Modelithics®
World's Best RF & Microwave Simulation Models

Application Note 072

A Filter Design Case Study with Nuhertz, Cadence AWR Design Environment, and Modelithics Models

Nearly every radar and communications system includes filters that must perform in accordance with the overall system requirements. Today's system manufacturers have the option of simply purchasing a filter from a vendor when one is needed. A range of filter vendors are currently on the market that could potentially provide a solution for a given requirement. Alternatively, a system manufacturer could choose to design its own filter rather than purchase one from an outside vendor.

Buying or designing a filter is a decision that must be determined by various factors like cost, lead times, design tools, etc. One company that recently had to make such a decision is Applied Radar, Inc. Located in North Kingstown, R.I., Applied Radar manufactures defense-based radar, communications, and electronic-warfare (EW) systems as well as products for commercial applications.

This application note presents a case study of a bandpass filter that Applied Radar designed with first-pass success. Both Ansys Nuhertz Filter Design software and the Cadence® AWR Design Environment® were used for the design process. In addition, passive-component models from the Modelithics COMPLETE Library™ were utilized to simulate all of the lumped components in the design.

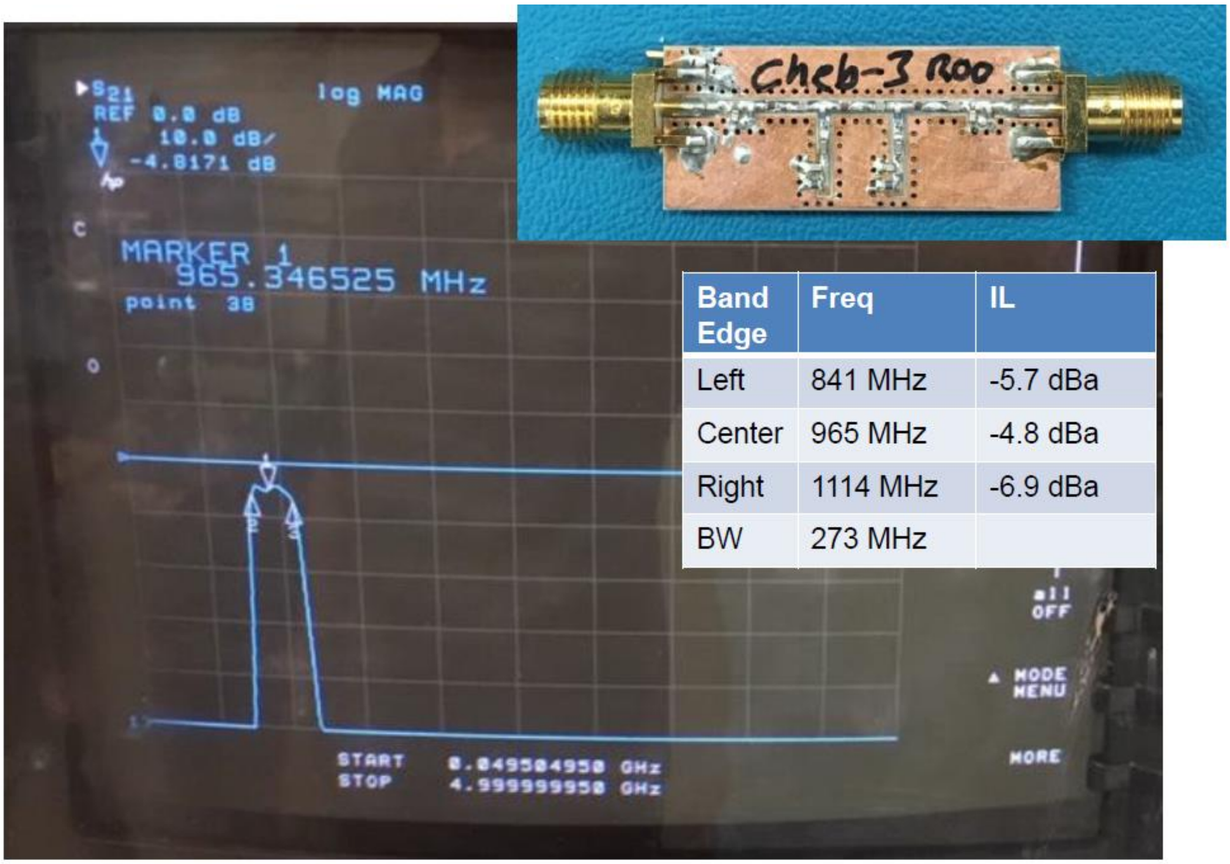
Choosing to Design Rather Than Buy

Applied Radar builds many frequency converters, many of which must provide a final intermediate-frequency (IF) range that is specifically determined by the customer. In addition, many newer systems require wideband instantaneous-bandwidth performance that often ranges from 500 MHz to 1 GHz. Of course, the proper filters are needed in these systems to achieve the desired performance.

A recent project required a bandpass filter with a passband frequency range of 950 to 1,450 MHz. This filter was needed for use in the IF stage of a frequency converter. For this requirement, rather than purchase a filter from a vendor, the company decided to design one itself. One motivating factor for this decision is that purchased filters are often long-lead items that can cause bottlenecks in the manufacturing process. Hence, the company chose to design a filter that would be realized by mounting surface-mount inductors and capacitors on a printed-circuit board (PCB). The chosen substrate for this filter was 20-mil-thick Rogers RO4003C.

Using the Right Design Tools

Without question, designing a filter that achieves the desired specifications with only one design iteration requires the use of the proper simulation tools. As mentioned, for this filter, Applied Radar used a combination of three design tools: Cadence AWR Design Environment software, Nuhertz FilterSolutions, and Modelithics Microwave Global Models™. Applied Radar made the decision to utilize these design tools after failing to achieve success with a "cookbook-based" design approach that the company attempted earlier.[1] This unsuccessful approach resulted in designs that failed to achieve the desired center frequency and bandwidth by considerable amounts. Specifically, the measured center frequency was about 20% less than the desired performance, while the measured bandwidth turned out to be roughly 50% less than the goal (Fig. 1).



