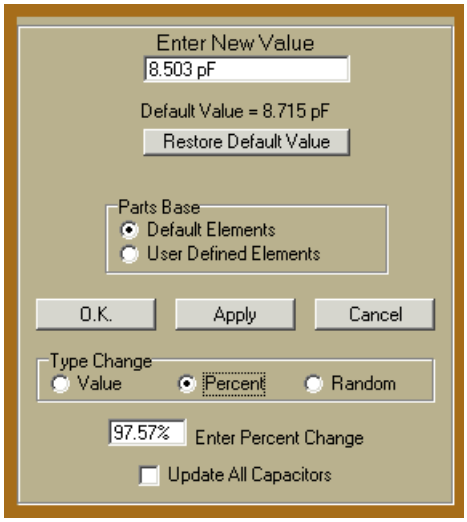
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



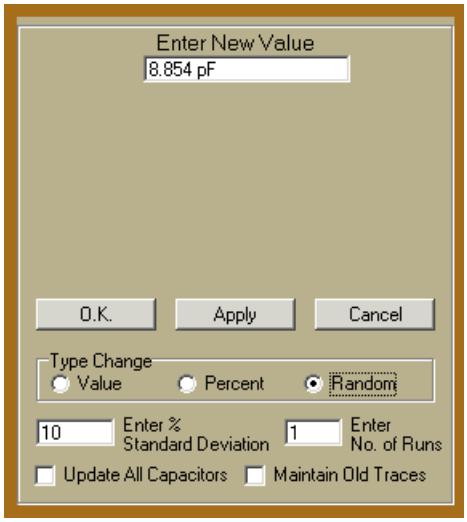
The colors of red and blue may be changed with the Color Control Panel.

Change Control Panel

Click the left mouse key to bring up the change control panel



Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The Change control panel entries are defined in the following table:

Enter New Value	Enter the desired new value of the component. This value will be calculated if "Percent" or "Part" is selected in the Type Change selection.
Restore Default Value	Restores the element value to the original unmodified value.
OK	Closes the control panel and updates the element value in the circuit.
Apply	Update circuit and graphical displays in real time without closing the Change Control Panel.
Cancel	Closes the control panel and does not update the element value.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

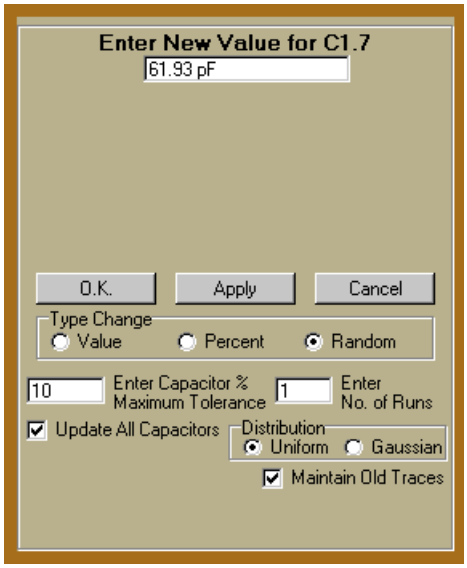
Enter Number of Runs	Enter the number of random simulations to be run at a time for Monte Carlo analyses.
Maintain Old Traces	Retains old random traces for visual Monte Carlo analysis.
Parts Base	Make the desired updates on the default or displayed element values.
Type Change	Specify to update element to a fixed value, percent change in value, or update to the nearest standard part.
Update All...	Check to update all like parts in the filter.

Monte Carlo Analysis

A Monte Carlo sensitivity analysis may be performed with the use of the **Random** selection. Enter the desired standard deviation in percent, and FilterSolutions will automatically update the selected element, or all like elements, by a random number from a Gaussian (or Normal) distribution. The mean of the distribution is 0%, and the standard deviation is that which is entered by the user.

All graphical analysis traces are automatically updated each time the **Apply** button is depressed. Check the **Maintain Old Traces** checkbox to prevent old graphical traces from being overwritten.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Random updates are particularly useful in investigating the robustness of a filter to element errors.

Net Lists

This chapter contains the following topics:

- Net Lists

Switched Capacitor netlists

To help retrieve the designed active circuits, FilterSolutions will output you filter in a netlist. From the control bar of any displayed active circuit, select **Netlist**, and the Netlist box will be displayed on the left side of the filter window. The number of significant digits used on the netlist will match the number in the "Digs" box on the right side of the controls bar. The netlist contains entries for Transient analysis with a Pulse source, including graph scaling. Pulse sources are also used to simulate the positive and negative clocks for the switched capacitors. No AC analysis is available for switched capacitor filters.

Component Indexing

netlist component indexes use three or four digits. The first two digits represent the filter stage the component is in, the third and fourth digit represents the index within that stage. This simplifies the task of finding a specific component within a filter netlist. Component values may be copied directly to the window clipboard from the netlist.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Switched Capacitors

Models of switched capacitors are created and used. The calling of the model uses the capacitance value and the positive and negative clock nodes. A separate model exists for positive switched capacitors and negative switched capacitors.

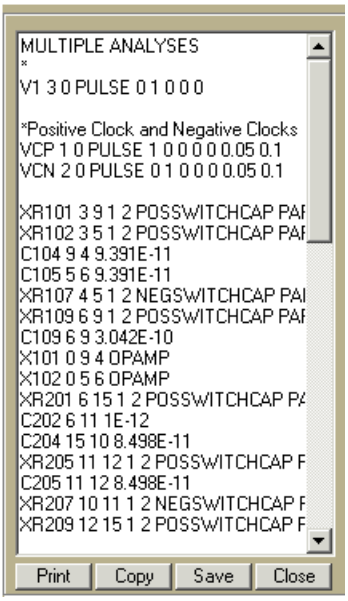
Op Amps

All opamp models are ideal opamps. The gain is set to a very large value to simulate the infinite gain of an ideal opamp.

Dummy Resistors

To insure a DC path to ground, dummy resistors of very high impedance are occasionally inserted into the netlist. They are not expected to hinder the simulation performance in any way.

The following is an example of a FilterSolutions active netlist:



Note that he bottom parts of the netlist are hidden below the text window. The scroll bar may be used to scroll up and down the netlist to observe these hidden parts.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Print	Sends the netlist to a printer.
Copy	Copies all or part of the netlist to the windows clipboard. Partial text may be copied by highlighting the desired text. All text may be copied by selecting all the text or none of the text.
Save	Saves the netlist to a text file.
Close	Closes the netlist window.

Exporting to Other Software

This chapter contains the following topics:

- Touchstone Files

Touchstone Files

You may easily export designed filters to Y or Z parameter Touchstone files that me be imported into other circuits analysis programs. Each filter schematic contains an **Export** function in the upper left that, when selected, offers Touchstone Z and Y parameter file exports.

Circuits analysis programs usually include an element called "N-Port", "Ideal element", etc. that have the capability to read or import Touchstone files that contain Z or Y parameter data, and effectively model the designed filter when they read a FilterSolutions generated Touchstone file.

Port Definitions

2 port filters:

Port 1 = Input.

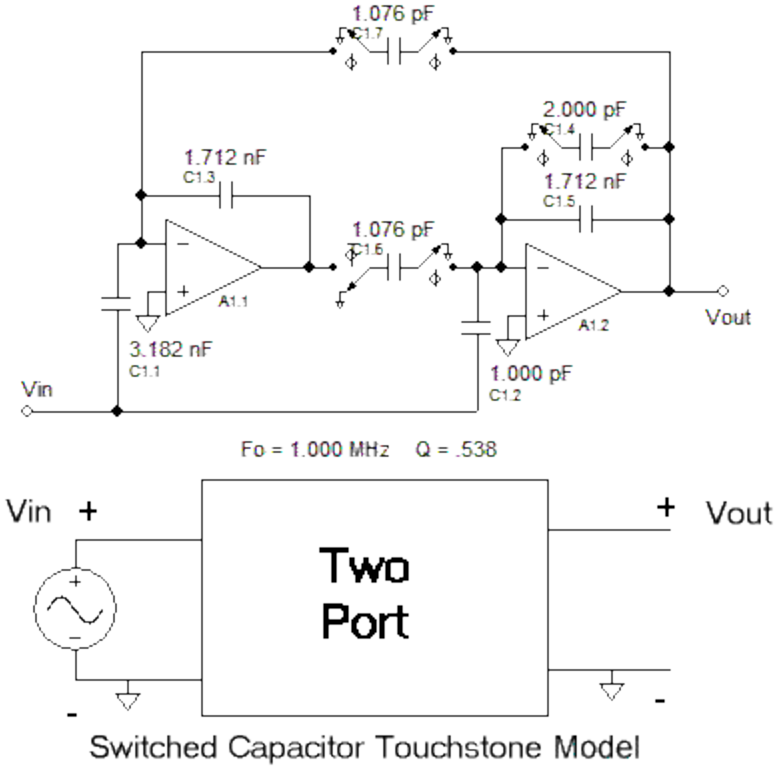
Port 2 = output

Tips:

- Z and Y parameters model only the filter network, and do not include any source or load terminations.
- Beginning and ending frequencies are determined by the FilterSolutions Min Freq and Max Freq entries.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Illustrations:



Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16 - Digital Filters

This chapter contains the following topics:

- IIR Filters
- FIR Filters
- Z Transform Output
- Standard Cascade, and Parallel Forms
- Finite Precision Analysis
- Digital Filter Construction
- Sin(x)/x Sampling Compensation
- Generate Automatic C Code
- Digital Overview Video

IIR Filters

This chapter contains the following topics:

- Digital Transformations
- Frequency Prewarping
- FIR Approximations of Analog Filters
- Multiple Point
- Broadband Matching

Digital Transformations

The answer to which digitization technique to use depends on the application. None of the techniques are perfect, but in general the Bilinear transformation maps the entire frequency response of the desired filter, the Impulse Invariant transformation preserves the impulse time response, and the Step Invariant preserves the step time response. So, if you need frequency performance, such as filtering a variable in a dynamic frequency sensitive environment, the best bet is to use the Bilinear transformation. If you need time performance, such as modeling the time response of a disturbance to a plant, the best bet is to use the Impulse Invariant or Step Invariant transformation. The matched Z transform is very basic and has no distinguishing features. When in doubt, go with the Bilinear transformation

Derivations

The derivations of the types of transformations are as follows:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Bilinear Transformation:

The S in the continuous transfer function is directly substituted with the Z expression below to create the Z transform.

$$S => \frac{2}{T}\frac{Z-1}{Z+1}$$

Where T is the sample period.

Matched Z Transform

This simple transformation transforms the continuous poles and zeros with the following transformation:

$$S + A => \frac{Z - e^{-AT}}{Z}$$

Where:

T is the sample period

A is the pole or zero of the continuous transfer function

The gain must be set to match the continuous gain.

Impulse Invariant Transformation:

The Impulse Invariant transformation preserves the impulse response of the continuous filter. The continuous transfer function is broken down into a sum of residues. Each residue term in the continuous transfer function is transformed as shown below. The transformed residues are summed together to create the Z transform.

$$\frac{K}{S + A} => \frac{KTZ}{Z - e^{-AT}}$$

Where T is the sample period

A is the pole in the residue.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Step Invariant Transformation

The Step Invariant transformation preserves the step response of the continuous filter. The Step Invariant transformation does this by performing an Impulse Invariant transformation on the step response of the continuous transfer function, and then dividing the step back out of the resulting Z transform by multiplying by the Z transform derivative function, (Z-1)/TZ.

$$F(Z) = \frac{Z}{Z-1} Z_{II}\left(\frac{F(S)}{S}\right)$$

Where

Zii() is the Impulse Invariant Transformation Function

F(S) is the continuous transfer function

F(Z) is the Step Invariant discrete transfer function

Modified Impulse and Step Invariant Transformations

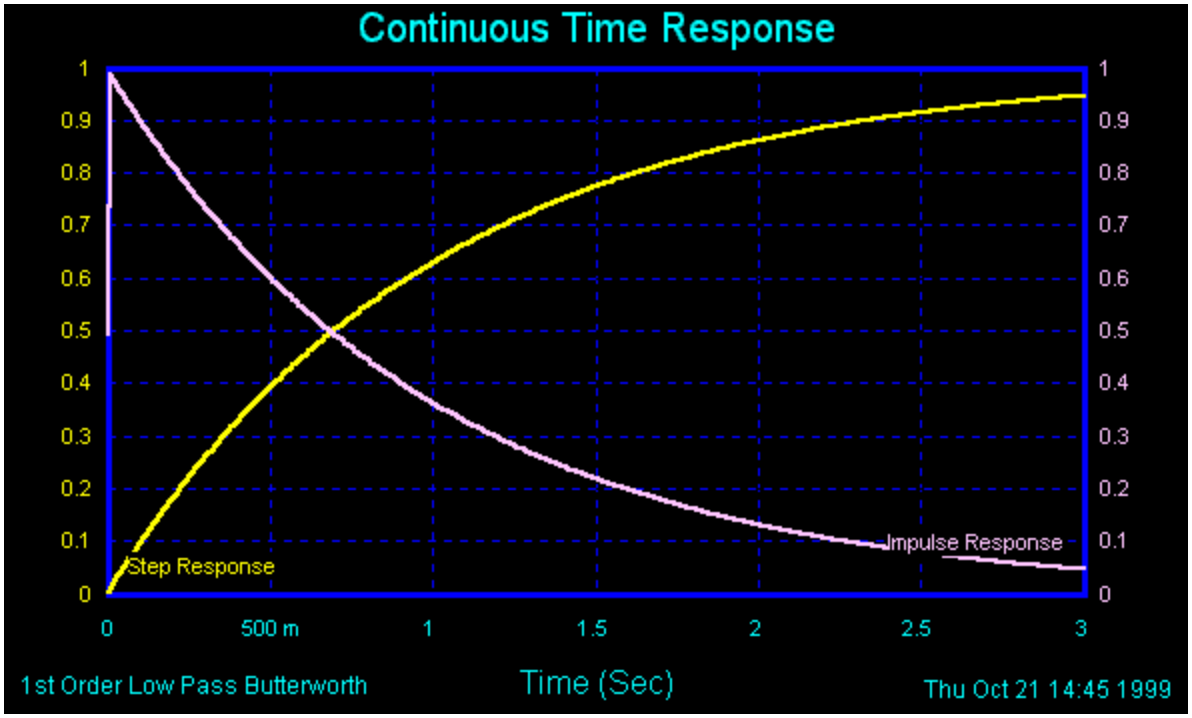
Checking the digital "Modified Invariant" box or selecting the switched capacitor digital entries for the modified transformations produces a modification of the impulse and step invariant transformations. The algorithms mentioned above are executed on the cascaded continuous transfer function instead of the parallel. The result is that some time based error is introduced, but the resulting Z Transform may be more desirable in some applications, such as to minimize capacitor spread in switched capacitor filters.

Illustrations

To illustrate the time characteristics of these digitization techniques, here is the step trace of a first order one radian/second Butterworth filter followed by a Bilinear transformation and an Impulse Invariant transformation sampled at 2 Hz.

