

NCT iNSTRUMENTS

Stripline Filter Tool 1.0 User manual

Index

- 1 Installation 3
 - 1.1 License file..... 3
 - 1.2 Demo mode 3
- 2 Introduction 4
 - 2.1 Striplines..... 4
- 3 Building an elliptic filter 5
 - 3.1 Finding the component values 5
 - 3.2 Converting to stripline..... 5
 - 3.3 Simulation..... 6
 - 3.4 Trace widths and substrate height 6
- 4 Using the Stripline Filter Tool..... 8
 - 4.1 Report..... 10
 - 4.2 Error messages 10
- 5 A practical example..... 11
 - 5.1 Component values..... 11
 - 5.2 Testing a real filter..... 14
 - 5.2.1 Results for the '1.2mm filter' 15
 - 5.2.2 Results for the '0.2mm filter' 17
 - 5.3 Conclusion..... 18

1 Installation

Installing the Stripline Filter Tool is matter of running the installer called 'Stripline Filter Tool 1.0 installer.exe'. In order to run the installer administrator privileges are required. Make a note in which directory the program is installed because the license file needs to be copied into that directory later on.

1.1 License file

After purchasing a license a license file will be sent by e-mail. This file needs to be copied manually into the directory where the program is installed. On Windows XP and 32 bit Windows 7/8 the default directory is: C:\Program Files\NCT instruments\SFT .On 64 bit versions of Windows 7/8 the software is installed in C:\Program Files(x86)\NCT instruments\SFT .

Please note that a license is personal and cannot be transferred to other persons or entities.

1.2 Demo mode

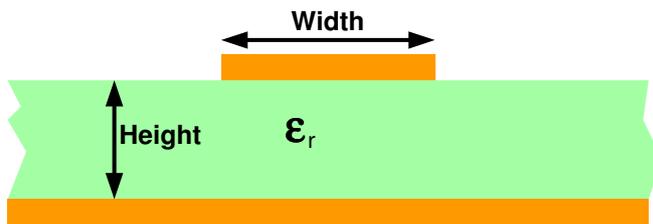
When no license file is available the Stripline Filter Tool will run in demo mode using the values from the examples. Only the width of the traces can be altered.

2 Introduction

The NCT instruments Stripline Filter Tool is a piece of software which can be used to calculate the geometry for 3rd, 5th or 7th order elliptic (Cauer) filters made from stripline PCB traces. These filters are very useful because they offer a wide choice in pass-band / stop-band ripple and sharpness. It is even possible to make these filters behave like Butterworth or Chebyshev filters. A downside is that the math required calculating the stripline geometry is quite extensive and difficult to solve. The Stripline Filter Tool intends to make the design of elliptic stripline filters a piece of cake.

2.1 Striplines

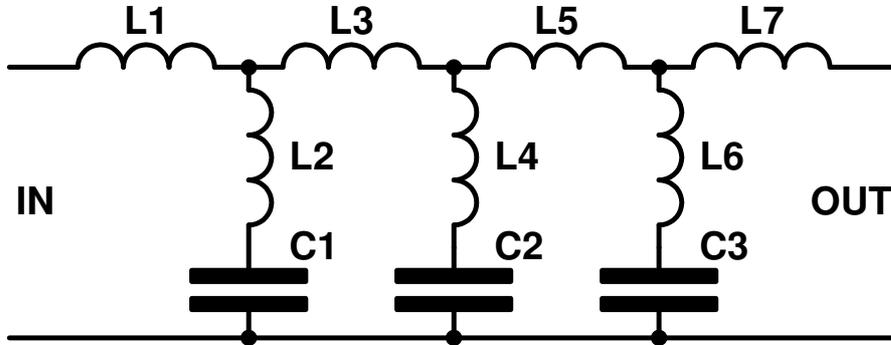
At high frequencies (say above 500MHz) even short PCB traces act as transmission lines which have a characteristic impedance depending on the width of the trace, the thickness (height) of the substrate and their dielectric constant (ϵ_r).



If a PCB trace matches the impedance of the source (driver) and load (receiver) it behaves like a resistor so there are no reflections and the transfer of energy is at its maximum. In reality there will be some frequency dependent losses. A fairly good online stripline calculator can be found at:
<http://wcalc.sourceforge.net/cgi-bin/microstrip.cgi>

3 Building an elliptic filter

An elliptic filter can be built in two ways. The first way is using shunt capacitors and parallel resonators. The second way is using series inductors and series resonators. The Stripline Filter Tool is based on converting the latter into a stripline equivalent. The schematic of a 7th order elliptic filter with series inductors and series resonators is shown below:



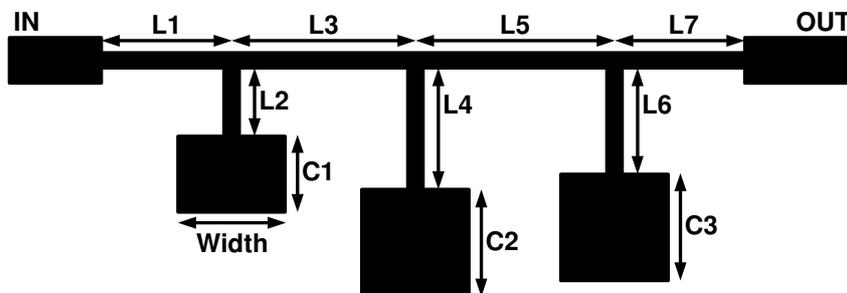
3.1 Finding the component values

Finding the component values for a filter takes quite some math. Back in the old days when computers were expensive people compiled a whole bunch of tables to create various types of filters. With these tables it is easy to create a filter. The trick is that when the input and output impedance is fixed the filter values can be expressed as normalized frequency independent impedance coefficients. Finding the values for the inductors and capacitors is simply a matter of selecting the table entry which matches the desired filter type, order, attenuation and ripple parameters and use the coefficients to calculate the components values from the cut-off frequency and impedance. There are a lot of tools available for this purpose so the Stripline Filter Tool doesn't include this step. A simple and free to use program called 'SVC filtertool' which can be downloaded from <http://tonnesoftware.com/svcfilter.html> is used for the examples later on.

