

Chapter 2

Computer Simulation

Introduction

There are many computer simulation programs available. The most accurate ones use Spice models, which include all relevant parameters. At lower frequencies programmes like the commercial “Multisim” and the free “Microcap” are useful, but for RF circuits computer simulation tools that incorporate transmission lines as elements, and are specially designed to handle high frequency effects are preferred. Typical examples are ADS by Agilent [1] and Microwave Office by AWR [2]. For these notes AWR Design Environment Version 8.03 [3] is used extensively. This is a very powerful computer simulation suite. At James Cook University, students have found this to be easier to become proficient at than ADS, which we used earlier.

Microwave Office is very good for RF simulations, in that many commercial RF transistors and diodes can be simulated using both linear models and non-linear (harmonic balance) models.

The linear models treat any passive and active devices as “linear” devices and use conventional circuit analysis techniques to solve the equations for the circuit to be simulated. As a result the simulation is fast but will not deal with non-linear effects such as those that occur in a mixer, oscillator or amplifier. Applying optimisation techniques to the linear simulation of the circuit allows the best performance to be obtained quickly.

Non-linear simulation uses a full Spice model to analyse the non-linear behaviour of a circuit. As examples, it allows the conversion loss of mixers to be calculated, and allows the variations of gain with signal amplitude in an oscillator to be calculated and it allows all the harmonics produced by the oscillator to be determined. Using the appropriate non-linear amplifier models, it allows the intermodulation distortion of an amplifier to be determined as a function of output power. Because this simulation is much more detailed, it takes significantly longer to complete. As a result, most designs are initially done using linear simulation models and then the final design is optimised using the harmonic balance models.

Computer simulation is a powerful tool for ensuring that a circuit has been designed to perform correctly. Normally one would firstly check the circuit under the ideal design conditions and then one would check that the circuit still performs correctly for all the component tolerances that are possible. Since the production of a PCB containing an RF circuit is quite expensive it is always good practice to fully simulate the circuit first.

The computer simulation process is illustrated by some examples.

Example 1: Low Pass Filter

Design a 4th order low pass filter and a high pass filter to have a cut off frequency of 250 MHz and an impedance level of 75 Ω , and in the second example combine these filters to form a VHF-UHF TV antenna diplexer.

For Butterworth filters the design equations are well known [4] and the components are:

$$C_i = \frac{1}{\pi F_c R} \sin\left(\frac{(2i-1)\pi}{2n}\right) \quad i = 1,3,5\dots \quad \text{Eqn. 1}$$

$$L_i = \frac{R}{\pi F_c} \sin\left(\frac{(2i-1)\pi}{2n}\right) \quad i = 2,4,6\dots \quad \text{Eqn. 2}$$

Where R is the desired load impedance and F_c is the cut off frequency. These equations can easily be entered into a spreadsheet, to give the results for the 250 MHz filter as follows:

Impedance	75		
Freq MHz	250		
Order	4		
Element			
	Normalised	Lowpass	Highpass
	Rs, Rn	R	R
	1 0.765367	75	75
	1 0.765367	C 6.497E-12	L 6.23838E-08
	2 1.847759	L 8.822E-08	C 4.59382E-12
	3 1.847759	C 1.568E-11	L 2.58402E-08
	4 0.765367	L 3.654E-08	C 1.10905E-11

Table 1. Butterworth filter element values

The highpass filter is obtained using the standard lowpass to highpass transformation described in most filter textbooks. We now enter these values into MWO, to do this we must create the required project as follows:

Open Microwave Office. This will then open up a blank project as shown in figure 1.

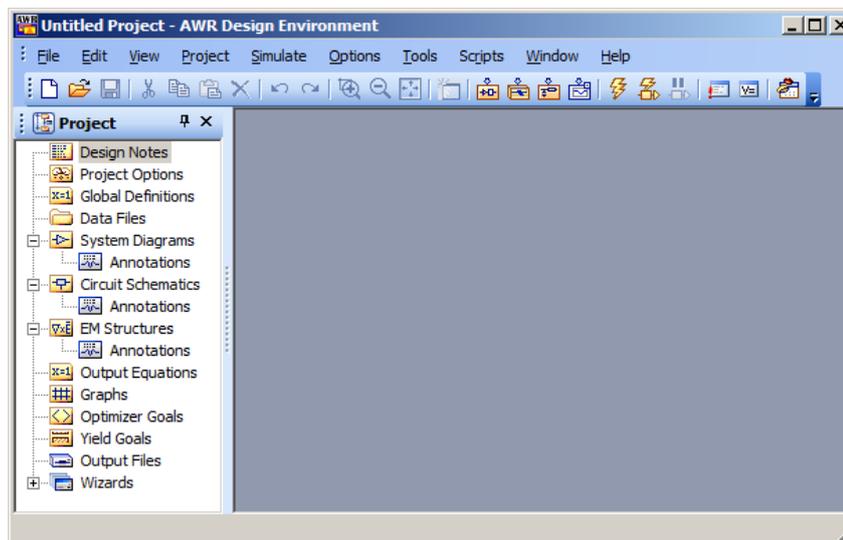


Figure 1. MWO blank project screen.

Select Project ⇒ Add Schematic ⇒ New Schematic and name the Schematic “LowPass”. Then select the Element window at the bottom left of the Design Environment window and select Lumped Elements ⇒ Capacitor and drag the CAP symbol into the circuit schematic as shown in figure 2.

Now place all the other capacitors and inductors in the correct location. When placing the component, a right mouse click rotates the component. Draw \Rightarrow Add Port allows the input and output ports to be added to the circuit. Draw \Rightarrow Add Ground allows a ground connection to be added. Double Left Click on a component allows the required value for that component to be entered. Repeat this for all the other components to give the final circuit as shown in figure 3.

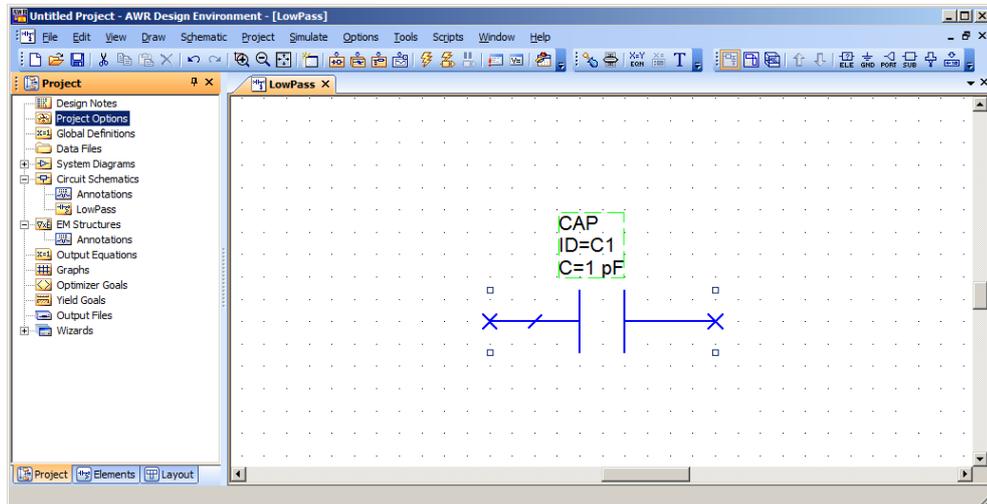


Figure 2, Capacitor placed.

Note that normally filters are designed with the smallest number of inductors, so that odd order filters with the end elements being capacitors are far more common. Since the filters are to be used as part of a diplexer consisting of a high-pass and a low-pass filter in parallel, the high-pass filter must not short out the low-pass filter at low frequencies and the low-pass filter must not short out the high-pass filter at high frequencies, so that an open circuit input impedance is required in the stopband of these filters. This is achieved with the first (series) element being an inductor for the low-pass filter, and the first element being a capacitor for the high-pass filter.