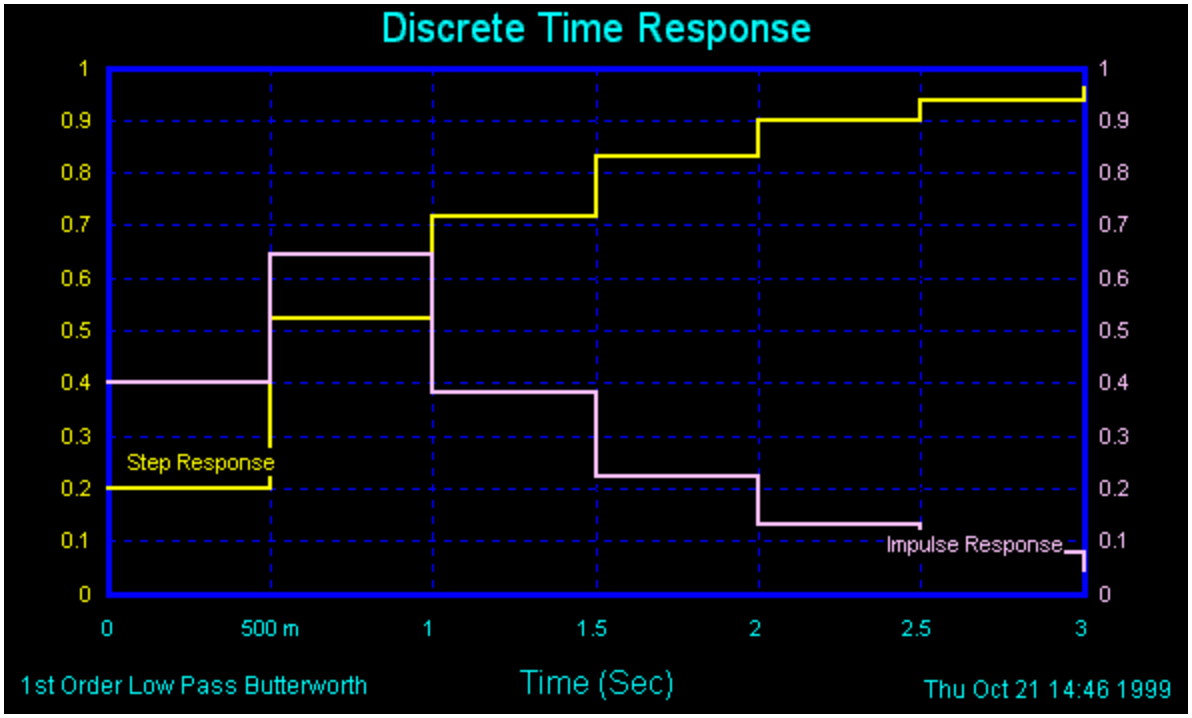
The continuous time response is a nice exponential curve starting at zero and decaying asymptomatically as shown below.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



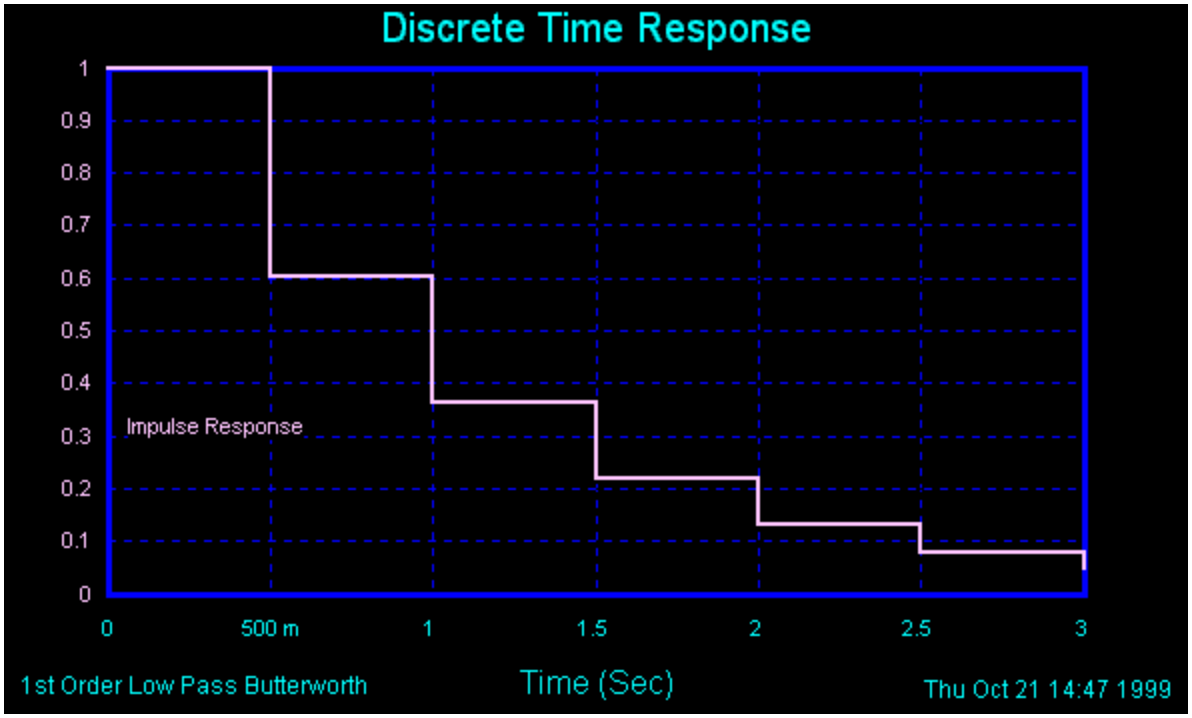
The step response and impulse response of the Bilinear transformation are shown below. Neither of these traces matches the continuous trace very well! The Bilinear maps the frequency spectrum and is not designed to preserve time response.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



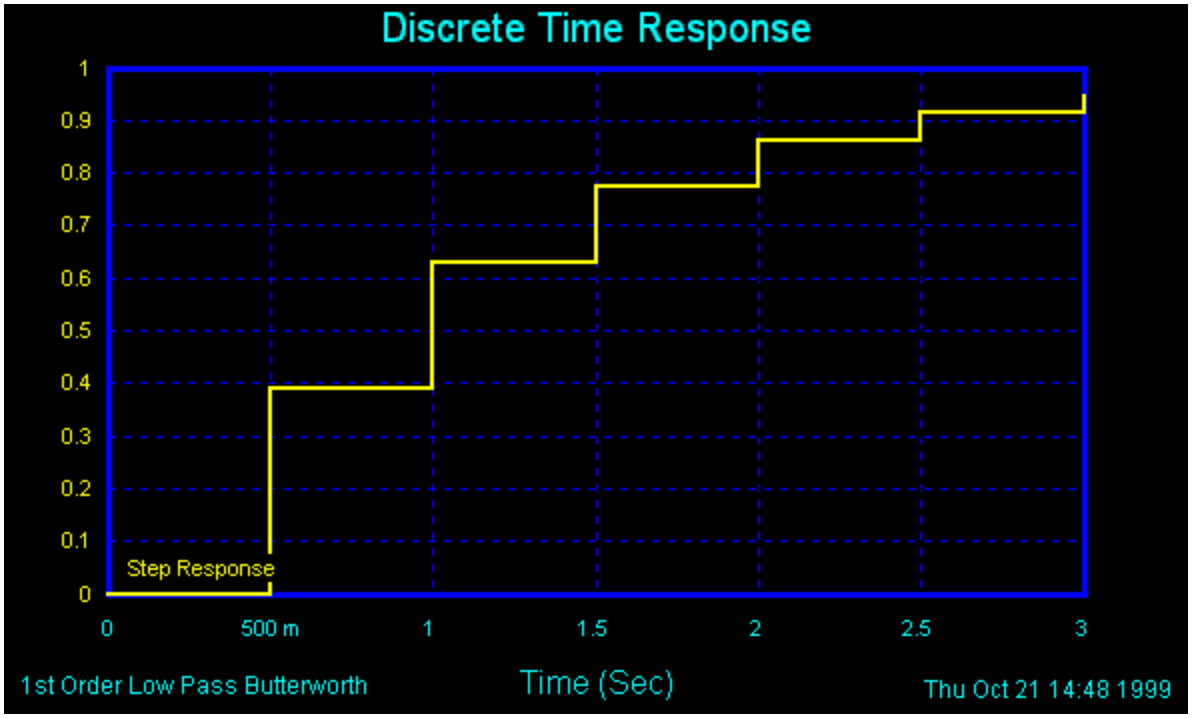
With the Impulse Invariant transformation shown below, the impulse transient response is nicely reproduced. Every sample point matches the continuous impulse trace exactly.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



With the Step Invariant transformation shown below, the step transient response is nicely reproduced. Every sample point matches the continuous step trace exactly.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Prewarping

The discrete sampling of digital filters produces aliasing for frequencies at or exceeding half the sample rate. In the case of the Bilinear transformation, the entire continuous frequency spectrum is mapped to the discrete unit circle. This mapping warps the frequency response and subsequently produces errors

To Correct Warping of the Frequency Axis

For **low pass** and **high pass** filters, it is possible to correct aliasing error at the cutoff frequency by mapping the desired cut off frequency of the Z plane over to the corresponding cut off frequency of the S plane. The easiest way to do this is to modify the sample period T used in the computation of the Bilinear transformation is modified. The modification is:

$$T' = \frac{2}{\omega_c} TAN\left(\frac{\omega_c T}{2}\right)$$

Where:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- T is the sample period
- T' is the modified sample period
- Wc is the filter cut off frequency

For **band pass** and **band stop** filters, both upper and lower corner frequencies must be corrected. To do this, we need the error expression:

$$\omega_s = \frac{2}{T} TAN\left(\frac{\omega_d T}{2}\right)$$

Where:

- T is the sample period
- Ws is a continuous frequency
- Wd is a discrete frequency

Since we know the lower and upper corner frequencies, it is possible to redesign the continuous filter with new upper and lower corner frequencies so that the Bilinear transformed filter has the desired upper and lower corner frequencies.

Prewarping Illustration

The following illustration in Figure 1 shows the effect of the Bilinear transformation warping of the frequency response and prewarping of the continuous frequency response to correct this error on a band pass filter with corner frequencies at 9 and 10Hz. First, the continuous response is shown:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

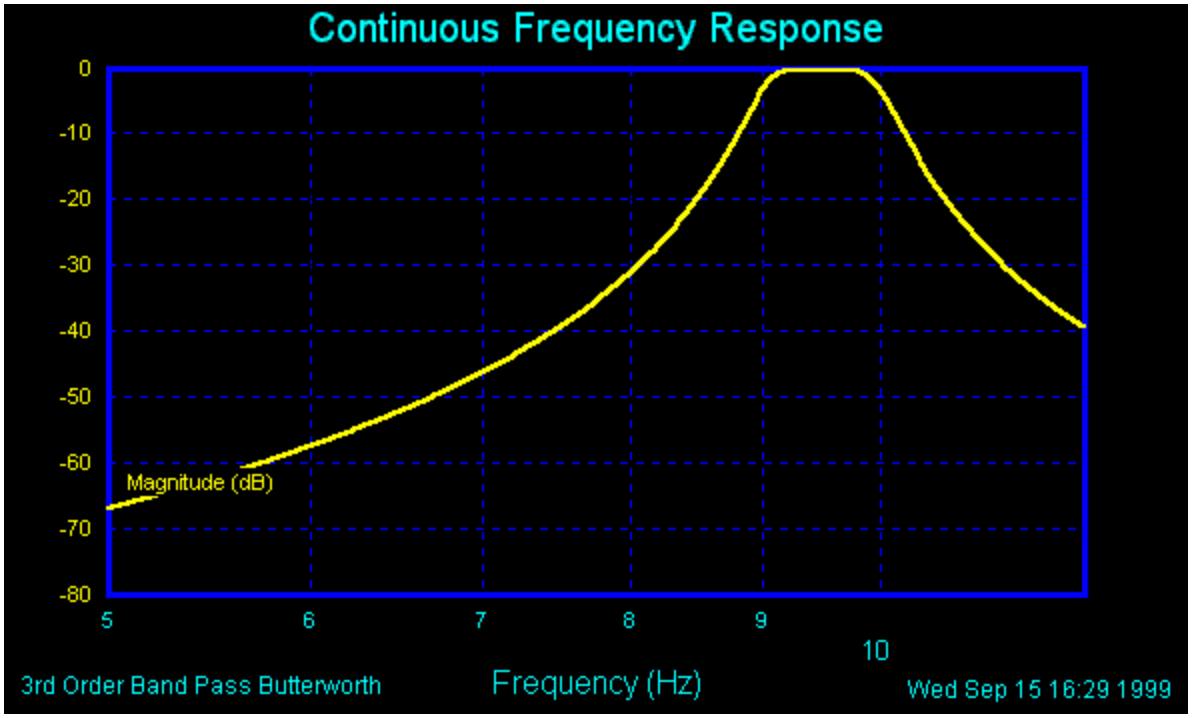


Figure 1: Continuous Band Pass Response

When digitized with a Bilinear transformation, and sampled at 25Hz, the frequency response is warped as shown in Figure 2. Note below that the center frequency is now less than 7 Hz and the band width is about 1/3 Hz. This is a significant error when compared to the original 9 and 10Hz corner frequencies!

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

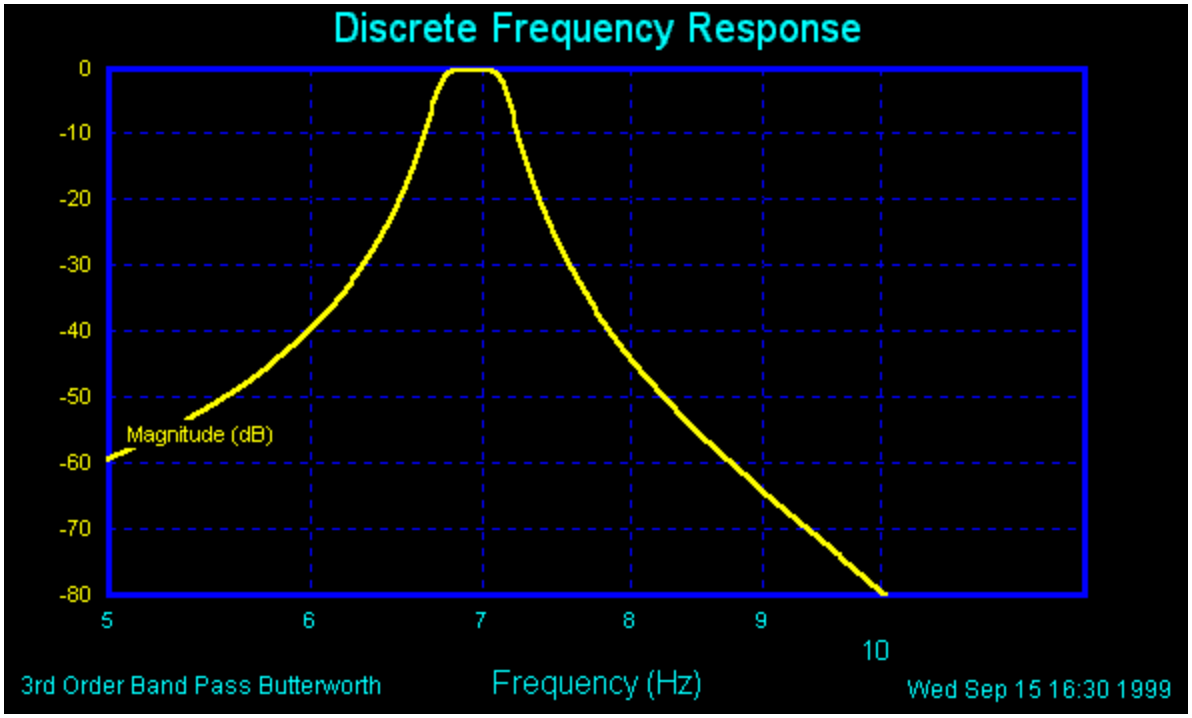


Figure 2: Bilinear Response sampled at 25Kz

By prewarping the continuous frequency response, the original pass band frequency response can be preserved as seen below in Figure 3. Note the corner frequencies are back at 9 and 10Hz where we wanted them!