3.2 Converting to stripline

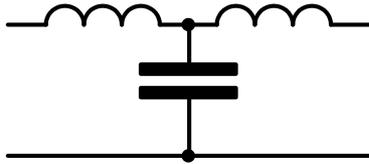
At higher frequencies wide PCB traces are mostly capacitive and thin PCB traces are mostly inductive. This feature can be used to translate a filter made out of real components (lumped elements) into a stripline equivalent. There are two ways to do this. The first is by using Kuroda identities and Richard's transform. With this method the impedance coefficients are used to calculate the width of pieces of PCB trace with a fixed length of $\lambda/8$. The disadvantage of this method is that the risk of having to use PCB traces which are so thin they cannot be manufactured is very high. All in all this method has a very limited use.

The second way is to choose large trace width for capacitors and a small trace width for inductors. By exploiting the fact that an open piece of PCB trace looks like a capacitor to ground the schematic from above can be converted into this structure:

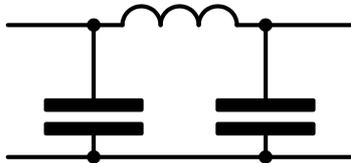


Because the amount of inductance or capacitance can be expressed per unit of length (say 0.5pf per mm or 0.7nH per mm) finding the values could be a simple matter of dividing the required component values by the capacitance or inductance per unit length.

Unfortunately there is a snag. Even a wide PCB trace where the capacitance dominates has some inductance and vice versa. A wide 'capacitive' PCB trace actually looks like this:



A thin 'inductive' PCB trace looks like this:



The extra capacitances and inductances need to be taken into account in order to keep the resonance frequencies where they belong. The resulting equations for calculating the length of the traces are quite difficult to solve even with the use of a computer. This is exactly why the Stripline Filter Tool has been created.

3.3 Simulation

When the filter geometry has been determined it is good practise to simulate the filter to see if the response matches the design goals. The examples include Sonnet EM simulation project files which can be used with Sonnet Lite which is available for free from <http://www.sonnetsoftware.com/>

3.4 Trace widths and substrate height

Because the stripline filter is an approximation of the filter made of ideal components the frequency response will be different. The stop band attenuation is usually less and because of a side effect of elliptic stripline filters there is peaking at higher frequencies (see the chapter covering the example). The peaking could be mitigated by adding an extra filter to the elliptic filter to suppress the high frequencies. When choosing the width for inductive traces the choice depends on the thinnest trace which can be manufactured. The thinner the trace the better the attenuation at high frequencies. On the other hand, a trace close to the smallest width a PCB manufacturer can make will have a relatively large error.

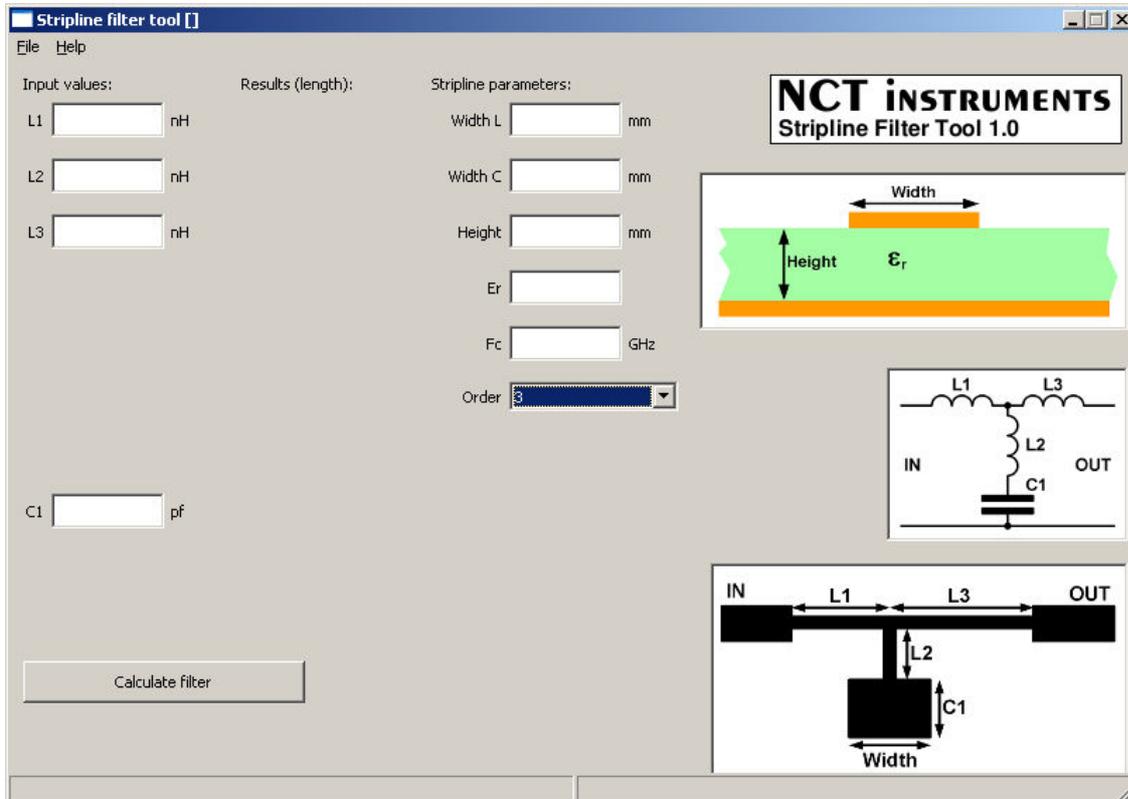
When choosing a trace width for the capacitive traces keep in mind that there are limits to the width of the trace. If a capacitive trace gets very short ($w/l < 0.2$) the trace will no longer behave as a capacitor. Another problem may occur when the trace is too wide relative to the wave length of the cut-off frequency and resonance frequency. The Stripline Filter Tool will warn when these situations occur and provide a hint on what to do to correct the problem.