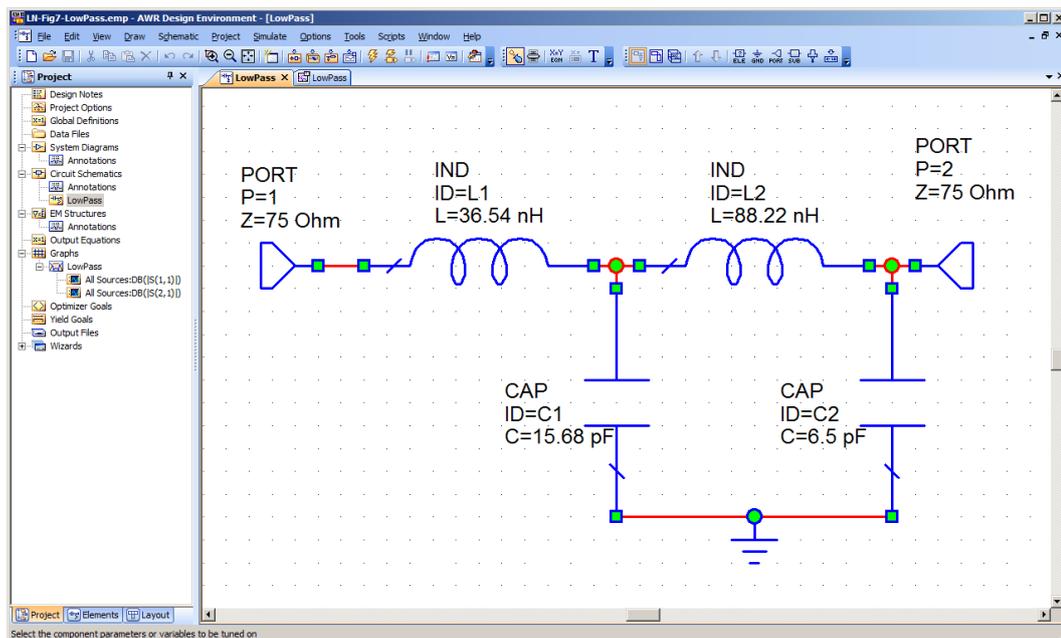


Figure 3. Low-pass filter circuit.

Select the project window again and select Graph and right mouse click and select add measurement. RF devices are normally specified by the Scattering (S) parameters, with S11 and S22 being the input and output return loss, indicating how well the device is matched and S21 and S12 being the forward and reverse gains. Select Linear Measurement type \Rightarrow Port Parameter and S parameter to obtain S parameter measurements of the filter, as shown in figure 4. Save the project as “LowPassFilter” to keep these values. Ensure that the impedance of the ports is changed from the default value of 50 Ω to 75 Ω .

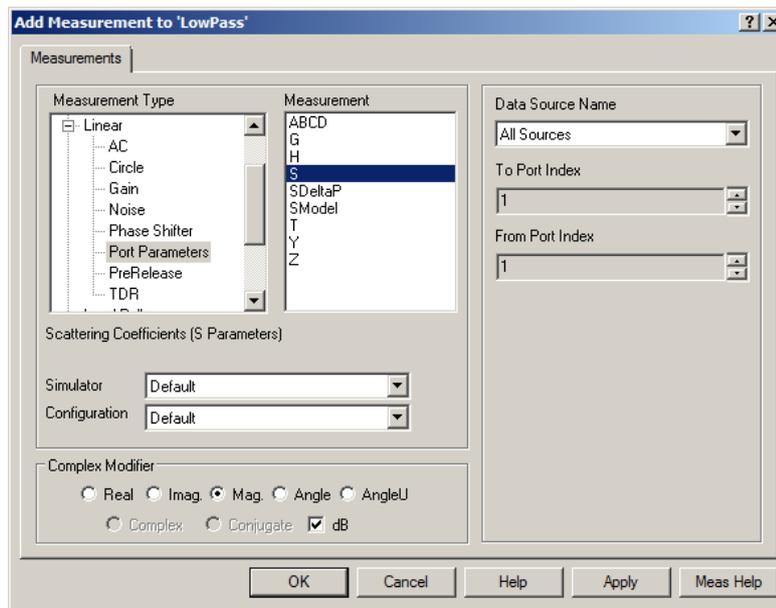


Figure 4. “Add Measurement” window.

Now set the operating frequency range using the Project options entry in the left bar. Set the Global Units frequency to MHz and then set the operating frequency range in the Frequencies window as shown in figure 5.

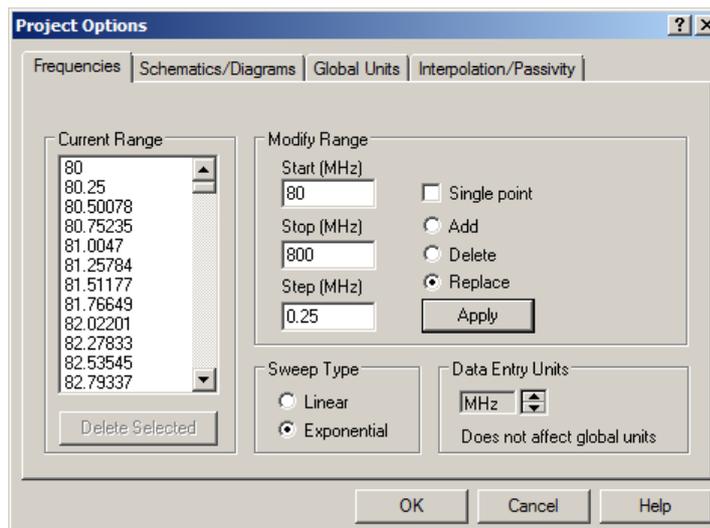


Figure 5. Project frequencies.

Select Simulate \Rightarrow Analyse to give the required frequency response as shown in figure 6. The scales, line thickness and other parameters of the graph can be set from inside the

Graph Property toolbar, (the left element of the right toolbar shown the blue area in figure 6).

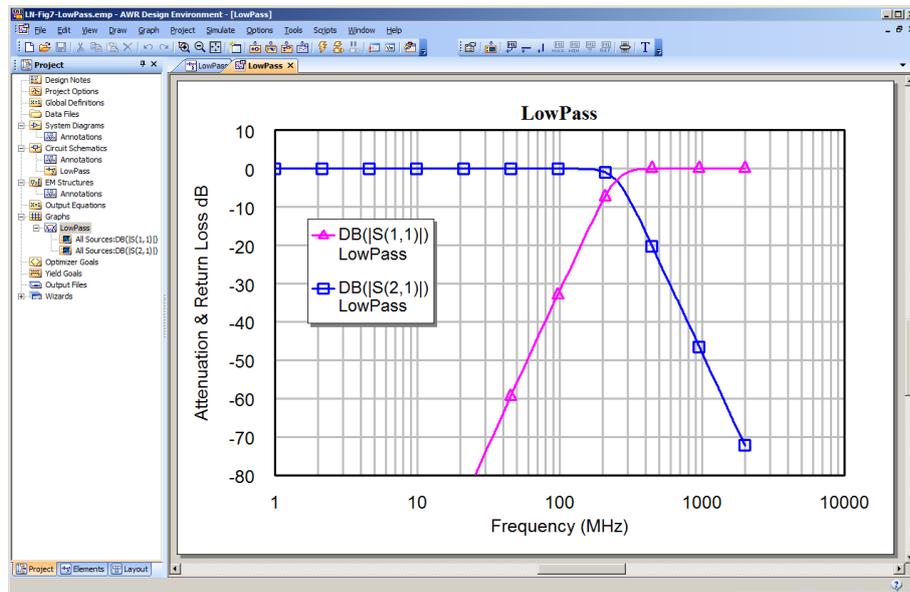


Figure 6. Frequency response of the low-pass filter.

The blue graph is the transfer function S_{21} and the magenta one is the input reflection coefficient S_{11} .

