ADS Fundamentals - 2001

LAB 6: Filter Design with E-syn, Momentum and Transient Simulation

Overview - This chapter shows the fundamentals of creating filters in ADS and using the Transient simulator. The E-syn interface is used to build a lumped component filter and Momentum is used to test a microstrip filter.

OBJECTIVES

? ***Build a 200 MHz IF low pass lumped filter using E-syn***

? ***Buile a 1900 MHz RF bandpass filter in microstrip.***

? ***Simulate the filter in Momentum***

? ***Perform a Transient analysis***

? ***OPTIONAL - DAC***

Agilent Technologies

TABLE OF CONTENTS

PROCEDURE

1. Change projects.

a. Go to the ADS Main Window and click File > Open Project.

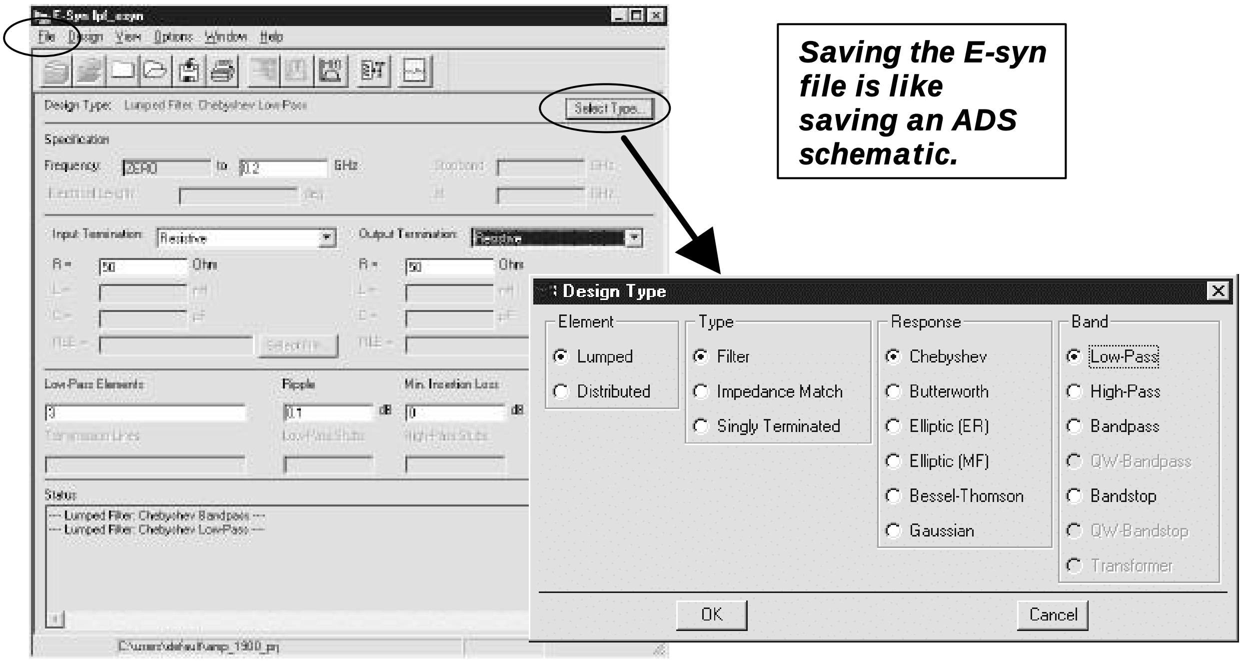
b. Answer Yes to All if you are prompted to save all your current work and open your earlier project: system_prj.

c. Create a new schematic named: filter_esyn.

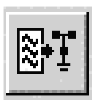
2. Use E-syn to create a 200 MHz Low Pass filter.

a. In the new schematic, click: Tools > E-syn > Start E-syn.

b. When the E_syn window first appears, click File > Save As and give the E-syn file a name: lpf_esyn. Then click the Select Type button and select a Lumped, Filter, Chebyshev, Low-Pass and click OK..

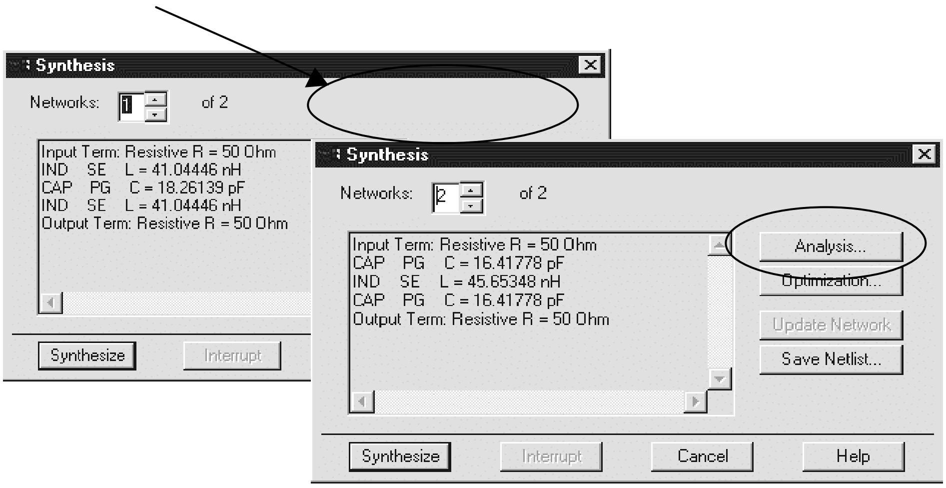


c. In the E-syn main window, set the Frequency spec: ZERO to 0.2 GHz. It will be used in the RF system output for 100 MHz IF.

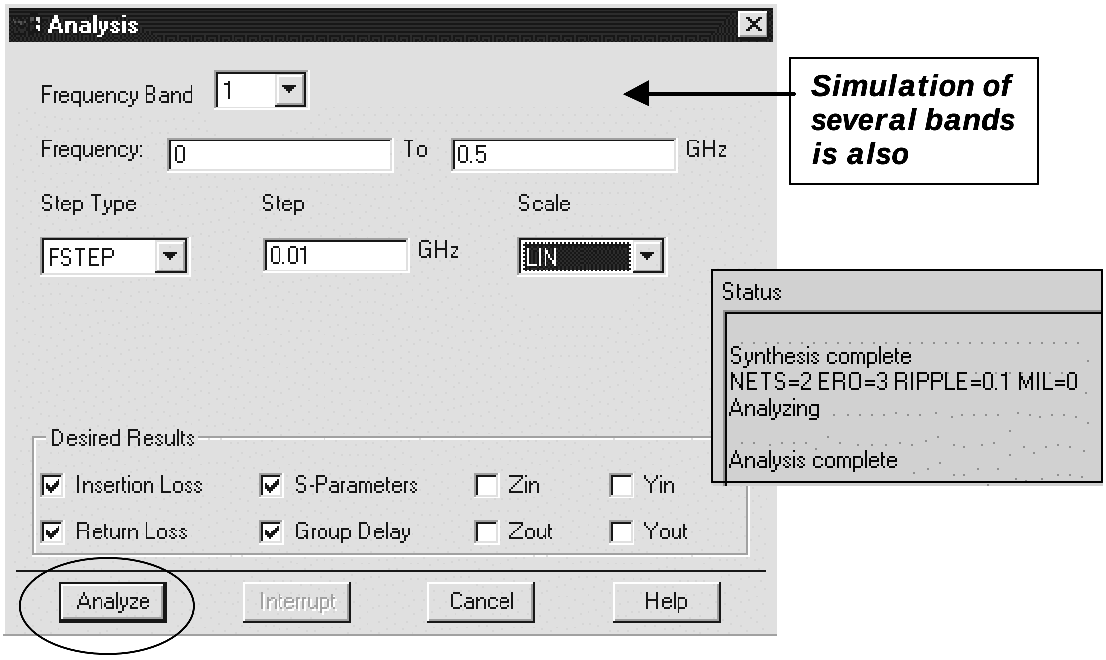


d. Start the E-syn synthesizer by clicking the menu command Synthesis or click the synthesis icon shown here.