Another factor is the thickness of the substrate. On a very thin substrate a 50 Ohm trace which typically connects the filter to other components can be 0.3mm to 1mm wide. Such a thin trace is easy to route but there is a downside. Choosing a very thin substrate causes the thin inductive traces to still have a lot of parasitic capacitance which could force very thin traces for the inductive parts of the filter. Using thin traces may lead to problems because the thinner the trace the larger the effects are of tolerances in the manufacturing (etching) process.

Last but not least there is also variation in the ϵ_r of the substrate. For FR4 the ϵ_r may vary from 4.3 to 4.8. This does affect the filter characteristic so a real world design must factor in these variations unless the kind of FR4 used can be tightly controlled. The graph below shows the simulation results for the 1GHz filter with 0.2mm 'inductive' traces from the examples which was simulated for an ϵ_r of 4.3 and an ϵ_r of 4.8. The graph shows both the frequency response and the VSWR.

4 Using the Stripline Filter Tool

When the program is started the screen looks like this:



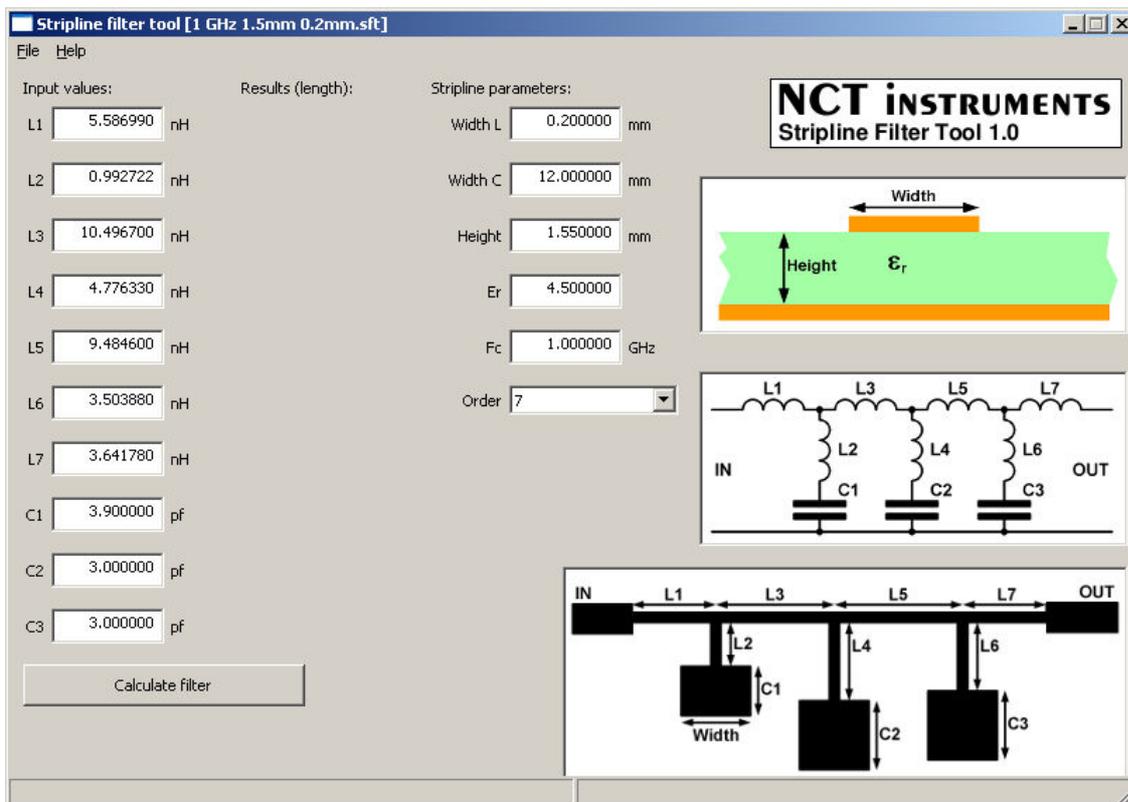
By changing the order more or less inductor / capacitor fields will be shown, the schematic and geometry drawing will change accordingly.