Equations in MWO

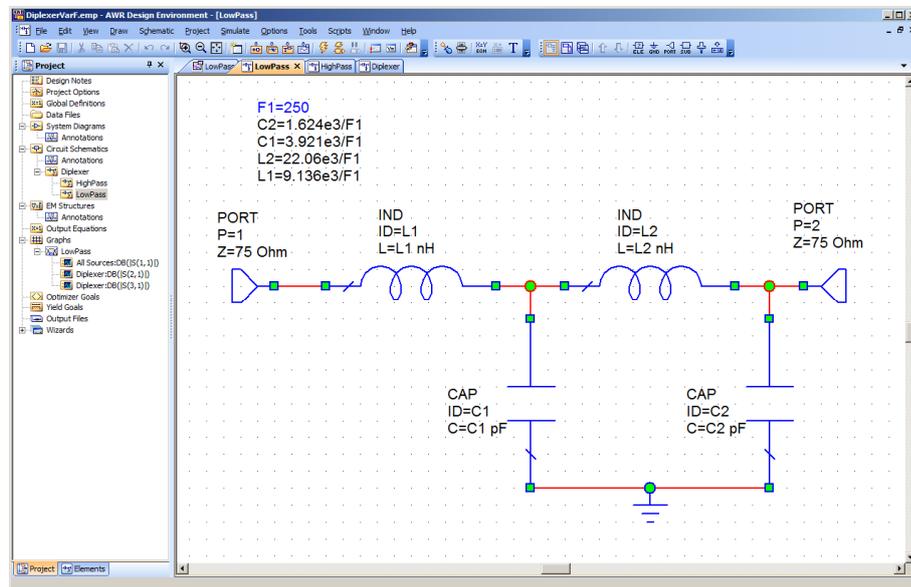


Figure 7. Tuneable filter.

It is possible to use variables and equations as part of the circuit diagram. This allows greater control over the filter parameters. For instance by making the cut off frequency a variable, the filter can easily be tuned to achieve a required cut off frequency. The cut off frequency can be tuned or optimised to meet required specifications. The resulting circuit diagram is shown in figure 7. The variables can be placed in any order, provided that any variable specification is above the variable usage. The frequency variable F1

must thus be on top, since that is used by C1, C2, L1 and L2. C1 must be higher than the capacitor using the variable C1 and so on. Unknown variables are shown in red. Any corrective action may simply be the moving around of a variable to satisfy this top to bottom flow requirements.

Example 2: Diplexer

It is required to design a Diplexer to permit VHF and UHF TV signals to be received by separate antennae and then combined to be fed to a TV. The corner frequency is chosen to be 250 MHz. The same circuit can be used as a splitter, to separate FM radio from UHF TV. A diplexer consists of a low pass filter selecting the low frequency components and a high pass filter selecting the high frequency components.

The low-pass filter can be the Butterworth low-pass filter of figure 7. A Butterworth High pass filter with a 250 MHz cut off frequency can be designed and entered into MWO.

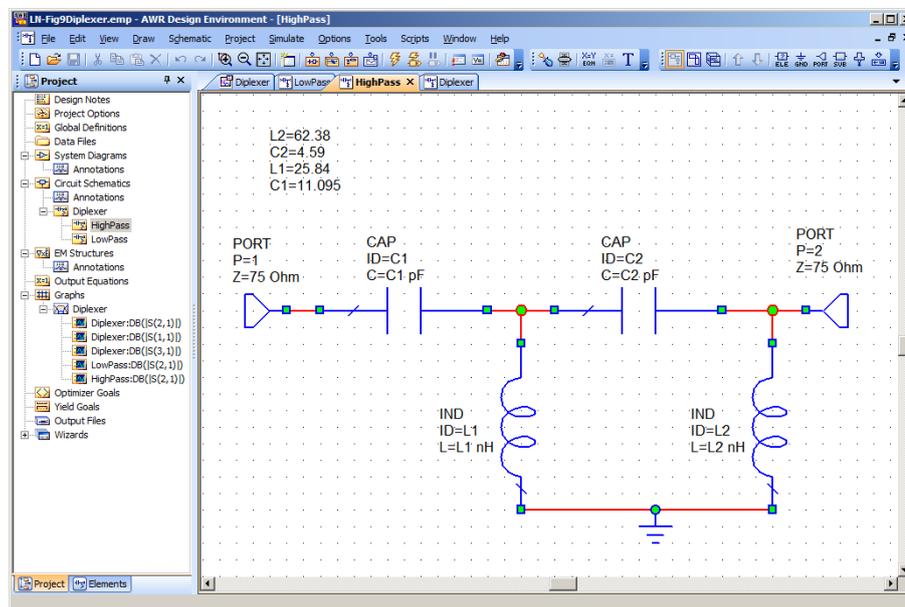


Figure 8. High-pass filter.

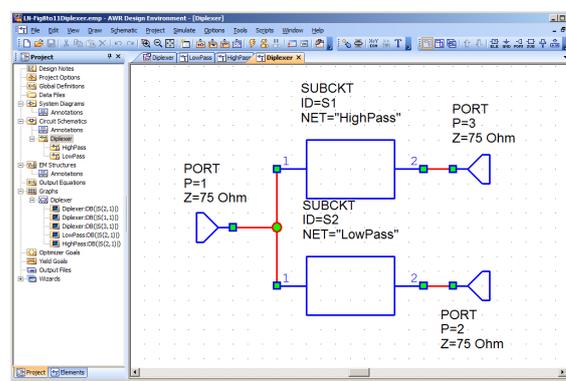


Figure 9. Diplexer using low-pass and high-pass sub-circuits.

The easiest way to produce the diplexer is to produce a separate high pass filter and then use that and the low pass filters as sub-circuits to produce the diplexer. The High pass filter uses the component values from table 1 and is shown in figure 8. The full diplexer

is shown in figure 9, and the resulting S parameters are shown in figure 10. Since the high pass filter and low pass filter are in parallel, the low pass filter must have an open-circuit input impedance at high frequencies and the high pass filter must have an open circuit input impedance at low frequencies. This is achieved by using filters that have the first element as a series element for the input.

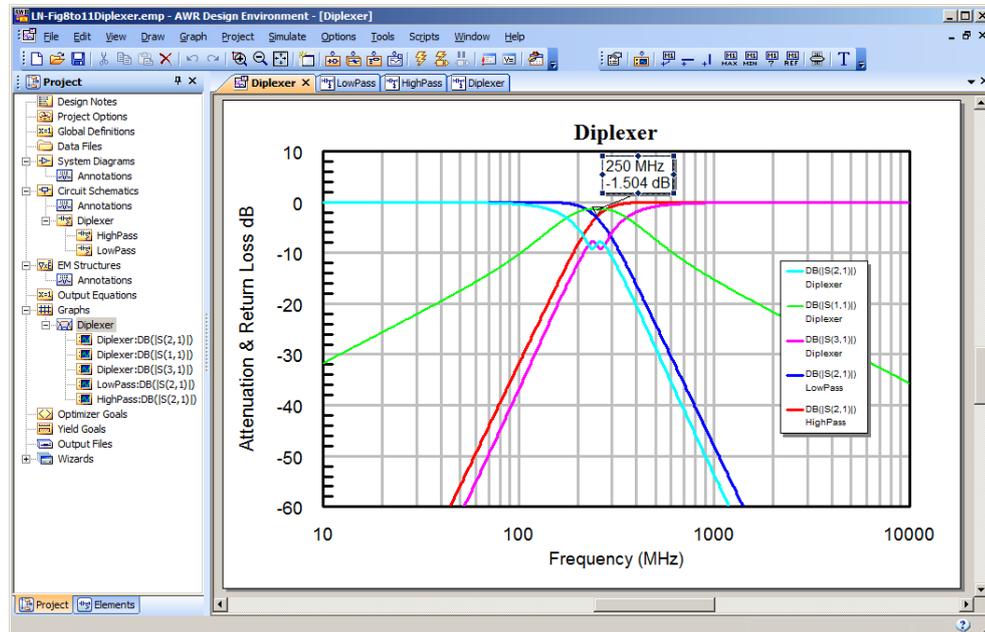


Figure 10. Frequency response of the diplexer.