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

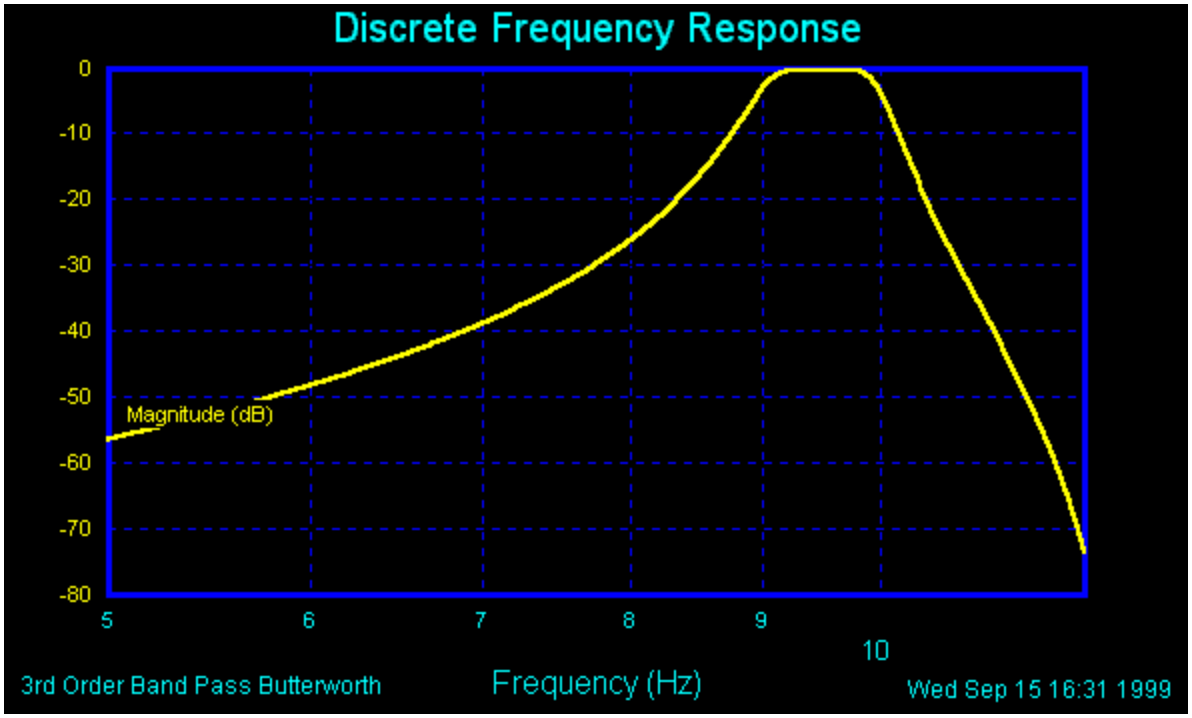


Figure 3: Bilinear Response Prewarped to Maintain Corner Frequencies

Multi-Band Filters

The prewarping option is also available to multi-band filters. With multi-band filters, all low pass, high pass, band pass, and/or band stop corner frequencies are restored to their original position. Below is an example of a low pass multi-band filters with corner frequencies at 1,2,3,4,5,6, and 10 KHz. Figure 4 is the continuous response. Figure 5 is the Bilinear, prewarped response sampled at 30KHz. It is easily seen that all corner frequencies are preserved.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

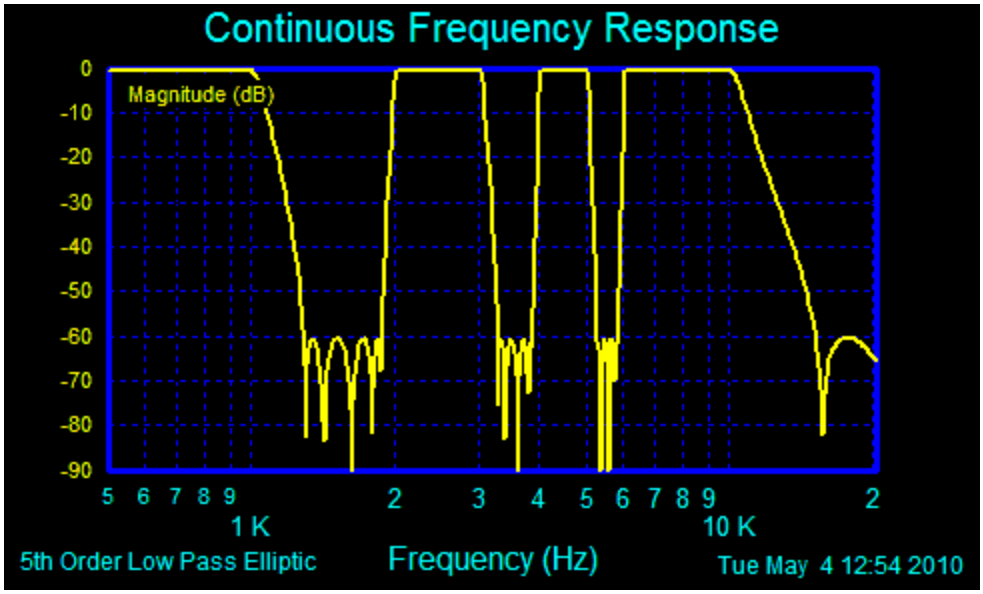


Figure 4: Continuous Low Pass Multi-Band Response

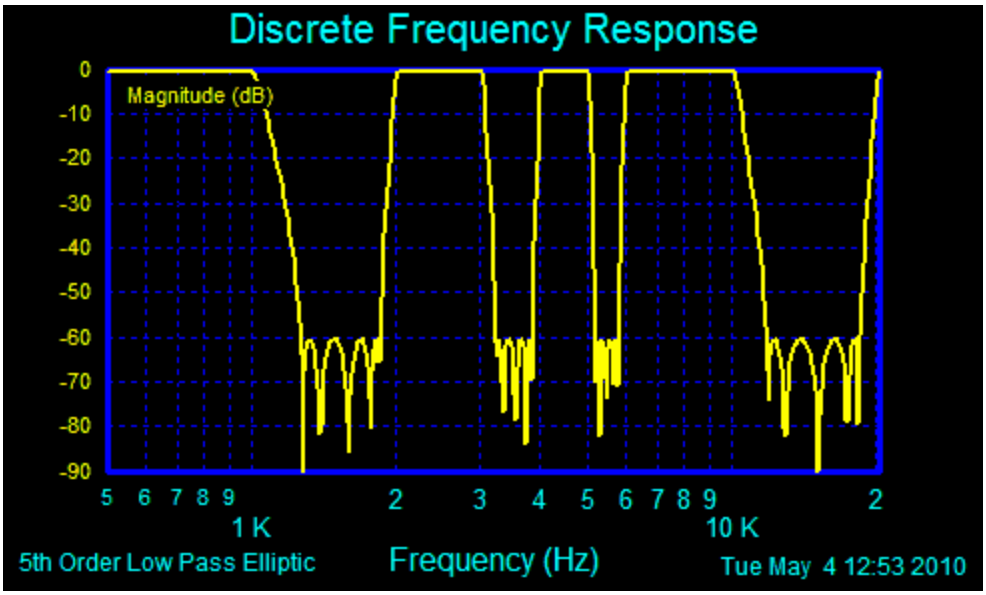


Figure 5: Bilinear with Prewarping Low Pass Multi-Band Response

FIR Approximations of Analog Filters

Any analog filter may be approximated to an arbitrary degree of accuracy with an FIR implementation. This is advantageous when only a limited number of bits for precision are

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

available to create a desired IIR filter. IIR filters, especially those of Standard Form implementation, are more susceptible to finite precision error than FIR filters. IIR filters may go unstable if not enough precision is available to build the desired filter. FIR filters are **always** stable. FIR filter coefficients are always between +1 and -1, which may allow for greater ease when computing with fixed point arithmetic.

The FIR filter is created by directly encoding the sampled analog filter impulse response directly into the FIR Z transform. FilterSolutions allows the entry of the desired sample rate and FIR tap number to create an FIR approximation of any analog filter supported.

Increasing the sample rate has the effect of increasing the digital alias frequency. Increasing the number of taps has the effect of increasing the accuracy of the FIR approximation.

FIR Filters

This chapter contains the following topics:

- General Information
- FIR Window Filters
- Optimization Options
- Iterative FIR Solutions
- FIR Communications Filters
- FIR Matched and Delay Filters
- FIR Approximations of Analog Filters

FIR General Information

FIR, or Finite Impulse Response, filters have the distinctive trait that their impulse response lasts for a finite duration of time as opposed to IIR, or Infinite Impulse Response, filters whose impulse response is infinite in duration. This trait is due to the fact that there are no feedback paths in the filter. All Z transform terms are in the numerator. Z transforms written in the Z**-1 notation only contain a "1" in the denominator. Z transforms written in the Z**+1 notation, as in FilterSolutions, only have a Z**N in the denominator, where N is the number of states in the filter.

FIR filters offer several advantages over IIR filters:

- Completely constant group delay throughout the frequency spectrum.
- Complete stability at all frequencies regardless of the size of the filter.

FIR filters also come with some disadvantages as well:

- The frequency response is not as easily defined as it is with IIR filters.
- The number of states required to meet a frequency specification may be far larger than that required for IIR filters.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FilterSolutions allows up to 10,000 taps to create the FIR filter.

FIR Window Filters

General Information

The FIR filters Rectangular, Bartlett, Hanning, Hamming, Blackman, Blackman-Harris, Kaiser, and Dolph-Chebyshev are all Window FIR filters. The name "Window" comes from the fact that these filters are created by scaling a sinc (SIN(X)/X) pattern with a window to produce the desired frequency effect. All FilterSolutions FIR Window filters create the same sinc pattern, which is the Fourier Transform of a square pulse. The FIR selection then determines the window to use to scale the sinc pattern and thereby create the desired filter. If you look closely at the windows shown below, they are all similar, but unique. The Kaiser and Dolph-Chebyshev are adjustable. It is the uniqueness of each window that creates a unique FIR Window filter.

Types of Windows

Rectangular	Constant, Essentially no window
Bartlett	Triangular window W(n) = 1.-2*abs(n-N/2.)/(N+2)
Hanning	W(n) = [1-COS(2PI * n/(N-1))]/2
Hamming	W(n) = .54 - .46*COS(2PI * n/(N-1))
Blackman	W(n) = .42 - .50*COS(2PI * n/(N-1)) + .08*COS(4PI * n/(N-1))
Blackman-Harris	W(n) = .35875 .48829*COS(2PI * n/(N-1)) + .14128*COS(4PI * n/(N-1)) -

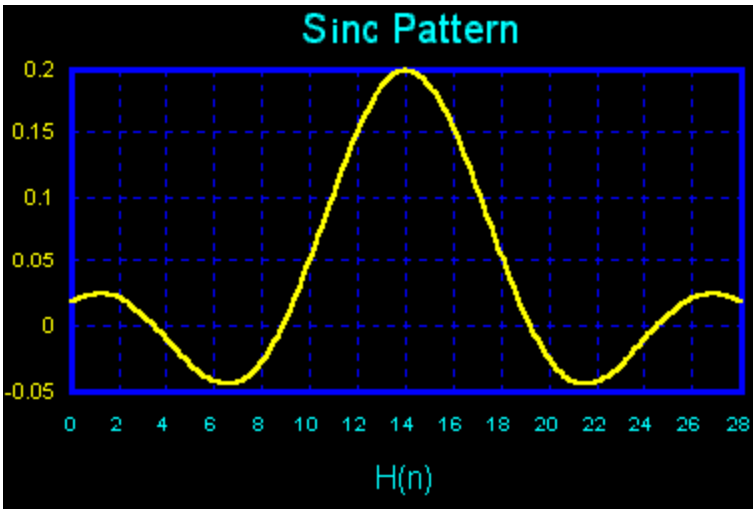
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	.01168*COS(6PI * n/(N-1))
Kaiser	Let: Ka = Kaiser Constant Na = (N-1)/2 Wa = Ka/Na W(n) = Io[Wa*SQRT(Na**2 - (n-Na)**2)] / Io(Ka) Where Io() is the zeroth order Hyperbolic Bessel function
Dolph-Cheby	Let: X = desired stop band attenuation in dB B = COSH[(1/N)*ACOSH(10**[X/20])] Dn = COS[N*ACOS(B*COS[PI*k/N])] Dd = COSH[N*ACOSH(B)] W(k) = [(-1)**k] * (Dn/Dd) W(n) = FIr[W(k)] Where: FIr[] is the real part of the Inverse Fourier Transform The argument of the ACOS may exceed unity. Therefore, complex arithmetic applies for all Dolph-Cheby calculations

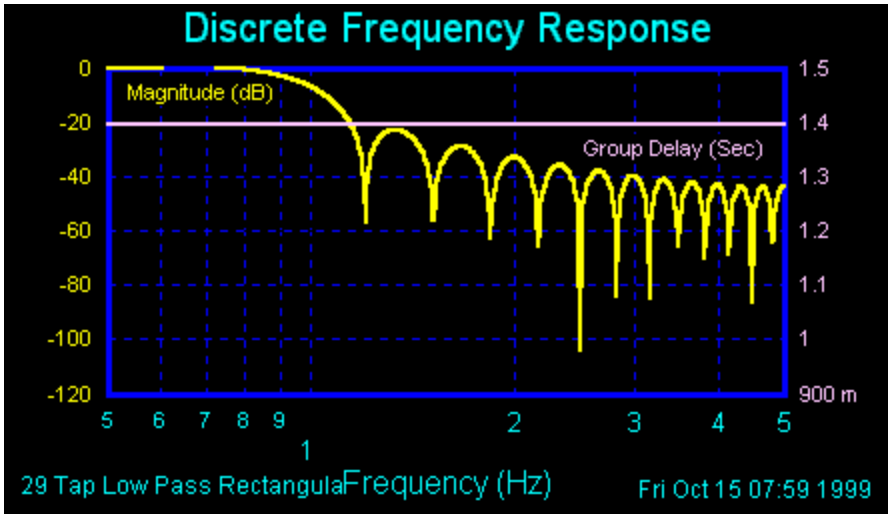
Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Illustration of FIR Construction

This is a sinc pattern for the 29 tap low pass filter below. It is used in the initial step of creating the FIR filter. The number of taps is the order of the filter plus 1 to account for the 0 term. The value of the taps at the sinc curve is scaled by the window and then is installed directly into the Z transform numerator.

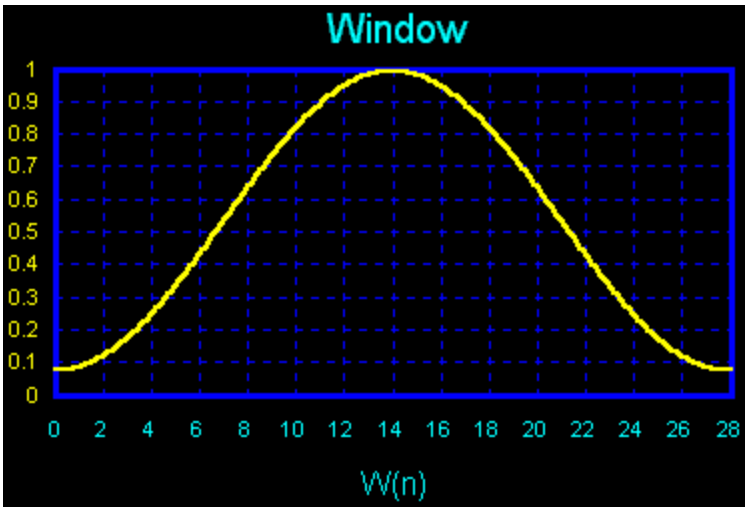


The frequency response of the Sinc pulse Z transform appears in the picture below. Note the constant group delay.