Band Edge	Freq	IL
Left	841 MHz	-5.7 dBa
Center	965 MHz	-4.8 dBa
Right	1114 MHz	-6.9 dBa
BW	273 MHz	

Figure 1. Measured performance of one of the original filters designed using a "cookbook-based" approach.

After failing to achieve design success with the initial method, Applied Radar turned to the aforementioned design approach that included the AWR Design Environment, Nuhertz FilterSolutions, and Modelithics passive-component models. The design process began in FilterSolutions, which is a tool used to design various types of filters, such as lumped-element and distributed implementations. Figure 2 shows the FilterSolutions user interface in which Applied Radar specified a seventh-order, lumped-element Chebyshev filter. The lower and upper corner frequencies are set to 950 and 1,450 MHz, respectively. The design is also specified to have equal inductors. Figure 3 shows the filter schematic with ideal component values. In total, the filter contains 7 inductors and 13 capacitors.

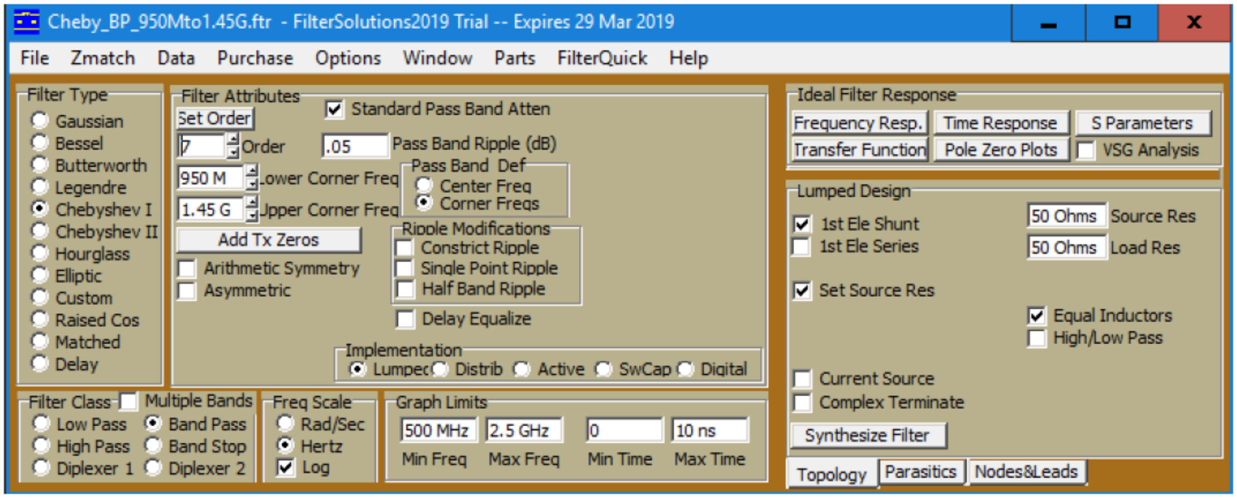


Figure 2. FilterSolutions user interface that represents the beginning of the design process.

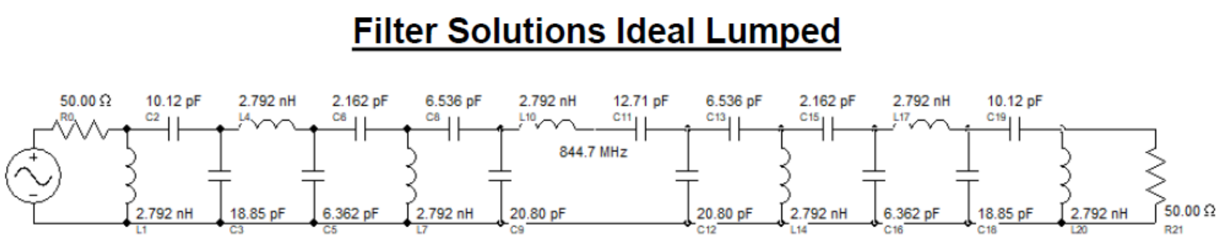


Figure 3. Generated filter schematic with ideal component values.

Next, Modelithics Microwave Global Models are specified for use (Fig. 4). Microwave Global Models represent entire families of inductor and capacitor components. These models scale with part-value, substrate, and solder pad dimensions, among other designer-friendly features. For this filter design, the AVX 08051A series of surface-mount capacitors is selected for all capacitors, while the Johanson L-07 wirewound series is selected for the inductors. FilterSolutions automatically sets the part values of the Modelithics Global models. Once the user is finished selecting the synthesis options and component families, the design can be pushed from FilterSolutions to AWR Design Environment software for further optimization.