e. ***When the dialog appears, click the Synthesize button and E-syn will build one or more filters. In the dialog box, you can see that 2 filters have been created and 1 of 2 is shown as LCL filter. Go ahead and click to select the 2nd filter which is a CLC topology.***

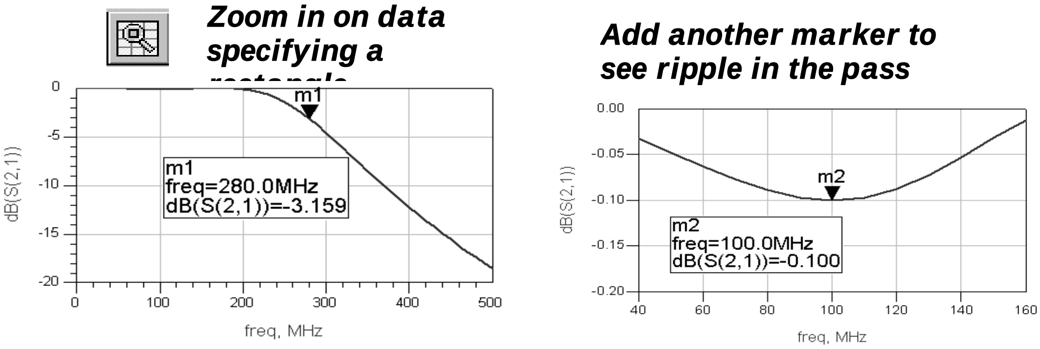


f. ***Also in the Synthesis dialog, click the Analysis button and a new Analysis dialog will appear. As shown, set up the simulation for one Frequency Band 1 - from from 0 to 0.5 GHz, FSTEP = 0.01 GHz (10 MHz). Also, select S-parameters and Group delay. Click Analyze to simulate the filter response. Then look in the E-syn main window to see if the analysis is***



complete.

g. In the E-syn main window, click the Data Display icon. When the untitled display opens, select the lpf_esyn dataset. Plot dB of S21 in one plot and then copy it using Ctrl C Ctrl V to get a second plot and zoom into part of pass band around the 100



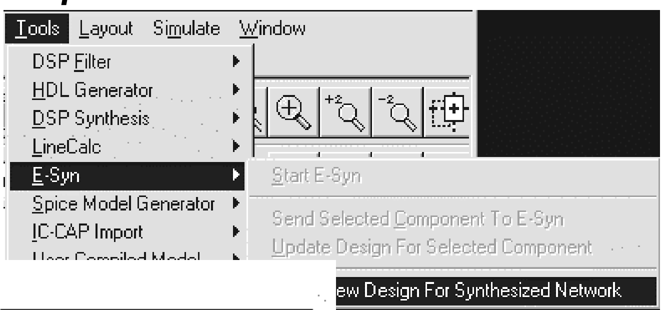
MHz IF to see the ripple.

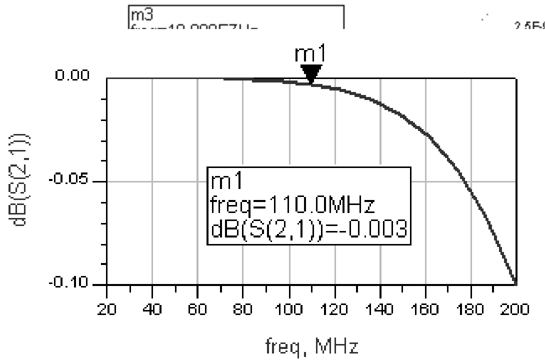
h. Plot S-11 in a Smith Chart and plot GD (group delay) in a rectangular plot. Here, S11 at 100 MHz is not 50 ohms and GD is not very flat.

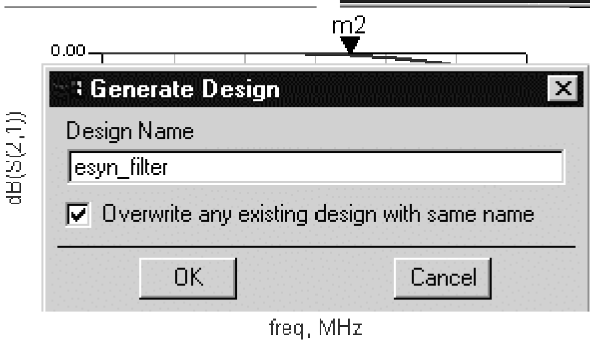
i. Go back to the E-syn main window and click the Save icon. Change the Type = Butterworth. Click Synthesize and Analyze the CLC filter: 8.5pF and 42.5 nH. The performance should improve as shown.

j. Save the data display with the name: filters.

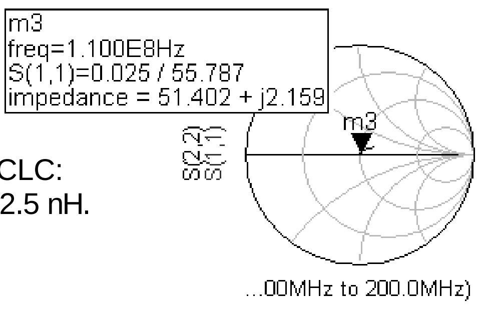
k. To make it a useable component (sub-circuit),

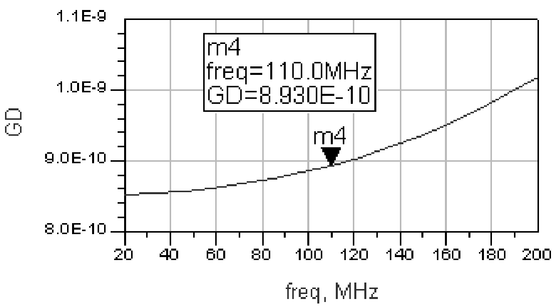






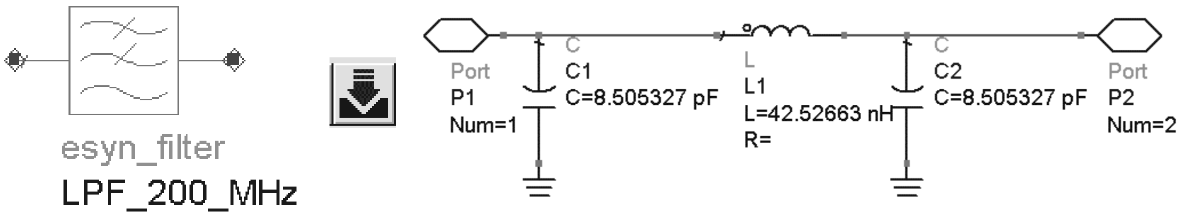
Butterworth CLC:
8.5 pF and 42.5 nH.





go back to the schematic window (filter _esyn) and click: Tools > E-syn > Place New Design For Synthesized Network – this menu command only appears after synthesis. A dialog box will appear for you to name the filter. Type in a name and click OK.

l. ***The E-syn component will be automatically attached to your cursor – place on the schematic in an open area, select it, and push into it to see the lumped element sub-circuit. Afterward, push out and back to the schematic. Name the instance on the schematic if desired, shown here.***



m. ***In the schematic window, click the library icon and verify that the filter is a sub-circuit (Sub-network) in your project.***

n. ***Close all the E-syn windows and save the schematic window.***

Note on using this filter – Later on in the class, you can easily remove the behavioral model from system design and replace it with this one.