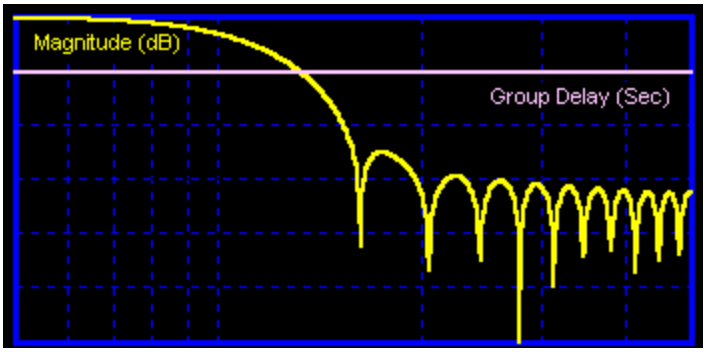


Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A typical window that scales the sinc pulse looks like this:



A window has the effect of improving the stop band performance as shown below:



FIR Optimization Options

High pass, band pass, and band stop FIR filters are all derived from low pass FIR filters. The translation from the low pass filter provides some flexibility in optimizations for the final filter design. Two optimization options are provided and may be selected in the FIR control panel as Option 1 or 2. Neither option is considered better or worse. The requirements of the filter application will determine which one to use.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FIR Optimization Option 1

This option optimizes high pass and band pass filters for full attenuation at zero hertz. The trade-off for this less attenuation between zero hertz and the high pass cutoff frequency or lower band pass cutoff frequency.

FIR Optimization Option 2

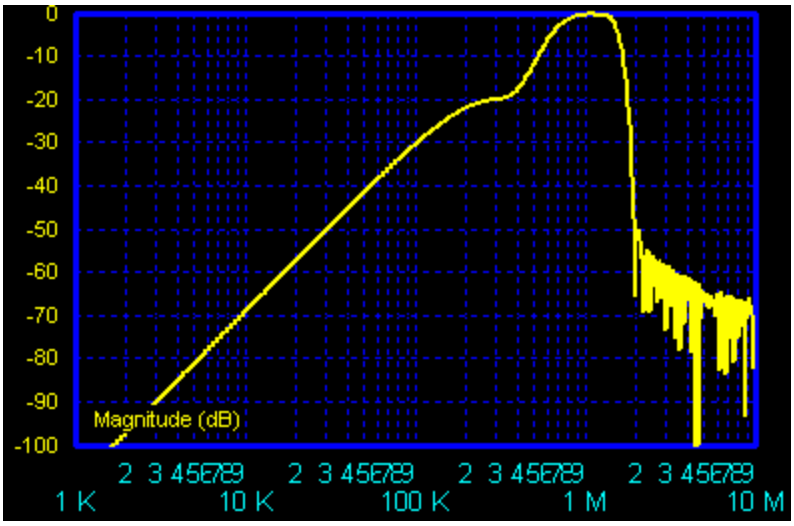
This option optimizes high pass and band pass filters for more attenuation at finite frequencies below the high pass cutoff frequency or lower band pass cutoff frequency. The tradeoff for this is incomplete attenuation at zero hertz.

Band Stop Filters

Band stop filters are affected in that they may be derived using synthesis techniques from either Option 1 or Option 2. Each option produces a slightly different frequency trace.

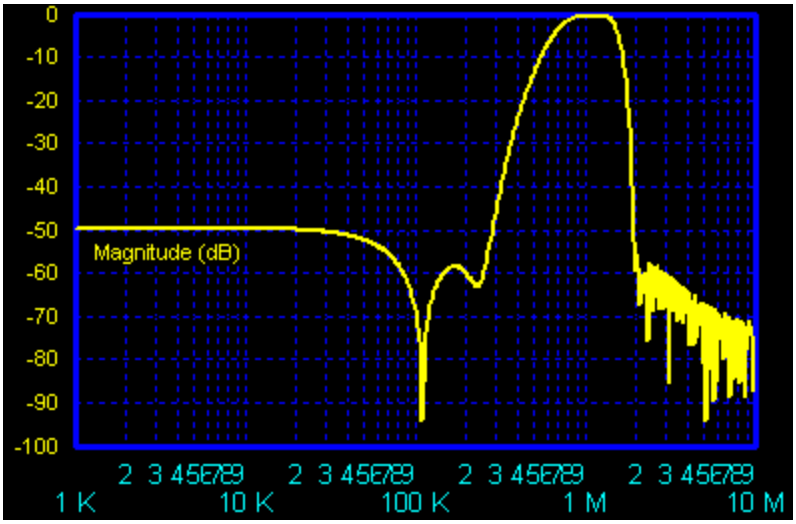
Example:

Below is an example of a Hamming band pass filter with a center frequency of 1 MHz and a bandwidth of 1 MHz. The sample rate is 20MHz, and the number of taps is 100. It is easily seen the Option 1 attenuation approaches infinity as the frequency approaches zero, and that the Option 2 attenuation is superior for finite frequencies of the stop band.



FIR Optimization 1: Infinite Attenuation at DC

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

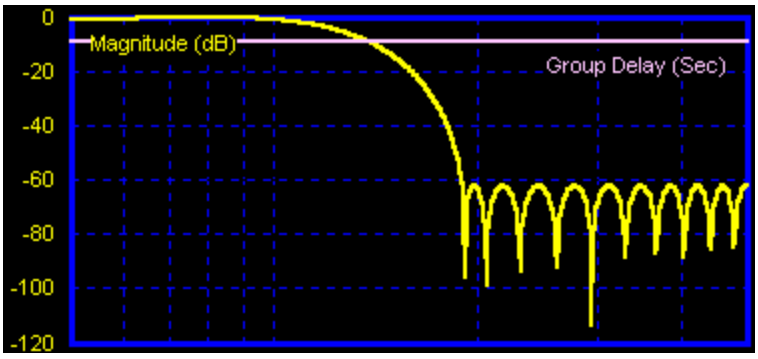


FIR Optimization 2: Increased Attenuation in the Finite Stop Band

Iterative FIR Solutions

FilterSolutions offers the Remez filter as an iterative solution FIR filter. The Remez is not a window filter, but is generated with iterative error-reducing algorithms designed to reduce the passband error. In addition to allowing stop band ratio and frequency definition, the Remez filter allows the "Ripple Ratio" to be defined as well. The ripple ratio is defined to be the arithmetic passband ripple divided by twice the arithmetic stop band ripple. The Remez filter always contains an odd number of taps. If an even number is entered, FilterSolutions will reduce the tap count by one, and create the filter with an odd number of taps.

The following frequency trace is a Remez filter with 27 taps, 1Hz pass band frequency, ripple ratio of 50, and stop band frequency of 1.9 Hz.



Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FIR Communications Filters

The FIR filters of Raised Cosine, Root Raised Cosine, Cosine, and Sine are generally for use in communications applications, and supported only in the low pass form.

Raised Cosine Filter

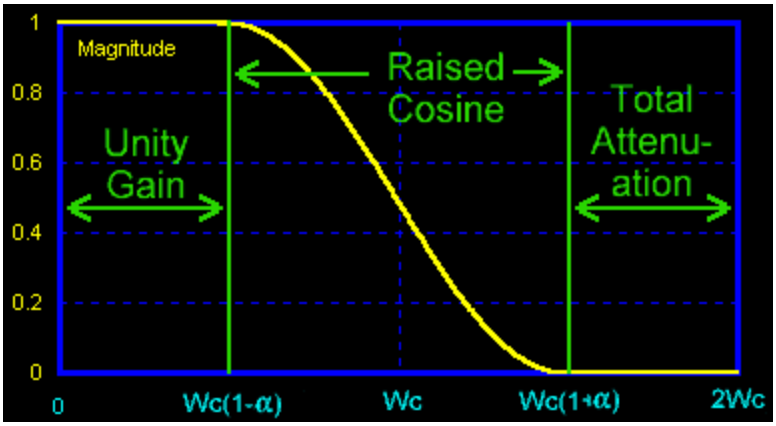
The ideal raised cosine filter frequency response consists of unity gain at low frequencies, a raised cosine function in the middle, and total attenuation at thigh frequencies. The width of the middle frequencies are defined by the roll off factor constant Alpha, (0<Alpha<1). In FilterSolutions, the pass band frequency is defined as the 50% signal attenuation point.

When the pass band frequency of a raised cosine filter is set to half the sample rate, then the impulse response Nyquist's first criteria is satisfied in that the impulse response is zero for T = NTs, where N is an integer, and T is the sample rate.

Mathematically, the frequency response may be written as:

$$F(\omega) = \begin{cases} 1 & \text{For } \omega < \omega_c(1-\alpha) \\ 0 & \text{For } \omega > \omega_c(1+\alpha) \\ \dfrac{1 + COS\left(\dfrac{\pi(\omega - \omega_c(1-\alpha))}{2\alpha\omega_c}\right)}{2} & \text{For } \omega(1-\alpha) < \omega_c < \omega(1+\alpha) \end{cases}$$

The ideal raised cosine filter frequency response is shown below:



Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ideal Raised Cosine Frequency Response

Root Raised Cosine Filter

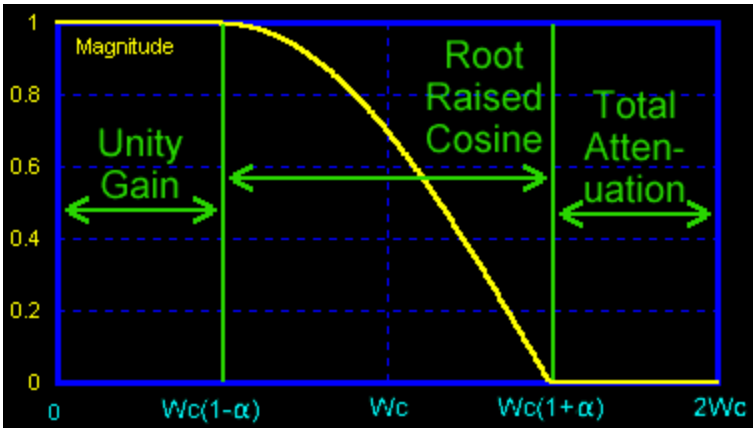
The ideal root raised cosine filter, frequency response consists of unity gain at low frequencies, the square root of raised cosine function in the middle, and total attenuation at thigh frequencies. The width of the middle frequencies are defined by the roll off factor constant Alpha, (0<Alpha<1). In FilterSolutions, the pass band frequency is defined as the .707 half power point.

The root raised cosine filter is generally used in series pairs, so that the total filtering effect is that of a raised cosine filter.

Mathematically, the frequency response may be written as:

$$F(\omega)=\begin{cases}1 & \text{For } \omega < \omega_c(1-\alpha)\\ 0 & \text{For } \omega > \omega_c(1+\alpha)\\ \sqrt{\dfrac{1+COS\left(\dfrac{\pi(\omega-\omega_c(1-\alpha))}{2\alpha\omega_c}\right)}{2}} & \text{For } \omega(1-\alpha) < \omega_c < \omega(1+\alpha)\end{cases}$$

The ideal root raised cosine filter frequency response is shown below:

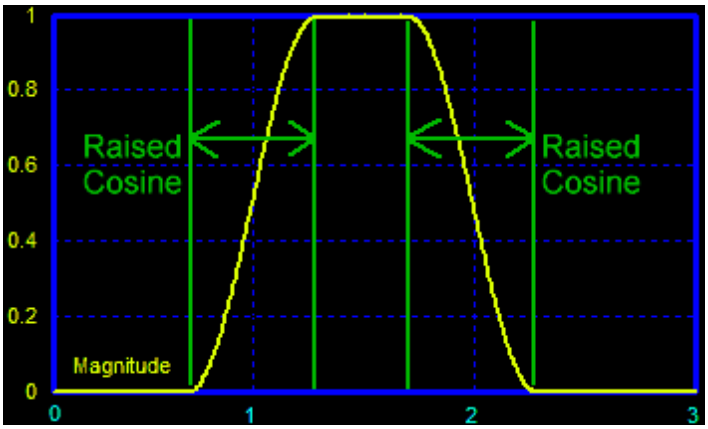


Ideal Root Raised Cosine Frequency Response

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Band Pass Raised and Root Raised Cosine Filters

Band pass raised cosine filters and root raised cosine filters are modified version of low pass raised cosine and root raised cosine filters where the low frequencies are also filtered out with the use of a raised cosine or root raised cosine response, which creates a band pass filter. **The alpha entry determines how flat the top of the pass band is, 1 for full round, 0 for brick wall**.



Ideal Band Pass Raised Cosine Response (Alpha = 0.5)

Cosine Filters

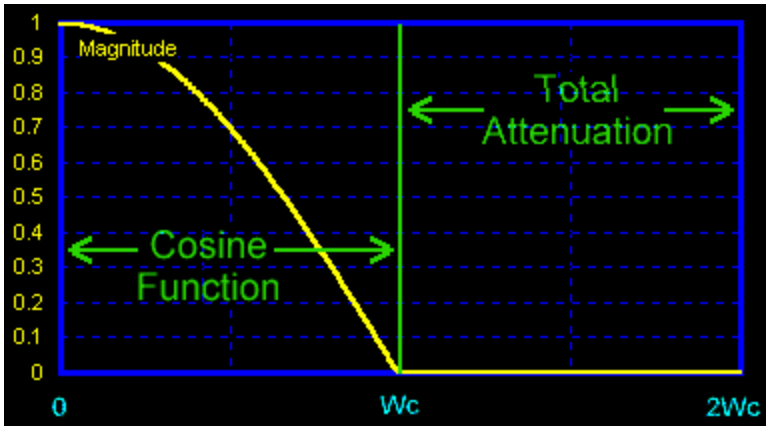
The ideal cosine filter, sometimes referred to as a duobinary filter, frequency response consists of a cosine function at low frequencies, and total attenuation at thigh frequencies. In FilterSolutions, the pass band frequency is set to the infinite attenuation point. Nyquist s first criteria may be satisfied by the cosine filter when used in conjunction with bipolar signaling.

Mathematically, the frequency response of the cosine filter may be written as:

$$F(\omega) = \begin{cases} 0 & \text{For } \omega > \omega_c \\ COS\left(\frac{\pi\omega}{2\omega_c}\right) & \text{For } \omega < \omega_c \end{cases}$$

The ideal cosine filter frequency response is shown below:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ideal Cosine Filter Frequency Response

Sine Filters

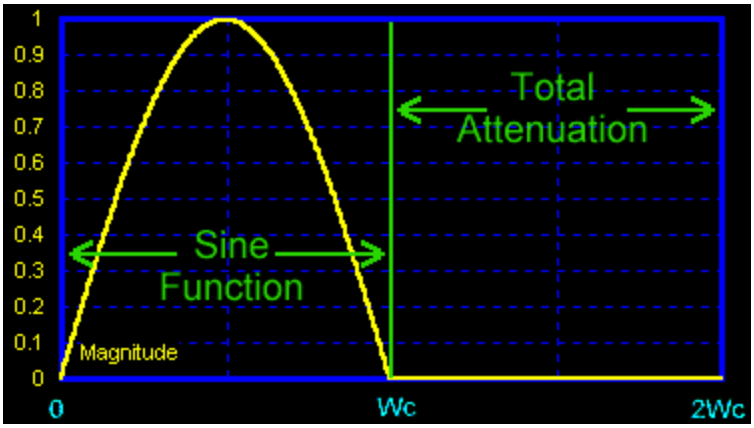
The ideal sine filter frequency, sometimes referred to as a modified duobinary filter, response consists of a sine function at low frequencies, and total attenuation at thigh frequencies. In FilterSolutions, the pass band frequency is set at the infinite attenuation point. Nyquist s first criteria may be satisfied by the sine filter when used in conjunction with bipolar signaling. The sine filter has the advantage over a cosine filter in that the PSD is null at DC.