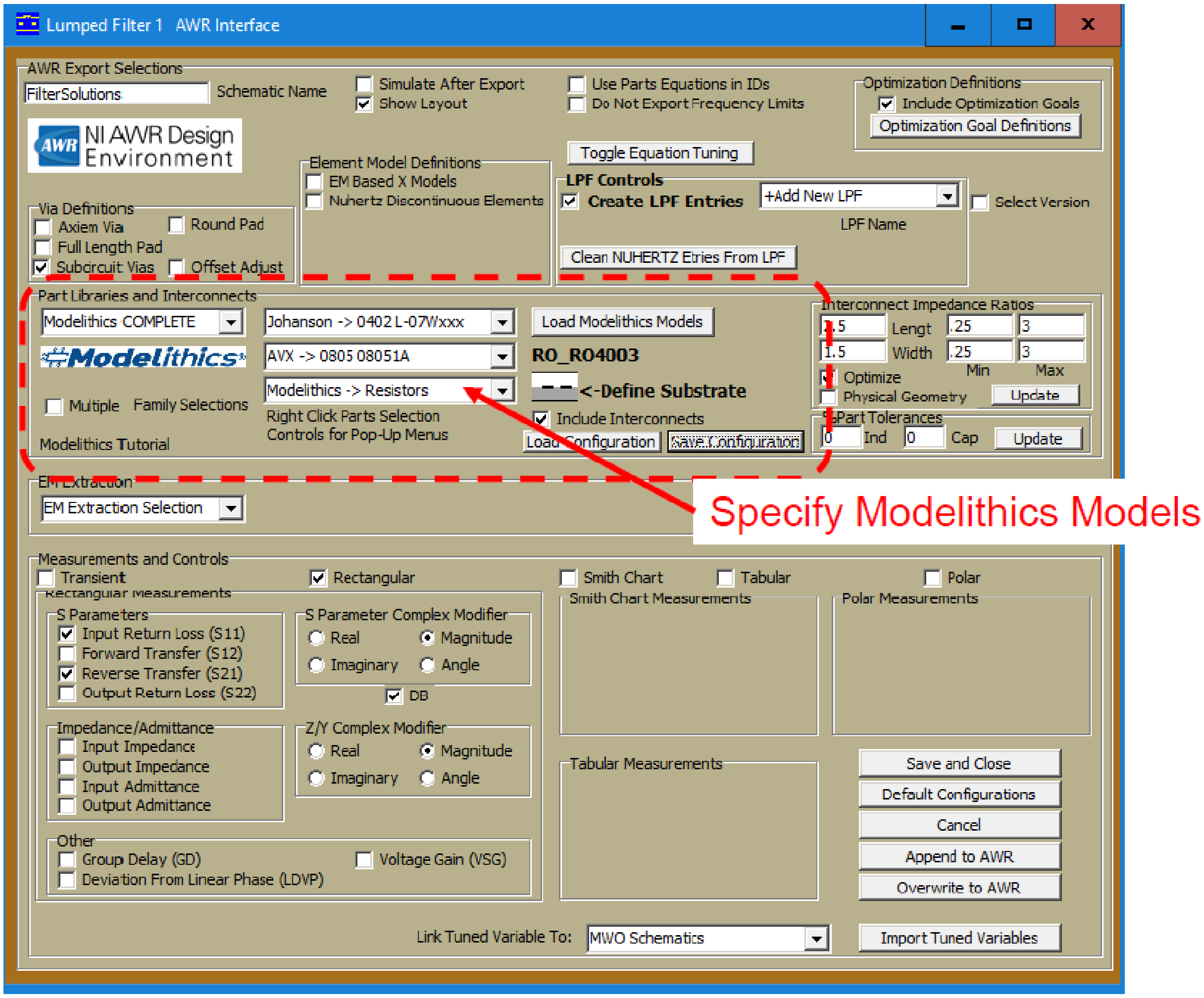


Figure 4. The Modelithics models are specified here.

In addition, before the design is pushed from FilterSolutions to AWR Design Environment software, the dimensions of the microstrip interconnects can be specified. Note that the interconnects are selected for optimization, meaning that their dimensions can later be optimized in AWR Design Environment software. Thus, both the part values of the component models and the interconnect dimensions are configured to be optimized.

The next step is to export the filter to the AWR Design Environment software. Figure 5 shows the schematic of the filter in AWR Microwave Office software after exporting from FilterSolutions. Notice how the microstrip interconnect lines together with the microstrip T-junction elements have been automatically placed in the filter schematic. Figure 6 shows a closer look at one section of the filter.

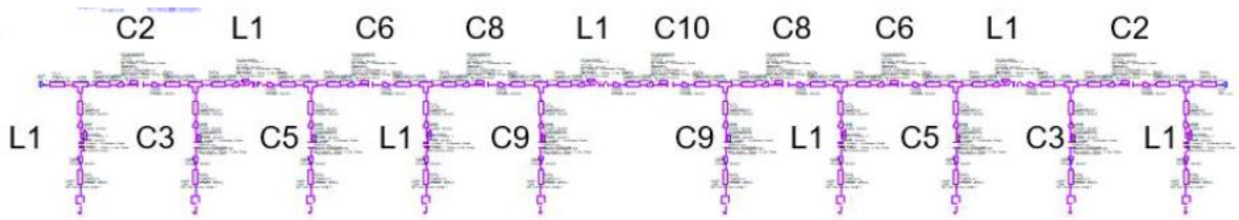


Figure 5. Filter schematic in Microwave Office after exporting from FilterSolutions.

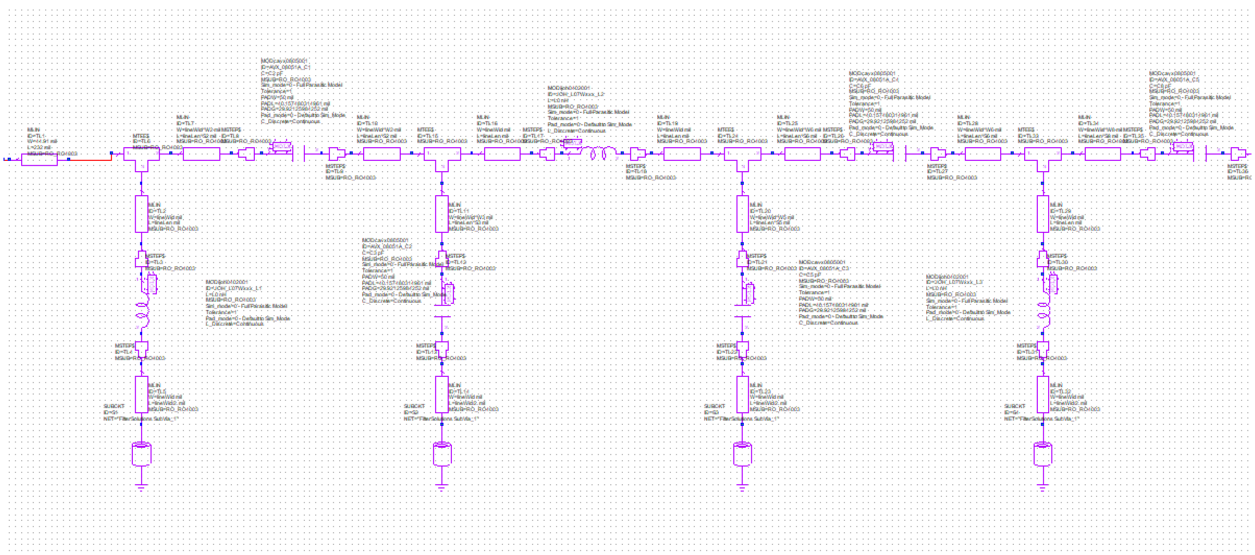


Figure 6. Close-up look at one section of the filter schematic in Microwave Office.

