Chapter 5

Microwave Design

In this chapter :

- Modern design of band pass filters made from coupled lines
- Using TRL85 for synthesis and analysis of microwave problems

One of the things that deters amateurs from constructing microwave equipment is that at times it seems to be a black art. At lower frequencies the physical layout of a circuit has less influence on performance and components such as capacitors and inductors can be recognised. Microwave circuits are more complex and until recently design was not easy. There are now many CAD packages available to assist with the design and implementation of microwave circuits. This chapter contains some articles that introduce how to use some of these products using worked examples. Even if you do not intend to design microwave circuits, an understanding of how they are designed will help when a kit or ready built circuit does not work as expected and needs "tweeking".

Modern design of band pass filters made from coupled lines, Gunthard Kraus, DG8GB

Nowadays there are many aids available to any electronics developer. Even for development work in the area of high frequency engineering, there is some very powerful software in existence, some of which is available without charge on the Internet. The use of modern design and simulation tools is described below by means of examples.

Introduction

Some years have passed since the series of articles entitled "Design and realisation of microwave circuits" in [1], where this subject was dealt with comprehensively. In the intervening period, the options for finding information and for circuit simulation have multiplied greatly. In addition, the analysis options are more precise, thanks to continuous improvements in the CAD field.

A "test version" or "student version" of almost any modern CAD or simulation programs can now be obtained from the Internet, including the original manuals. These are usually complete textbooks in themselves - and mountains of application notes on almost any subject. The real problem becomes how to make a suitable choice. "Know-where and know-how" are also important, because all test versions of what are usually very expensive programs have some kind of limitations. And there's nothing more irritating than to slog away familiarising yourself with a new program and then suddenly realising that the program available just cant go any further with the problem you're working on.

So the idea here is to demonstrate the correct and successful design of stripline band-pass filters, together with their implementation in practice. We shall compare not only the procedures but also the degrees of success, using the tried and tested CAD program "PUFF" (Version 2.1) and the ultra-modern student version of ANSOFT Serenade.

A glance at the technology

Band pass filters serve to separate out a specific frequency range, while simultaneously suppressing, as far as possible, all undesirable signals outside this range. The following filter models can be considered for the microwave range in this context:

- Waveguide filters (for very high power levels)
- Coaxial filters
- Helix filters
- Filters made from ceramic resonators
- SAW filters
- Inter-digital band-pass filters
- Stripline filters with coplanar structures
- Microstrip filters made from coupled lines
- Hairpin filters, etc.

If we also lay down additional requirements, such as:

- DIY manufacture as simple printed circuit board at lowest possible cost
- easily convertible to other frequencies without high costs or problems
- no smoothing
- absolute reproducibility

then the two last types are usually given preference. In this context, hairpin filters represent a modified version of the standard stripline filter for shortening the construction length and increasing the edge sharpness. The disadvantages of larger dimensions must be taken into account.

Principles of stripline band pass filters made from coupled lines

We are using "coupled lines", i.e. two striplines which are running in parallel and close together. Due to running closely together in parallel, we obtain not only a capacitive coupling (via the electrical field) from one line to the other, but also a magnetic coupling. The magnetic field of one line induces an electrical voltage in the second line and thus transfers electrical power. The remarkable thing here is that the different waves triggered through this coupling from one line onto the other are added together in one direction only. But in the other direction they are in anti-phase and try to cancel each other.

This is precisely the behaviour of a directional coupler and it is also the main applications area for this line structure. This behaviour can be used to separate forward and return waves in a system!

However, the description of such a component for simulation can be expanded further, Due to the fact that some of the waves triggered in the second line are in-phase and some are anti-phase, it is necessary to specify two different impedance levels, namely

- the EVEN impedance (or: in-phase impedance)
- the ODD impedance (or: anti-phase impedance)

The EVEN impedance level is always higher than the system impedance (usually 50Ω), whereas the ODD impedance is essentially lower than the system impedance used.

The relationship of the three impedance levels to one another always depends on the formula:

$$Z^2 = Z_{EVEN} \cdot Z_{ODD}$$

Note:
As soon as one impedance is specified, many CAD programs react in the following way. If the impedance exceeds the system impedance, Z, then it is indicated as the EVEN impedance, and the missing ODD value is calculated in accordance with the above formula - and vice versa.

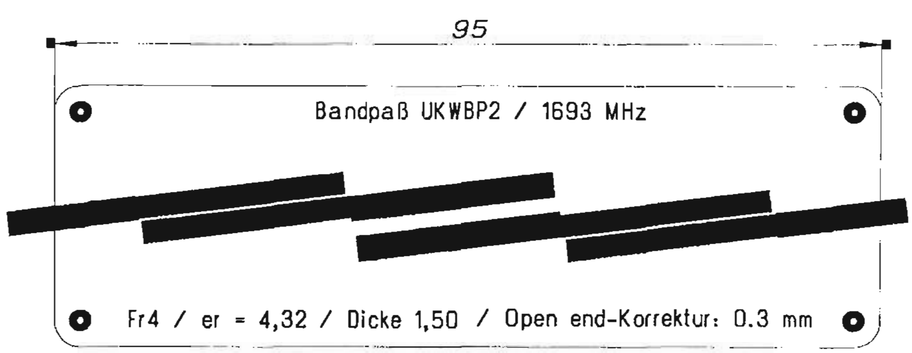

Fig 1: PCB layout for a 1693MHz band pass filter.

Now for stripline band-pass filters

If several such coupled line pairs, with 90 degree electrical length, are connected together in series, the line sections act as resonators and the input signal is transmitted from input to output, only in the range around this frequency. Thus the desired band-pass behaviour is obtained. Unfortunately, this is repeated at the odd multiples, i.e. for example the triple frequency, etc.

For the practical implementation, see Fig. 1.

It can be seen that, in addition to the three coupled line pairs, the 50Ω striplines are also used as connection to the SMA sockets. The underside of the printed circuit board is a continuous earth surface.

The design technique: from the standard low pass to the stripline band pass filter

Preliminary work

The circuit developer is initially faced with the following decisions:

Which type of filter is the correct one? The choice will be, for example, Bessel, Butterworth, or Chebyshev filters:

- Chebyshev filters display ripple in the pass band range, but as against that they can offer good edge sharpness for the transition into the filter stop band.
- If, in contrast, we need better group delay behaviour and no ripple in the transmission range, we go for Butterworth filters, though their edge sharpness in the filter stop band is markedly lower than that of the Chebyshev type.
- If the filter has to remain as gentle and as smooth as possible at all points, that leaves only the Bessel filter. Mind you, we pay for this "gentle" behaviour with a very "tired" transition from the pass band to the stop band (in order to keep phase distortion as low as possible). Thus, there is scarcely any edge sharpness to speak of in the filter stop band.

Then comes the question of the degree of filtration, N, which for normal low pass filters directly corresponds to the number of components needed. A greater degree of filtration brings about sharper edges in the filter stop band, but in practise the attenuation in the pass band is also increased, due to the greater number of components and their losses.

In practice, the type of filter used very frequently is the Chebyshev, with N between 3 and 5. For this reason, a filter from this group is taken as an example here.

The next decision relates to the system impedance (usually 50Ω). Moreover, especially for Chebyshev filters, the maximum pass band ripple, the reflection factor, etc, must be determined.

It should be borne in mind that the variables:

- Pass band ripple (oscillations of S21 and / or the transmission loss)
- reflection factor r
- voltage standing wave ratio VSWR
- S11
- reflection attenuation a_r

are inseparably associated with each other in the Chebyshev type! The following relationships apply:

- Between the reflection factor r and the voltage standing wave ratio VSWR:

$$r = \frac{VSWR - 1}{VSWR + 1}$$

- Between the reflection factor r and the pass band ripple (maximum transmission loss in dB)

$$a_{MAX} = 10 \cdot \log \frac{1}{1 - |r|^2}$$

- Between S11 and / or S22, the reflection factor r and the reflection attenuation a_r

  With correct matching, S11 and / or S22 correspond precisely to the reflection factor of the filter, but are normally specified in dB:

$$S11 = S22 = 20 \cdot \log|r|$$

  The reflection attenuation is then simply the "negative dB value of S11 or S22"! Correctly:

$$a_r = 20 \cdot \log \frac{1}{|r|}$$

The following summary table (drawn up in accordance with the above formula) is intended to serve as a small design aid:

Reflection factor |r|	Reflection attenuation a_r	S11 or S12	Chebyshev ripple of transmission loss
50%	6dB	6dB	1.25dB
20%	14dB	14dB	0.177dB
10%	20dB	-20dB	0.436dB
5%	26dB	-26dB	0.01dB
2%	34dB	-34dB	0.0017dB
1%	40dB	-40dB	0.00043dB
0.5%	46dB	-46dB	0.0001dB

(In practise, a maximum reflection factor between 5 and 10 % is usually sufficient...)

Designing a GPS bandpass filter

A bandpass for GPS with the following data is intended to serve as a design example:

Filter type:	Chebyshev
Mean frequency	f_0 = 1575MHz
Lower limiting frequency	f_{min}=1550MHz as ripple limiting frequency
Upper limiting frequency	f_{max}=1600MHz as ripple limiting frequency
Degree of filtration	N = 3
System impedance	Z = 50Ω
Max. reflection factor	|r| = 10%
Max. ripple	a_{max} = 0.0436dB
Reflection attenuation in pass band:	a_r = 20dB
S11 in pass band, if at all possible less than	S11 = -20dB

The selected reflection factor r = 10% gives a maximum ripple of 0.0436dB in the pass band.

This means that S21 can fall as low as -0.0436dB, whilst S11 and S22 never exceed 20dB.

Note:
The following calculation technique is taken from the book, Microwave Engineering by David Pozar [5], Page 484.

Additional note:

The degree of filtration, N, should always be selected to be odd (i.e. 3, 5, 7...), because the source resistance and the load resistance is identical under these conditions. Apart from this, make sure that the number of coupled line pairs is always 1 more than the selected degree of filtration. For N=3, there must thus be four line pairs.

And now to the individual design steps:

1st step:

First we need the filter coefficients of a single low-pass filter for this case. For this we can, for example, use the "faisyn" program (obtainable, for example, from http://www.rfglobalnet.com).

The above filter data are entered in succession when the program makes the corresponding requests, and the option "Parallel Capacitor" is selected. Thus the following table is finally obtained, with the 4 coefficients required for the calculation (Fig. 2):

g1 = cap1 = 0.8532
g2 = Ind1 = 1.1038
g3 = g1 = cap2 = 0.8532
g4 = normalised load resistance=1.00

2nd step:

Specification of fractional bandwidth of pass band:

$$\Delta = \frac{f_{\max} - f_{\max}}{f_{mean}} = \frac{1600-1550}{1575} = \frac{50}{1575} = 0.031746$$

```
Normalized Lowpass Prototype Filter Components:
Wc=1 rad/sec, Normalized Source Resistance=1
CAP 1  0 C=     0.8532
IND 1  2 L=     1.1038
CAP 2  0 C=     0.8532
The Normalized Load Resistance= 1
Press any key to continue...
```

Fig 2: Filter coefficients from Faisyn.