Optimisation

As can be seen from figure 10, return loss (S_{11}) is only -1.4 dB near the cut off frequency of the filters so that the input is poorly matched. By comparing the blue curve of the low pass filter by itself, with the cyan curve of the low pass filter as part of the diplexer, it can be seen that there is a significant interaction between the lowpass and highpass filters. There is a similar degradation in performance of the highpass filter, as shown by the red and magenta traces. One can use the optimisation capability to improve the performance of the diplexer at the crossover frequency range. The first task is to specify the optimisation goals. For this design the requirements are that S_{11} is less than -20 dB over the entire VHF and UHF TV frequency range. In Australia the VHF TV frequencies range from 46 MHz to 230 MHz and the UHF TV frequencies range from 527 MHz to 820 MHz. Similar frequency ranges are used in other countries. For the example here 250 MHz is used as the boundary between VHF and UHF. As a result S_{21} (low-pass transfer function) is to be greater than -3 dB for frequencies below 250 MHz and S_{31} (high-pass transfer function) is to be greater than -3 dB for frequencies above 250 MHz.

These optimisation goals are entered in the optimisation goal function as shown in figure 11. Double click on the elements to be optimised and selecting the “opt” checkbox in the element options window, for the component or the variable linked to a component as shown in figures 7 and 8, to ensure that the element can be optimised. In addition some attenuation limits at 100 MHz and -40 dB and 625 MHz and -40 dB are added, to ensure a steep rolloff for the diplexer. The weighting of the optimisation limits for S_{21} and S_{31} are made 1000 since we want to ensure that these limits are satisfied.

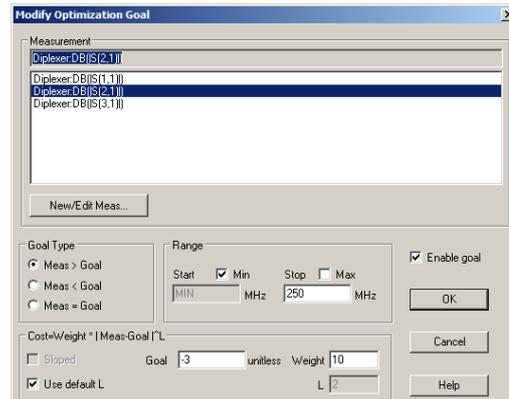


Figure 11. Specification of optimisation goals.

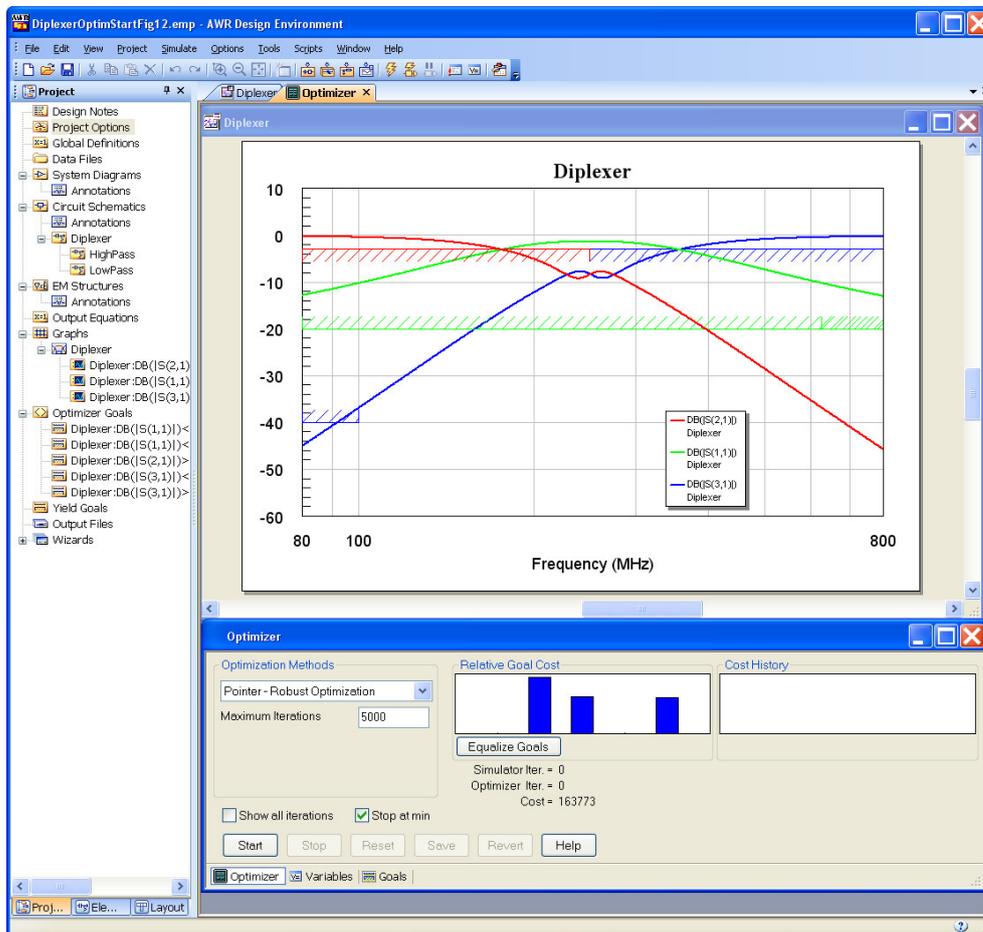


Figure 12. Optimisation criteria and the diplexer performance before optimisation.

Simulate \Rightarrow Optimise is selected to start the optimisation. For the first part of the optimisation Pointer-Robust optimisation is a good optimisation method to use. Once the requirements are met, the S_{11} limit is progressively reduced from -20 dB to -35 dB. Figure 12 shows the optimisation starting performance with the optimisation goals and the Optimizer panel

When the Pointer-Robust optimisation becomes stuck, so that the cost function does not decrease, it is worthwhile to change some conditions since the optimisation may be stuck in a local minimum rather than the global minimum for the cost function. Changing the weight for one or more of the goals or changing the optimisation function

may get the optimisation out of the local minimum. The “random local” optimiser is good for a complex optimisation function that will exist close to the global optimum condition. Figure 13 shows the optimisation criteria and the performance after the optimisation has been completed. It can be seen that the diplexer meets all the required specifications and that it is even possible to meet a -35 dB return loss on S_{11} .