Screen elements:

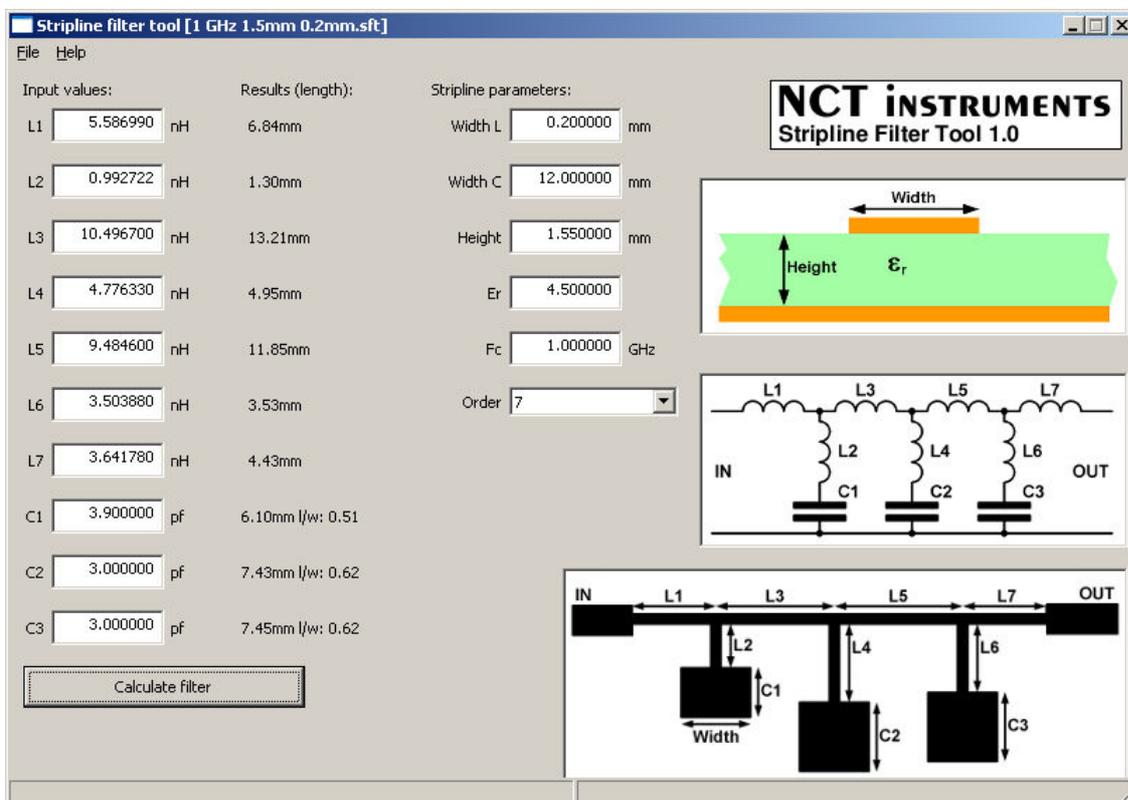
Name	Purpose
Lx	Inductor values in nH
Cx	Capacitor values in pf
Width L	Width of inductive traces (as thin as possible) in mm
Width C	Width of capacitive traces (wide traces) in mm
Height	Thickness of the substrate in mm
Er	Relative dielectric constant
Fc	Cut-off frequency in GHz
Order	Order of the filter (choice between 3, 5 or 7)

Projects can be opened and saved using the 'File' menu.

The Stripline Filter Tool comes with two examples of a 1GHz filter which can be used to become familiar with the software:



The above shows the screen with the '1 GHz 1.5mm 0.2mm.sft' project. When the button 'Calculate filter' is pressed the Stripline Filter Tool will start to solve the filter geometry. This may take more than just a few seconds on a slow PC. When it is finished the results will be shown:



The l/w values indicate the shape of the capacitors.

4.1 Report

The results can be saved into a comprehensive report which is a text file:

```

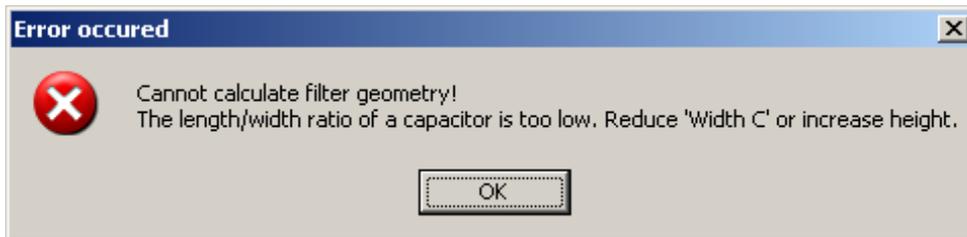
Elliptic filter report
*****
Material parameters:
Er: 4.50
Substrate thickness: 1.55 mm
Inductors trace width: 0.20 mm
Capacitors trace width: 12.00 mm

*****
Filter parameters:
Order: 7
Cut-off frequency: 1.0000 GHz
L1 value: 5.58699 nH length: 6.84 mm
L2 value: 0.99272 nH length: 1.30 mm
L3 value: 10.49670 nH length: 13.21 mm
L4 value: 4.77633 nH length: 4.95 mm
L5 value: 9.48460 nH length: 11.85 mm
L6 value: 3.50388 nH length: 3.53 mm
L7 value: 3.64178 nH length: 4.43 mm

C1 value: 3.90000 nH length: 6.10 mm
C2 value: 3.00000 nH length: 7.43 mm
C3 value: 3.00000 nH length: 7.45 mm
*****
    
```

4.2 Error messages

There are several safeguards built into the Stripline Filter Tool to prevent impossible results. In such an event an error message will be shown:



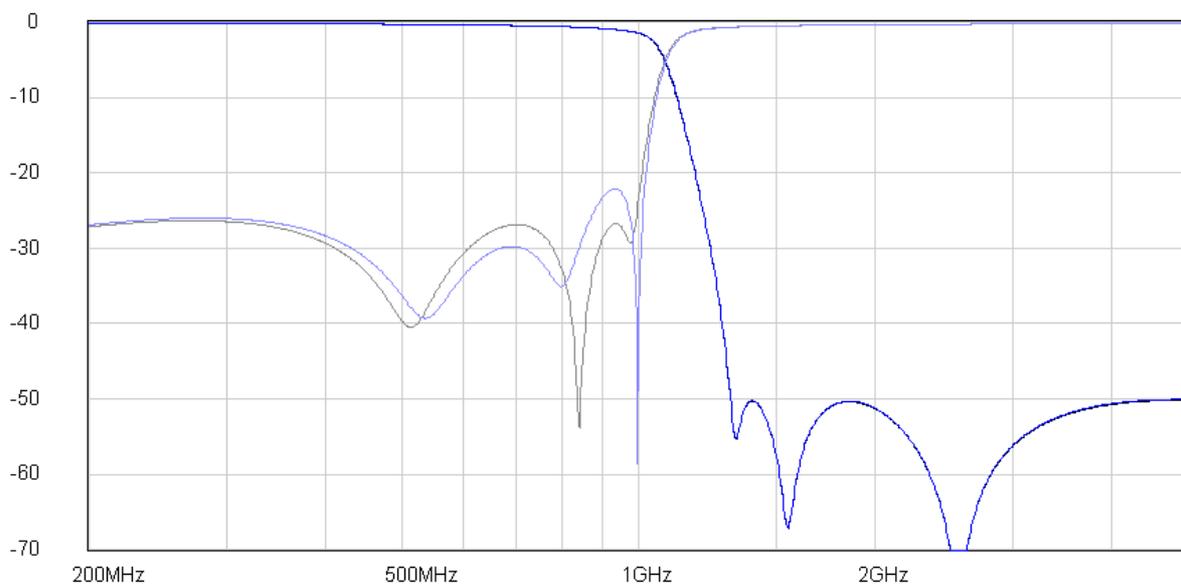
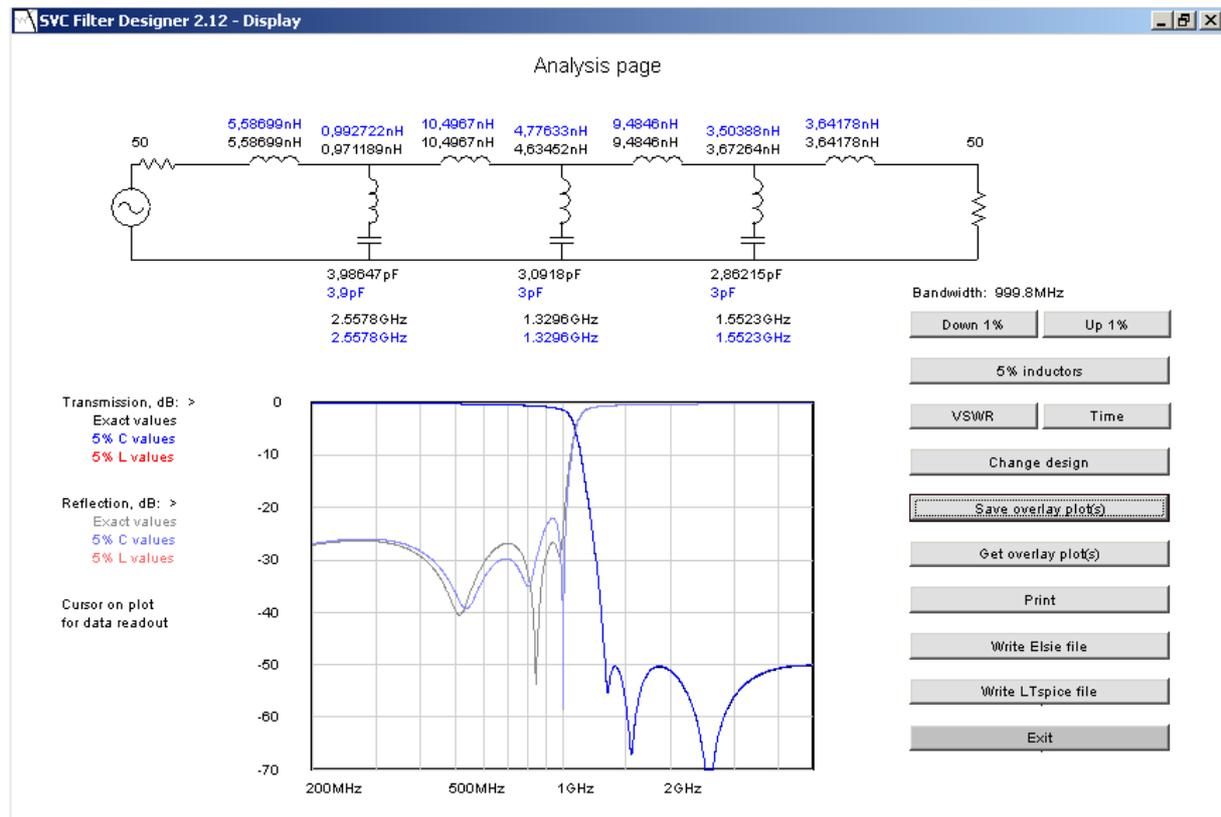
Most errors can be 'fixed' by changing the width of the capacitive traces or inductive traces. See 'Trace widths and substrate height'.