3rd step:

Now we come to the admittance inverter constants for the four line pairs:

1st line pair:

$$Z_0 \cdot J_1 = \sqrt{\frac{\pi \cdot \Delta}{2 \cdot g_1}} = \sqrt{\frac{\pi \cdot 0.031746}{2 \cdot 0.8532}} = 0.24175$$

2nd line pair:

$$Z_0 \cdot J_2 = \frac{\pi \cdot \Delta}{2 \cdot \sqrt{g_1 \cdot g_2}} = \frac{\pi \cdot 0.031746}{2 \cdot \sqrt{0.8523 \cdot 1.1038}} = 0.05138$$

3rd line pair:

$$Z_0 \cdot J_3 = \frac{\pi \cdot \Delta}{2 \cdot \sqrt{g_2 \cdot g_3}} = \frac{\pi \cdot 0.031746}{2 \cdot \sqrt{1.1038 \cdot 0.8532}} = Z_0 \cdot J_2 = 0.05138$$

4th line pair:

$$Z_0 \cdot J_4 = \sqrt{\frac{\pi \cdot \Delta}{2 \cdot g_3 \cdot g_4}} = \sqrt{\frac{\pi \cdot 0.031746}{2 \cdot 0.8532 \cdot 1.00}} = Z_0 \cdot J_1 = 0.24175$$

4th step:

The EVEN and ODD impedances of a line pair are generally specified in accordance with the following formulae:

$$Z_{EVEN} = 50\Omega \cdot \left[1 + Z_0 \cdot J_N + (Z_0 \cdot J_N)^2\right]$$

$$Z_{ODD} = 50\Omega \cdot \left[1 - Z_0 \cdot J_N + (Z_0 \cdot J_N)^2\right]$$

For the first and fourth line pairs we obtain:

$$Z_{EVEN} = 50\Omega \cdot \left[1 + 0.24175 + 0.24175^2\right] = 65\Omega$$

$$Z_{ODD} = 50\Omega \cdot \left[1 - 0.24175 + 0.24175^2\right] = 40.8\Omega$$

For the second and third line pairs the values are:

$$Z_{EVEN} = 50\Omega \cdot \left[1 + 0.05138 + 0.05138^2\right] = 52.7\Omega$$

$$Z_{ODD} = 50\Omega \cdot \left[1 - 0.05138 + 0.05138^2\right] = 47.56\Omega$$

Use of PUFF

Simulation of ideal circuit using PUFF

First start "PUFF" And load the SETUP file. Then press the "F4" key and enter the following values for the Rogers material R04003:

- thickness 0.032"
- impedance level z_d = 50Ω,
- design frequency f_d = 1575MHz,
- dielectric constant ε_r = 3.38
- board thickness h = 0.813mm.

The printed circuit boards size "s" should be 200mm, and 50mm. is a sufficient distance between the connections (Fig. 3).

Then move into field "F3" and successively enter there the data for the two coupled line pairs required. Please transfer them precisely as shown in Fig. 4!

Press the "F1" key to make the layout window appear (Fig. 5). And now please pay attention, first move the cursor as far to the left as you can. Then press the shift key for upper case lettering and keep pressing the "Cursor Left" key until you get to the desired location. Press "1" and port 1 is connected immediately.

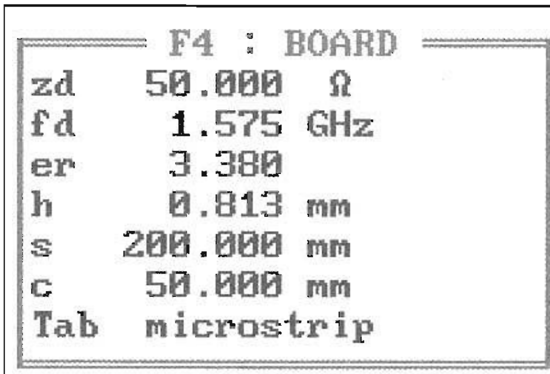

Fig 3: PUFF starting parameters.

Fig 4: PUFF data for successive line pairs.

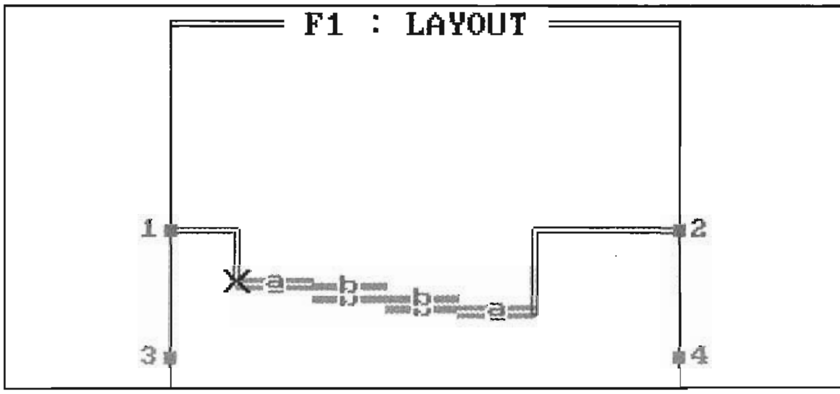

Fig 5: PUFF layout window.

Press letter “a” on the keyboard, followed by “Cursor Right”. This positions the first line pair. Then press “b” and next “Cursor down”, which connects up line pair “b”.

Now press “Cursor down” again and the third pair is already sitting there on the screen. Finally press “a”, “Cursor down” again and then “2” and port 2 is likewise connected to the exit of the circuit.

Use “F2” to go to the simulation window. Using “Cursor Up” or “Cursor Down”, you can move, not just in the top left-hand “Plot window”, but also along the axes of a diagram in the bottom right-hand corner (“linear plot”) (Fig. 6).

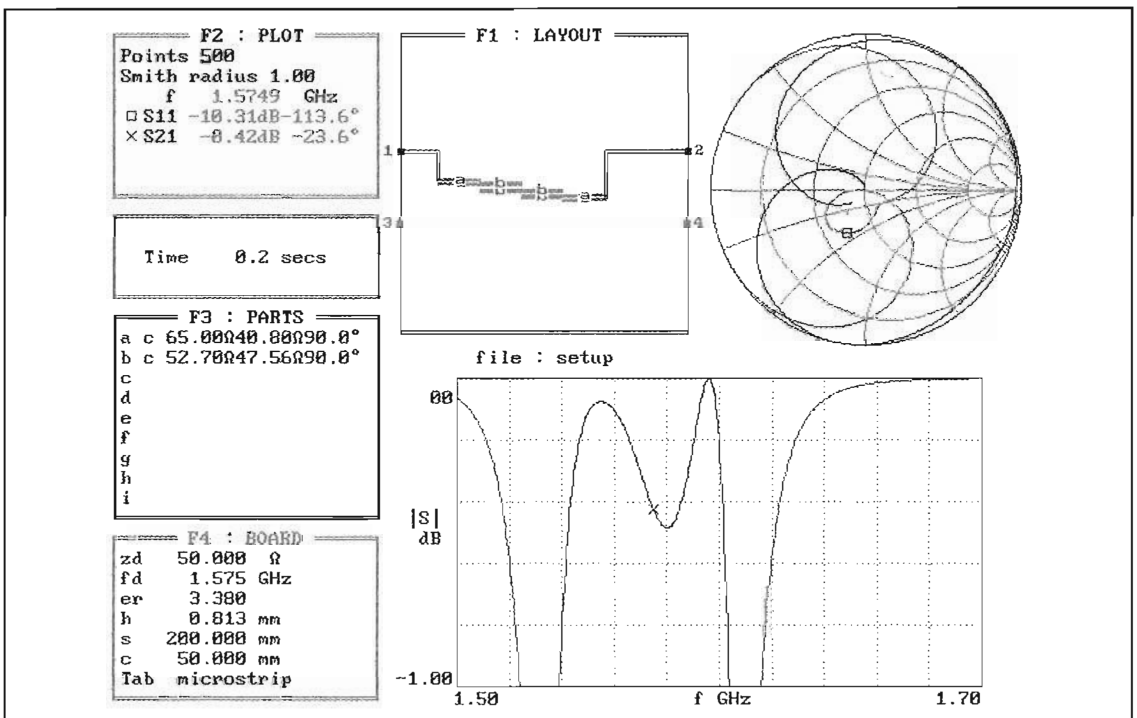

Fig 6: PUFF simulation window.

Here you pre-set, for example:

- 500 simulation points
- Smith radius = 1
- Representation of S11 and S22
- Horizontal scales in the ratio: 1.5 ...1.7GHz
- Vertical scales in the ratio: -1.00...0dB

And now please press "p", and you can watch the computer at work. If you think thats too slow, press "Q", and then all the calculations are done and the image is built up off screen, and things go considerably faster!

If you look at the result now, you'll undoubtedly be a little disappointed. It is nowhere near a ripple with maximum 0.0436dB. It is greater by more than a factor 10 with the value being 0.5dB. But don't worry, we can still get round that. We just need to make some very slight corrections to the line data!

We need only reduce the electrical length of the first and fourth line pairs (part "a" in the parts list) by approximately 1 degree and increase the design frequency by 2MHz in order to obtain the theoretical curve (Fig. 7)!

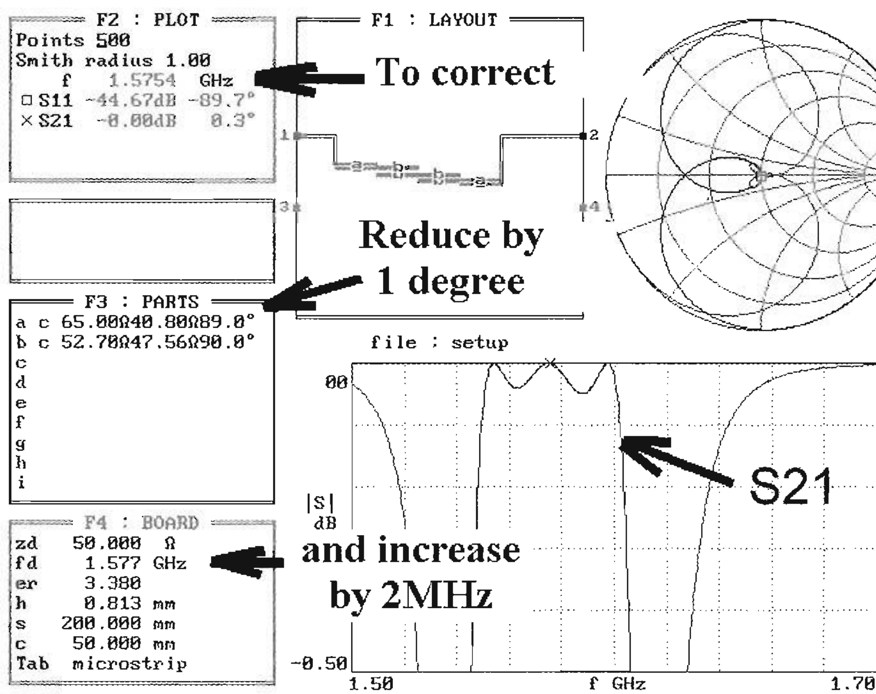

Fig 7: PUFF simulation after slight corrections.

Simulation of the real circuit with PUFF

First call up the "SETUP.PUF" file from the PUFF directory into a text editor and then enter the remaining printed circuit board data. Thickness of copper layer = 35 micrometres and surface roughness for a strip conductor polished until it gleams with grinding paste or scouring powder approximately 2 micrometres. Loss factor "l_t" of the board material R04003 at this frequency max. 0.001):

zd	50,000Ω {normalizing impedance. 0<zd}
fd	1.575GHz {design frequency. 0<fd}
er	3.380 {dielectric constant. er>0}
h	0.813mm {dielectric thickness. h>0}
s	200.000mm {circuit-board side length. s>0}
c	100.000mm {connector separation. c>=0}
r	0.010mm {circuit resolution, r>0, use Um for micrometres}
a	0.000mm {artwork width correction.}
mt	0.035mm {metal thickness, use Um for micrometres.}
sr	2.000µm {metal surface roughness, use Um for micrometres.}
lt	1.0E-0003 {dielectric loss tangent.}

The amended setup file is loaded back into PUFF and then the exclamation mark for each pair of coupled lines is entered in field "F3". This switches to "real modelling with all side effects" (Fig. 8).

If we now place the cursor in field "F3" on part "a" and enter the equals sign, the actual data of the coupled lines are immediately placed into the dialogue field (Fig. 9).

Now we have to keep changing the values entered under "a" until the data determined above are displayed in the dialogue field:

Z_e = 65Ω
Z_o = 40.8 Ω
l = 89 degrees

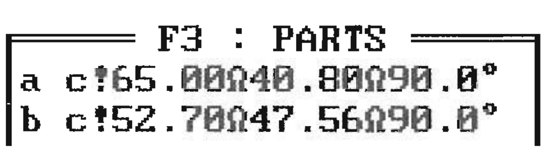

Fig 8: Exclamation mark changes to real modelling mode.