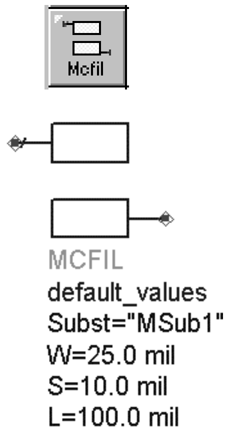
3. Build a microstrip 1900 MHz bandpass filter circuit

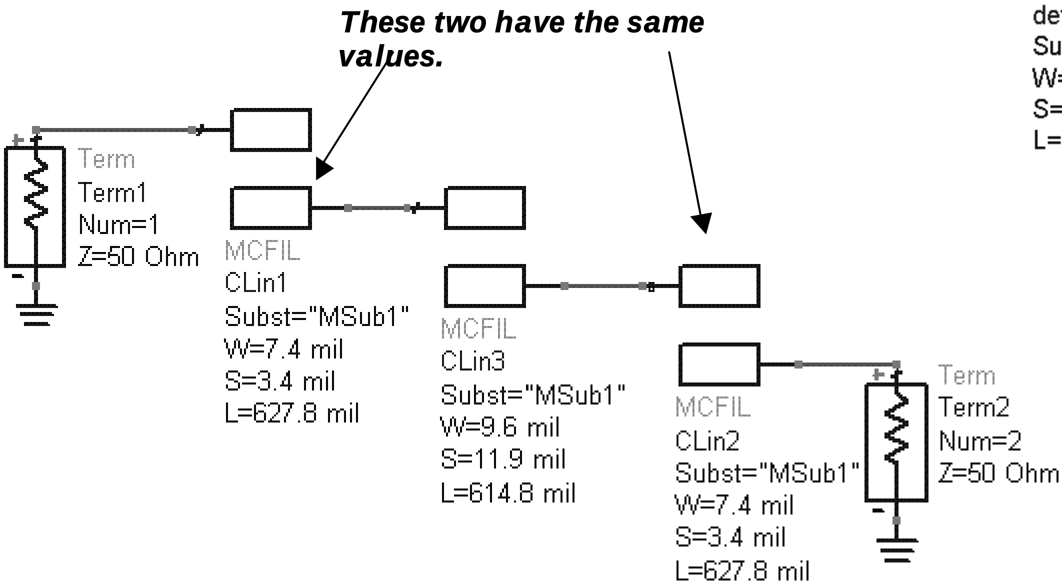
This step will create the system bandpass filter using ADS microstrip coupled lines and simulated with the ADS circuit simulator. Afterward, you will transfer it into layout and simulate with Momentum for a comparison of the results. This step is only intended to briefly show how Momentum works in the most simple application.

a. ***Create a new schematic named: filter_ms. This will be the filter designed in schematic.***

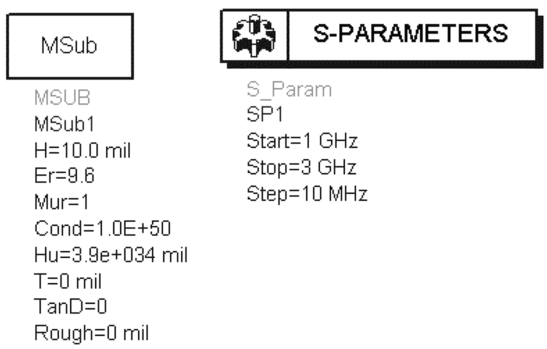
b. ***From the Tlines-Microstrip palette, insert and connect the microstrip coupled lines (MCFIL) as shown –***

   because the two end components are symmetrical (CLin1 and CLin2) so you can save time by setting up Clin1 and then copying it. Also, be sure to connect the terminations with grounds as shown.

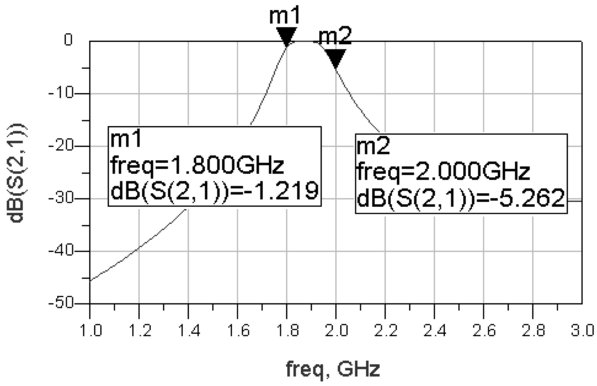




c. ***Also from the Microstrip pallete, insert the substrate definition MSUB as shown - no other settings are required for this lab.***



d. Set up an S-parameter simulation from 1 to 3 GHz in 10 MHz steps.



e. ***After the simulation, plot S21 to verify the response.***

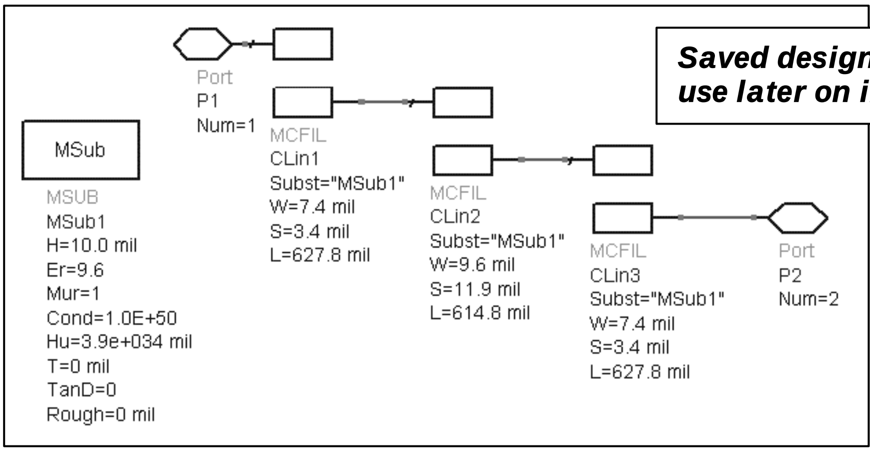
f. ***Save the design and data display.***

g. ***Save the schematic with a new name: filter_1900. This is necessary because you will use this filter (filter_1900) for your final system simulation.***

h. ***In the schematic of filter_1900, click: File > Design Parameters. When the dialog appears, select the ADS built-in symbol for a bandpass filter: SYM_BPF and click OK.***

i. ***In filter_1900, remove the simulation controller, the terms, and grounds. Insert port connectors as shown here but leave the MSUB. Now, the circuit is ready to be used as a sub-circuit.***

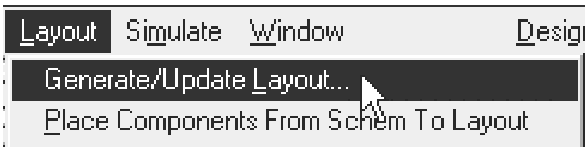
j. ***Save the design again so that all the modifications are also saved.***



Saved design: filter_1900 for use later on in the system.