5 A practical example

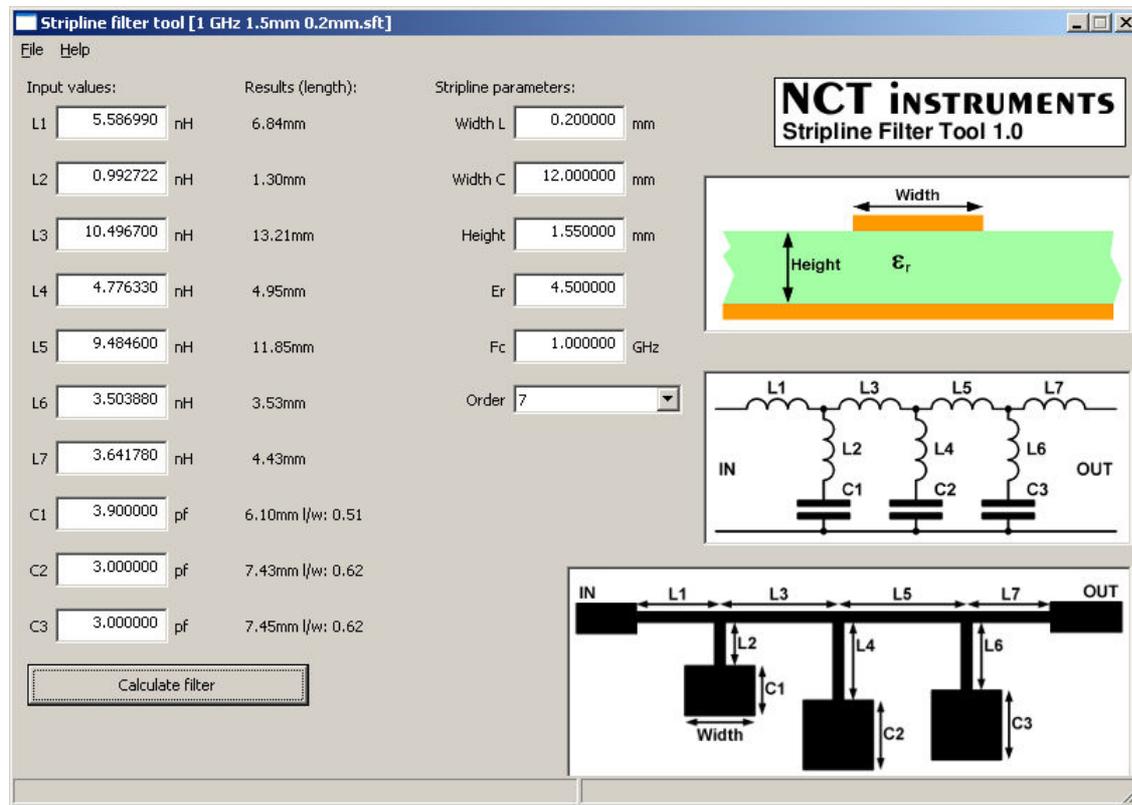
This chapter shows the step-by-step design flow. The design requirements are to design a 7th order filter with a cut-off frequency at 1GHz, 0.01dB pass band ripple and 40dB stop band attenuation.

5.1 Component values

Because the stop band attenuation is expected to be less than achieved by ideal components the stop band attenuation is chosen at 50dB. The screen dump below shows the values found by SVC filter and the frequency response:



For comparison the filter will be built with 0.2mm and 1.2mm inductive traces on an FR4 PCB using a substrate which is 1.55mm in height. It is assumed the PCB has an Er of 4.5. Both filters will use capacitive traces which are 12mm wide. The screen dump below shows the input values and the results for the filter made from 0.2mm inductive traces.



The results lead to the following reports for the two trace widths:

Elliptic filter report

Material parameters:

Er: 4.50
 Substrate thickness: 1.55 mm
 Inductors trace width: 0.20 mm
 Capacitors trace width: 12.00 mm

Filter parameters:

Order: 7
 Cut-off frequency: 1.0000 GHz
 L1 value: 5.58699 nH length: 6.84 mm
 L2 value: 0.99272 nH length: 1.20 mm
 L3 value: 10.49670 nH length: 13.21 mm
 L4 value: 4.77633 nH length: 4.95 mm
 L5 value: 9.48460 nH length: 11.85 mm
 L6 value: 3.50388 nH length: 3.53 mm
 L7 value: 3.64178 nH length: 4.43 mm

 C1 value: 3.90000 nH length: 6.45 mm
 C2 value: 3.00000 nH length: 7.43 mm
 C3 value: 3.00000 nH length: 7.45 mm

Elliptic filter report

Material parameters:

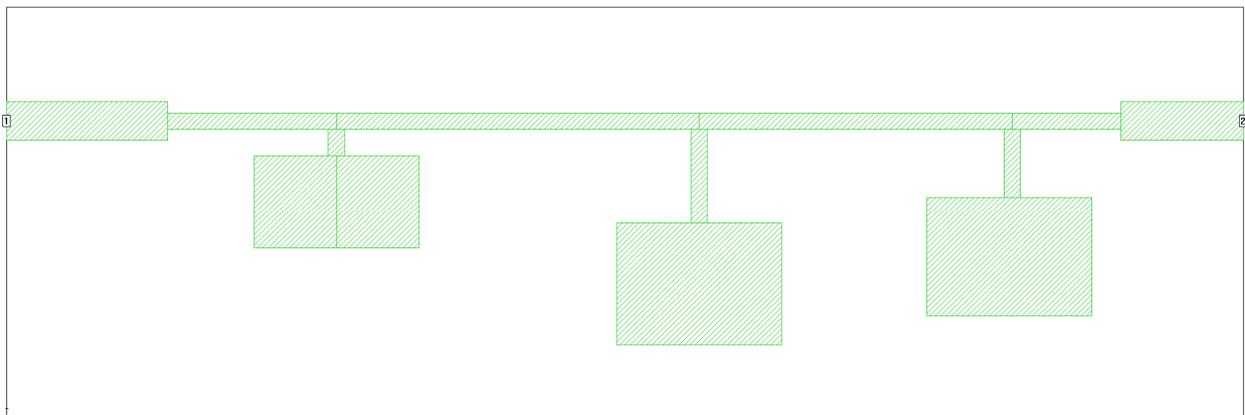
Er: 4.50
 Substrate thickness: 1.55 mm
 Inductors trace width: 1.20 mm
 Capacitors trace width: 12.00 mm

```

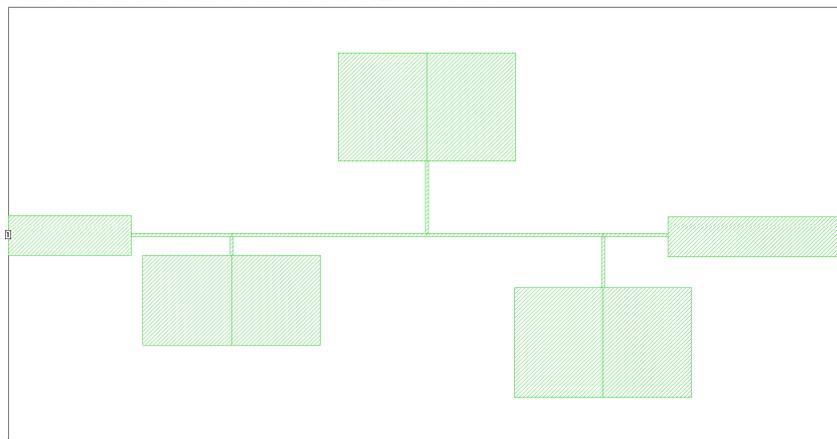
*****
Filter parameters:
Order:                7
Cut-off frequency:    1.0000 GHz
L1 value:  5.58699 nH  length:  12.30 mm
L2 value:  0.99272 nH  length:   1.90 mm
L3 value: 10.49670 nH  length:  26.36 mm
L4 value:  4.77633 nH  length:   6.92 mm
L5 value:  9.48460 nH  length:  22.81 mm
L6 value:  3.50388 nH  length:   5.03 mm
L7 value:  3.64178 nH  length:   7.85 mm

C1 value:  3.90000 nH  length:   6.77 mm
C2 value:  3.00000 nH  length:   8.90 mm
C3 value:  3.00000 nH  length:   8.72 mm
*****
    
```

Based on the reports both filters were input into Sonnet Lite. Below is the filter with 1.2mm inductive traces:



And this is the filter based on 0.2mm traces

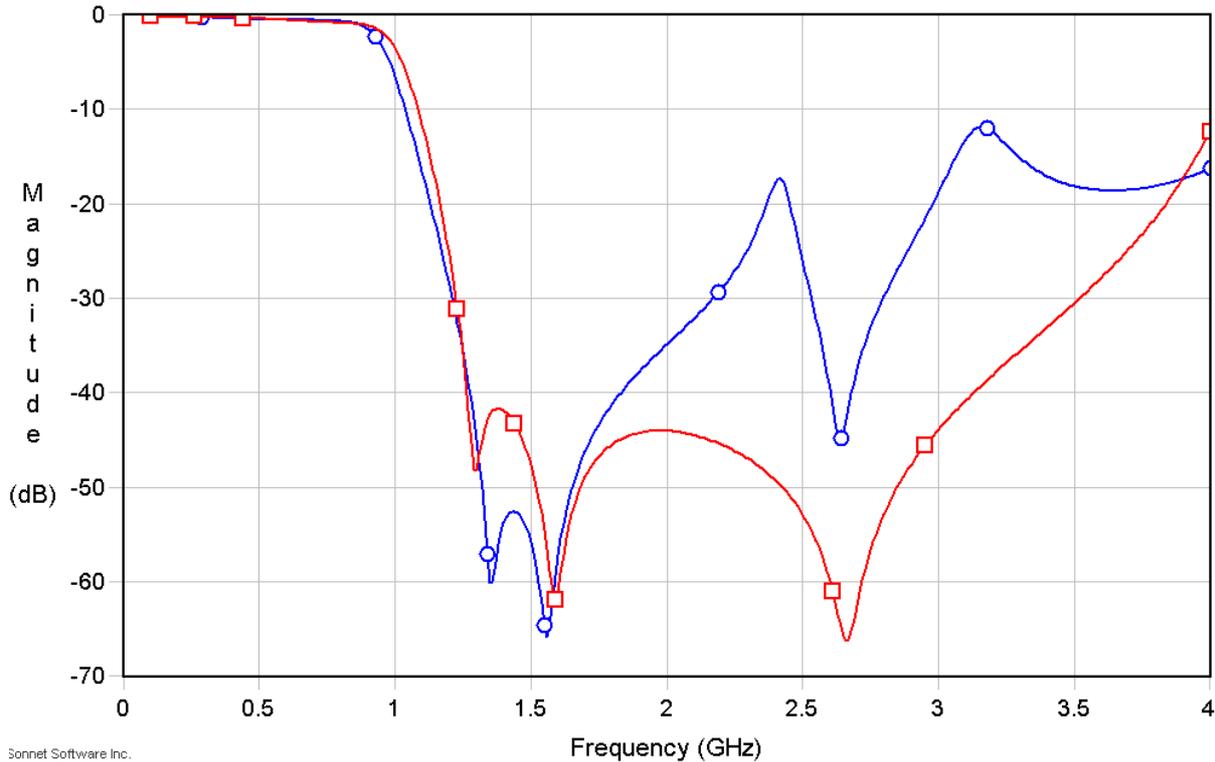


It is clear that the filter with 0.2mm traces is much shorter than the one with the 1.2mm traces. Actually the filter is so much shorter there was no room to place the 'capacitors' next to each other.