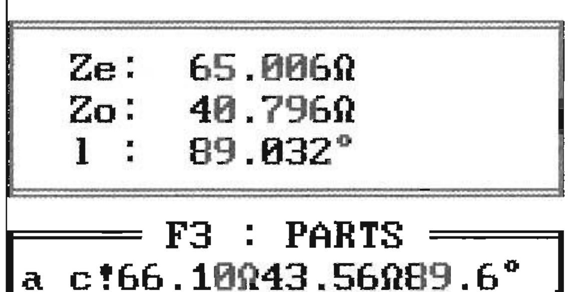

Fig 9: PUFF data for part "a".

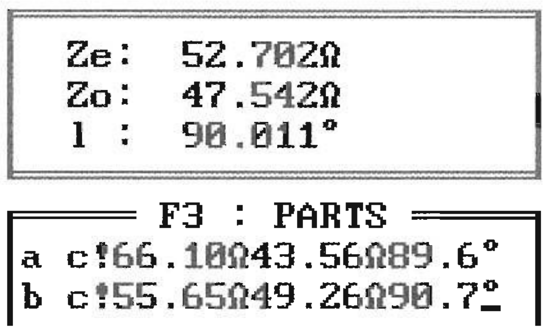

Fig 10: PUFF data for part "b".

It can be seen that to bring this about the entry for "a" has to be

c! 66.1Ω 43.65Ω 49.6°

This procedure must be repeated for part "b" (Fig. 10). The target is to obtain this display in the dialogue field:

Z_e = 52.7 Ω
Z_o = 47.56 Ω
l = 90 degrees

For this, finally, we need the entry:

c! 55.6Ω 49.26Ω 90.7°

Fig. 11 shows the result of the circuit simulation if the losses are taken into account.

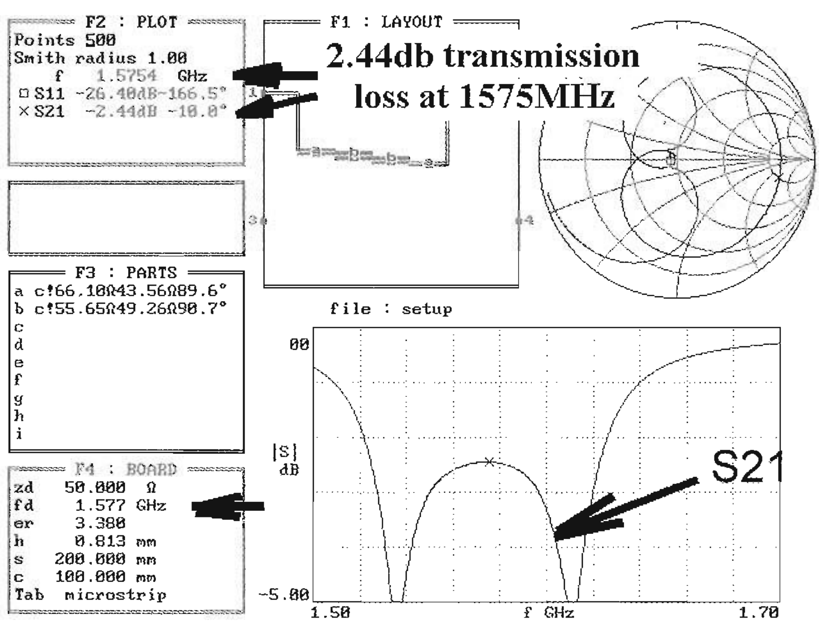

Fig 11:PUFF results with losses taken into account.

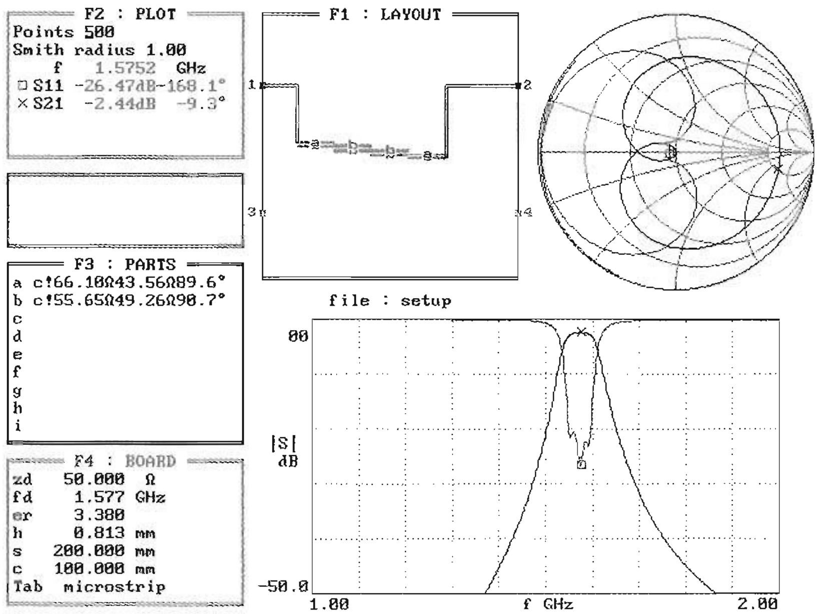

Fig 12: puff simulation results in the range 1 - 2GHz.

The design frequency continues to remain at 1577MHz, but following the simulation use "Page Down" to move the display cursor to 1575MHz. We now have a transmission loss of approximately 2.5dB.

If we correspondingly switch the value range in the two axes of the lower diagram, we can take another look at the long range selection, i.e. the behaviour in the range between 1 and 2GHz (Fig. 12).

Determination of mechanical, uncorrected line data with PUFF

In order to get at the dimensions of the coupled lines, we move back again into field "F3" and delete the exclamation mark for part "a" behind the letter "c" (for coupled lines). As soon as we key in the equals sign behind here, we obtain the desired values in the dialogue field (Fig. 13):

Length l	= 29.34mm
Width w	= 1.59mm
Gap s	= 0.31mm

Repeat this for part "b", i.e. the two central pairs of coupled lines, and we correspondingly obtain Fig. 14:

Length l	= 29.15mm
Width w	= 1.82mm
Gap s	= 1.84mm

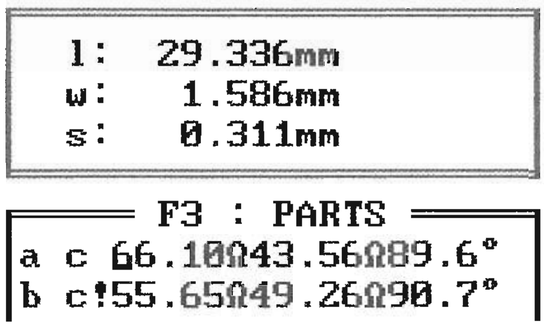

Fig 13: PUF results for line pair "a".

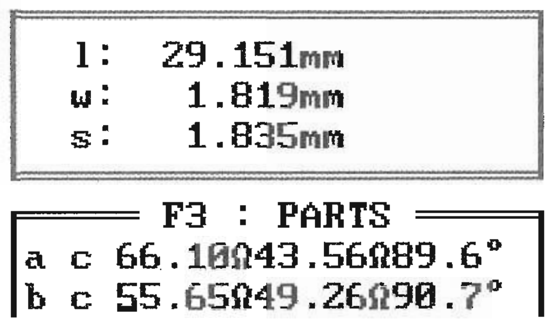

Fig 14: PUF results for line pair "b".

Then, as a preliminary to the board design, determine the width of the 50Ω feed likewise in the same way. It is modelled as "lossy transmission line with 90 degree length" and, as part "c", supplies a required width, w = 1.84mm, following the deletion of the exclamation mark (Fig 15).

Necessary layout corrections

Here we are dealing either with striplines open at the end or with the junction of two striplines that are of different widths. In both cases, the familiar open-end correction is required, due to fringing, but one peculiarity should be taken into account here at the open ends of coupled lines:

The two line pairs are coupled to each other both electrically and magnetically. It is true that the electrical field lines project beyond the open ends of the striplines (so we need to do some shortening....), but the magnetic coupling decreases linearly in this area right down to zero.

For this reason, calculate in only ***half*** the "open-end-extension" which would otherwise be normal and shorten the line correspondingly!

Apart from this, we now need several tools to create the layout:

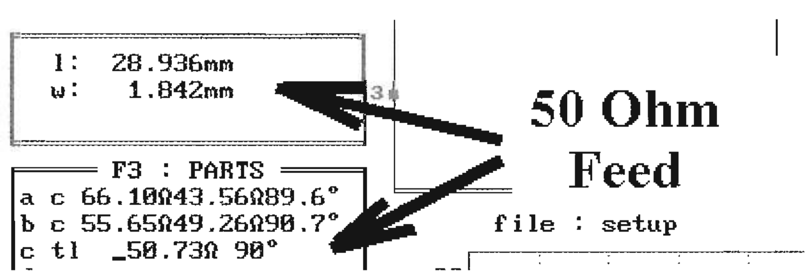

Fig 15:PUFF determining the width of the 50Ω line.

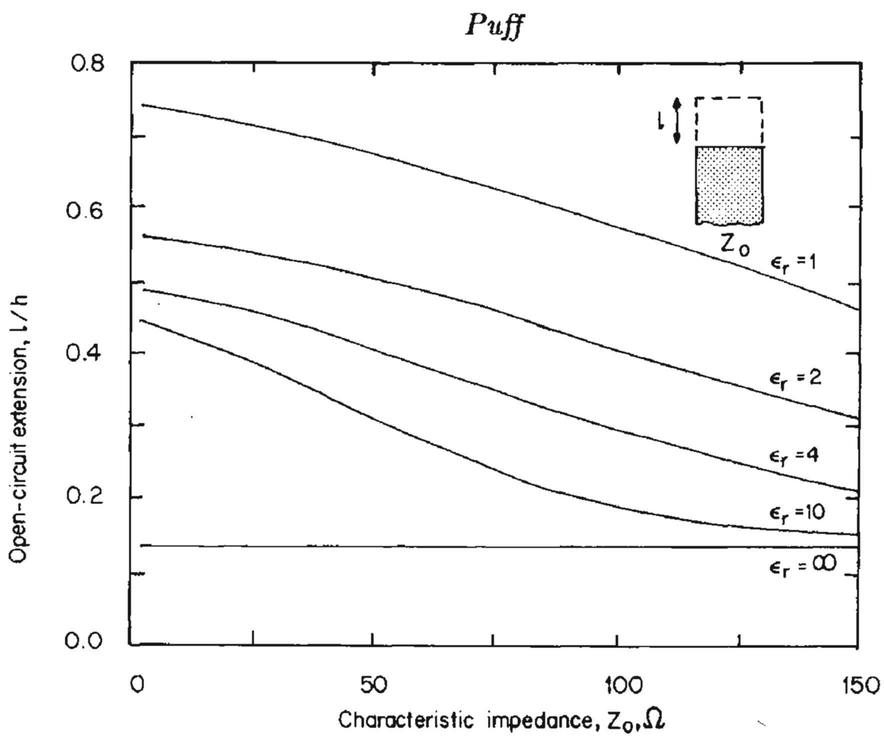

Figure 7.2 The open-circuit end correction in microstrip, plotted from (7.2). The artwork length correction in a parts list should be *negative.*

Fig 16: Diagram for determining the open end extension of striplines, from PUFF manual.

- The tried and trusted diagram for determining the open-end-extension from the PUFF manual (Fig. 16).
- A simple hand drawn sketch (Fig. 17) with the electrical data of the individual line pairs already determined. Enter the necessary corrections.
- A printed circuit board CAD program, which simultaneously makes it possible to solve tricky construction problems (e.g. GEDDY-CAD, tried and trusted for such microwave tasks for many years).