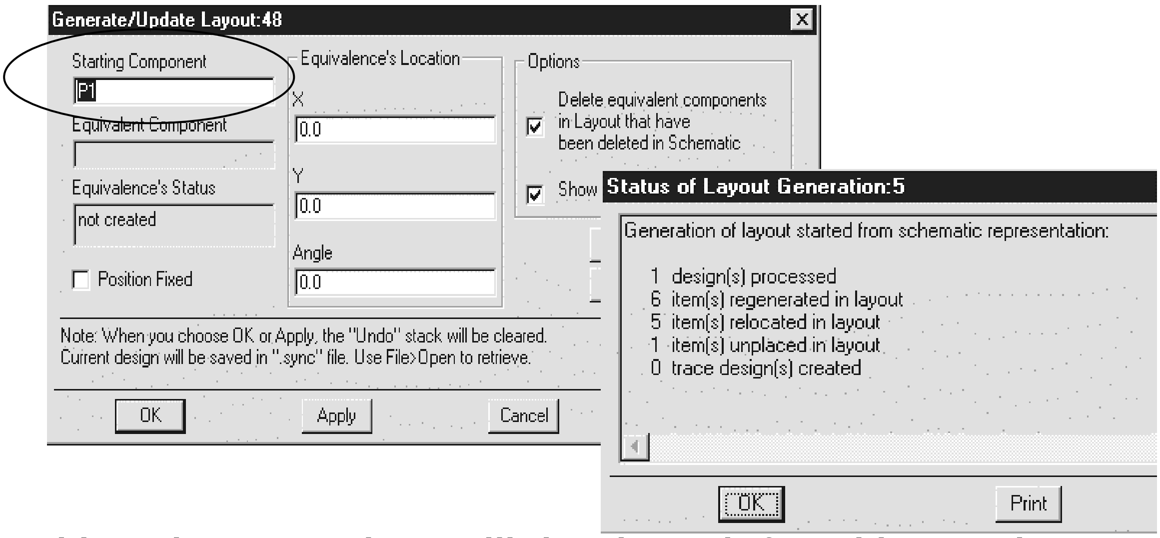
k. ***Save the filter_1900 with a new name for the Momentum simulation which comes next - save it as: filter_mom.***

4. Momentum simulation from layout



a. Transfer the filter_mom schematic to layout by clicking the schematic window command: Layout > Generate/Update Layout.

b. When the next dialog appear, be sure the Start Component is P1 (port connector 1 from schematic) so that the left-to-right layout will be correctly generated. Click OK and you should see another dialog indicating that all components have been created in the layout window.

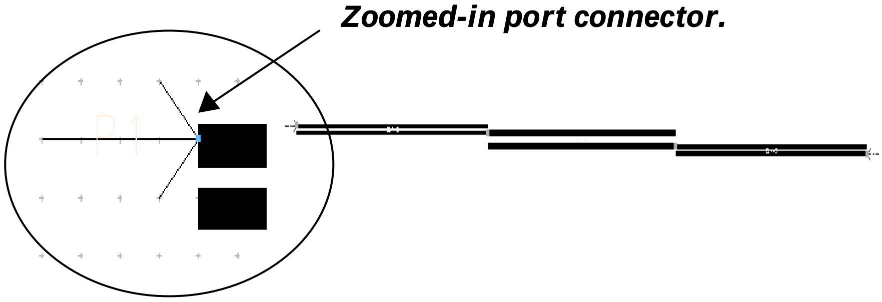


NOTE: Without the MSUB, 5 items will show instead of 6. Either way is OK.

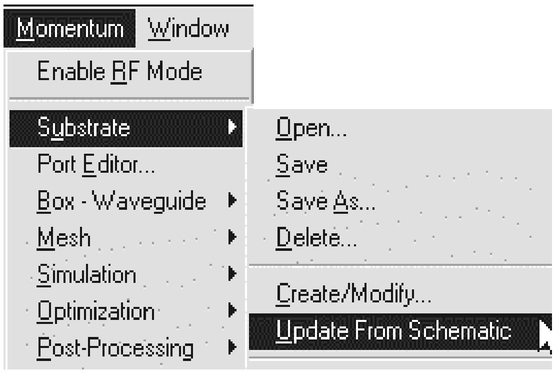
c. When the layout opens, verify that you have the long coupled lines shown here. If you zoom in on a port, you will see that it is connected to the edge of the metal. For most Momentum solutions, it is not necessary for the port to connect to the middle of the line – it only needs to be on the edge.

Zoomed-in port connector.

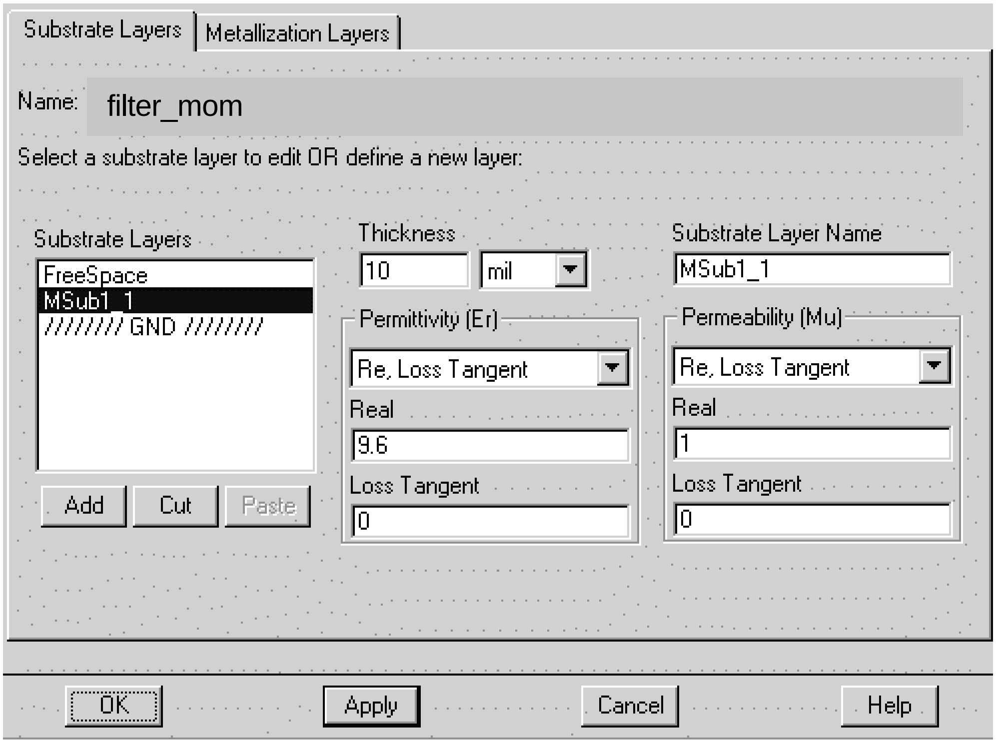
Momentum ports are usually inserted in layout if the drawing is created in layout.