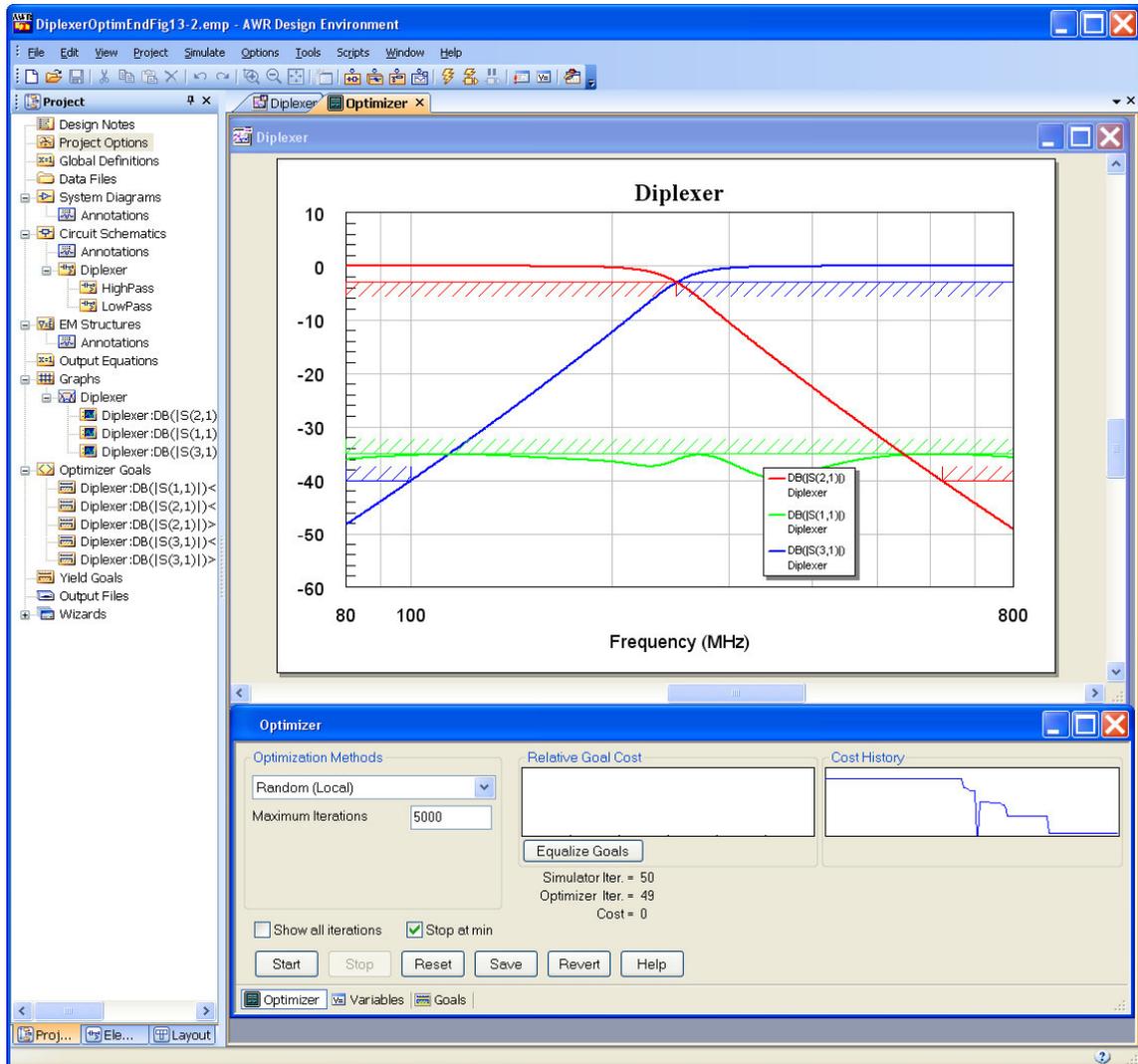


Figure 13. Optimisation criteria and the diplexer performance after optimisation.

The component values of the filter after optimisation for the low-pass sub-circuit are shown in figure 14 and those for the high-pass sub-circuit are shown in figure 15. Note that the component values are significantly different from the starting values using the Butterworth filter table values. The component values are very practical, so that the diplexer can easily be made

It is possible to replace the inductors using microstrip lines as outlined in the lecture notes on RF filters. That allows this diplexer to be made on a low cost FR3 or FR4 PCB using only capacitors and microstrip lines. If this is done then the length and impedances of the microstrip lines must be further optimised to meet the same specifications as this diplexer.

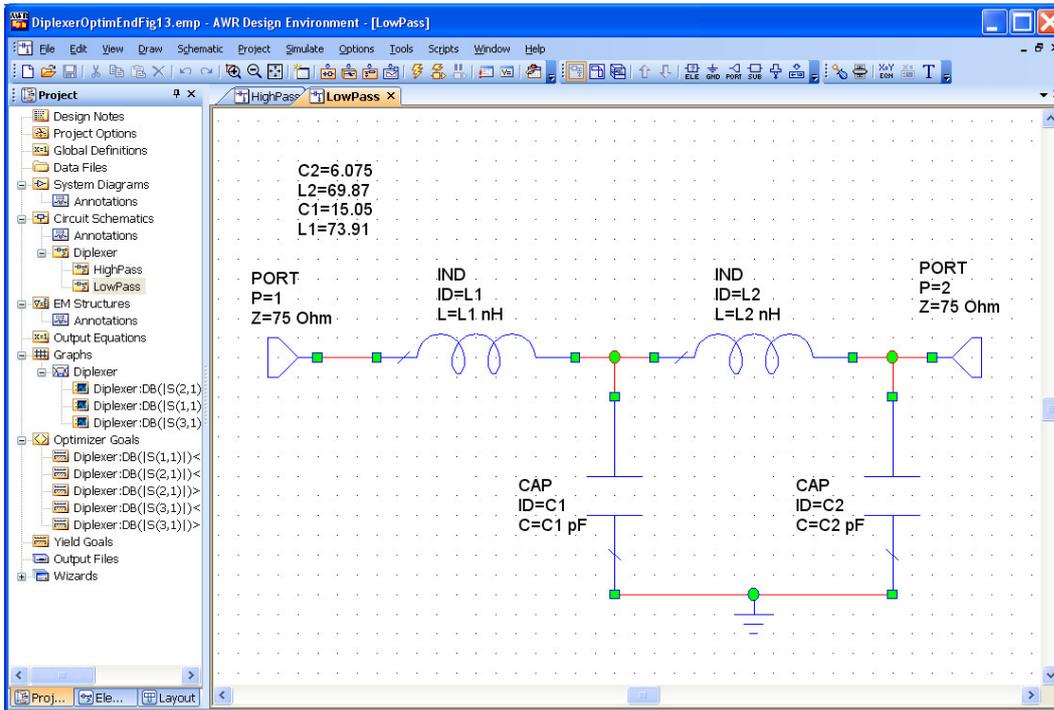


Figure 14. Final low-pass filter for diplexer.

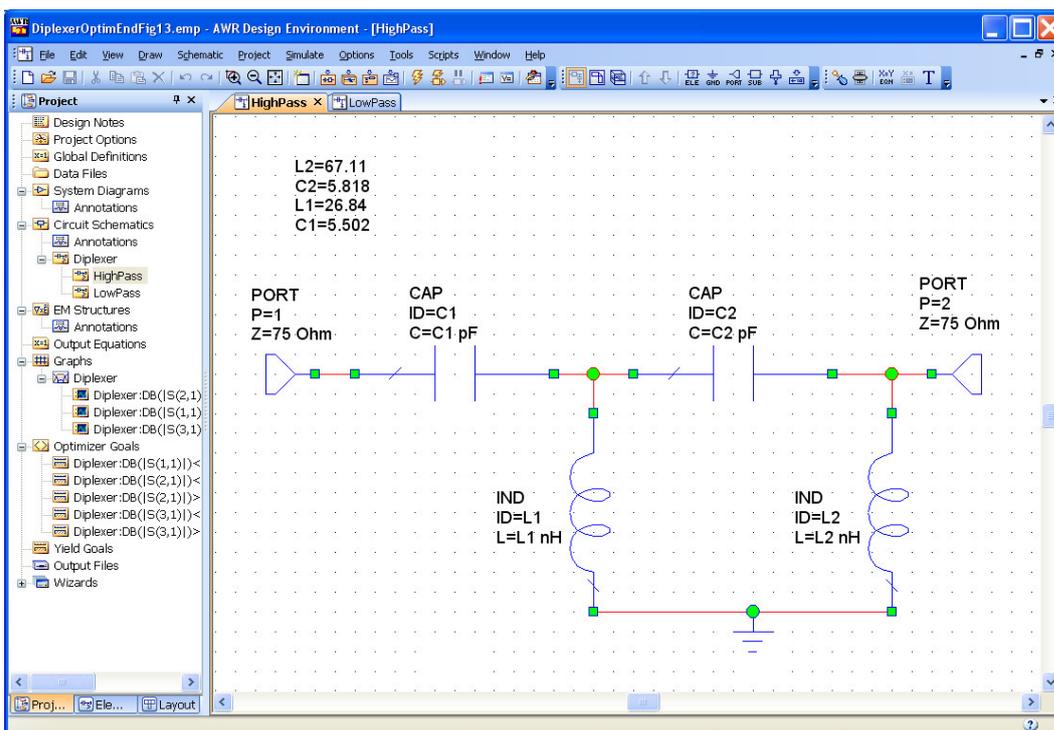


Figure 15. Final high-pass filter for diplexer.