1st step:

The first and fourth line pairs consist of two striplines each with a width of 1.59mm. With the help of PUFF we obtain the impedance level for the pre-set printed circuit board data:

The result gives us: Z = 54.6Ω

So we go to the above diagram from the PUFF manual. For this task, it supplies (with ε_r = 3.38) an open-end extension Δl/h of approximately 0.45. So these section pieces must be reduced by

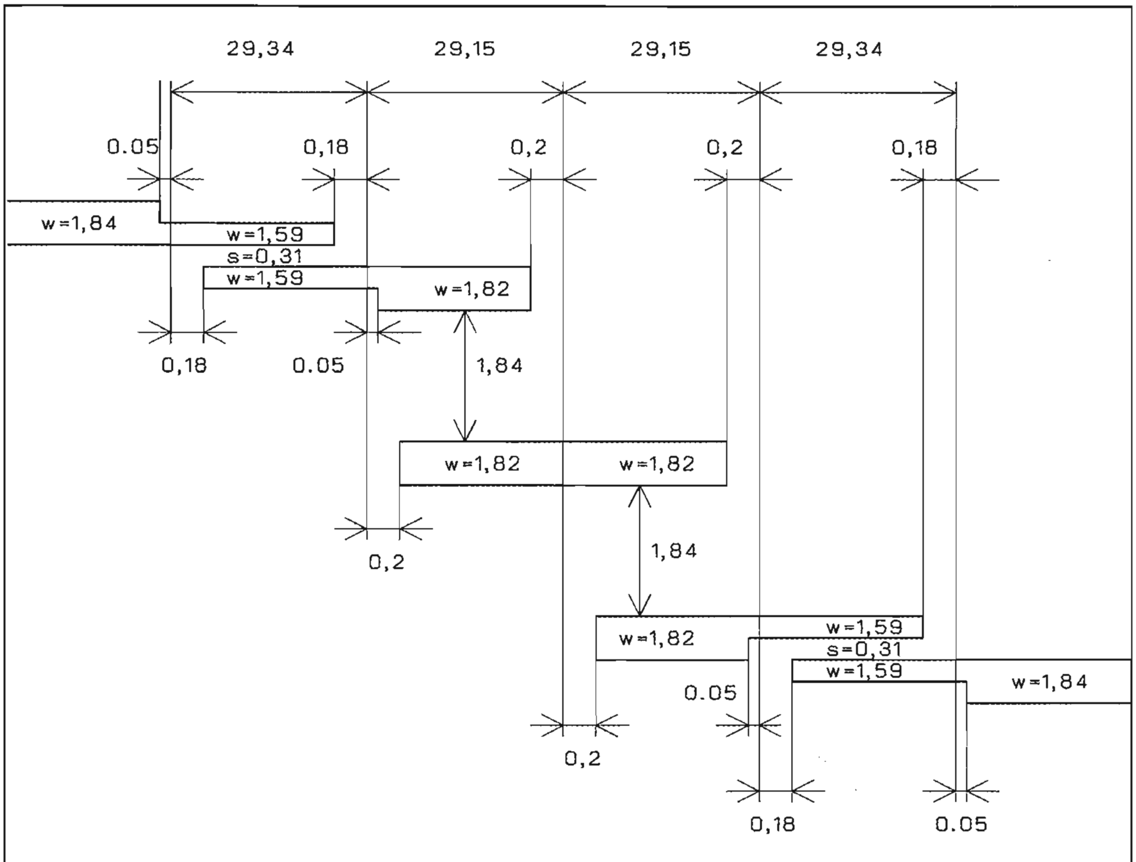

Fig 17: Hand drawn sketch of the filter.

half of 0.45 x 0.813mm = 0.18mm at all open ends!

2nd step:

At the start and end of the bandpass, the 50Ω feed is connected up with a width of 1.84mm and turns into the (narrower!) stripline with a width of 1.59mm. So the narrower line must be extended by a little piece measuring (1 1.59mm/1.84mm) x 0.45 x 0.813mm = 0.05mm.

***3rd step*:**

The two central line pairs have conductor widths of 1.82mm. The associated impedance level (according to PUFF) is 50.4Ω and at ε_r = 3.38 requires an open-end correction of 0.48 x 0.813mm = 0.39mm.

Again, only half of this value, i.e. approximately 0.2mm, needs to be cut off from the two ends.

4th step:

When the first line pair meets the second and the third meets the fourth, there is a correction of (1 1.59 mm / 1.82 mm) x 0.48 x 0.813 mm = 0.05 mm.

The wider line must be shortened and the narrower line must be lengthened by this amount.

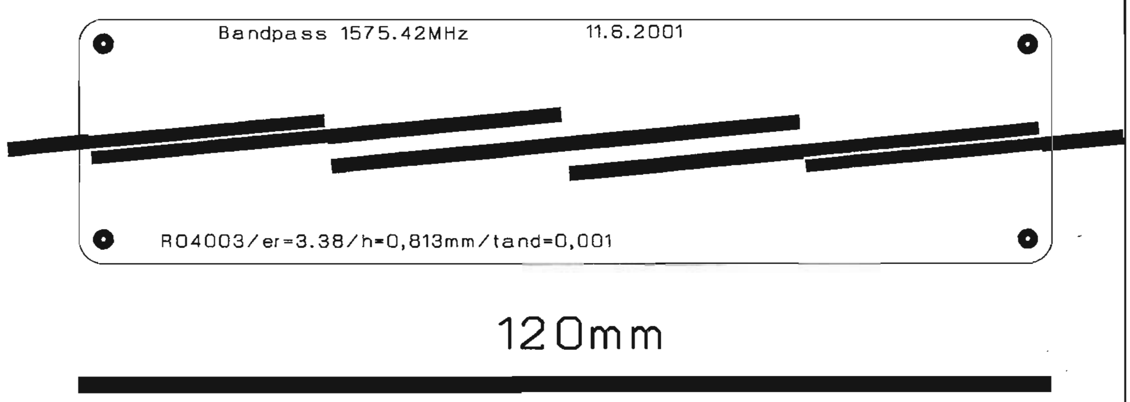

Fig 18: The completed layout using PUFF.

You should never omit entering all these details on a sketch because of laziness. It is an obligatory stage in the layout design (Fig. 17) and you need to take considerable trouble just to produce the drawing. But this is as nothing in comparison with the time and expense which will be wasted if the behaviour of the prototype produced inexplicably displays big differences from the simulation and you have to laboriously check every individual dimension on the completed printed circuit board. Its really very annoying if a gross error in the board layout turns out to be the reason for this.

Normally if you've followed all the instructions in this article the divergences between the simulation and prototype are max. 1 - 2%.

The finished layout is designed to fit into a milled aluminium trough with internal dimensions of 30 x 120mm, as shown in Fig. 18.

One more tip for those who don't know why there is a thick 120mm long line under the printed circuit board. We need this to set up the board manufacturing equipment, unless there is a photo plotter available that can be used to photographically set the correct dimension. This is the only way we can find the correct scale and be able to handle the manufacturing problem posed by the narrow gap between line pairs 1 and 4.

Use of TRL85 Stripline Calculator

To determine the data for fitting the circuit into a screened housing, the "TRL85" stripline calculator from Ansoft is used.

Ansoft are known for supplying very high quality and expensive RF CAD programs, but have also always had their eye on instruction and training! So on their Homepage on the Internet [3] we find a student version of the microwave CAD package "Serenade" which can be downloaded free.

Following installation a very good stripline calculator called "TRL85.EXE" together with excellent online help is in a separate directory. It can be used separately at any time or copied and transferred to other computers. You will very soon learn to appreciate its WINDOWS user interface.

If we now compare the simulation results from "TRL85" with the values from PUFF, we can make the following statements:

In normal cases, the data determined by PUFF and TRL85 for single and coupled striplines are practically identical.

In addition to this, TRL85 offers the advantage that screening can be brought into the simulation in the form of the "Cover Height" (the distance between the cover and the board).

With TRL85 all data which is of interest (impedance level, losses, broken down into dielectric loss, conductor loss and total loss... etc.) can be determined directly for a specific design frequency. Unfortunately, PUFF cant provide this in the same way, although you have to carry out calculations using these values, they are not displayed.

The "TRL85" program was described in a separate article by the same author and is the next article in this chapter.

Table 1 below shows the differences produced by TRL85 microstrip simulations for operation without and with screening.

Another tip:

TRL85, unfortunately, won't automatically apply the open-end correction either. So again we have to revert to the use of the diagram from the PUFF manual when a line end is open.

Table 1: TRL85 simulations with and without screening.

Bandpass Specification	TRL85 witout screening	TRL85 with screening distance = 13mm
50Ω Microstrip	Width = 1.84mm	Width = 1.84mm
First line pair:		
E = 89 1575.42MHz Z_e = 65Ω Z_0 = 40.8Ω	Ladder width = 1.58321mm Spacing = 0.33225mm Length = 29.2968mm	Ladder width = 1.5769mm Spacing = 0.32877mm Length = 29.3133mm
Second line pair:		
E = 90 1575.42MHz Z_e = 52.7Ω Z_0 = 47.56Ω	Ladder width = 1.8216mm Spacing = 1.87272mm Length = 29.186mm	Ladder width = 1.81525mm Spacing = 1.82327mm Length = 29.2009mm

Repetition of design using ANSOFT-SERENADE

Simulation of ideal electrical circuit

Since "SERENADE" is already installed on the PC, because "TRL85" stripline calculator has been loaded, we can repeat the design to see how our filter is simulated by this very modern program. Naturally, we are interested, above all, in what improvements it can give us, in terms of ease of operation or precision.

Procedure

Launch the SERENADE software and start a new project (e.g. "BP1575_1").

Then look for the "ideal coupled line" (Fig. 19), position it on the screen four times (see preceding chapter!). In the "Property Editor", enter the EVEN and ODD impedances, the electrical length of 90 degrees and the operating frequency of 1575.42MHz. When the component is positioned, the editor opens automatically. If it does not, just double click on the left hand mouse button on the circuit icon in the circuit diagram. Our ports are connected up at the input and output, but thats not enough yet!

The "Harmonica" circuit simulator is the problem, two of the four connections are open circuit for each coupled line pair and this is prohibited.

We could use a very high value resistance (e.g. 10MΩ) to such open connections. However unloaded ideal line sections (Stubs) are considerably better and have lower losses, with an electrical length of zero at 1575MHz and with an impedance Z = 50Ω. They cause no additional losses, nor do they alter the data for the coupled lines.

Next the frequency block is reset and the range between 1GHz and 2GHz is represented in 1MHz steps. Rogers R04003 material is again used as substrate, with a thickness of only 32 MIL (0.813mm), as this gives the filter structure smaller dimensions. The other data are as follows:

Dielectric constant	ε_r = 3.38
Metallisation Met1:	Copper with thickness 35µm
Surface roughness	RGH = 2µm
Loss factor	TAND 0.001

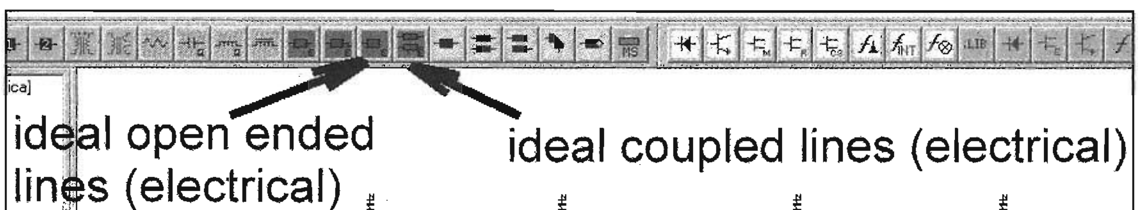

Fig 19: Serenade toolbar.