All that is needed at this point is to perform an optimization, in which both the part values of the components and the microstrip dimensions are automatically adjusted to achieve optimal performance. As mentioned earlier, FilterSolutions automatically handles the configuration, simplifying the process for the designer. Figure 7 shows the results of the AWR Microwave Office circuit simulation after optimization.

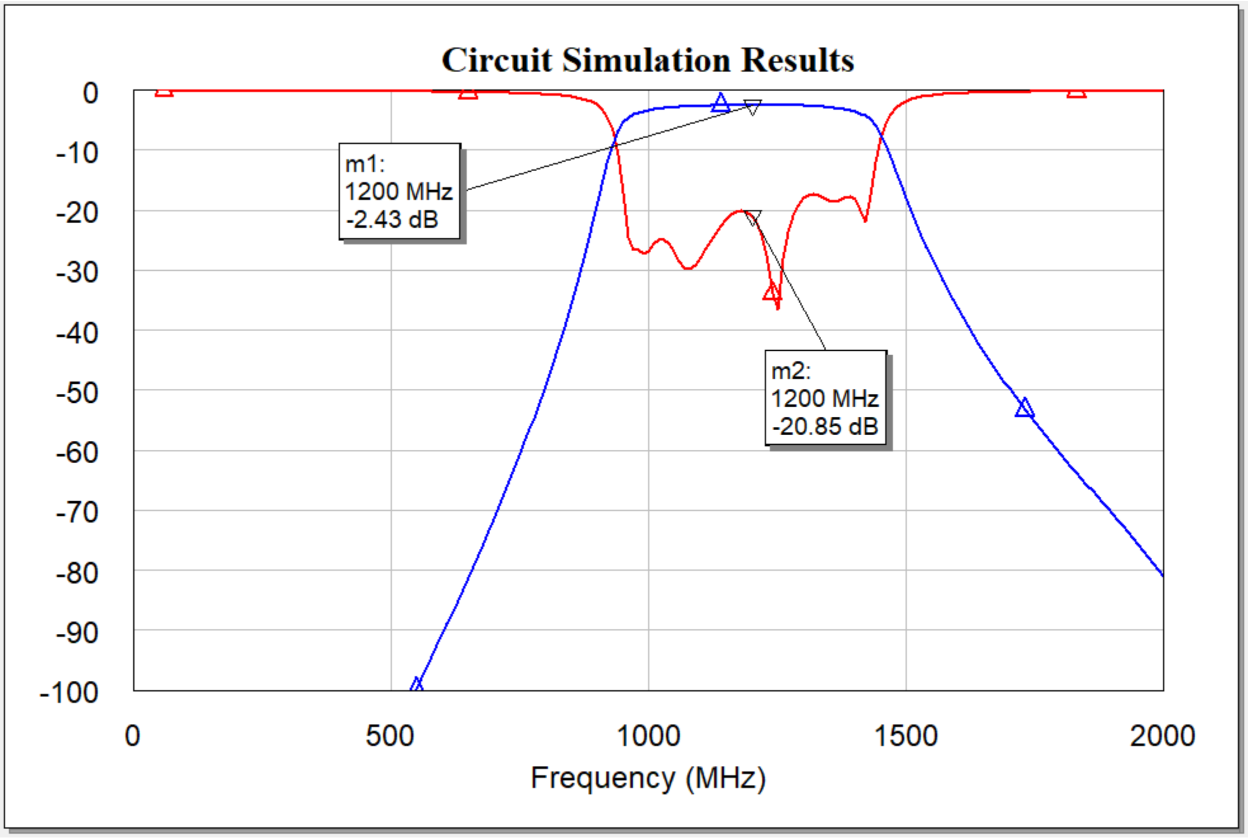


Figure 7. Circuit simulation results of the filter. The blue trace depicts S_{21}, while the red trace depicts S_{11}.

While not performed in the original design process for this particular filter example, designers may also want to use the AWR AXIEM 3D planar electromagnetic (EM) simulator to perform an EM/circuit co-simulation. In an EM/circuit co-simulation, the microstrip interconnects and vias would be analyzed with AWR AXIEM software, while the components would still be analyzed within the Microwave Office circuit simulator. For comparison purposes, we can show the EM/circuit co-simulation results here. Figure 8 shows the EM structure of the filter that will be analyzed with AXIEM. Note that for both the circuit simulation already presented and the EM/circuit co-simulation, the Sim_mode parameter of all models is set to 0. This setting enables the models to account for all real-world parasitic, substrate, and solder-pad effects.

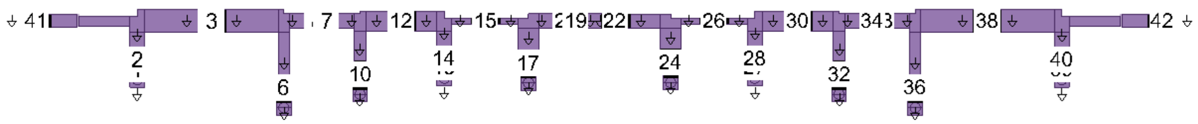


Figure 8. EM structure of the filter.

Figure 9 shows both the circuit simulation and EM/circuit co-simulation results of the optimized filter. The results reveal that the EM/circuit co-simulation results do not stray too far from the results of the circuit simulation. Therefore, EM/circuit co-simulation may not be necessary for this design. Moreover, Table 1 shows the final part values of the filter.