Example 3: Amplifier

For the second example an amplifier is designed and its performance is determined. The design is to use a 2N4124 transistor. The transistor data can be obtained from the manufacturer's (Fairchild) web site. Since the 2N4124 transistor is not included in the standard MWO parts list, the parameters for the transistor model must be entered manually. If the full model is entered, the quiescent voltages can be determined by

MWO using the harmonic balance simulation. This is very useful to ensure that the biasing is correct. Figure 16 shows the circuit of a simple amplifier using the 2N4124 transistor and the manufacturer's data. Many manufacturers of RF transistors now give full transistor models for MWO or ADS from their web site.

The most critical of any transistor design is the biasing. By right clicking on the Transamp circuit schematic and selecting "Add Annotation" and then selecting "DCVA_N" will provide voltages at all the nodes in the circuit, to enable the biasing voltages to be checked and compared with the design values. The node voltage annotation is shown in figure 16. Similarly currents in and out of all the nodes can also be shown. If needed the effect of supply voltage and component variations can be investigated, to ensure that the design works properly for all expected component tolerances.

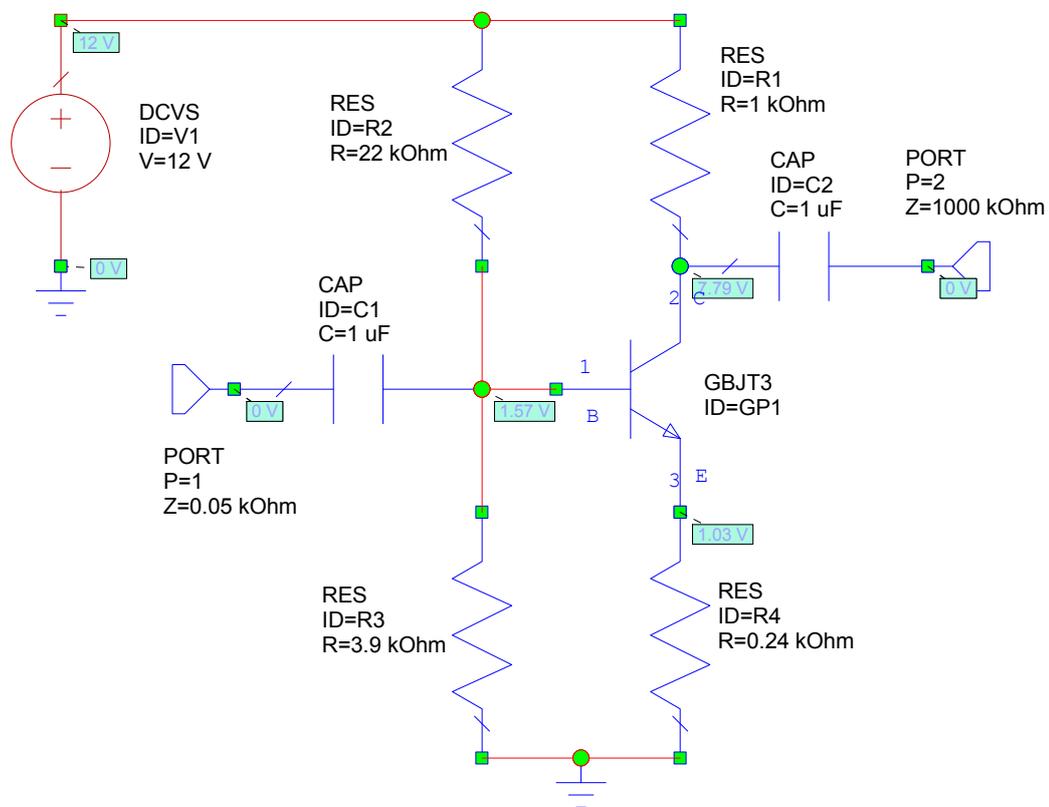


Figure 16. MWO transistor amplifier schematic diagram.

The resulting frequency response of the amplifier is obtained by selecting Graph \Rightarrow New Graph \Rightarrow Rectangular. Then right clicking in on that graph label in the Project Browser and selecting Add Measurement to add the measurement required. In this case VTG (Voltage gain from input to output terminal) with the magnitude response in dB is selected. The frequency range over which the measurements are made are set by double clicking on "Project Options" in the Project Browser and then specifying the frequency range. In this case The data entry units are set to kHz and the start frequency is set to 0.03 kHz and the stop frequency is set to $3e5$ kHz and the exponential step size is set to 0.03 kHz. The resulting frequency response from the amplifier is shown in figure 17. The results from MWO are virtually identical to those from Microcap. Selecting Graph

⇒ Marker ⇒ Add Marker, allows markers to be added to the graph. These markers can search for the maximum value of the voltage, or for specific attenuation values or frequencies. Using these markers, the amplifier has a gain of 11.9 dB and an upper 3dB point of 94.4MHz.

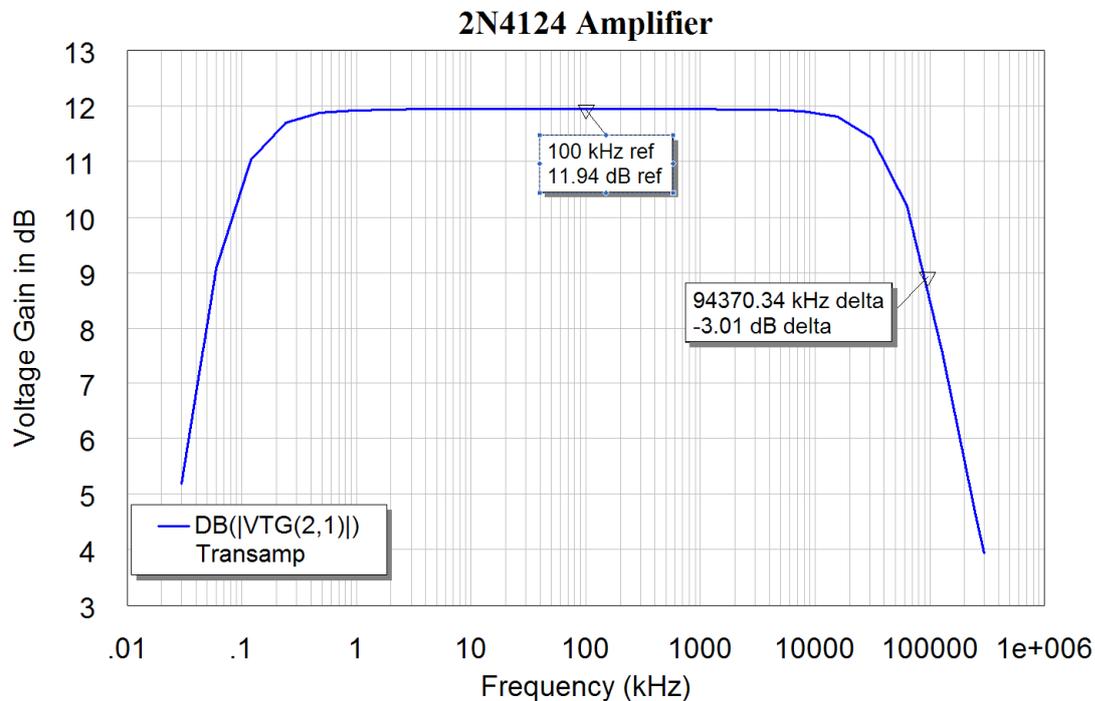


Figure 17. Frequency response from Microwave Office.