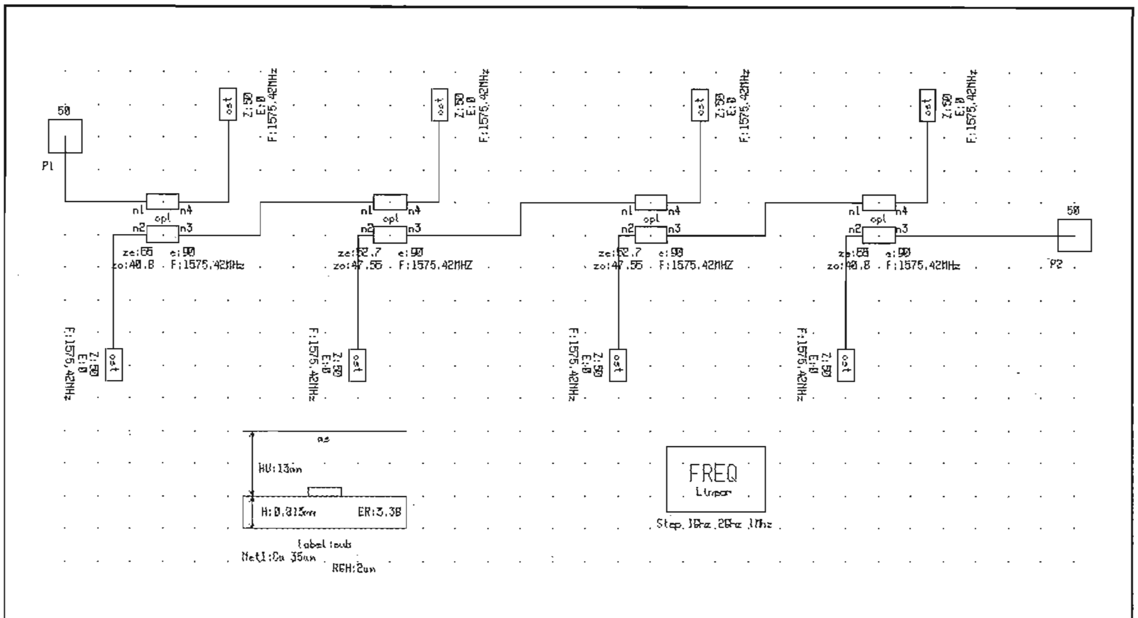

Fig 20: Circuit used for Serenade simulation.

The entire circuit, as used for simulation, is shown in Fig. 20. Even for those who don't know the program yet, the components just produced can be easily identified. Frequency and substrate control blocks certainly need no further explanation.

Following a precise check of the circuit, the simulation can begin (button with gears) and the Report Editor can be activated (turquoise / grey button). Select, for example, S11 and S21 in dB representation and examine the result (Fig. 21).

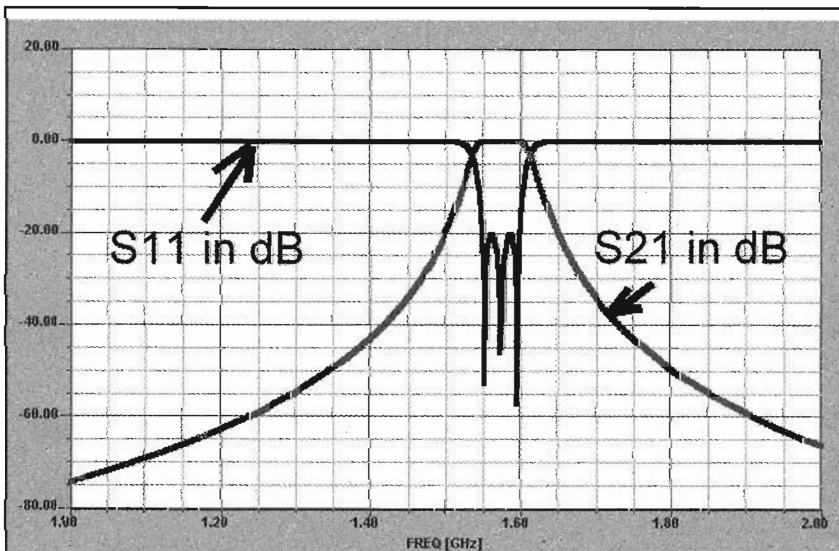

Fig 21: Serenade simulation results showing S11 and S21 curves.

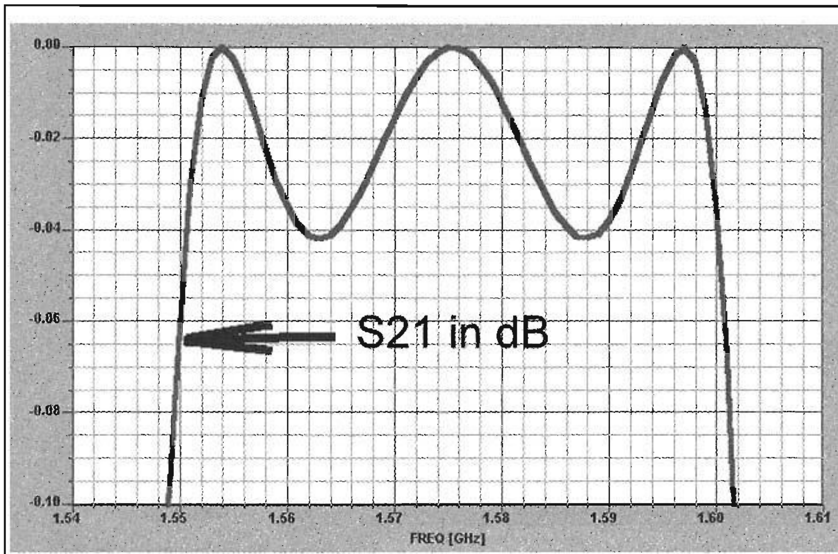

Fig 22: Serenade simulation results showing a closeup view of the S21 curve.

The result looks promising and S11 is never worse than the intended value in the pass band 20dB.

Use the right hand mouse button and click on "zoom in" repeatedly to bring out the precise sequence of S21 in the range from 0 to 0.1dB between 1550 and 1600MHz.

Only in this way can we assess whether the design technique from previous simulations really supplies the correct values desired.

Fig. 22 shows a perfect and well-formed filter curve.

In practice, both the mean frequency (1575MHz) and the minimum ripple value (0.041dB) are in accordance with the design requirements.

Simulation of physical circuit using HARMONICA

Apart from simulating the pass band with "electrical components", HARMONICA also offers the option of a structure made up of "physical circuits". This requires the conductor width, interaction gap and conductor length to be entered, the dimensional being in "mm". These values can be obtained using the TRL85 stripline calculator, which can even be called up from the operating screen by pressing a button.

Here only the values for the first and fourth line pairs are needed, (Z_e = 65 Ω, Z_o = 40.8 Ω, E = 90 degrees). The track data is copper with 35μm thickness and a roughness of 2μm and the printed circuit board and housing data are board thickness H = 0.813mm, ER = 3.38, cover height above board, HU = 13mm, TAND = 0.001.

If you then press the "Synthesis" button, you obtain a representation corresponding to Fig. 23, thus:

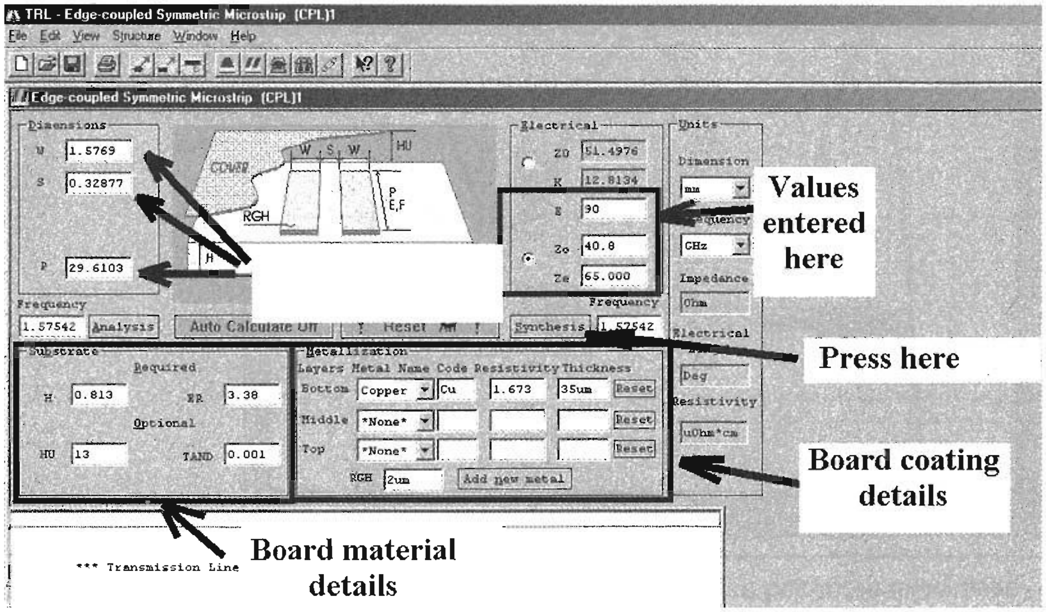

Fig 23: Serenade simulation screen showing details for synthesis.

Conductor width	W = 1.58mm
Interaction gap width	S = 0.33mm
Circuit length	P = 29.61mm

This procedure is repeated for the second and third line pairs (Z_e = 52.7 Ω, Z_o = 47.56 Ω, E = 90 degrees), then we obtain:

W = 1.82mm
S = 1.82mm
P = 29.20mm

It is interesting to compare this with PUFF, although it should be remembered that the part involving the housing and the distance of 13mm between the board and the cover is not included in this calculation!

Moreover, for simulation using PUFF, the electrical length for the first and last line pairs is presumed to be 89 degrees, whereas for SERENADE it is 90 degrees. The uncorrected PUFF values are as follows:

Line pairs 1 + 4:

W = 1.58mm
S = 0.31mm
P = 29.34mm

Line pairs 2 + 3:

W = 1.82mm
S = 1.84mm
P = 29.15mm

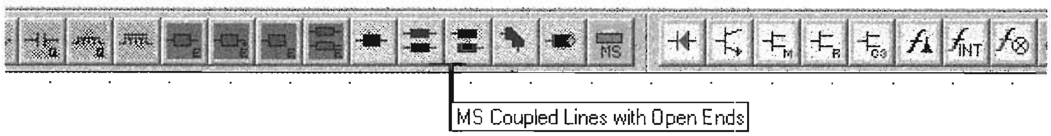

Fig 24: Serenade toolbar showing coupled lines with open ends tool.

It can be seen that the differences between the two simulations are not so devastating to make one of them seem completely unusable.

But let us simulate the pass band with the physical TRL85 values again, consider the result and ponder on:

- where the differences to the simulation using PUFF come from and
- how can we arrive at the correct values.

To do this, though, we must draw a new circuit diagram, and first we must delete the old one completely.

When we create the new circuit diagram, things move forward splendidly. The coupled line pair is actually there with the two open ends as a completed component (Fig. 24). This naturally makes the work considerably easier.

The screen is very tidy, even when the necessary data are entered (Fig. 25).

The S parameters after simulation using HARMONICA are shown in Fig. 26.

If we zoom into the representation of the pass band (Fig. 27), then several points strike us:

- The lowest transmission loss is predicted to be similar by both programmes (PUFF: approximately 2.5dB, HARMONICA approximately 2.8dB).

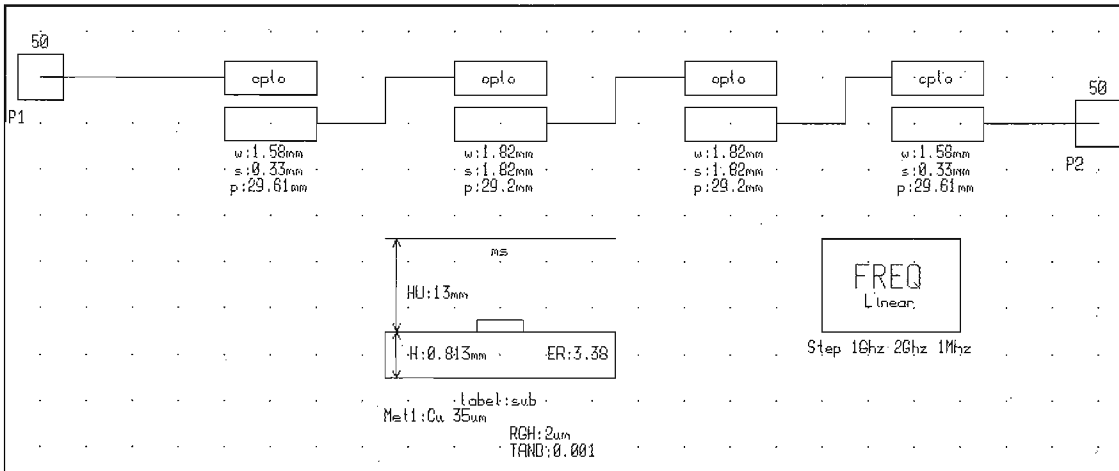

Fig 25:The revised circuit diagram for Serenade.