Mathematically, the frequency response of the cosine filter may be written as:

$$F(\omega) = \begin{cases} 0 & \text{For } \omega > \omega_c \\ SIN\left(\frac{\pi\omega}{\omega_c}\right) & \text{For } \omega < \omega_c \end{cases}$$

The ideal sine filter frequency response is shown below:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ideal Sine Filter Frequency Response

FIR Matched and Delay Filters

FIR Matched and Delay filters are unique in that they are defined with their time domain traits as opposed to their frequency domain traits. The required update rate is calculated by FilterSolutions to generate the filter design.

Matched Filters

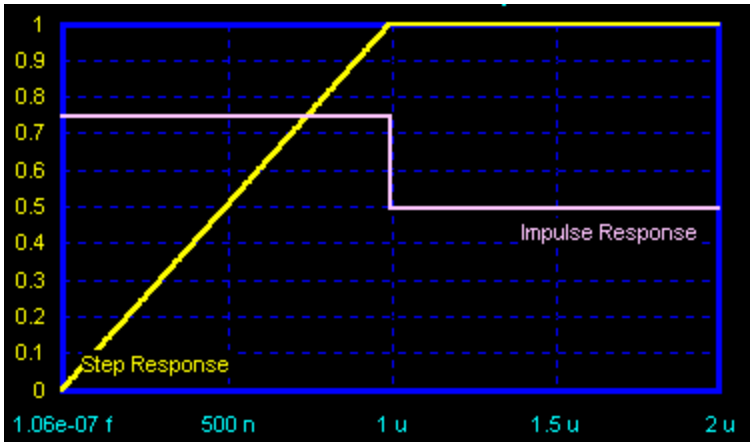
In addition to FIR matched filters, FilterSolutions supports passive, active, and digital IIR matched filters. It is possible, however, to create much more accurate FIR matched filters than those other methods.

The distinguishing characteristic of a Matched Filter is the step response approximating a ramp, and the impulse response approximates a pulse. The purpose of the Matched Filter is to maximize the signal to noise ratio and to minimize the probability of undetected errors received from a signal.

FilterSolutions allows one to define the Matched Filter by setting the rise time of the ramp. **FilterSolutions will automatically set the sample rate to the value needed to create the matched filter with the order and rise time you define**. The proper use of the matched filter is to set the rise time to be equal to the pulse width of the pulses in a bit stream.

The ideal matched filter step and impulse response is shown below.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Ideal Matched Filter Time Response

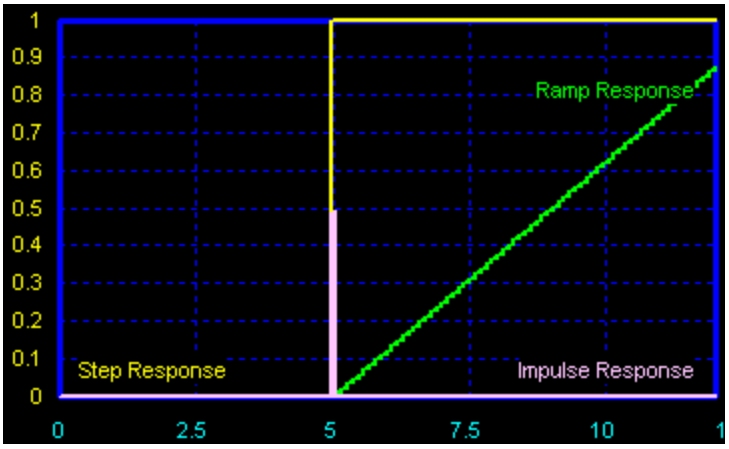
Delay Filters

In addition to FIR delay filters, FilterSolutions supports passive, active, and digital IIR delay filters. It is possible, however, to create much more accurate FIR delay filters than those other methods.

The distinguishing characteristic of a delay Filter is time response is undistorted. The input signal undergoes only a pure time delay.

FilterSolutions allows one to define the Delay Filter by setting the delay time. **FilterSolutions will automatically sets the sample rate to the value needed to create the delay filter with the order and delay time you define**.

The ideal delay filter step, ramp, and impulse response is shown below:



Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ideal Delay Filter Time Response

FIR Approximations of Analog Filters

Any analog filter may be approximated to an arbitrary degree of accuracy with an FIR implementation. This is advantageous when only a limited number of bits for precision are available to create a desired IIR filter. IIR filters, especially those of Standard Form implementation, are more susceptible to finite precision error than FIR filters. IIR filters may go unstable if not enough precision is available to build the desired filter. FIR filters are **always** stable. FIR filter coefficients are always between +1 and -1, which may allow for greater ease when computing with fixed point arithmetic.

The FIR filter is created by directly encoding the sampled analog filter impulse response directly into the FIR Z transform. FilterSolutions allows the entry of the desired sample rate and FIR tap number to create an FIR approximation of any analog filter supported.

Increasing the sample rate has the effect of increasing the digital alias frequency. Increasing the number of taps has the effect of increasing the accuracy of the FIR approximation.

Z Transform Output

This chapter contains the following topics:

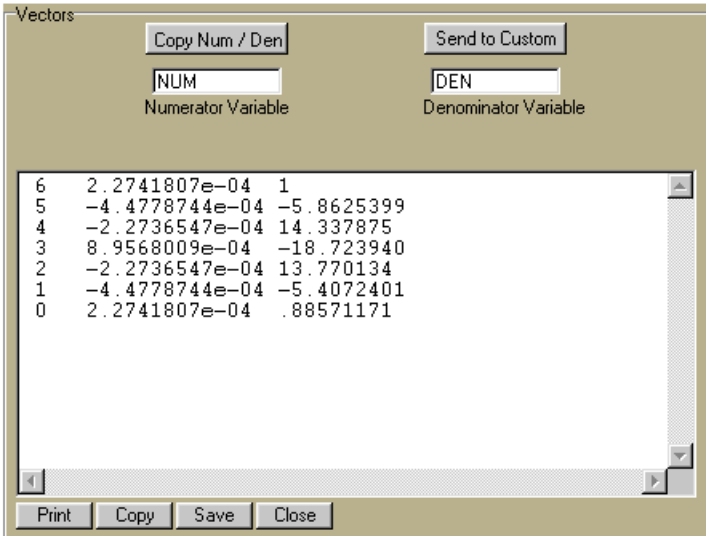
- Vector Outputs
- Digital Formats
- Single Point
- Multiple Point
- Broadband Matching

Z Transform Vectors

FilterSolutions provides an easy means to retrieve the Z transform vectors from a text window. The vector form, Standard, CasCade, and Parallel, is preserved. The vector text window appears directly on the Z transform.

For the Standard form, the following style of vector window appears:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

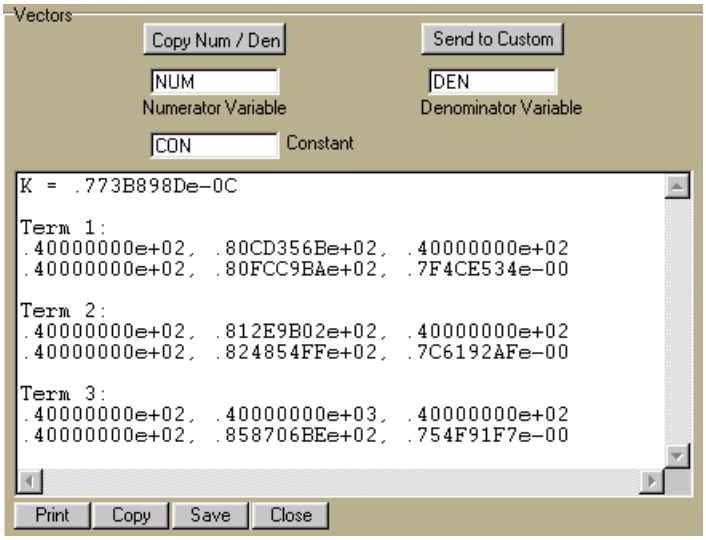


Copy Num/Den	Translates the Z transform vectors to Z**-1 form, which is compatible with Matlab, then places them in the windows clipboard. NUM and Den are the Matlab numerator and denominator variable names.
Send to Custom	Installs the Z transform into the Custom window using the same format as selected in the Custom form. This allows staged filters to be constructed. Look under Custom filters and Staged filters for more information on this subject.

The columns of data from left to right represent the coefficient power, numerator coefficient, and denominator coefficient respectively. The digital format of the Z transform, hex, decimal, signed, etc. is maintained.

For Cascade and Parallel forms, the following style window appears:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Copy Num/Den	Formats the Z transform vectors to a form compatible with Matlab, then places them in the windows clipboard. NUM and Den are the Matlab numerator and denominator variable names. FilterSolutions will index NUM and DEN with an integer to maintain biquad uniqueness. CON is the name of the variable containing the fixed constant, which is a product for Cascade form and is a sum term for Parallel.
Send to Custom	Works as discussed in the preceding paragraphs.

In the text window, K is the constant term, followed by all the biquads. Each biquad is represented as a ratio of first or second order polynomials is order of descending powers.

Button Selections

Print	Sends the contents of the text window to the printer.
Copy	Sends the contents of the text window to the Windows clipboard. If only part of the data is to be copied, highlight the data with the cursor and left mouse key, then select **Copy**. To send all text contents to the clipboard, do not highlight any data or highlight all the data, then select **Copy**.
Save	Places the all the text window data into a file of the designer's choosing.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Z Transform Digital Formats

FilterSolutions allows great flexibility in digital output formats. All simulations performed under the "Test" button use the selected digital format to do the tests.

Decimal

The easiest form for visual interpretation is decimal format. Each term of the Z transform is a simple base ten number with the number of significant digits chosen by the user.

Signed Floating Point

Z transform terms are represented in twos complement hex encoded binary floating point. The exponent is an unsigned hexadecimal number that represents the number of left binary shifts of the decimal point.

Unsigned Floating Point

Z transform terms are represented in unsigned hex encoded binary floating point. The exponent is an unsigned hexadecimal number that represents the number of left binary shifts of the decimal point. The sign of each number must be represented by a physical "+" or a "-" to indicate whether the number is positive or negative.

Signed Fixed Point

Z transform terms are represented as a twos complement hexadecimal integer. Signed fixed point notation requires a numerator and denominator scale in addition to the integer. A range selection is available just below the digital format selection box when Signed fixed point is selected. When the integer is set to its maximum value, EFF..., the value of the signed fixed point is the range less the resolution. When the integer is set to its minimum value, 800..., the value of the fixed point is the negative of the range. The resolution is R/[2**(N-1)] where R is the range and N is the significant bits.

Example:

If:

Scale = 4;

Sig Bits=16;

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Then:

7FFF = 3.99987793

0001 = 0.000122070

FFFF = -0.000122070

8000 = -4.00000000

Unsigned Fixed Point

Z transform terms are represented as a unsigned hexadecimal integer. Unsigned fixed point notation requires a numerator and denominator scale in addition to the integer. A range selection is available just below the digital format selection box when Unsigned fixed point is selected. When the integer is set to its maximum value, FFF..., the value of the signed fixed point is the range less the resolution. When the integer is set to its minimum value, 0, the value of the fixed point is zero. The resolution is R/[2**N] where R is the range and N is the significant bits. The sign of each number must be represented by a physical "+" or a "-" to indicate whether the number is positive or negative.

Example:

If:

Scale = 4;

Sig Bits=16;

Then:

FFFF = 3.999938965

0001 = 0.000061035

0000 = 0.000000000

Justification

Z Transforms using significant bits not divisible by 4 present a problem in what to do about left over bit in the Hex numerical display. The default solution is to left justify the significant bits, and place zeros on the right. An alternative selectable solution is to right justify the significant bits and place leading zeros on the left. See the descriptive example below:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6 Significant bits:

Justification: Left Justify Right Justify

Binary: 1101 0100 0011 0101

Hexadecimal D4 35

Standard Cascade, and Parallel Forms

This chapter contains the following topic:

- Standard Cascade, and Parallel Forms

Standard, Cascade, and Parallel Forms

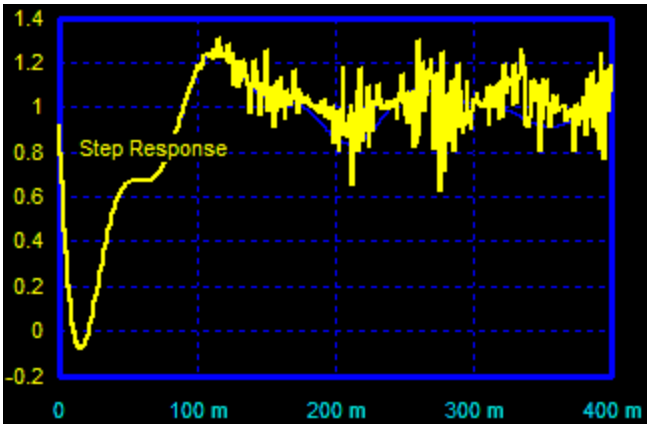
With ordinary 64 bit arithmetic, a low pass or high pass filter should work with the standard form if the order is 10 or less. For a band pass or band stop filter, the standard form should work if the order is 5 or less. For filters larger than this, or with fewer bits of precision available, the cascade or parallel form may be necessary. The reason for this is that although the standard form is simpler and easier to build, it introduces and magnifies rounding errors for large filters. The cascade and parallel forms are more complex, but do not introduce as much rounding error. The rounding error increases as the arithmetic precision of the filter decreases. In general, both the parallel and cascade forms provide more stability due to less pole wandering than the standard form. For Bilinear transformations and Matched Z transformations, the cascade form tends to provide better performance than the parallel form. For Impulse Invariant and Step Invariant transformations, the parallel tends to provide better performance than the cascade form. Large FIR filters using small registers may be improved with the cascade form. **Use the "Test" button in the Digital Transfer Functions page** to view the effects of finite precision error on the filter in the various forms and precisions available.

The parallel form is not available for filters with repeated poles, especially FIR filters.

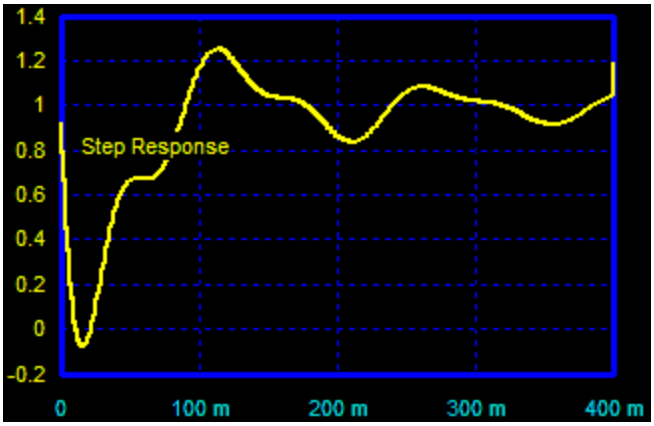
If you are using FilterSolutions autocode to build the filter, it is just as easy to build a parallel or cascade filter as it is a standard filter. Since there is no additional computer overhead to pay, you may as well use the more complex forms whenever you are in doubt about which to use. Filters with repeated poles, such as FIR filters with multiple poles at zero, are not supported in parallel form.