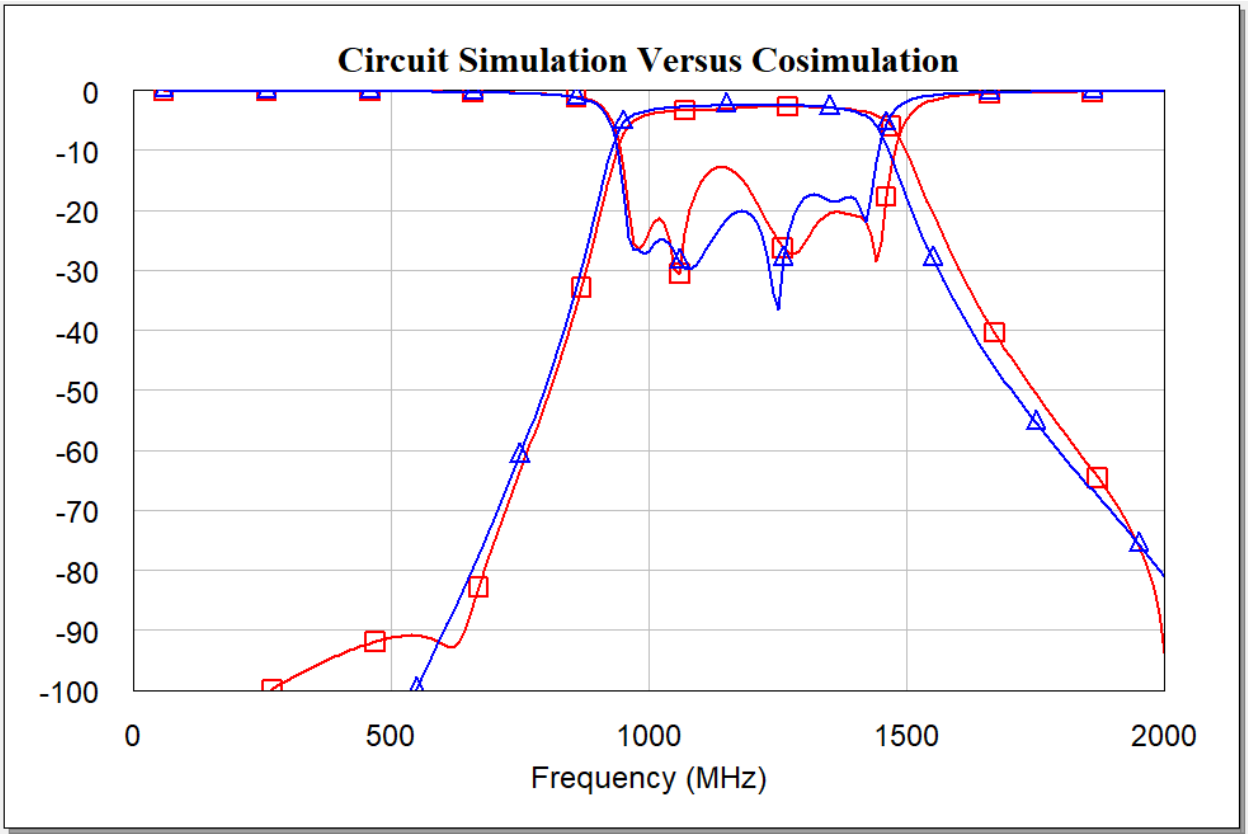


Figure 9. Circuit simulation and EM/circuit co-simulation results of the optimized filter are depicted by the blue traces (with triangles) and red traces (with squares), respectively.

All inductors	2.7 nH
C2	2.7 pF
C3	1.8 pF
C5	2.0 pF
C6	1.8 pF
C8	1.5 pF
C9	1.8 pF
C10	560 pF

Table 1. Final component values of the filter. Refer to Figure 5 for reference designators.

Comparing Real and Simulated Performance

Following the design process, the filter was built and tested with the same components from the final simulation (Fig. 10).

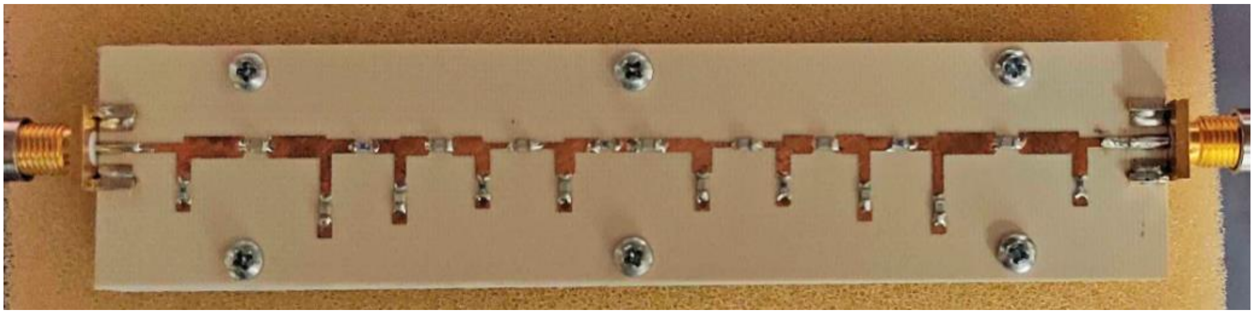


Figure 10. Fully assembled bandpass filter.

Figure 11 shows the measured data of the filter along with the circuit simulation and EM/circuit co-simulation results. One can see that the measured data agrees well with the results from both simulations. Note that while SMA connectors were used to measure the filter, these connectors were not incorporated into the simulations. Therefore, the slight difference between measured and simulated S_{11} could be partially attributed to the connectors not being simulated.