d. ***The next step is to define the substrate in Momentum. To do this, use the Layout commands shown here to transfer the schematic MSUB definition: Momentum > Substrate > Update From Schematic.***



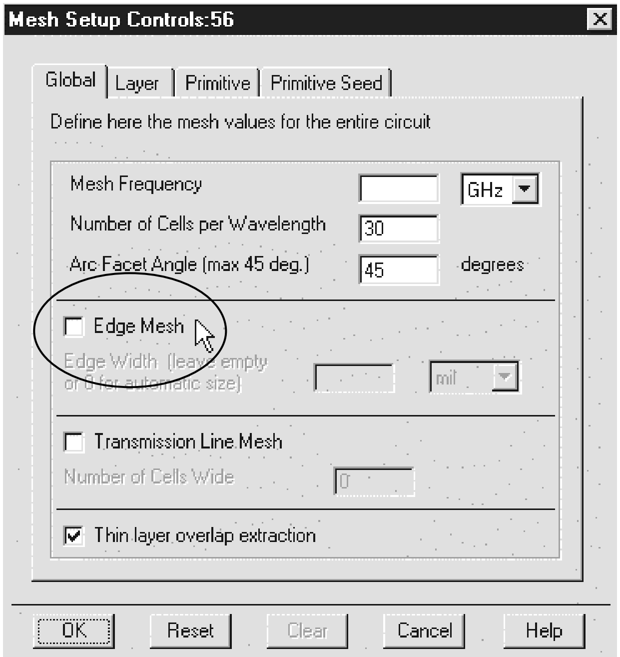
e. ***Verify that the 10mil substrate definition is now in Momentum. Click: Momentum > Substrate > Create/Modify and you should see the MSUB values in the dialog. Click OK if the settings are***



correct.

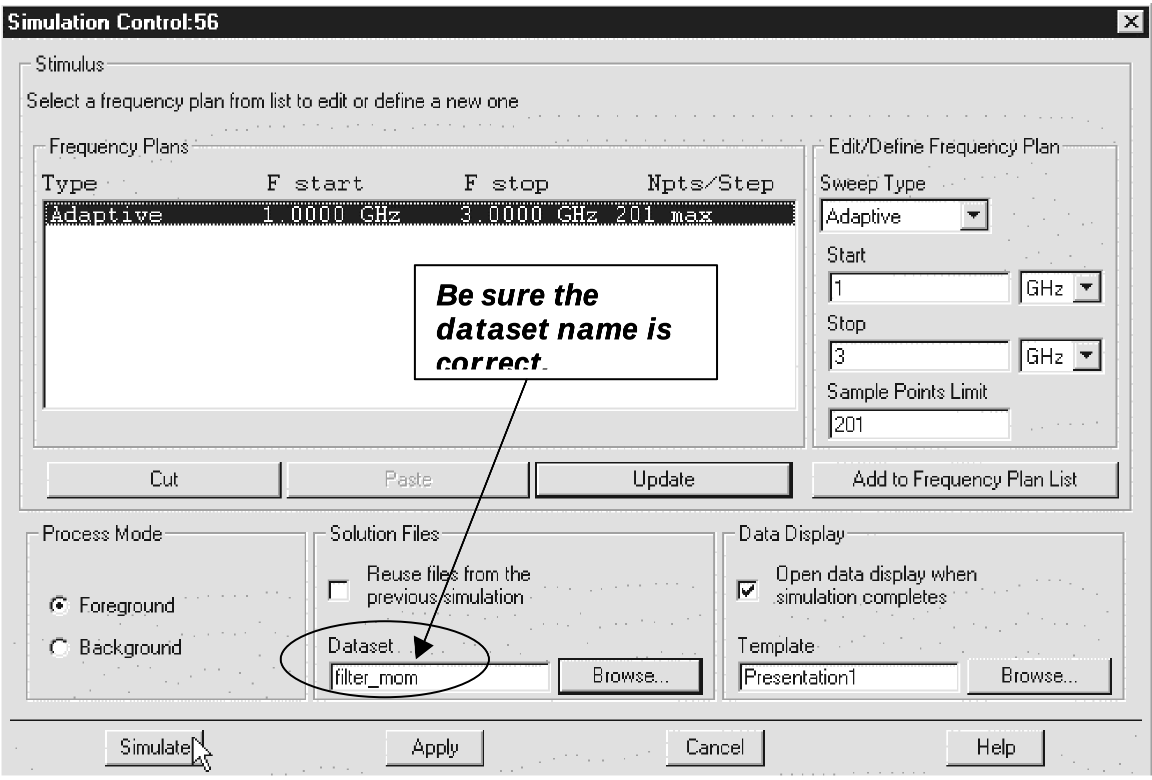
f. ***You can also look at the Metalization Layers tab to see how the drawing layers in layout are mapped into the substrate in Momentum. But DO NOT change anything.***

g. ***Before simulating, turn off Momentum's edge mesh feature to get a faster solution. Click: Momentum > Mesh > Setup. When the dialog appears, notice there are many mesh features. But for now, turn off the Edge Mesh (uncheck box) and click OK.***



NOTE on the Momentum process - the order of steps is: compute the substrate definition (Green's functions), create the mesh or grid pattern (cells), and then simulate. However, you can go directly to the Simulation and Momentum will automatically compute the substrate and mesh the circuit as the cell size equal to the wavelength of the highest frequency.

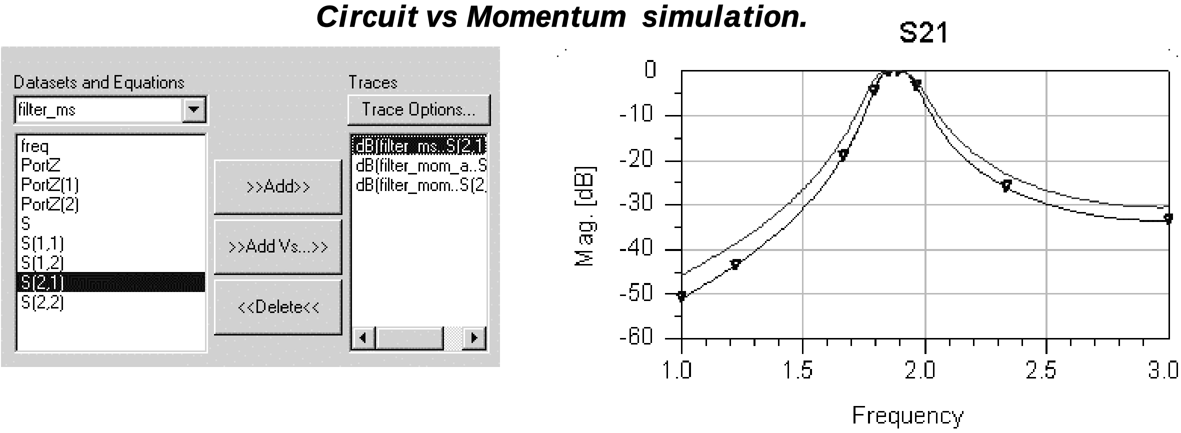
h. ***To simulate, click: Momentum > Simulation > S-parameters. When the dialog appears, setup the simulation: 1 to 3 GHz using 201 points as the limit and click Update. Then click Simulate and watch the status window. The Adaptive sweep type is the default and will save time with its curve fitting- like***



algorithm.

i. When the ADS Data Display opens, notice that the Momentum template looks slightly different – this is OK. Zoom in to the S21 plot and <u>add the S-21 trace from the filter_ms circuit simulation</u> as shown here. As you can see, there is some difference between the two results because Momentum considers the coupling effects using the Method of Moments technique.