To illustrate these concepts, a 6th order band stop filter step response from a standard form Z transform, 64 bit arithmetic, is shown below.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This filter does not achieve our desired filter effects! The same filter in cascade form is shown below



The cascade form response above represents a significant improvement in filter performance.

Finite Precision Analysis

This chapter contains the following topic:

- Finite Precision Analysis

Finite Precision Analysis

General Information

The digital function selection features on the face of the Main Control Panel assume perfect infinite precision arithmetic. Your digital filter may reside in a low cost processor with very limited

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

precision resulting in computational errors. Therefore, the digital filter you designed may not perform as well in the target processor as it did in FilterSolutions Main Control Panel. To help determine how the designed filter will perform, FilterSolutions allows one to do a finite precision analysis using finite precision arithmetic and different digital forms. The digital Transfer Function display resulting from selecting **Transfer Function** on the Main Control Panel when in digital mode has a button called "Test". **Test** brings up a menu for time response, frequency response, and pole zero plot. All three of these items will perform the indicated task on the displayed Z transform in the same precision and form as it is shown. The ideal digital filter function is displayed in the background for comparison purposes.

If the precision of the target processor is known, you may be estimate what number of base 10 significant digits is equivalent to the target precision. This number should then be placed in the **Sig Dig** box in the upper right of the Transfer Function display. The range of this number must be from 1 to 20. If the digital precision of the processor is known to match one of the digital format selections available in FilterSolutions Z transform page, signed and unsigned floating and fixed point, you may select this format, place the number of bits in the **Sig Bits** box, 1 to 64, and the finite precision analysis will be performed with this exact digital format.

FilterSolutions executes Z transform simulations using the Z transform that is formatted as specified in the digital format selection. Time Test simulations are executed using finite precision arithmetic that matches the selected digital format. Real world IIR filter internal computations may have a different resolutions than FilterSolutions calculates for the Z transform, so **care must be taken in interpreting the IIR fixed point simulation results**.

Of the three form selections available, Standard, Cascade, and Parallel, the Standard form is the least accurate. Errors due to finite precision errors may make the filter go unstable, result in excessive noise, or diminish attenuation. The Cascade form usually works better than the Parallel form for Bilinear transformations and Matched Z transformations, and the Parallel form usually works better than the Cascade form for Impulse Invariant and Step Invariant transformation. However, you since this is not always the case, you should test the filter in both the Parallel and Cascade mode to determine the best form selection. Since FilterSolutions is capable of producing C code for the Cascade and Parallel forms just as easily it can for the Standard forms, the Standard form is not recommended except for very simple filters. FIR filters are not as susceptible to finite precision errors as IIR filters. The Cascade form is supported for FIR filters, but occasionally the Standard form works better in FIR filters.

Finite Precision Error Example

You wish to place a third order band pass Butterworth filter in the target processor that has roughly three digits of precision. The corner frequencies of the filter are at 1 and 2 Hz, and the sample rate is 50 Hz. You have chosen the Bilinear transformation with prewarping. Here is the transfer function for each form.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Standard Form

$$\frac{8.648e\text{-}04*Z^6 - 2.595e\text{-}03*Z^4 + 2.595e\text{-}03*Z^2 - 8.648e\text{-}04}{Z^6 - 5.528*Z^5 + 12.94*Z^4 - 16.42*Z^3 + 11.91*Z^2 - 4.684*Z + .7803}$$

Cascade Form

$$\frac{8.648e\text{-}04\ (Z^2 - 1)\ (Z^2 - 1)\ (Z^2 - 1)}{(Z^2 - 1.823*Z + .8825)\ (Z^2 - 1.941*Z + .9655)\ (Z^2 - 1.764*Z + .9158)}$$

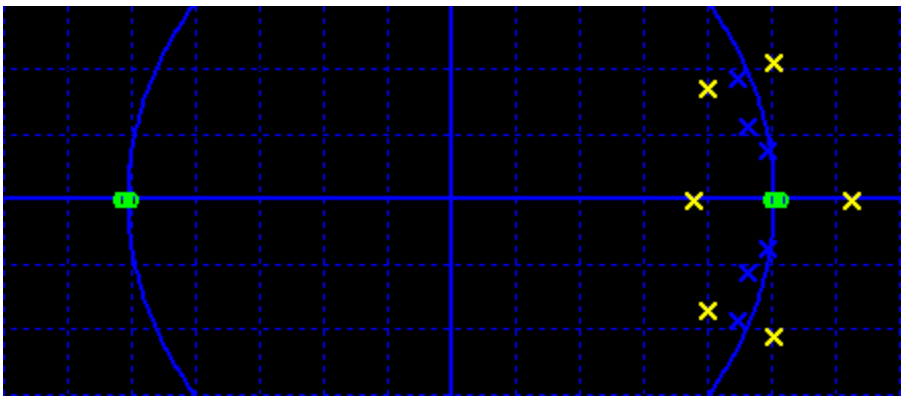
Parallel Form

$$8.648e\text{-}04 + \frac{.1071*Z - .1106}{Z^2 - 1.823*Z + .8825} + \frac{-3.573e\text{-}02*Z + 3.505e\text{-}02}{Z^2 - 1.941*Z + .9655} + \frac{-6.663e\text{-}02*Z + 7.975e\text{-}02}{Z^2 - 1.764*Z + .9158}$$

Pole Zero Plots

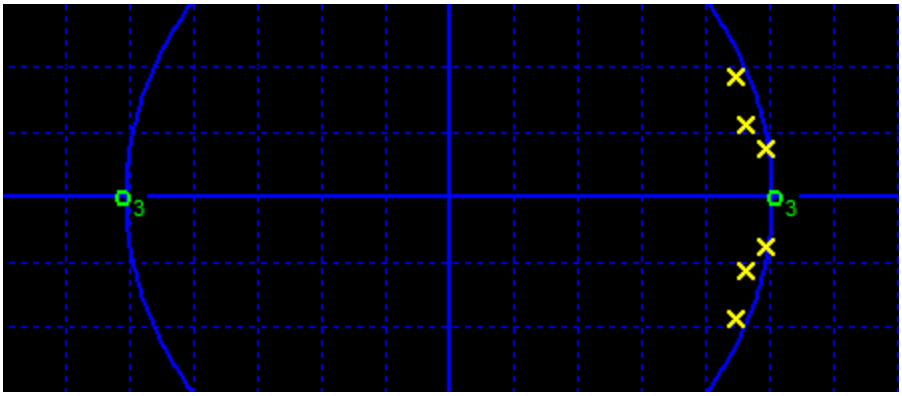
Below are the Pole Zero plots for each form at three significant digits. The dark blue poles and zeros indicate the ideal pole zero placement for this filter The Standard form is totally unacceptable! Three of the six poles reside outside the unit circle indicating a very unstable filter. The cascade form succeeds in placing poles and zeros very close to the ideal location. The Parallel form secedes in positioning the poles properly, but misses some with the. The Cascade form is clearly the ideal choice for this filter.

Similar exercises may be performed using the frequency performance and time performance to determine the best form to use.

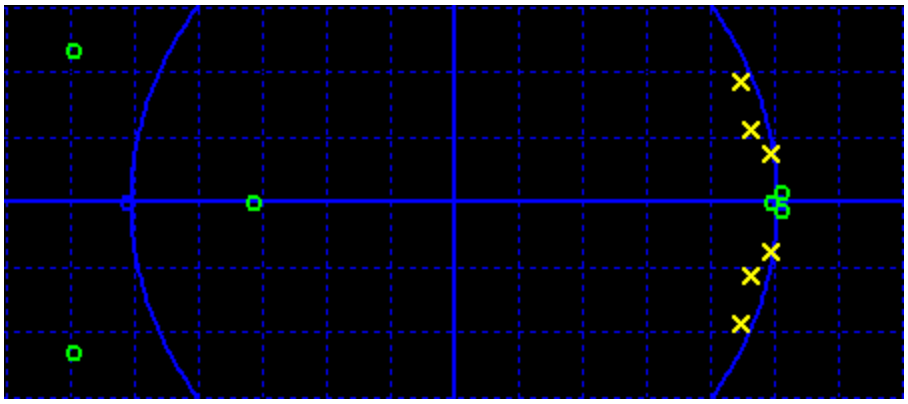


Standard Form Pole Zero Plot

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Cascade Form Pole Zero Plot



Parallel Form Pole Zero Plot

Digital Filter Construction

This chapter contains the following topic:

- Digital Filter Construction

Digital Filter Construction

Deriving Digital Filters From Z Transforms

FilterSolutions provides a Z transform and C autocode to create the digital filter. The following illustrates how to create a digital filter from a Z transform.

The term "Z" refers to inverse past value. Hence Z raised to the -1 power refers to a past value, Z raised to the -2 refers to two past values, etc. Sometimes Z transforms are written in terms of Z to

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the -1 rather than Z. This requires more notation, but makes it easier to visualize the digital filter. The denominator of the Z transform refers to feedback paths. The numerator refers to feed forward paths. Feed forward paths are positive; feedback paths are always negative. For FIR filters, all the feedback terms in the denominator are zero.

The following figures illustrate the conversion of a Z transform from textual to graphical representation for the standard, cascade, and parallel forms. Computer code may be easily written from the graphical implementations.

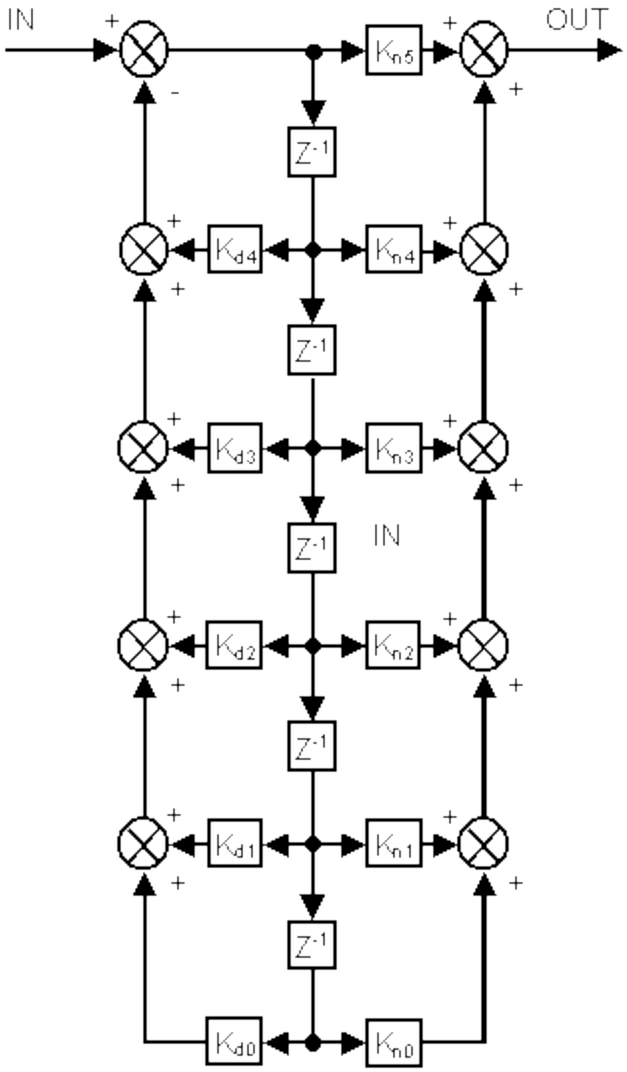
The Standard Form

The general fifth order Standard Z transform is:

$$\frac{K_{n5}Z^5 + K_{n4}Z^4 + K_{n3}Z^3 + K_{n2}Z^2 + K_{n1}Z + K_{n0}}{Z^5 + K_{d4}Z^4 + K_{d3}Z^3 + K_{d2}Z^2 + K_{d1}Z + K_{d0}}$$

A simple graphical implementation for this is:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note that an FIR Z Transform only has a Z**n in the denominator. This translates graphically to zero gains in all the feedback paths above. FIR filters have no feedback paths in their construction. Because there are no feedback paths in an FIR filter, the activity of the filter following the reception of an impulse is gone after n updates of the filter. The impulse response of an FIR filter is therefore finite in length.

Cascade Form

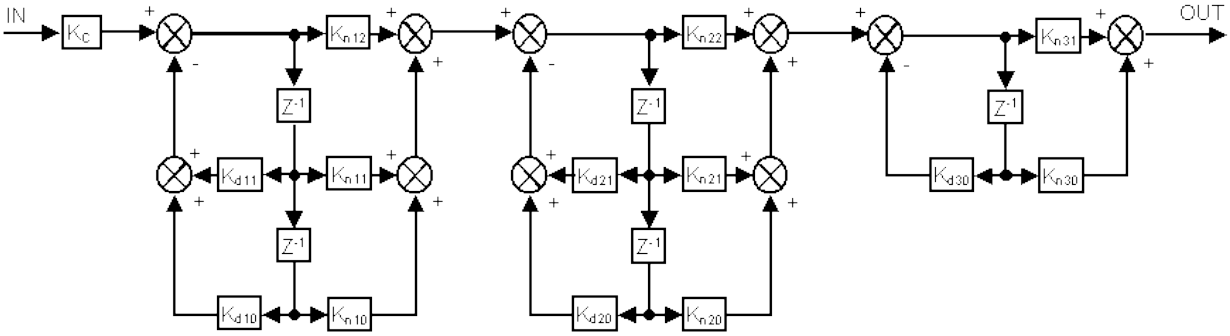
The general fifth order Cascade Z transform is:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\frac{K_c(K_{12}Z^2 + K_{n11}Z + K_{n10})(K_{n22}Z^2 + K_{n21}Z + K_{n20})(K_{n31}Z + K_{n30})}{(Z^2 + K_{d11}Z + K_{d10})(Z^2 + K_{d21}Z + K_{d20})(Z + K_{d30})}$$

Note that the cascade form is simply the product of first and second order Z transform plus a constant gain. The highest order numerator terms are usually 1 or 0.

A graphical implementation for the cascade form is:



FIR cascade filters have no feedback paths in the cascaded section.