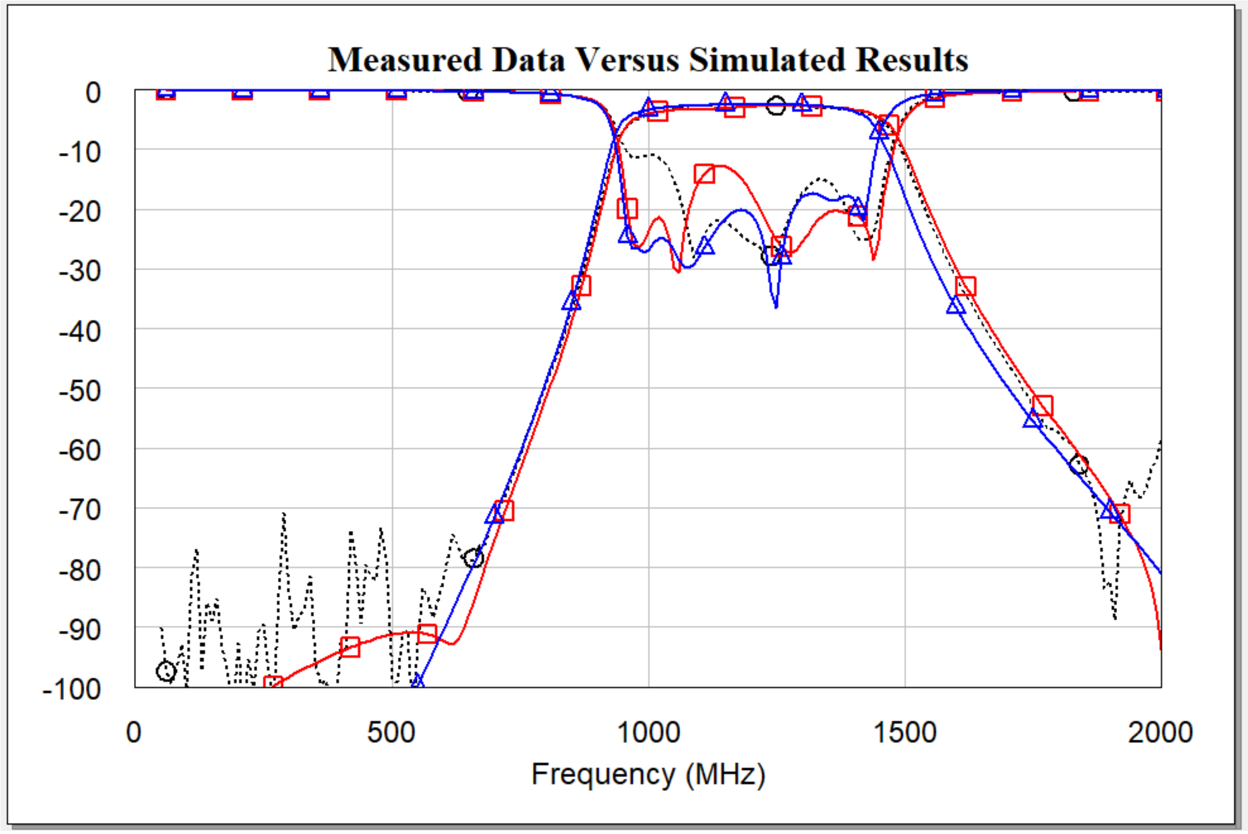
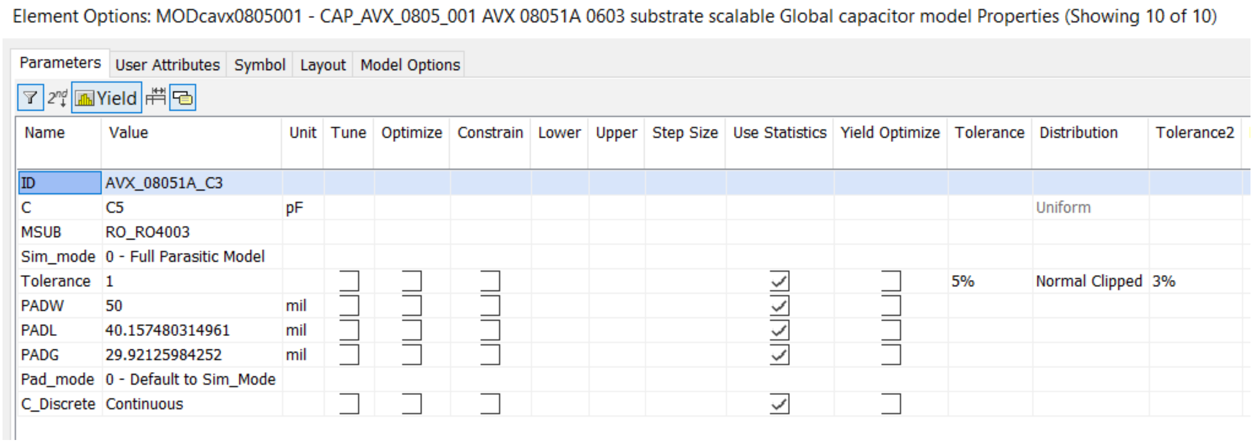


Figure 11. Measured data (dashed traces), circuit simulation results (solid blue lines with triangles), and EM/circuit co-simulation results (solid red lines with squares). Both S_{21} and S_{11} are shown.

Component tolerances may also be contributing to the slight discrepancy between measured and simulated S_{11}. We can perform a yield analysis to further explore this aspect. The Modelithics models include a "Tolerance" parameter that can be utilized for yield simulations. To perform a yield analysis, users must check the "Use Statistics" checkbox next to the "Tolerance" parameter and then specify the tolerance in percentage as well as the distribution (Fig. 12).