Circuit vs Momentum simulation.



The next step shows Momentum's power.

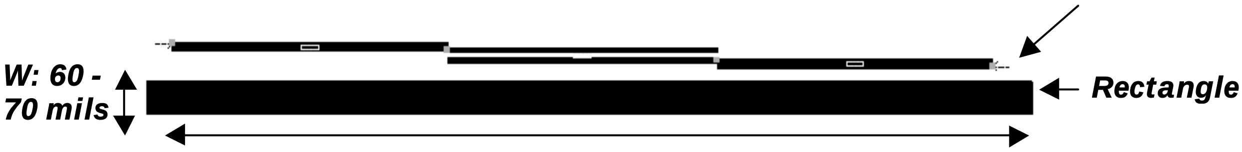
j. In the layout, notice that the filter now has the mesh pattern on it. Zoom in to look at it.

k. Next, draw a simple crude rectangle (click icon) about 60 or 70 mils and about 2000 mils long along the length of the coupled line - this will represent a trace or some other metalization near the filter as shown here.

When you draw the rectangle, notice that the cursor begins drawing at x-y 0,0 as shown here. Also, you can roughly measure using the cursor by clicking in layout and watching the

MS Filter with wide trace along side for Momentum simulation of coupling effects. To measure,

Spacing between the filter and the rectangle is about 10 mils on this end.

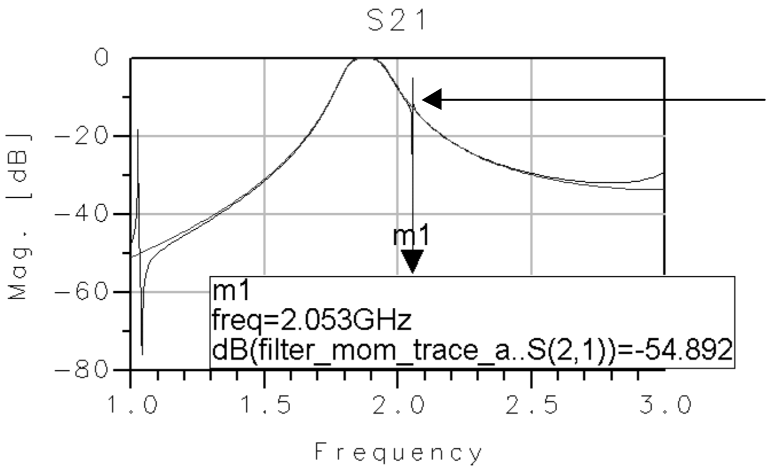


L : about 2000 mils or slightly longer then the filter

values change here.

l. With the filter and rectangle on the layout, perform another Momentum Simulation with a new dataset name such as: filter_mom_strip. Then click: Momentum > Simulation > S-parameters and when the dialog appear, simply change the dataset name as shown here and click Apply and Simulate.

m. This simulation will take a little longer (several minutes) because there are now more cells and therefore more computation time is required. When the data display appears, you should see that there is a resonance somewhere near the bandpass or its edges, depending upon your rectangle as shown here. This is the type of simulation that can only be accomplished with Momentum. Afterward, close the layout and data display windows.



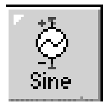
Narrow resonance at 2.05 GHz due to coupling effects of rectangle.

5. ***Transient Analysis on the filter***

   a. ***Save the current filter schematic design and data display.***

   b. ***Open the schematic of microstrip filter_ms and then save it with a new name: filter_trans.***

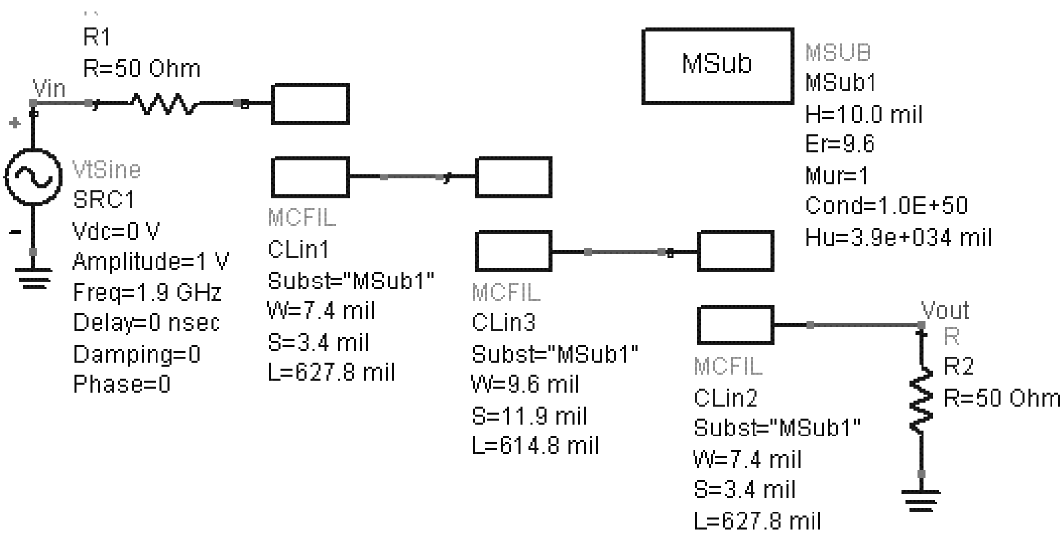
   c. ***Modify the schematic to look like the one shown below starting by deleting the S-parameter simulation controller and the S-parameter terminations (Terms). Keep the MSUB.***



Sources-Time Domain:

   d. ***From the Sources-Time Domain palette insert a VtSine at the input. It is recommended to use Time Domain sources for Transient analysis.***

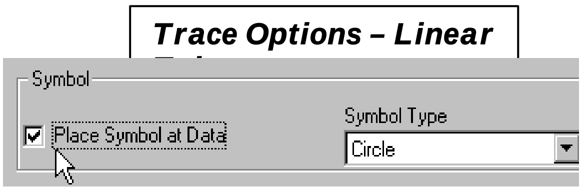
   e. ***Insert a 50 ohm resistor at the input and on the output. While you could leave the Term there, it is always better to set up circuits and simulations that have commonly used components. Also, label pin/nodes Vin and Vout as shown.***

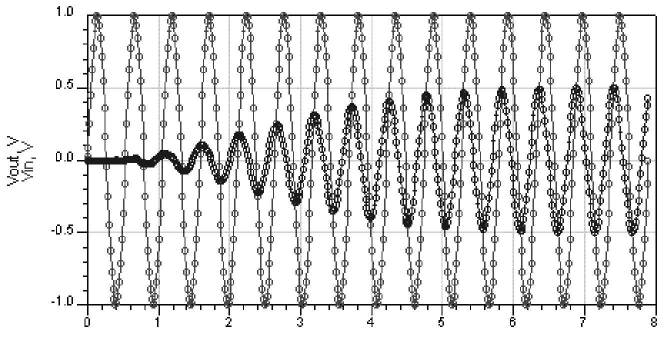


   f. ***From the Simulation-Transient palette, insert a Transient simulation controller and set the Stop Time and MaxTimeStep as shown. Here, the simulation will begin at time zero (default) and stop after fifteen periods of the input signal (8 ns). In addition, the time step will sample at twice (Nyquist rule) the rate***