The Parallel Form:

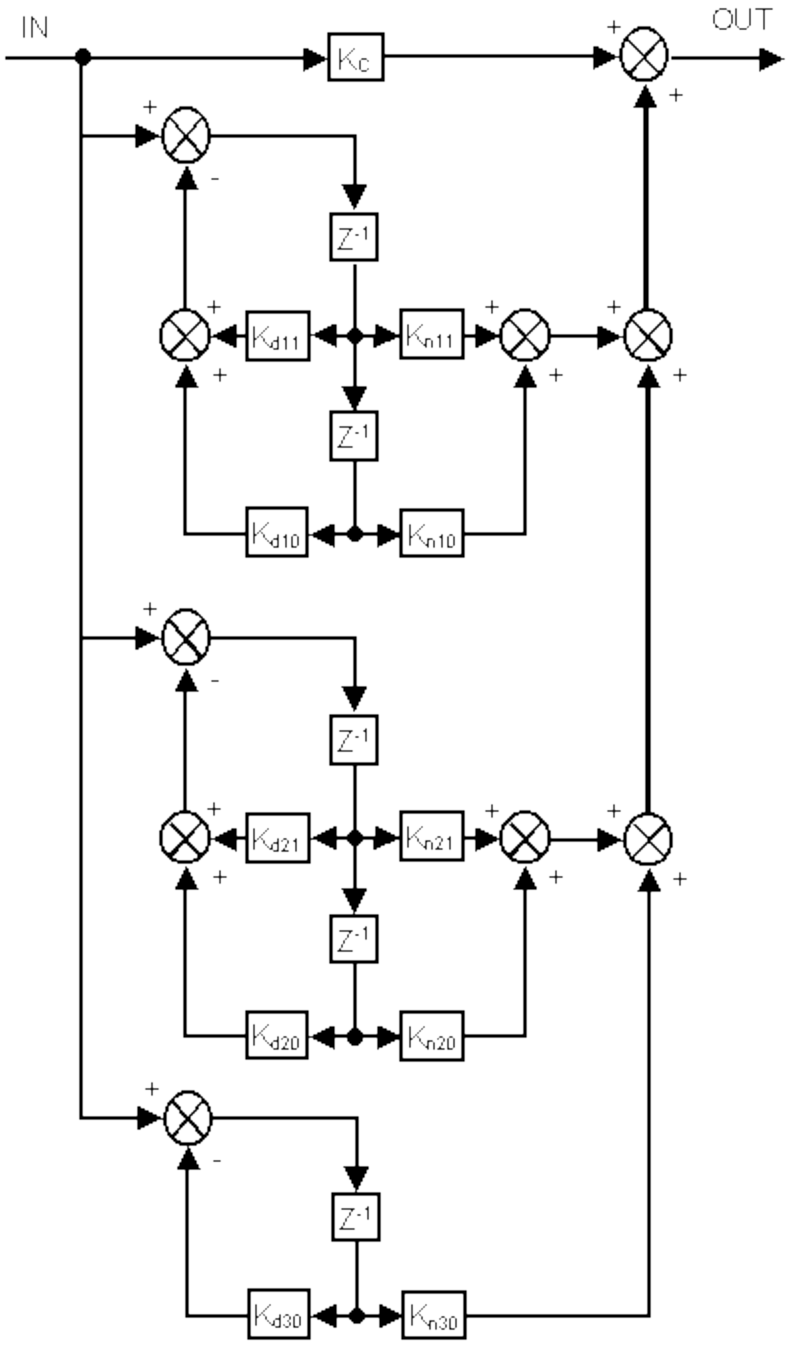
The general fifth order parallel Z transform is:

$$K_c + \frac{K_{n11}Z + K_{n10}}{Z^2 + K_{d11}Z + K_{d10}} + \frac{K_{n21}Z + K_{n20}}{Z^2 + K_{d21}Z + K_{d20}} + \frac{K_{n30}}{Z + K_{d30}}$$

Note that this Z transform is simply the sum of first and second order Z transforms plus a constant offset. Filters with repeated poles, especially FIR filters, may not always be realized by the Parallel form as defined. FilterSolutions does not support Parallel forms for filters with repeated poles.

A graphical implementation for the parallel form is:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Sin(x)/x Sampling Compensation

This chapter contains the following topic:

- Sin(x)/x Sampling Compensation

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sin(x)/x Sampling Compensation

Due to discrete sampling, the transfer function of a digital process is altered by the sampling spectrum according to the following expression:

$$|H(\omega)| = \frac{2\sin\left(\frac{\omega T}{2}\right)}{\omega T}$$

To maintain the frequency response of a digital process in light of this sampling corruption, it is necessary to alter the desired digital process by the inverse of the sampling frequency response according to the following expression:

$$\left|G\left(e^{J\omega}\right)\right| = \frac{1}{|H(\omega)|} = \frac{\omega T}{2\sin\left(\frac{\omega T}{2}\right)}$$

Sinx/x compensation

References:

Rabiner & Gold, "Theory and Application of Digital Signal Processing", 1975, pages 300, 301

Oppenheim & Schaffer, "Discrete Time Signal Processing", 1999, page 199

In reality, an approximate solution that is reasonably accurate up to half the sample rate is an acceptable solution. The following expressions provide this approximate accuracy:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\text{IIR}: \quad \left|G\left(e^{J\omega}\right)\right| \approx$$

First Order Second Order Third Order

$$\frac{1.16Z}{Z + 0.16} \approx \frac{1.15Z^2}{Z^2 + 0.17Z - 0.02} \approx \frac{1.15Z^3}{Z^3 + 0.172Z^2 - 0.028Z + 0.006}$$

$$\text{FIR}: \quad \left|G\left(e^{J\omega}\right)\right| \approx$$

First Order Second Order Third Order

$$\frac{1.16Z - .16}{Z} \approx \frac{1.15Z^2 - 0.2Z + 0.05}{Z^2} \approx \frac{1.153Z^3 - 0.2Z^2 + 0.067Z - 0.02}{Z^3}$$

The higher order expressions are more accurate than the lower order expressions. The FIR solution is slightly less accurate than the IIR solution, but is useful when the FIR characteristic of a filter must be maintained.

The table below shows the ideal correction, FilterSolutions IIR correction, and FilterSolutions FIR correction as a function of frequency.

Frequency	Ideal	IIR 1	IIR 2	IIR3	FIR 1	FIR 2	FIR3
Fs/10	1.02	1.02	1.01	1.02	1.04	1.01	1.02
Fs/5	1.07	1.09	1.07	1.06	1.12	1.06	1.06
Fs/3	1.21	1.25	1.22	1.21	1.25	1.24	1.22
2Fs/5	1.32	1.33	1.33	1.33	1.29	1.34	1.34
Fs/2	1.57	1.38	1.42	1.45	1.32	1.40	1.44

Where Fs is the sample rate = 1/T

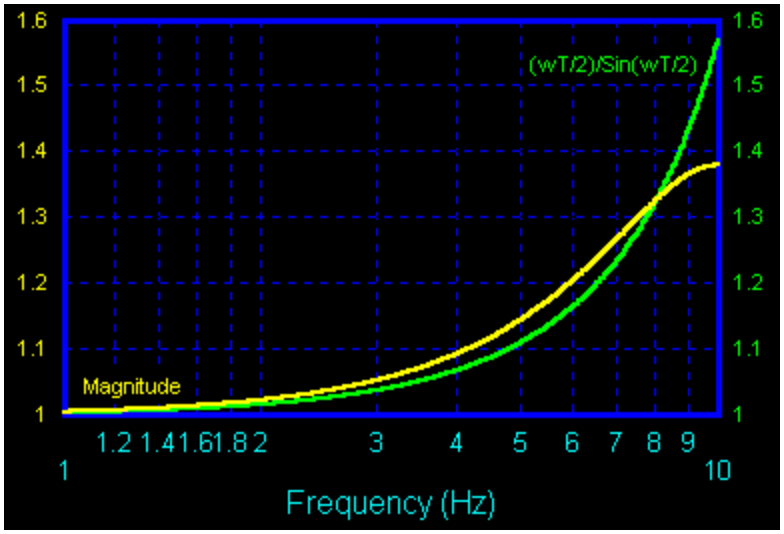
When the **Correct for Sinx/x** box is checked in the digital selections box, the IIR or FIR Z transforms are automatically augmented by one of the sin(x)/x correction Z transforms above. The **Order** box selection determines first or second order augmentation. The effect of these

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

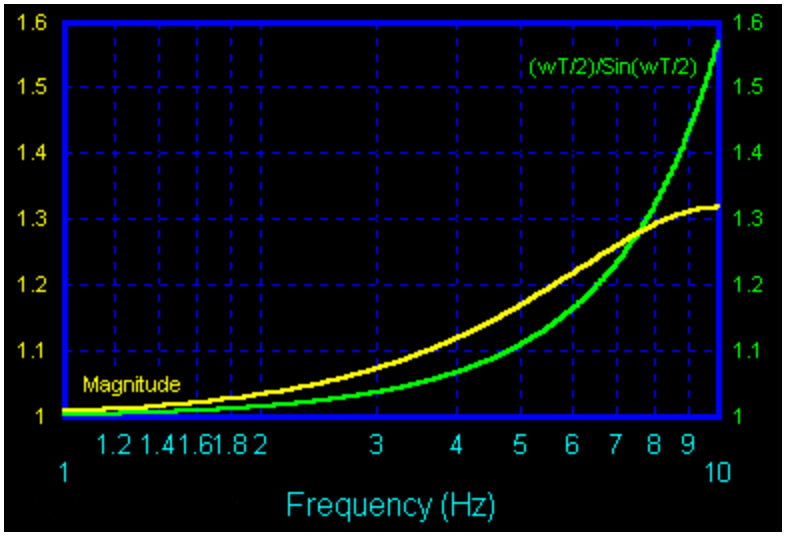
corrections can be observed by viewing the magnitude frequency response up to half the sample rate of a high pass or band stop filter with and without the sin(x)/x correction.

Graphical Comparison

The graphs below show each sinx/x correction trace superimposed on the ideal correction. This will help determine which correction is best to use for the digital filter design. The sample rate is 20 Hz on all graphs below.

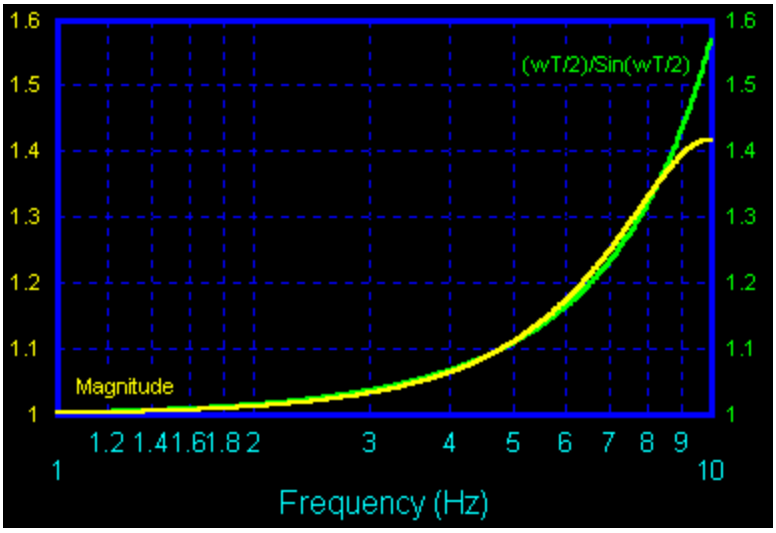


IIR First Order Correction

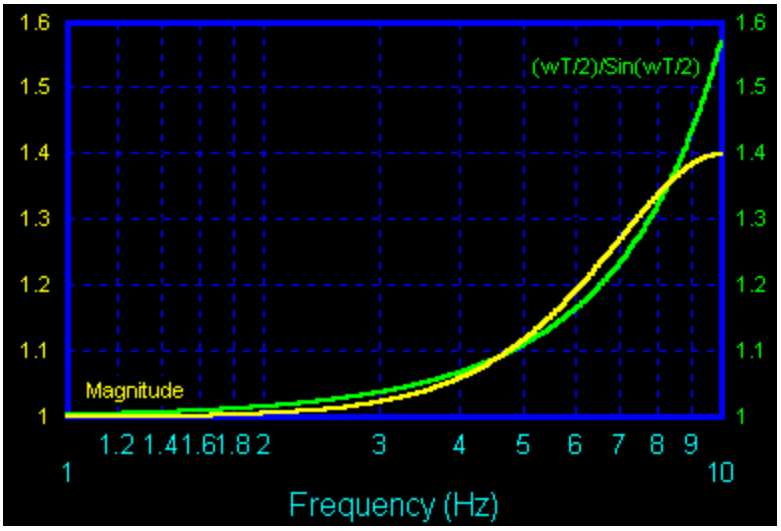


Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FIR First Order Correction

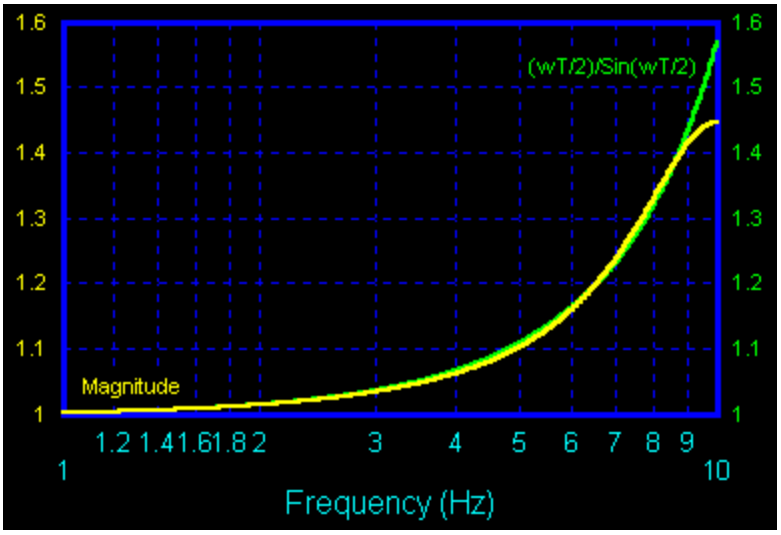


IIR Second Order Correction

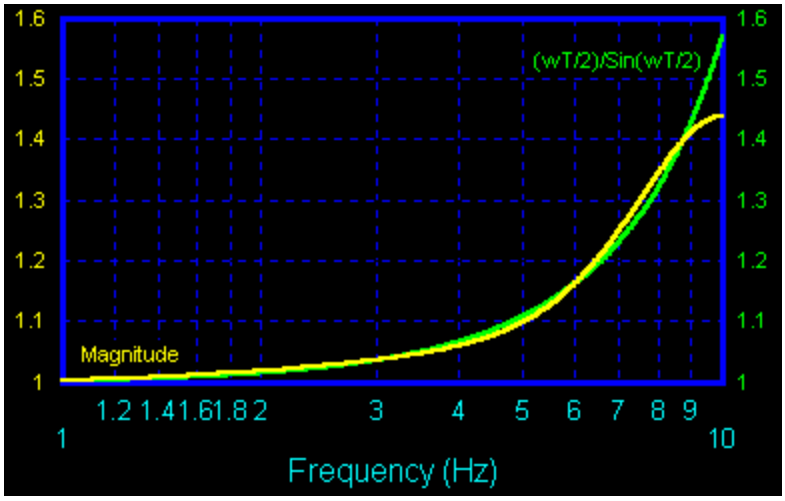


FIR Second Order Correction

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



IIR Third Order Correction



FIR Third Order Correction

Generate Automatic C Code

This chapter contains the following topics:

- C Autocode
- Calling Code

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

C Autocode

General Information

FilterSolutions provides the capability to generate C code for the digital filter in the same form and precision, up to 16 digits, that is specified in the Z transform. If a floating or fixed point notation is selected, the number of digits will be the number of bits divided by 4. The code is compatible with any standard C or C++ compiler. Your may initialize the filter at any point in time to any value you choose. There is no requirement to ever initialize. The filter starts with an initial value of zero at the first call. The variable types used in the code will be float or double, or long double, depending on the number of significant digits or bits selected

Generating Code

To generate the C autocode:

1. Display the digital filter transfer function using the **Transfer Function** button on the Main Control Panel.
2. Select as many significant digits or bits as the compiler and target will allow.
3. Select the form you wish to use (Standard, Cascade, or Parallel).
4. Select the Ccode button.

FilterSolutions will generate the C code in a text window on the monitor. You may edit the C code if desired. The default function name of DigFil may be changed to a name of the choosing.

The C code text window allows one to Print, Copy, or Save the code.

Print	Sends the C code text the printer.
Copy	Copies all or part of the code to the windows clipboard. To copy part of the code, highlight the desired code with the cursor and left mouse key, then select "Copy". To copy all of the C code, leave all C code text unhighlighted, or highlight all the C code text, and select "Copy".
Save	Places the C code text in a directory of the choosing. If you are using C++, you will need to change the file extension from ".c" to ".cpp".