Element Options: MODcavx0805001 - CAP_AVX_0805_001 AVX 08051A 0603 substrate scalable Global capacitor model Properties (Showing 10 of 10)

Parameters | User Attributes | Symbol | Layout | Model Options

Name	Value	Unit	Tune	Optimize	Constrain	Lower	Upper	Step Size	Use Statistics	Yield Optimize	Tolerance	Distribution	Tolerance2
ID	AVX_08051A_C3												
C	C5	pF										Uniform	
MSUB	RO_RO4003												
Sim_mode	0 - Full Parasitic Model												
Tolerance	1		☐	☐	☐				☑	☐	5%	Normal Clipped	3%
PADW	50	mil	☐	☐	☐				☑	☐			
PADL	40.157480314961	mil	☐	☐	☐				☑	☐			
PADG	29.92125984252	mil	☐	☐	☐				☑	☐			
Pad_mode	0 - Default to Sim_Mode												
C_Discrete	Continuous		☐	☐	☐				☑	☐			

Figure 12. Model parameters that show the "Tolerance" parameter.

For scenarios in which tight-tolerance parts are selected from the middle of a manufacturing lot, the AWR Design Environment offers a "Normal Clipped" distribution setting.[2] With this distribution setting, a normal (Gaussian) probability density function is specified with the values near the edges removed. Users can then enter two values called "Tolerance" and "Tolerance2," respectively. "Tolerance" specifies the standard deviation of the distribution, while "Tolerance2" would be a lower value that determines what is actually kept in the distribution.[3] Figure 13 illustrates the "Normal Clipped" distribution, with the dashed line showing the full normal distribution for reference.

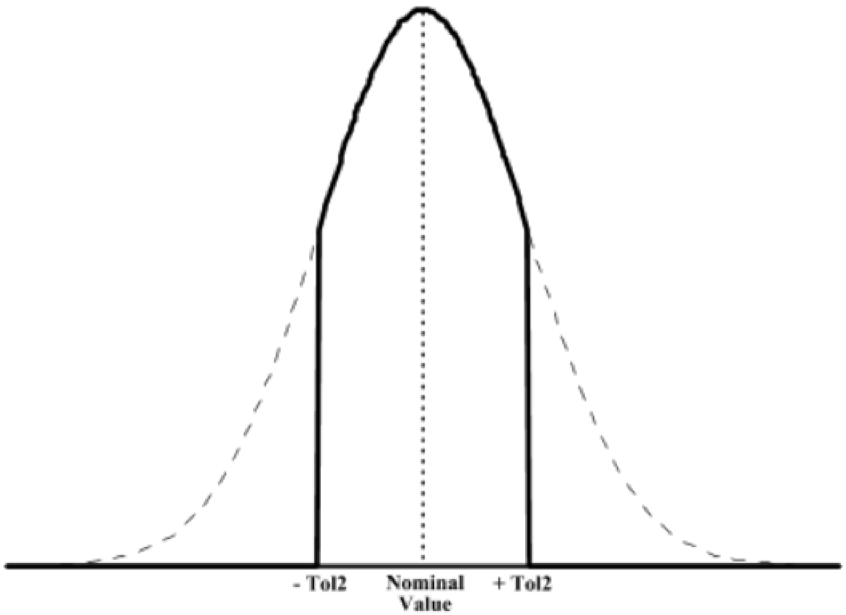


Figure 13. "Normal Clipped" distribution. Dashed line shows the full normal distribution for reference. Solid lines correspond to the "Tolerance2" value.

We can perform a yield analysis with the distribution setting of each model set to “Normal Clipped.” We can then set the “Tolerance” parameter of each model to a specific manufacturer tolerance and the “Tolerance2” parameter to a lower value to represent a tight-tolerance part. Figure 14 shows the results of the yield analysis. The measured results are also shown, demonstrating how it is reasonable to believe that component tolerances are a key contributor to the differences between measured and simulated S_{11}.