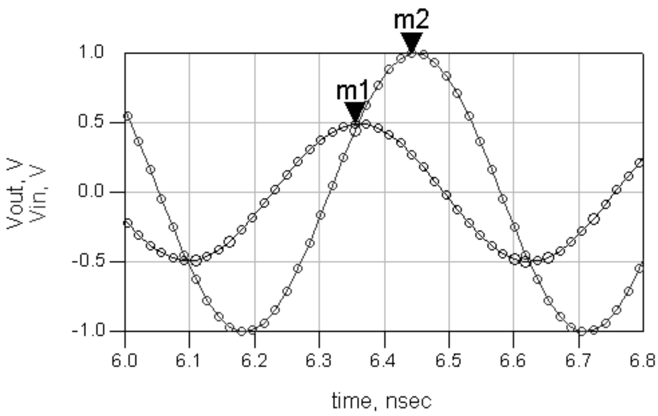
of the highest spectral component desired, here the 15th harmonic.

g. ***After the simulation, insert a rectangular plot of Vin and Vout and edit each trace (Trace Options) to place symbol at data – this will show you the time points taken in the simulation.***





h. ***Zoom into the plot <u>after 5 ns</u> and put the markers on the peaks of Vin and Vout as shown here.***



i. ***Write an equation as shown to calculate the delay through the filter: marker_difference. This calculates the X axis difference between markers using the indep function (independent variable = time).***

Eqn marker_difference = indep (m2) - indep (m1)

j. ***Insert a list of marker_val and use Plot Options to remove the independent data as shown. The value is the delay through the filter after start up (settling). Depending upon where you zoom in and where you place the markers, your value may differ slightly. The delay through the filter is about 90 pico seconds***

marker_difference
8.772E-11

as shown here.

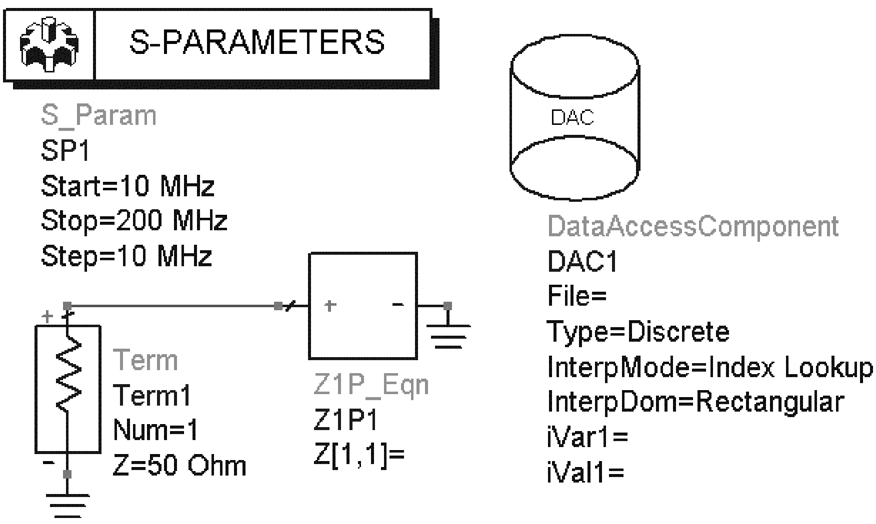
6. OPTIONAL - create an impedance response Data Access Component

A DAC component is a file-based component. It may contain various formats of data from measurements, listed data, or any other valid file type that ADS can read. In this step, you will create a simple file with complex impedance values that vary over a frequency range. Keep in mind that the DAC can be used to create such models as: frequency sensitive impedances, varactors, step functions, bit sequences for sources, time domain sequences, and many other uses where a file is more efficient than typing in long equations or lists on your schematic. In general, the DAC works like this: it is a component that points to a file in the data directory. In this example you create a file that will be used for the impedance parameter of an ADS component - the simulator will simply read the file.

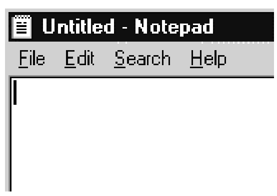
a. Open a new schematic with the name: Z_DAC.

b. Refer to the schematic shown here. Insert a termination with ground, an equation base linear Z1P_Eqn (Z1port) from the Eqn Based Linear palette. Then insert an S-parameter controller, and a DAC from the Data items palette in their default states (no setting yet).

c. Set the S-parameter simulation as if it was the LPF: 10 MHz to 200 MHz in 10 MHz steps as shown and save the schematic again.



d. Open the Windows Notepad program in Start > Programs > Accessories. Or use the ADS Main

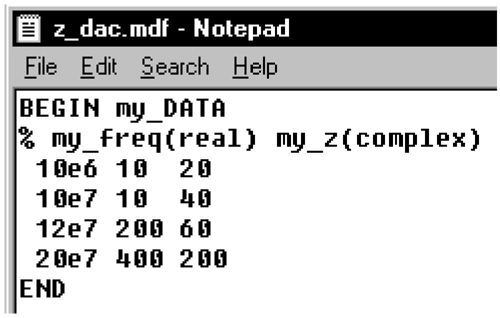


Window (Options >Text Editor) if Notepad is the default text editor.

NOTE on DAC text files - You must not use a program that has formatting - this is a must - such as Wordpad.

e. ***Write an mdf (multi-dimensional data file) file shown here and save it in the DATA directory as: z_dac.mdf.***

NOTE on file type .mdf - If necessary, use windows file explorer to change the name if it is saved as a .txt file. Also, the syntax in the first column is 4 frequency points, the second and third columns contain the real and imaginary impedance values at each frequency.

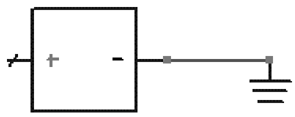

z_dac.mdf - Notepad
File Edit Search Help

```
BEGIN my_DATA
% my_freq(real) my_z(complex)
 10e6 10  20
 10e7 10  40
 12e7 200 60
 20e7 400 200
END
```

f. ***On schematic, edit the S-parameter controller. In Parameters tab, set to compute Z parameters not S. In the Display tab, check Sweep Var, Start, Stop and set them as shown to sweep freq from 10 to 200 MHz in 10 MHz steps. You will get interpolated data for all the steps.***

S_Param
SP1
SweepVar="freq"
Start=10 MHz
Stop=200 MHz
Step=10 MHz
CalcZ=yes

g. ***On schematic for the Z1P, type in the value of Z[1,1]= file{DAC1,"my_z"} where "my_z" refers to the complex values in the file.***