C code Options

Figure 1 below depicts the options that are available in generating C code:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

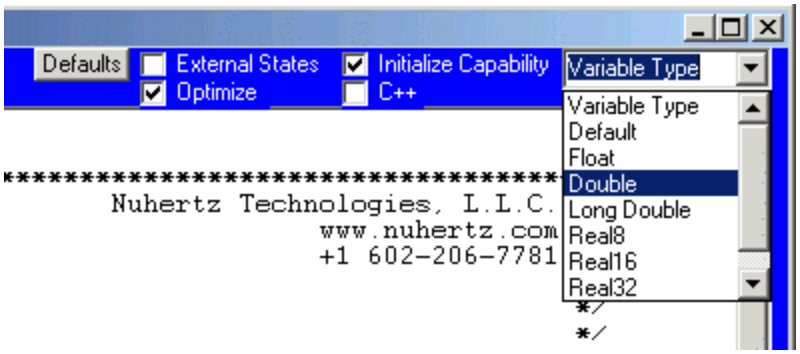


Figure 1 C Code Options

The options are remembered for the next C code generation, and are described below.

External States	Check to store states externally to the digital filter subroutine. A pointer to the string containing the states must be passed in the calling interface. Default state: OFF
Optimize	Check to optimize the code with a minimum number of variables and arithmetic operations. This includes eliminating repetitive Z transform constants and taking advantage of any symmetry that may exist. If the code is not found to optimizable, the Optimize checkbox is disabled. Default State: ON
Initialize Capability	Check to provide initialization capability to the code. An initialization value and trigger variables must be passed in the calling interface. (Band Pass filters assume an initialization value of zero only)

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Initialization capability requires Internal States. If External States is selected, the user is responsible to provide initialization to the external states. Default State: ON
C++	Check to create code with C++ syntax. Default State OFF
Variable Type	Select the desired variable type to use. The default selection selects a variable type based upon the number of significant digits in the filter Z Transform.
Defaults	Click to select the default options.

Calling C Code

Default Selections: (Initializable, no C++, Internal States, Optimized)

For low pass, high pass, and band stop filters, The first two lines of the C autocode (C++ NOT selected) are:

float DigFil(invar, initval, setic)

float invar, initval; int setic;

or

double DigFil(invar, initval, setic)

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

double invar, initval; int setic;

or

long double DigFil(invar, initval, setic)

long double invar, initval; int setic;

or

realN DigFil(invar, initval, setic)

realN invar, initval; int setic;

where N is 16, 32, or 64.

The variable types are a function of the number of significant digits.

For band pass filters, the first two lines are:

double DigFil(invar, setic)

double invar; int setic;

Where the variable type shown as "double" represents the variable type selected by the user.

The input variables are defined in the autocode header. invar is the input to the filter, and the code returns the output. The filter assumes the value of input variable initval whenever setic is input as true.

Low pass, high pass, and band stop digital filters will initialize to the value of "initval" when "setic" is true. Band pass filters will initialize to zero when "setic" is true. At all other times, the filter is free running.

Your calling code should look like this:

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

digout = DigFil(digin, digic, icnow);

or for band pass filters:

digout = DigFil(digin, icnow);

Where digfil is the name of the autocode function, digout, digin, and digic are all defined as the same type of the matching variables in the C function (float, double, or long double) and icnow is type int.

If you are using C++, the prototype should be:

double DigfFl(double, double, int);

or for band pass filters:

double DigFil(double, int);

If you have many digital filters throughout the program, it is possible to use just one digital filter subroutine by modifying the C autocode from Nuhertz Filter into a general purpose digital filter that accepts Z transforms as structures or classes. The Vectors feature of the Z transforms may be used to cut and paste the Z transforms into the source code. Deselect the "Initialize Capability" switch to generate general digital filter C code.

Other Selections (External States, C++, not Initializable, Optimized and Variable Type)

C++

If you are using a C++ compiler, you may select the C++ option for C++ syntax..

External States

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The state array may be stored externally and passed as a pointer. Minimum array dimensions are posted as a comment. The size of the external state array must be greater than or equal to the minimum posted dimensions.

No Initializable

If you have no need to ever initialize the filter, select this option to eliminate the initialization section.

Not Optimized

If you wish to see the underlying basic concept behind the optimized code, generate code for a general purpose Z transform, or do not wish to optimize for any reason, deselect "Optimized".

Variable Type

You may select the standard variable types of float, double, and long double for the C Code.. If you do not make a variable type selection, the default is a function of the number of selected significant digits.

Digital Overview Video

See the Online Help to view the Digital Overview video.

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Index

.

.CKT Files 14-75

A

active all pass stages 14-34

Active Capacitor Losses 14-42

active circuits display 5-100

active filter control panel 5-42, 5-47

Active Simulations of Passive Filters 14-18

Active Stage Gains 14-55

add elements 12-82

AEDT Circuits 13-113

Akerberg-Mossbeg Description 14-4

Akerberg-Mossbeg Equations 14-6

all pass 6-1, 6-4, 6-8, 12-60

alter transmission lines 13-80

amplitude equalizers 12-52

analog 5-23

Ansys Circuits 13-117

Ansys Circutis 13-115

Arithmetic Symmetry 7-1, 7-3, 7-6

Asymmetric Design 7-1

Asymmetrical Band Pass Filters 4-5

Automated General Parameter Synthesis 3-1, 3-5

Automated Parameter Synthesis 3-1, 3-5, 3-9, 3-12, 3-15, 3-16, 3-18

B

Balanced Passive Filters 12-63, 12-84

Band Pass 14-37

band pass filter 4-1, 4-2, 11-2

band stop filter 4-1, 4-2, 11-3

Bessel 4-9

Bridge T 14-12, 14-16

Bridged Resonator 12-36, 14-32

Bridged Tubular 12-36

BridgeT Topologies 12-45

Butterworth 4-11

C

Capacitor Approximations with Transmission Lines 13-8

Capacitor Transformer 12-88

Capacitor Tuning 12-80

capacitors 14-57

Cascaded High and Low Pass Topologies 12-49

Cascaded Quadruplets 13-40

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

change graph limits 5-89

Chebyshev Type I 4-14

Chebyshev Type II 4-15, 12-58

circuit analysis 12-2

circuit synthesis 12-2, 12-9, 12-16, 12-21, 12-29, 12-58

circuits terminations 12-2

Coilcraft Parts 12-75, 12-79, 13-79, 14-54

Combline 13-46

Complex Terminations 12-102

Conductors 13-52

constricted equiripple 3-1, 3-5

Continuous Transfer Function Display 5-66

control panel 5-2, 5-23, 5-31, 5-42, 5-47

Controlling Stub Widths 13-60

copy vectors to clipboard 5-70

Coupled Resonator 12-36, 13-66

coupled resonators 12-16, 13-50, 13-85

Coupled Transmission Lines 13-13

Cross Coupled 13-41

cross coupled resonators 12-29, 12-34

custom filters 4-18, 5-17, 5-70

D

Data Analysis 8-1

Data Files 8-3

Data Transmission Filters 4-24

DCR 12-6

Delay Equalization 12-36, 14-32

delay equalizer 5-78, 6-1, 6-4, 6-8, 14-34

Delay Filters 4-27

delete elements 12-82

Delta 13-10

Desktop 13-92, 13-94

Dielectric 13-52

Differential Stages 14-35

diplexer 12-9

Discrete Optimization 12-127

distributed control panel 5-31

distributed display 5-96

Double Op Amp Biquads 15-8

doubly 12-3

drag 5-76

DXF AutoCAD 13-189

E

Element Sensitivity Plots 12-109, 14-45

Element Sensitivity Tables 12-108, 14-45

Elliptic 4-16, 12-58, 13-40

Elliptic Leap Frog 14-20

End Effect 13-19

engineering notation of numbers 1-12, 5-65

equal inductors 12-39, 12-48

Equiripple Group Delay 4-9, 4-27

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ESR 12-6, 14-42

even order modification 12-58

Export 12-139, 13-190, 14-76, 15-24

F

FDNRs 14-30

filter classes 4-1, 4-2

finite Q 12-4

Fn 14-33

Fo 14-33

Fourth Order Single Amplifier Stages 14-10, 14-17

Frequency Asymmetric 3-20

frequency display 5-81

G

Gain 14-55

Gaussian 4-8

Geometry 13-62

Geometry Limitations 13-57

GIC 14-32

GIC Biquads 14-25

GIC Ladders 14-30

GIC: Third Order GIC 14-29

group delay 3-12, 6-1, 6-5, 6-8, 6-13

Group Delay Equalization 7-3, 7-5, 7-6

H

Hairpins 13-34

Half Band Ripple 3-9

HFSS 13-111

HFSS 3D Layout 13-101

high pass filter 4-1, 4-2, 11-2

Hourglass 4-16, 12-58

I

IIR 4-7, 4-9, 4-11, 4-13, 4-14, 4-15, 4-16, 4-16, 4-18, 4-25, 4-27, 5-17

Impedance Matching 12-44, 12-88

Impedance Simulations 14-30

Import 13-97

Increasing Frequency Symmetry 12-27

Inductor Approximations with Transmission Lines 13-6

Inductor Tuning 12-82

injector resistor 14-7, 14-12

Interdigital: Taps: Pins: Hairpins 13-43

Interlaced Resonators 13-4, 13-53

Inverse Chebyshev 4-15

Inverse Legendre 4-13

Inverted Microstrips 13-4

Inverted Substrates 13-105

IronPython 13-94

J

Jones Charts 5-87

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

K

Kuroda Transforms 12-93

L

Ladder 13-67

Layout 13-62

Layout Equations 13-99

Layouts and Schematics 13-61

Lead Inductance 12-7

Leap Frog 14-20

Leap Frog Active Filters 14-18

Legal 1-12

Legendre 4-13

Libraries 9-1, 9-5, 9-7, 9-7

Linear Phase 4-9, 6-13

low pass filter 4-1, 4-2, 11-1

Lumped Circuit Approximations 13-22

lumped circuits display 5-90

Lumped Coupled Resonator Simulations 13-27

Lumped Element Approximations 13-25

lumped elements 13-84

Lumped Port 13-107, 13-111

M

MacLaurin Series Expansion 4-8

Main Menu Selections Definitions 5-51

Matched Filters 4-25

Materials 13-52

Matlab 5-70

Maximally Flat 4-11

MFB 14-12, 14-16, 14-17

Micro strips 13-102

Microcsp 14-75

Microstrips 13-4

Miniaturized Hairpins 13-36

Minimizing Inductor Count 3-15

Minimizing Order Requirements 3-1, 3-5

Modelithics 12-128

modify active filter stages 14-48

modify circuits 12-66, 14-59, 15-17

modify elements 12-66, 14-59, 15-17

modify switched capacitor stages 15-15

modify transmission lines 13-69, 13-80

Monotonic Class L 4-13

Monte Carlo 12-66, 13-69, 14-60, 15-17

Monte Carlo Analysis 12-105, 14-43

Moving Average 4-25

Multi-Band 5-13, 11-1, 11-2, 11-2, 11-3

Multiband Filters 11-4

Multiconductor 13-62

multoconductor 13-33

mutual coupled 12-19

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

N

narrow notch 13-50

Negative Single Amplifier Biquads 14-12, 14-13, 14-16, 14-17

Negative Single Op Amp Biquads 15-10

Negative Switched Capacitors 15-3

net list 14-73, 15-22

net lists 13-88

Node Capacitance 12-7

Norton Transformations 12-85

notch 13-50

numbers 1-12, 5-65

numerical format 1-12, 5-65

O

Off-Axis Zeros 12-36

Off Axis Quadruplet and Dual Zeros 6-9

op amps 14-40, 15-15

Open Stubs 13-19

Optimization 13-99

Optimize 13-55

Outer LC Transformations 12-85

Overlap Stack Up 13-101

Overlay Capacitors 13-4, 13-52

P

Parallel Active Filters 14-23

Parallel Edge Coupled Resonators 13-30

Parallel Resonator Approximations with Transmission Lines 13-11

Parameters 13-97

Parametric 12-47

parasitic effects 12-82

pass band definition 4-2

passband 3-1, 3-5

passive 5-23

passive filter control panel 5-23

Phase Deviation 6-13

Pi to T Transformations 12-85

Planar 13-82

Polar Plots 5-86

pole zero display 5-75

poles zeros 5-78

Positive Feedback Enhancement 14-13

Positive Single Amplifier Biquads 14-7, 14-9, 14-10, 14-10

Positive Single Op Amp Biquads 15-11

Positive Switched Capacitors 15-2

Python 13-94

Q

Q 5-24, 12-4, 14-33

Quadruplet Zeros 12-36

Quadruplets 12-34, 13-40

R

Radial 13-10

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Raised Cosine 4-21

random error 12-66, 13-69, 14-59, 15-17

Reactive Element Parasitics 12-6

real time updates 5-78, 6-1, 6-20

reflection display 5-84

release information 1-13

Resistivity 13-52

Resistor Parasitic Capacitance 14-41

RGLC: Microstrip: and Striplines 13-3

Richards Transforms 12-91

Root Raised Cosine 4-23

S

S-Parameters 9-1, 9-5, 9-7, 12-79, 13-79, 14-54

S Parameters 12-75, 13-76, 14-52

Sallen & Key 14-7, 14-9, 14-10

sequencing transmission zeros 5-21

Series Resonator Approximations with Transmission Lines 13-14

Short Stubs 13-19

Shunt Node Capacitance Compensation 12-57

Shunt Stub Resonators 13-29

Single Point Ripple 3-5, 3-12

singly 12-3

Smith Charts 5-86

Spice 14-75

Spice Models 9-3, 9-5, 9-7

Split Stubs 13-60

Stack Up 13-101, 13-113

staged filters 4-18, 5-17, 5-74

standard parts 5-53, 12-72, 12-94, 14-62, 14-65, 15-17

Stepped Impedance Filters 13-15

stop band definition 4-2

stop band zeros 5-21

stopband 3-1, 3-5

Strip Line 13-102

Striplines 13-4

Subtrate Geometries 13-4

Suspended Substrate 13-52

Suspended Substrates 13-4, 13-101

Swap LC 12-27

Switched Capacitor Band Pass Architectures 15-14

Switched Capacitor Cascade Biquads 15-12

Switched Capacitor Parallel Biquads 15-13

Switched Capacitor Purpose and Definition 15-1

Switched Capacitor Z Transforms 15-5, 15-7, 15-7, 15-7

Symmetric Substrates 13-105

system requirements 1-13

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

T

T to PI Transformations 12-85

Taps 13-49

terminations 12-3, 12-41

terminology 1-4

Third Order Single Amplifier Stages 14-7, 14-12

Thomas Biquad Design Equations 14-3

Thomas Biquads With Additional Pole 14-4

time display 5-79

Touchstone 12-139, 13-190, 14-76

Touchstone Format Files 12-75, 12-79, 13-76, 13-79, 14-52, 14-54

transfer function vectors 5-70

transformations 13-84

Transitional Filters 4-8

Transmisison Line Parasitics 13-51

Transmission Line Equations 13-1

Transmission Line Prewarping 13-21

Transmission Line Signals 13-22

transmission lines 13-85, 13-88

Transmission Zeros 14-20

Trap Filters 11-4

triplexer 12-9

tubular 12-36

Tuned Stages 14-9

Tuning 13-82

Twin T Notch 14-10

Tx Zeros 3-16, 3-20

U

Unswitched Capacitors 15-5

User Data 8-1, 8-3

User Options 5-61

User Transformations 11-4

V

Variables 13-97

VCVS 14-7, 14-9

vectors 5-70

Via Holes 13-19

W

Wave Port 13-107, 13-111

wide notch 13-50

Wn 14-33

Wo 14-33

Z

zigzag filters 12-38

zoom 5-75

Ansys Electromagnetics Suite 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.