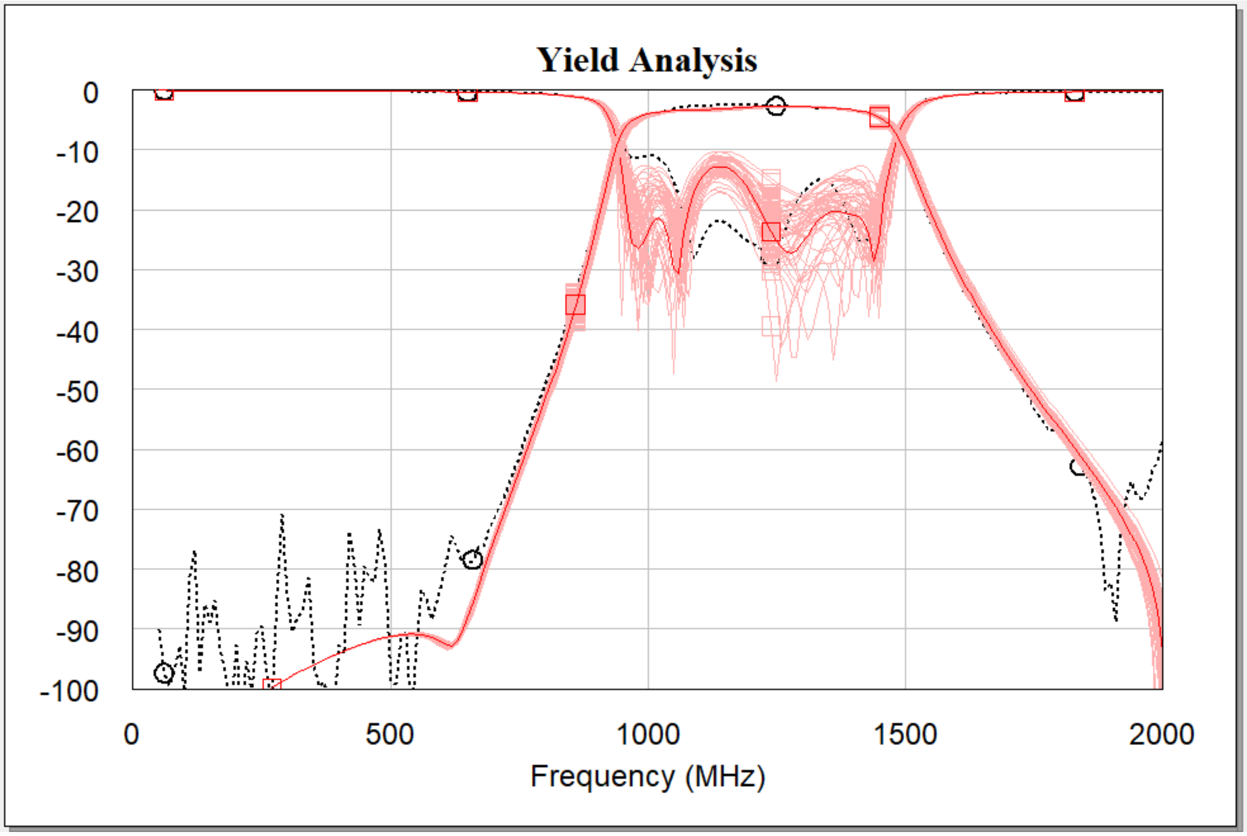


Figure 14. Yield analysis simulation results along with the measured data.

Finally, Figure 15 shows wideband measured data of the filter along with the final simulated results. These results illustrate how the simulations not only predicted the passband performance, but they also predicted the out-of-band performance well.

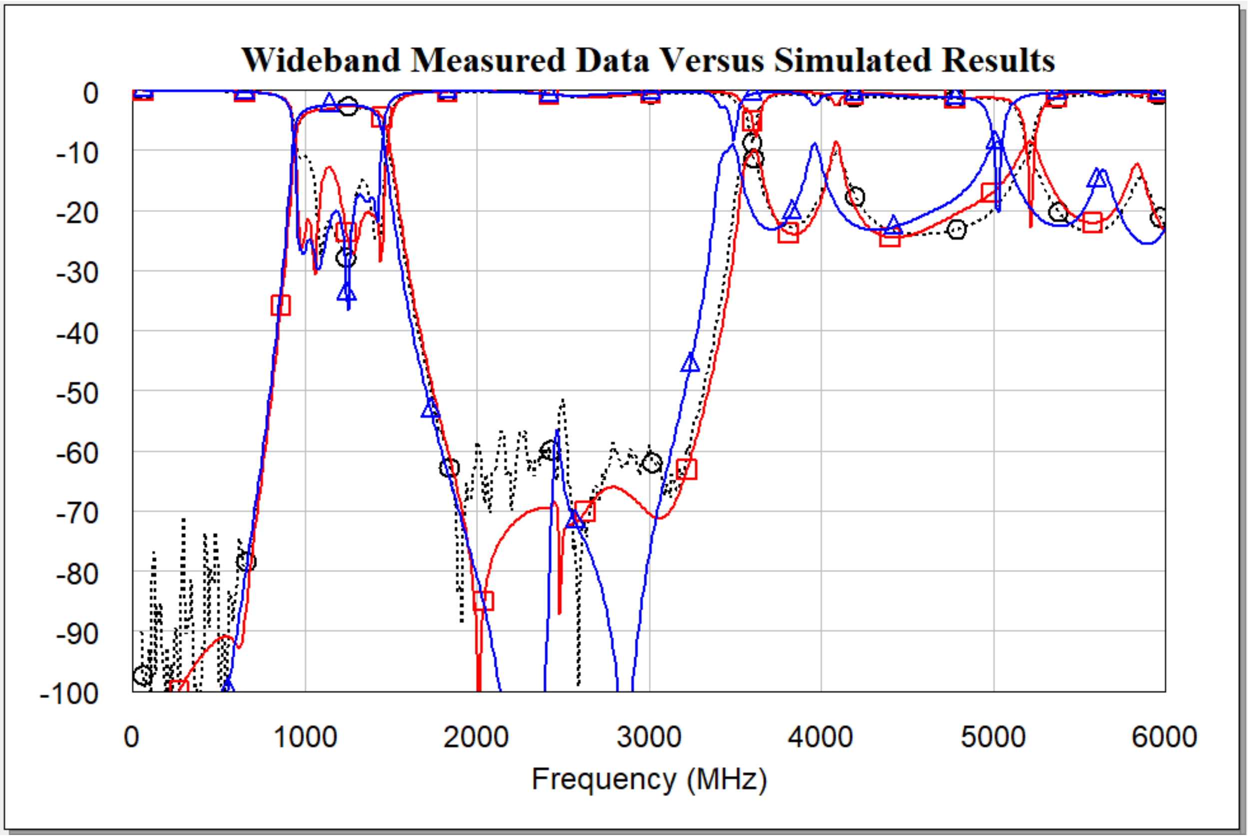


Figure 15. Wideband measured data (dashed traces) and simulated results (solid lines). Blue traces represent circuit simulation results, while the red traces depict EM/circuit co-simulation results.

In closing, the design flow presented here made it possible to achieve first-pass success with a solution that converged in minutes rather than hours or days. The combination of AWR Design Environment software, Nuhertz FilterSolutions, and Modelithics Microwave Global Models for lumped passive components allowed for a very direct workflow that produced excellent results. Those with similar needs may want to consider the tools and semi-automated design flow illustrated with this example.

References

1. G. Matthaei, L. Young, E.M.T. Jones, "Microwave Filters, Impedance-Matching Networks, and Coupling Structures," Artech-House, Norwood, MA, 1980.

2. Dr. L. Dunleavy and Lars van der Klooster, "Application Note 046: Improved Microwave Circuit Design Flow Through Passive Model Yield and Sensitivity Analysis." Modelithics literature: https://www.modelithics.com/Literature/AppNote.

3. AWR Knowledgebase, AWR Design Environment Simulation and Analysis Guide: https://awrcorp.com/download/faq/english/docs/Simulation/Simulation_Analysis.htm.

About This Note

This application note was developed by Bill Weedon of Applied Radar and Chris DeMartino of Modelithics.

Contact Information

For information about Modelithics' products and services, please contact us at:

Modelithics, Inc.
3802 Spectrum Blvd., Suite 130
Tampa, FL 33612
Phone: (813) 866-6335
Email: sales@modelithics.com
Web: www.modelithics.com