Transistor Models

Two types of transistor models can be used, firstly a very simple closed form model, like the Hybrid, Pi or T models used in basic electronic design subjects. For the simple closed form transistor model, MWO treats the transistor as a linear device and only AC analysis can be performed. The model used by MWO is as follows:

Equivalent Circuit

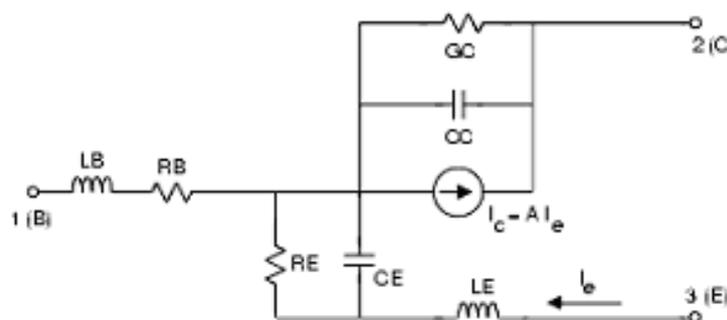


Figure 18. Circuit of the closed form transistor model from Microwave Office [5].

Parameters

Name	Description	Unit Type	Default
ID	Element ID	Text	B1
A	Magnitude of DC current gain (alpha)	None	0.8
T	Current gain time delay	Time	0 ns
F	-3dB frequency for current gain	Frequency	0 GHz
CC	Collector capacitance	Capacitance	0 pF
GC	Collector conductance	Conductance	0 S
RB	Base resistance	Resistance	1 Ohm
LB	Base inductance	Inductance	0 nH
CE	Emitter capacitance	Capacitance	0 pF
RE	Emitter resistance	Resistance	1 Ohm
LE	Emitter inductance	Inductance	0 nH

Figure 19. Parameters of the closed form transistor model from Microwave Office.

Alternately a more sophisticated Spice model can be used. Spice models for a wide variety of transistors can be obtained from Fairchild Semiconductors. The model for the 2N4124 transistor obtained from the Fairchild web site is as follows:

```
.model 2N4124 NPN(Is=6.734f Xti=3 Eg=1.11 Vaf=74.03 Bf=495 Ne=1.28
Ise=6.734f Ikf=69.35m Xtb=1.5 Br=.7214 Nc=2 Isc=0 Ikr=0 Rc=1 Cjc=3.638p
Mjc=.3085 Vjc=.75 Fc=.5 Cje=4.493p Mje=.2593 Vje=.75 Tr=238.3n Tf=301.3p
Itf=.4 Vtf=4 Xtf=2 Rb=10)
* Fairchild pid=23 case=TO92
* 88-09-08 bam creation
```

Circuit simulation software like Multisim, MicroCap and MWO all use the same basic Spice transistor model, with Multisim using exactly the same model as produced by Fairchild. Microcap uses some extra parameters. MWO uses a slightly more sophisticated model, which is better for high frequencies and includes:

- 1 Terminal inductances LB , LC , and LE ,
- 2 Complete collector-to-substrate diode parameters, CJS , VJS , MJS , ISS , NS , and TS ,
- 3 A different (but equivalent) formulation for nonlinear base resistance,

Equivalent Circuit

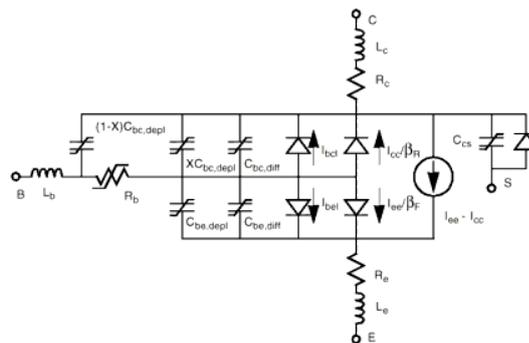


Figure 20. Circuit diagram of the Gummel-Poon Transistor model From Microwave Office.

The Gummel-Poon Transistor model [5] included in Microwave Office has the circuit diagram as shown in figure 20. The model is significantly more complex than the simple closed form model shown two pages previously. This model is however far more accurate. Note the transistor lead inductances, which have significant impedances at higher frequencies.

The parameters for the Gummel-Poon transistor model can be accessed by double clicking the transistor as shown in figure 21 and selecting secondary properties in the parameter window [6]. Using an existing transistor as a starting point, the model for another transistor can be produced by editing the secondary parameters and then saving that with a new part number.

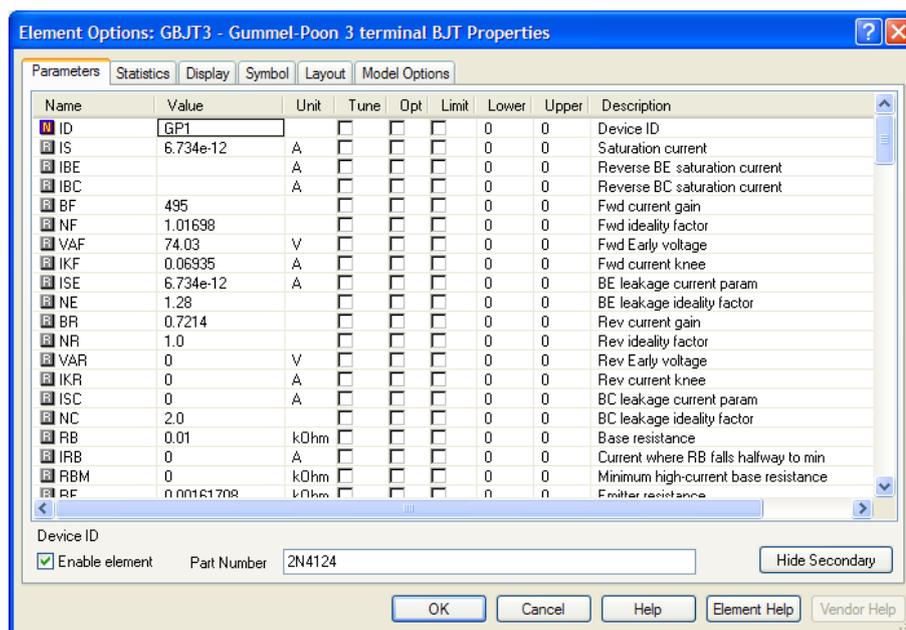


Figure 21. Parameters from the Gummel-Poon Transistor model From Microwave Office.

Determination of Line Parameters

In many applications a printed circuit layout using microwave printed circuit board materials needs to be made. The widths and lengths required can be determined from a program Txline, which is part of Microwave Office. ADS has a similar program called Linecalc. Both these programs do not cover all the different coupled lines or strip lines available. In many cases it is more convenient to use the schematic circuit realisation of the transmission line models to determine the required line parameters, rather than using Txline or Linecalc. This technique is illustrated by the following example of a quarter wavelength microstrip line with a 50 Ω impedance at 1 GHz.

The calculations for the width of a 50 Ω microstrip line using Txline for a Rogers RO4003 substrate, with a dielectric constant of 3.38, a loss tangent of 0.0027, a substrate thickness of 0.8128 mm and 35 micron (1 oz) copper cladding, is shown in figure 24. This shows that a quarter wavelength microstrip line with a 50 Ω impedance at 1 GHz requires a track width of 1.84308 mm and should be 45.9755 mm long.

The same line width determination can be done by making some very simple test circuits and optimising the relevant parameters. Figure 22 shows a schematic for determining the correct line width for a 50 Ω track on a RO4003 substrate. When the line is exactly 50 Ω , then the return loss (S_{11}) is very small regardless of frequency, as

described in chapter 4 of these notes [7]. The same PCB parameters that are used in Txline are used for the substrate element in the circuit schematic of figure 23. Tuning or optimising the line width to give the lowest return loss S_{11} results in a -60 dB return loss for a 1.851 mm line width. That width is close to the result from Txline.