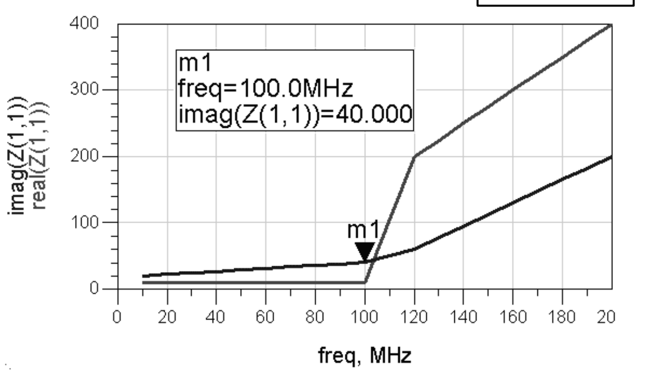
Z1P_Eqn
Z1P1
Z[1,1]=file{DAC1, "my_z"}

h. ***Edit the DAC as shown: Type in the file name, select the Type, etc. Also, iVar1 (my_freq) is the independent variable name and iVal1 (freq) is the independent variable value. As "freq" is swept, "my_freq" will be indexed in the file and the DAC will return complex values of "my_z" interpolated over the range.***

DataAccessComponent
DAC1
File="Z_DAC.mdf"
Type=Generalized Multi-dimensional Data
InterpMode=Linear
InterpDom=Rectangular
iVar1="my_freq"
iVal1=freq

Generic MDIF

i. ***Simulate and plot two traces, real and imag, of Z(1,1) as shown. As you can see, the Zport can be used wherever a frequency***



sensitive component is required. For multiple components, simply create different files and access them as required.

EXTRA EXERCISES:

1. ***Calculate the bandwidth of the filter in Transient simulation using the rule-of-thumb: BW= 0.35 / rise time. From time zero until about 6 ns should be the rise time. However, use the markers and the equations to do the calculation.***

2. ***Create a DAC for a frequency sensitive inductor and simulate it in a simple CLC filter. The DAC will contain the filter values at each frequency.***

3. ***GO back to the Momentum simulation and copy the design to another schematic – they try using a different substrate or simply use Momentum to create a drawing that can be simulated in Momentum – the drawing can be a spiral inductor or anything else.***

4. ***Try using E-syn for another circuit type or some other design.***

射 频 和 天 线 设 计 培 训 课 程 推 荐

易迪拓培训(www.edatop.com)由数名来自于研发第一线的资深工程师发起成立，致力并专注于微波、射频、天线设计研发人才的培养；我们于 2006 年整合合并微波 EDA 网(www.mweda.com)，现已发展成为国内最大的微波射频和天线设计人才培养基地，成功推出多套微波射频以及天线设计经典培训课程和 ADS、HFSS 等专业软件使用培训课程，广受客户好评；并先后与人民邮电出版社、电子工业出版社合作出版了多本专业图书，帮助数万名工程师提升了专业技术能力。客户遍布中兴通讯、研通高频、埃威航电、国人通信等多家国内知名公司，以及台湾工业技术研究院、永业科技、全一电子等多家台湾地区企业。

易迪拓培训课程列表：http://www.edatop.com/peixun/rfe/129.html

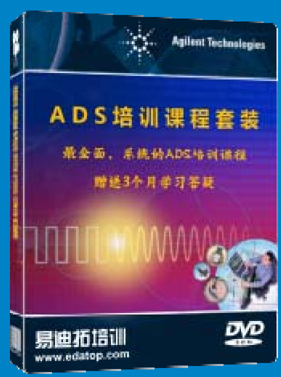
射频工程师养成培训课程套装

该套装精选了射频专业基础培训课程、射频仿真设计培训课程和射频电路测量培训课程三个类别共 30 门视频培训课程和 3 本图书教材；旨在引领学员全面学习一个射频工程师需要熟悉、理解和掌握的专业知识和研发设计能力。通过套装的学习，能够让学员完全达到和胜任一个合格的射频工程师的要求…

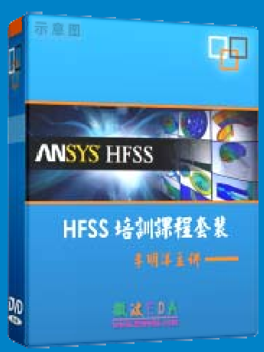
课程网址：http://www.edatop.com/peixun/rfe/110.html

ADS 学习培训课程套装

该套装是迄今国内最全面、最权威的 ADS 培训教程，共包含 10 门 ADS 学习培训课程。课程是由具有多年 ADS 使用经验的微波射频与通信系统设计领域资深专家讲解，并多结合设计实例，由浅入深、详细而又全面地讲解了 ADS 在微波射频电路设计、通信系统设计和电磁仿真设计方面的内容。能让您在最短的时间内学会使用 ADS，迅速提升个人技术能力，把 ADS 真正应用到实际研发工作中去，成为 ADS 设计专家...



课程网址： http://www.edatop.com/peixun/ads/13.html

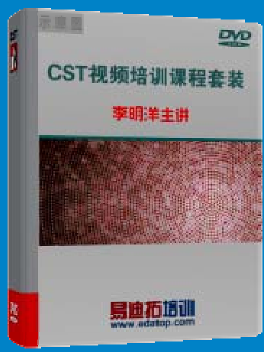


HFSS 学习培训课程套装

该套课程套装包含了本站全部 HFSS 培训课程，是迄今国内最全面、最专业的 HFSS 培训教程套装，可以帮助您从零开始，全面深入学习 HFSS 的各项功能和在多个方面的工程应用。购买套装，更可超值赠送 3 个月免费学习答疑，随时解答您学习过程中遇到的棘手问题，让您的 HFSS 学习更加轻松顺畅…

课程网址：http://www.edatop.com/peixun/hfss/11.html

CST 学习培训课程套装



该培训套装由易迪拓培训联合微波 EDA 网共同推出，是最全面、系统、专业的 CST 微波工作室培训课程套装，所有课程都由经验丰富的专家授课，视频教学，可以帮助您从零开始，全面系统地学习 CST 微波工作的各项功能及其在微波射频、天线设计等领域的设计应用。且购买该套装，还可超值赠送 3 个月免费学习答疑…

课程网址：http://www.edatop.com/peixun/cst/24.html

HFSS 天线设计培训课程套装

套装包含 6 门视频课程和 1 本图书，课程从基础讲起，内容由浅入深，理论介绍和实际操作讲解相结合，全面系统的讲解了 HFSS 天线设计的全过程。是国内最全面、最专业的 HFSS 天线设计课程，可以帮助您快速学习掌握如何使用 HFSS 设计天线，让天线设计不再难…

课程网址：http://www.edatop.com/peixun/hfss/122.html

13.56MHz NFC/RFID 线圈天线设计培训课程套装

套装包含 4 门视频培训课程，培训将 13.56MHz 线圈天线设计原理和仿真设计实践相结合，全面系统地讲解了 13.56MHz 线圈天线的工作原理、设计方法、设计考量以及使用 HFSS 和 CST 仿真分析线圈天线的具体操作，同时还介绍了 13.56MHz 线圈天线匹配电路的设计和调试。通过该套课程的学习，可以帮助您快速学习掌握 13.56MHz 线圈天线及其匹配电路的原理、设计和调试…

详情浏览：http://www.edatop.com/peixun/antenna/116.html

我们的课程优势：

※ 成立于 2004 年，10 多年丰富的行业经验，

※ 一直致力并专注于微波射频和天线设计工程师的培养，更了解该行业对人才的要求

※ 经验丰富的一线资深工程师讲授，结合实际工程案例，直观、实用、易学

联系我们：

※ 易迪拓培训官网：http://www.edatop.com

※ 微波 EDA 网：http://www.mweda.com

※ 官方淘宝店：http://shop36920890.taobao.com