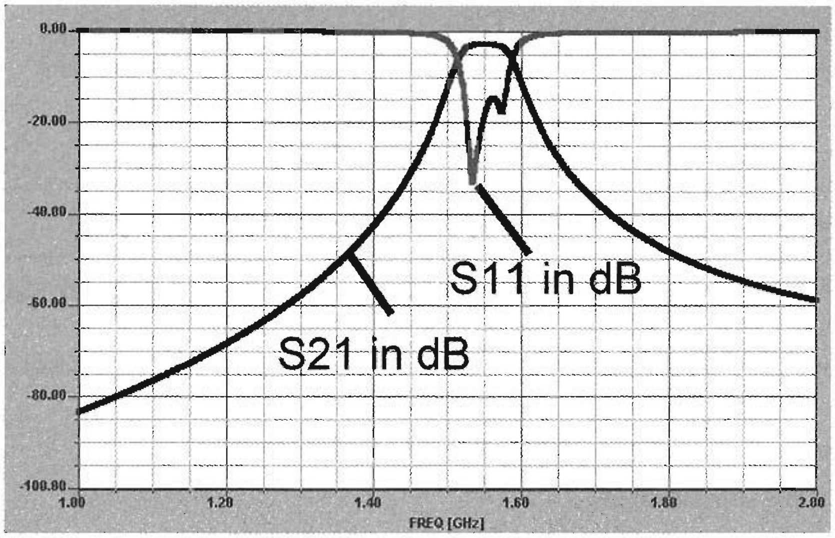

Fig 26: S Parameters from the Harmonica simulation.

- The mean frequency of the pass band according to the HARMONICA simulation reveals a divergence of 30MHz (approximately 2%) and is clearly too low. The program therefore does not make automatic open-end corrections!

Luckily, if we use Ansoft, we don't need to carry out the same changes to the diagram as listed in the PUFF manual to make corrections. Here we have something which is extremely useful.

What we actually do is to take specific values for the line pair through variables, pre-set maximum and minimum values for the S parameters at specific frequencies, and then let the optimiser do the job of reconciling.

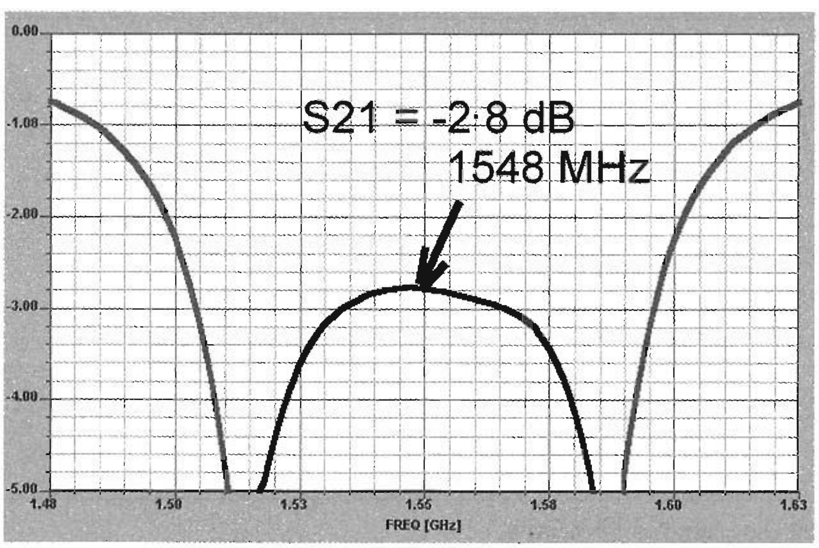

Fig 27: Closeup view of the S21 curve from the Harmonica simulation.

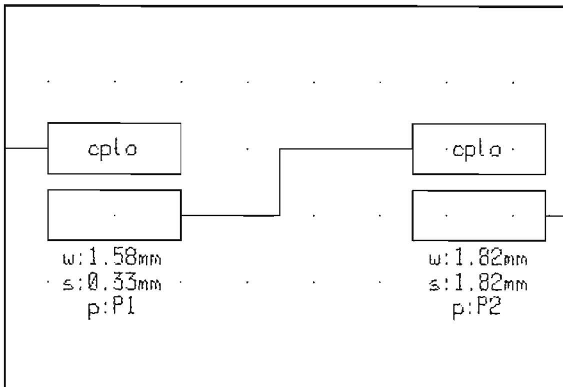

Fig 28: Variables P1 and P2.

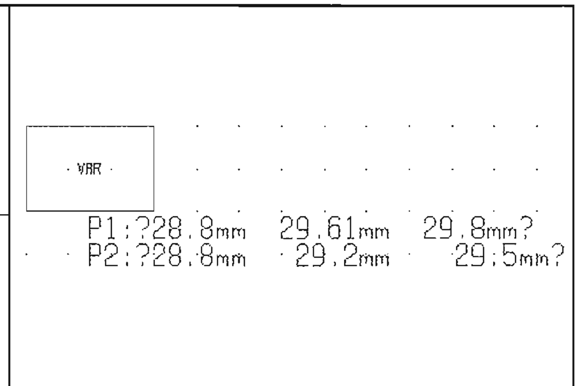

Fig 29: Entry of variable P1 and P2.

Here we have the following steps:

1st step:

In the first and fourth line pairs, the physical length, P, is replaced by a variable, P1. We correspondingly use variable P2 for the second and third line pairs (Fig. 28).

2nd step:

Call up a "variable control block" (Menu path: "Parts/Control Blocks/Variables") and enter the permissible variation range for P1 and P2 between question marks. In the middle is the original initial value (Fig. 29).

3rd step:

We now have to formulate the optimisation goals. There is an "original" button for this, in the form of a yellow and red practice target. In the student version a maximum of 3 optimisation goals are permissible, but this should just be enough at first.

The optimisation goals here are:

- In the "Frange1" range, from 1.55 to 1.6GHz, S21 should not fall below 3.3dB (Goal1)
- In the "Frange2" range, S11 should be lower than 20dB (Goal2) (Fig. 30).

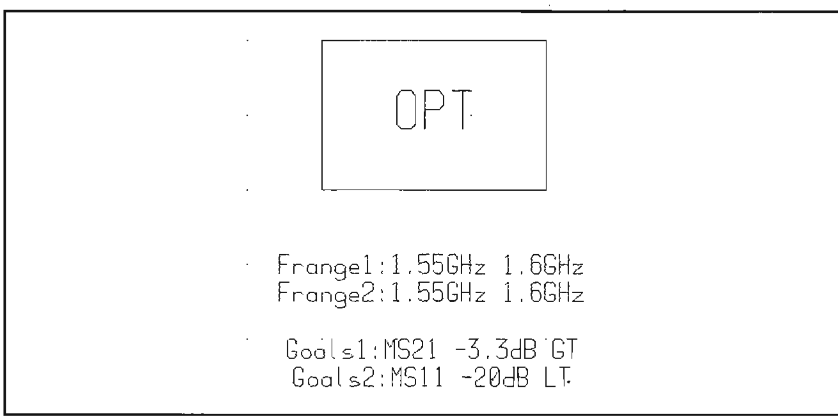

Fig 30: Goals for optimisation.

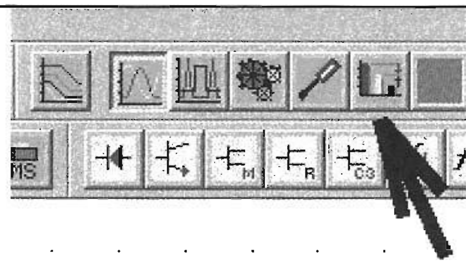

Fig 31: Press this button to start the optimisation.

4th step:

Now press the "Optimisation" button. The program normally indicates that it must first analyse the circuit, and asks for permission to do this. Grant it permission and also finally give it precise instructions on the number of optimisation searches, see Fig. 31.

5th step:

Please follow the sequence exactly:

1) Pre-set, for example, 2000 searches;

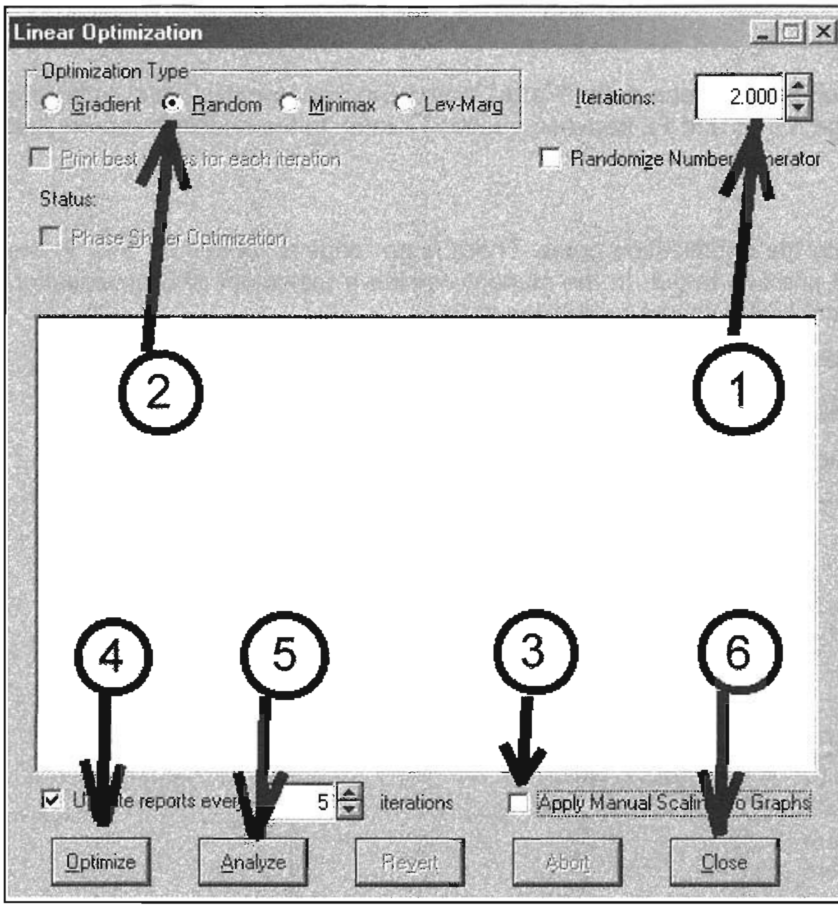

Fig 32: Optimisation screen showing the steps to be taken.

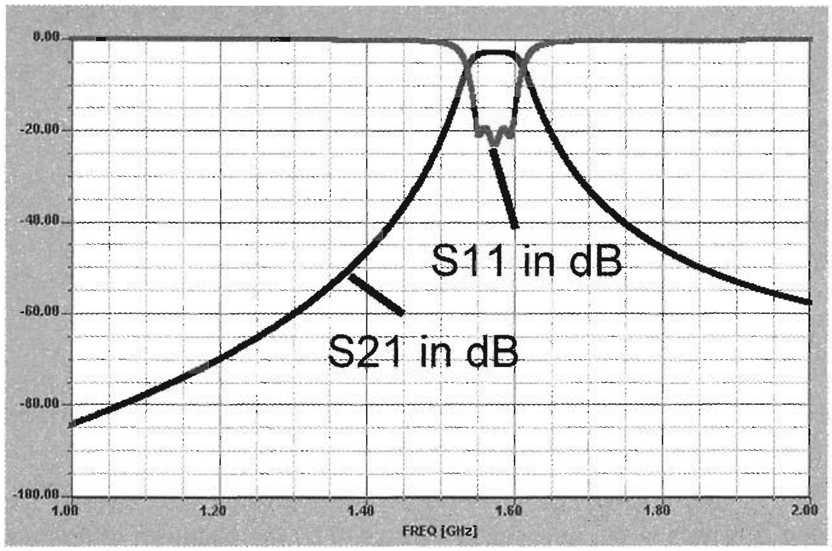

Fig 33: Results of optimisation.

2) Select "RANDOM" as optimisation type

3) Make sure this tick box is clear

4) Now press "Optimise" and wait until the program has found the best approximation to the conditions.

5) Now start another circuit analysis, as this is the only way to update the results diagram.