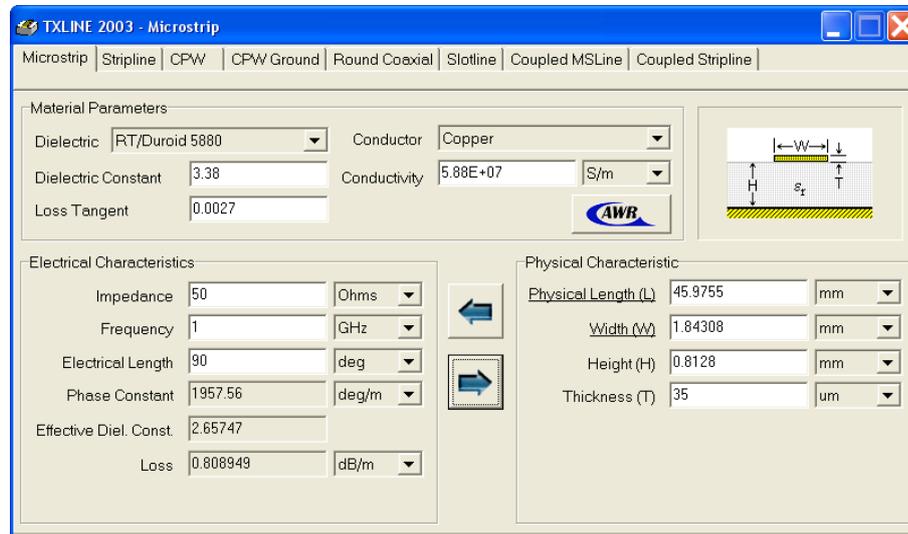


Figure 22. Txline program for calculating line parameters.

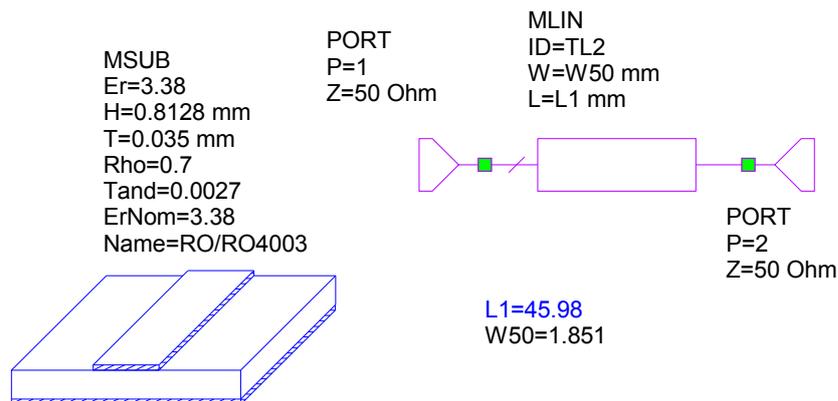


Figure 23. Circuit for determining Microstrip line widths.

When the length of a quarter wavelength long line needs to be determined, then the circuit of figure 24 can be used. The open circuited stub will reflect as a short-circuit when the length of the stub is one quarter wavelength. Using the width as determined in figures 23 or 24, the length is now tuned or optimised to produce a notch at the required frequency, as shown in figure 25. The T junction can be included to ensure that the actual layout, including the required junctions, performs exactly as required. This technique is very useful for obtaining the correct length when the stub has bends in it, as is the case for folded branch-line couplers as used in the first laboratory session for these notes [8].

For many circuits, simple test circuits like those of figures 23 to 25 can be used to determine the basic parameters of stripline or microstrip lines that can then be used as part of a more complex circuit. This technique for determining microstrip, or stripline,

parameters from circuit schematic measurements is used in several places throughout these lecture notes.

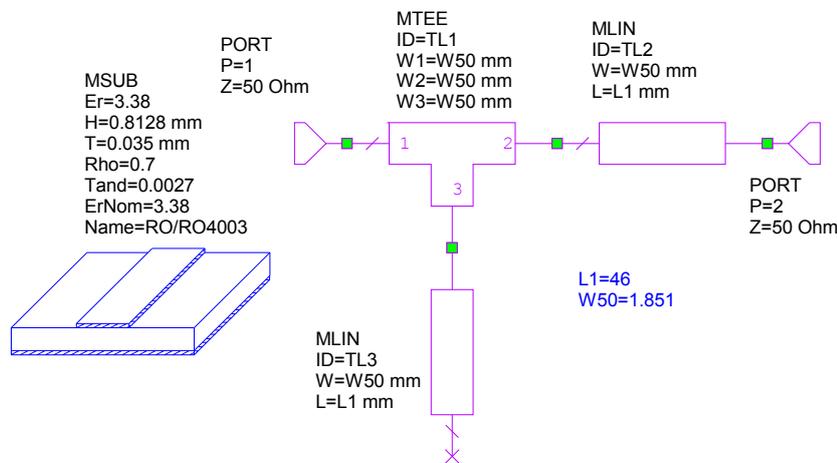


Figure 24. Circuit for determining the length of one quarter wavelength.

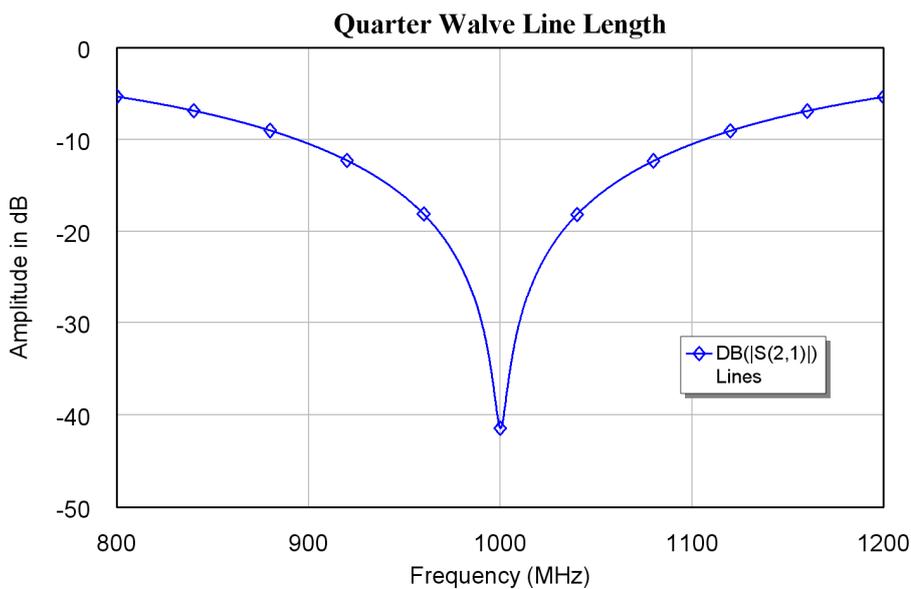


Figure 25. Frequency response of the circuit in figure 24.

Summary

This chapter has introduced some basic aspects of computer simulation using Microwave Office. During the rest of these notes, the computer simulation will be expanded with both linear and non-linear applications and examples.

MWO has a good interactive help capability, with specific information available for every circuit element. The interactive element help can be accessed by right clicking on the element and selecting “Element Help”. In addition the examples included with the AWR software will assist in understanding the operation of RF computer simulation software.

References

1. Agilent EEsof EDA <http://eesof.tm.agilent.com/>
2. AWR Corp: MWO, VSS Design Software <http://web.awrcorp.com/>
3. Microwave Office <http://web.awrcorp.com/Usa/Products/Microwave-Office/>
4. Zverev, A. I. "Handbook of filter Synthesis", Book Publisher John Wiley & sons 1967.
5. H. K. Gummel and R. C. Poon, "An Integral charge control model of bipolar transistors", Bell Syst. Tech. J. Vol 49, pp827-852, May-June 1970.
6. AWR Corp. "MWO Help files: MWO/AO Element Catalog>Nonlinear>Gummel-Poon BJT:GBJT"
7. Kikkert C. J. "RF Electronics : Transmission Line Transformers and Hybrids" Chapter 4, Available through AWR.
8. Kikkert C. J. "RF Simulation using Microwave Office" Practical Session 1, Available through AWR.