6) Now close this menu and obtain the diagrams with S11 and / or S21 in the foreground (Fig. 32).

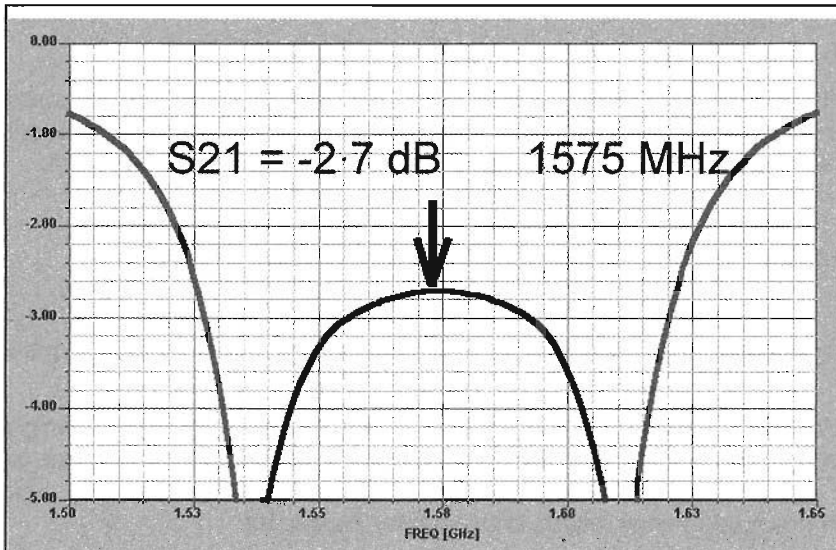

Fig 34: Closeup of the S21 curve after optimisation.

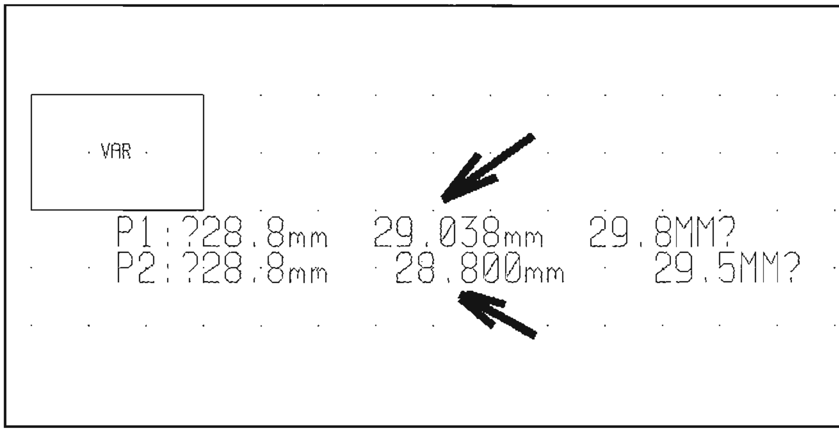

Fig 35: New circuit data in the variable block.

The results are shown in Fig. 33, and they look very satisfactory.

The data within the pass band have changed in accordance with Fig. 34; the result is useable.

The only question remaining is, where is the new circuit data used by the optimiser to produce the curves above?

It's very simple: you'll find them in the variable block instead of the initial values (Fig. 35)!

Finally, all data for the layout preparation in accordance with ANSOFTs physical variants:

First and fourth line pairs:

W = 1.58mm
S = 0.33mm
P = 29.04mm

Second and third line pairs:

W = 1.82mm
S = 1.82mm
P = 28.80mm

Now only one question remains: "Which of the two programs is really right"?

There's only one way to find the answer to this question: produce another printed circuit board using these dimensions, measure it under exactly the same conditions as for the PUFF product using the network analyser, and then cold-bloodedly analyse the results and compare them.

TRL85 microwave and analysis program, Gunthard Kraus, DG8GB

TRL85 is based on the same computing algorithms as PUFF or PCAAD, but is markedly more powerful.

Apart from the familiar WINDOWS tools, you will appreciate, for example, the option which allows you to take into account the incorporation of the circuit into a housing using "distance to cover". In addition, the relationships between losses and frequencies are carefully and separately logged, broken down into power losses and insulator losses, etc.

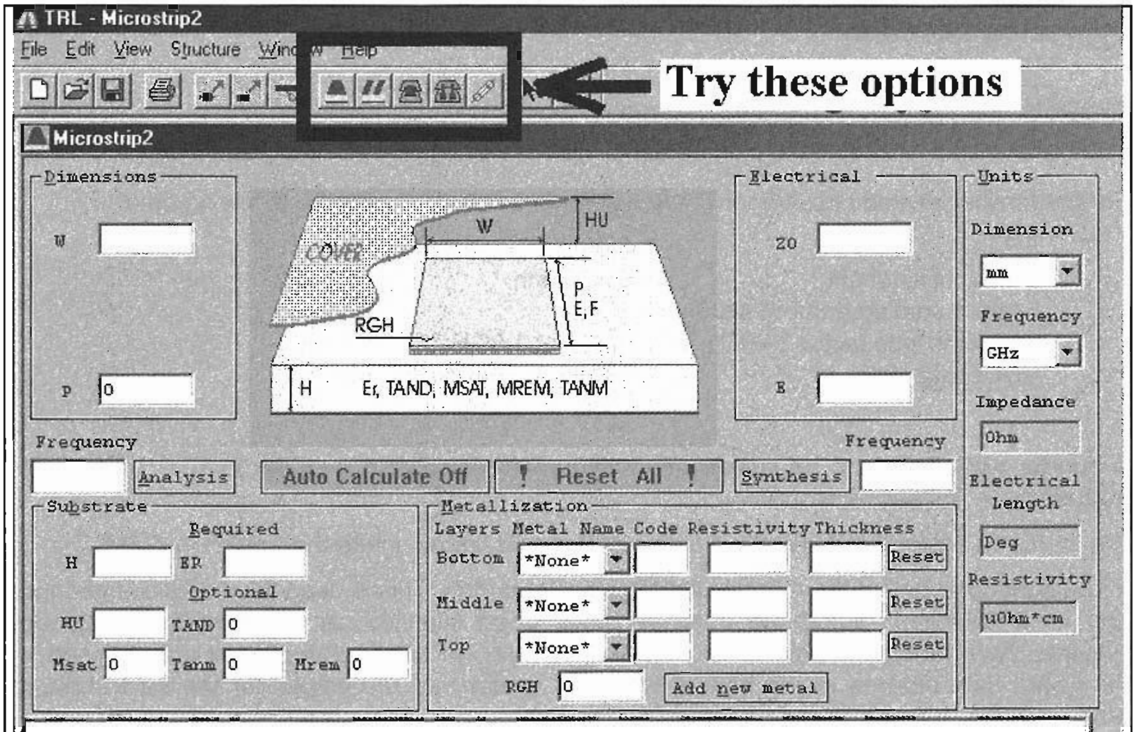

Fig 36: TRL85 screen display showing stripline calculation options.

Working with TRL85

The rest of this article assumes that a TRL85 is installed on the computer in use. You can set up a start button for the TRL85 program on the WINDOWS screen or simply move to the directory for TRL85 and click on the file "TRL85.exe".

On the first screen (Fig. 36) you should first click on each of the symbols in succession. In this way you can see the basic structure of the various modes of for yourself, together with the screen menus offered. The areas covered are:

- Microstrip
- Edge coupled Microstrip
- Stripline
- Edge coupled Stripline
- Coaxial cable

The following examples can be used to familiarise yourself with the program on a step-by-step basis.

Typical application

Analysis and synthesis of a 50Ω microstrip feeder

For the familiar printed circuit board material Rogers RO 4003 with the following data:

ER (ε_r)	= 3.38
Board thickness H	= 0.813 mm
Copper coating on both sides	= 35 µm
Dielectric loss factor TAND	= 0.001 at 1.6GHz
Screening plate	= 13 mm above the board

A quarter wave line is to be investigated, with an electrical length of 90 degrees at the GPS frequency f = 1575.42MHz.

We therefore select the "Microstrip" option button and first examine the screen in detail:

In the right hand half of the menu is the "Synthesis" button. This is used if a specific impedance and electrical circuit length (in degrees) have been selected, at the operating frequency required. As a result, the physical (mechanical) length and width of the circuit are obtained for the board data entered at the bottom left hand corner and the values for the track material entered at the bottom right hand corner.

In the left hand half of the menu is the "Analysis" button. This can be used to specify (from the mechanical dimensions) the electrical characteristics (the impedance level, the electrical length in degrees, the losses in dB / mm, even separated into dielectric and conductor losses, etc.).

The following steps should now be carefully carried out one after another (Fig. 37):

1) Enter a 50Ω impedance level here

2) Is the dimensional unit the mm?

3) Is the frequency already being measured in GHz?

4) Let the electrical length be 90 (the associated dimensional unit has already been set to degrees).

5) Enter frequency 1.57542GHz correctly.

6) The specifications for the track material [copper /thickness = 0.035mm /RGH = roughness = 5µm] must go in this field. Under "Bottom" select the option "Copper" and for "RGH" enter the proposed 5µm.

7) And here finally all data concerning the printed circuit board (thickness = 0.813mm / ER = 3.38 / cover distance from board = 13mm / TAND = 0.001)

When everything has been done, press the "Synthesis" button and the corresponding result appears on the screen (Fig. 38).In the top left hand corner of the menu, the two boxes for W (width) and P (physical length) are filled with the calculated values.

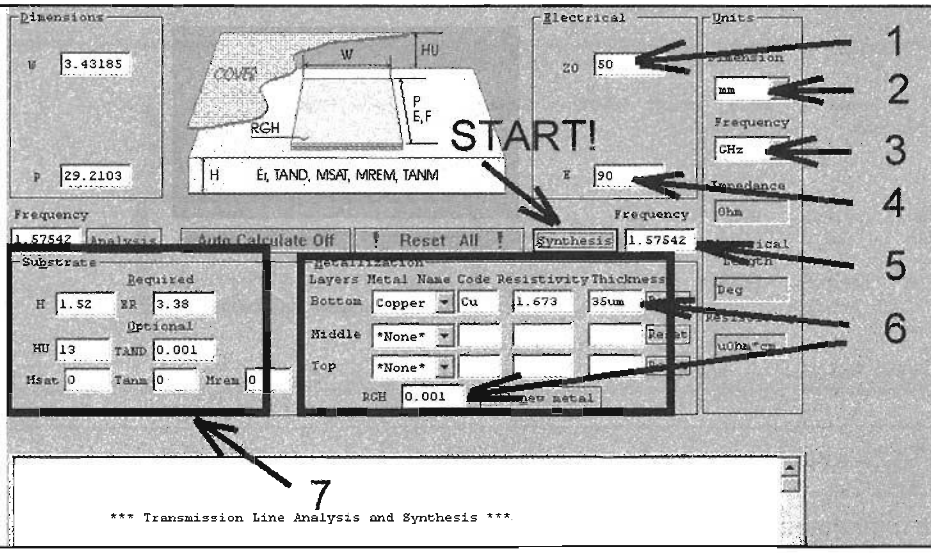

Fig 37: TRL85 steps for microstrip simulation.

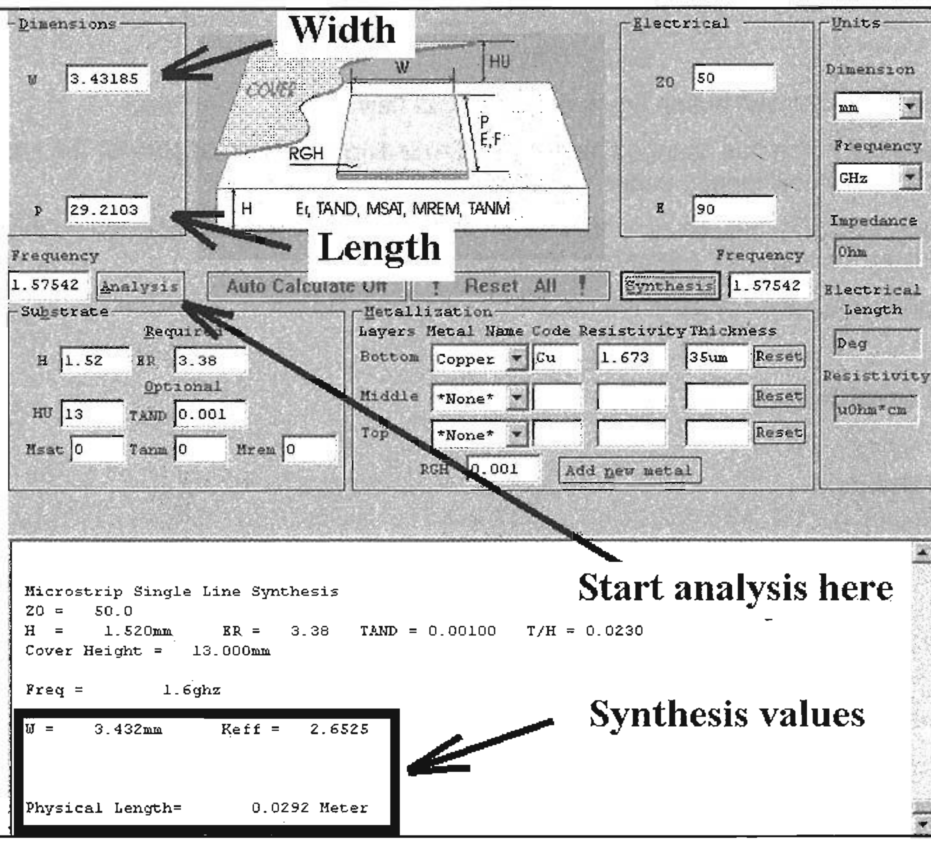

Fig 38: TRL85 results of synthesis on microstrip line.

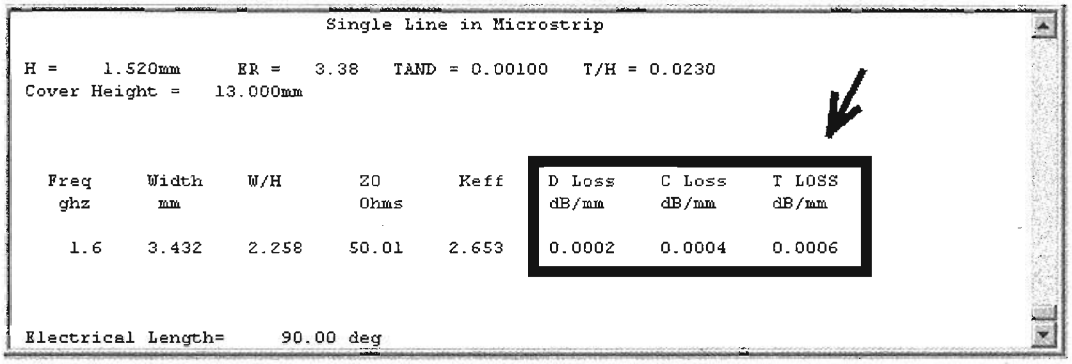

Fig 39: TRL85 analysis report for the microstrip line showing attenuation values.

Under the menu area, in a large separate field, the results are displayed. In the marked frame, here too we have the values for the length and width of the stripline, which are:

Width = 1.83mm
Length = 29.24mm
for Z = 50Ω

Now press the "Analysis" button again and see what new options are offered:

In the menu not much has changed, just that the rounding errors are shown for the impedance level and the electrical length, under "Reverse Calculation".

In the report field (Fig. 39), we now also have the attenuation value at 1.6GHz, and this divided into "D" (dielectric loss), "C" (conductor loss) and "T" (total loss).

You are given the option to save the results in a separate file when you leave TRL85.

Typical application:

Coupled line pair

With band pass filters made from coupled line pairs, once the filter design is complete we obtain the values for the EVEN and ODD impedances. They must first be converted into mechanical data before the printed circuit board is designed. "PUFF" can also do this but with "TRL85" we can also calculate fitting into a screed housing with the cover distance of, for example, 13mm.

The following data are assumed for the line pair in such a filter:

Z_{EVEN}	= 65Ω
Z_{ODD}	= 40.8Ω
Electrical length	= 90 degrees

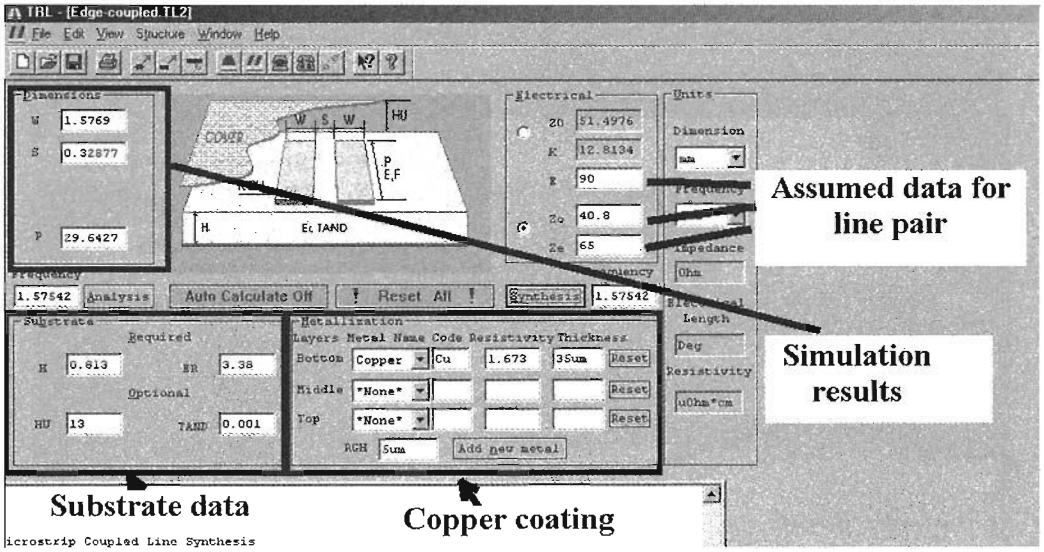

Fig 40: TRL85 data setup for simulation of a coupled line pair.

The same printed circuit board made of RO4003 is used again and the data to be entered for it are:

Relative permittivity	ER = 3.38
Printed circuit board thickness	H = 0.813mm
Copper coating on both sides	35µm
Dielectric loss factor	TAND = 0.001 at 1.6GHz

Screening plate 13mm above printed circuit board.

Now use the second button from the left to switch to "edge coupled microstrip" and re-enter the printed circuit board data in the "Substrate" field in the screen menu that appears.

We also need the data on metallisation again. So we select "copper" under "Bottom" and for "RGH" we select a surface roughness of 5µm.

Now things get interesting, because we can now finally enter the electrical length, E, the ODD resistance, Z_o, and the EVEN resistance, Z_e.

Press the "Synthesis" button once, and we at once obtain the following values in the left-hand half of the screen (Fig. 40):

Conductor width	W = 1.58mm
Interaction gap	S = 0.33mm
Physical circuit length	P = 29.65mm

If we now press the "Analysis" button, we can see how TRL85 determines the electrical length, E, together with the resistances Z_o and Z_e. But the most important thing is the results report in the lower half of the screen, for there we find the precise loss factors and other details.

But don't forget that unfortunately TRL85 can not perform open-end corrections, we have to do

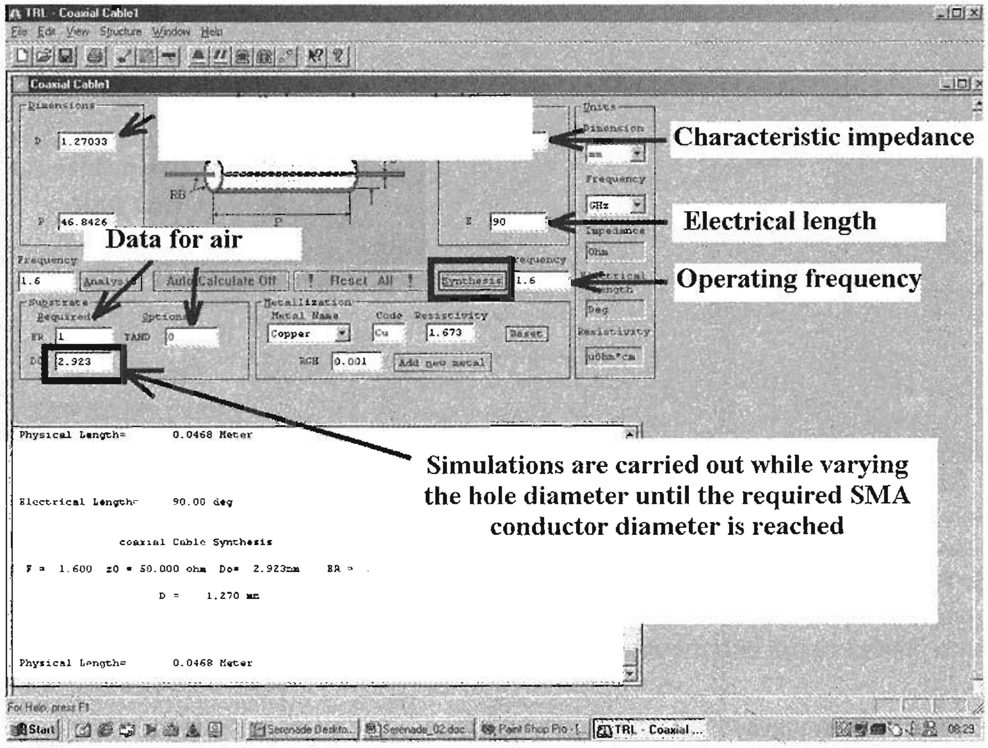

Fig 41: Using TRL85 to simulate a coaxial line problem.

that ourselves and determine the shortening required using, for example, the familiar diagrams in the "PUFF Manual", see Fig 16.

Coaxial line

Here we are not trying to go over the data for a coaxial cable but to solve the following problem:

The internal conductor of an SMA flanged bush with a diameter of 1.27mm is fed through the wall of an aluminium housing to the printed circuit board. This housing wall is 3mm thick and the hole should be selected in such a way that the impedance level, even inside the wall, is 50Ω. What drill diameter should be selected?The solution is shown in Fig. 41.

Call up "TRL85", select the "Coaxial cable" menu and enter an impedance level of 50Ω, an assumed electrical length of 90 (degrees), an operating frequency of 1.6GHz and the data for air (ER = 1, TAND = 0). The program then needs the external diameter and calculates the internal diameter for this. Here we can start with any value as the external diameter. Finally, press the "Synthesis" button.

We alter this external diameter and continue to create simulations until the internal diameter reaches the required value of 1.27mm. For this, we need a hole with a diameter of 2.9mm in the housing wall.

Summary

The "TRL85" stripline calculator provides an outstanding supplement to simulation using "PUFF" and is indispensable once the housing has to be included as well.

In other respects, "PUFF" and "TRL85" give practically identical results. Its ease of operation makes it a joy to use, and this is not the only way in which "PUFF" compares unfavourably. TRL85 also has direct modelling of all data (impedance level, splitting loss fractions, etc.) at the design frequency and a WINDOWS interface.

References

[1] Design and realisation of microwave circuits, Gunthard Kraus, VHF Communications, from issue 4/96, P. 244 - 250, at irregular intervals to issue 2/99.

[2] PUFF manual, original English version available from VHF Communications, http://www.vhfcomm.co.uk

[3] Ansoft-Serenade manuals (supplied when program is downloaded), www.ansoft.com

[4] APLAC manuals (supplied when program is downloaded), www.aplac.com

[5] Microwave Engineering by David Pozar (John Wiley & Sons, New York, ISBN 0-471-17096-8).

[6] Microwave Filters, Impedance-Matching Networks, and Coupling Structures by G. Mattaei, L. Young and E. M. T: Jones. (Artech House Publishers, ISBN 0-89006-099-1).

[7] Microstrip Lines and Slotlines by K: C. Gupta, Ramesh Garg, Inder Bahl and Prakash Bhartia. (Artech House Publishers, ISBN 0-89006-766-X).

[8] Software, manuals and tutorial are combined on an "ANSOFT-CD" and can be obtained from the author, provided the costs are reimbursed. Please E-mail me at: krausg@elektronikschule.de.