The grid in Sonnet is set to 0.1mm so the sizes of the traces were rounded to 0.1mm. A nice trick when drawing these kind of filters is to create the capacitive traces out of 2 identical rectangles. This way it is easy to connect the capacitive traces exactly in the middle of the inductive traces.

The thick traces are 2.8mm wide 50 Ohm traces which connect the filter to the outside world.

The simulation results are shown in the following graph:

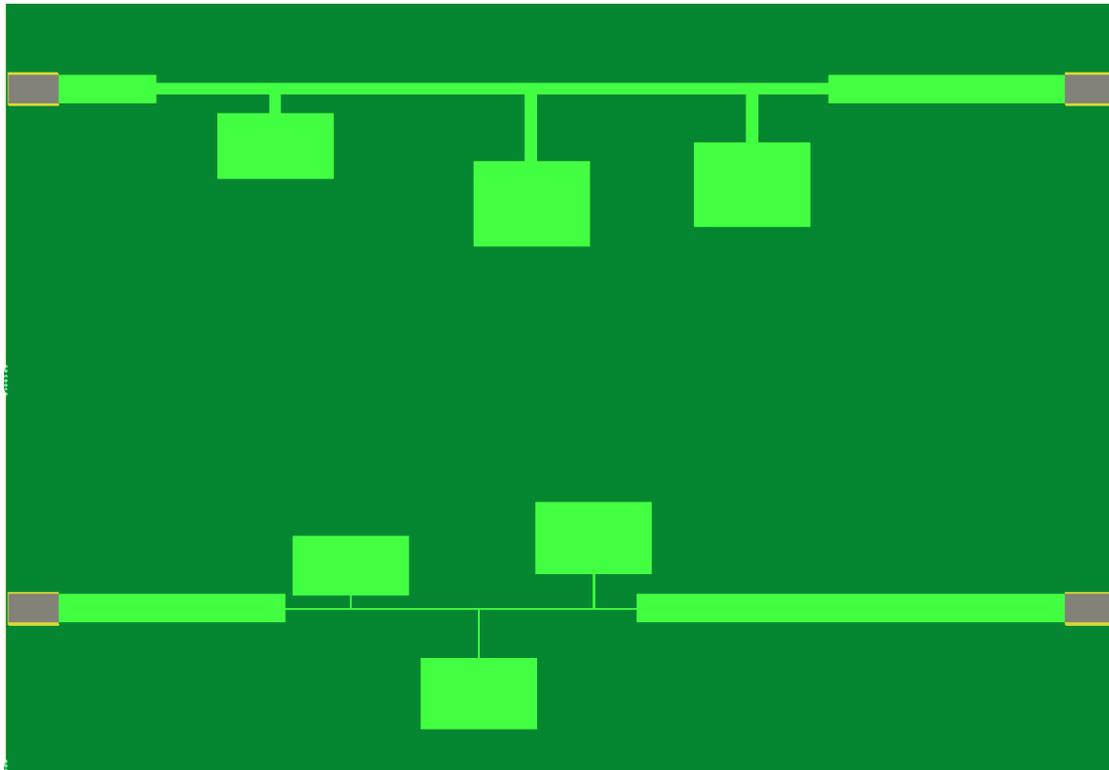


Sonnet Software Inc.

The blue line shows the result from the filter with 1.2mm inductive traces, the red line shows the result of the filter with 0.2mm inductive traces. The filter using 0.2mm traces shows promise to achieve the wanted 40dB attenuation.

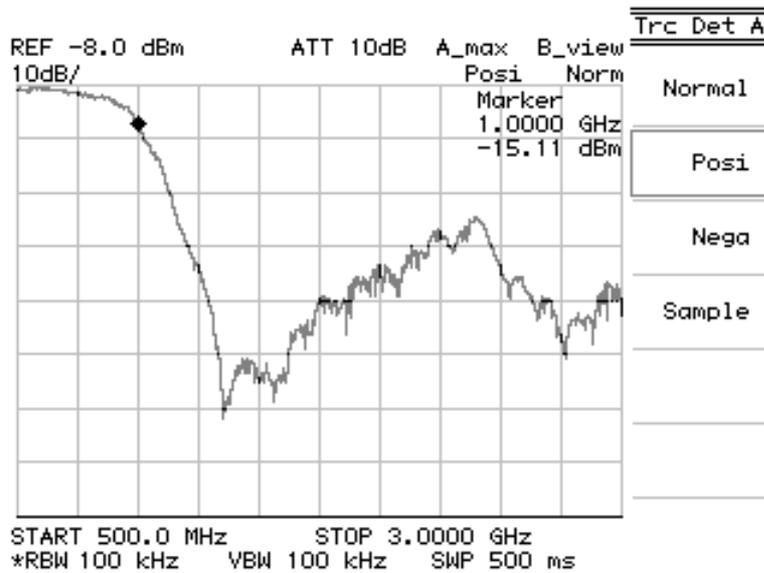
5.2 Testing a real filter

Of course the proof of the pudding is in the tasting. In order to verify the filter it was produced on a PCB. A picture of the layout is shown below:

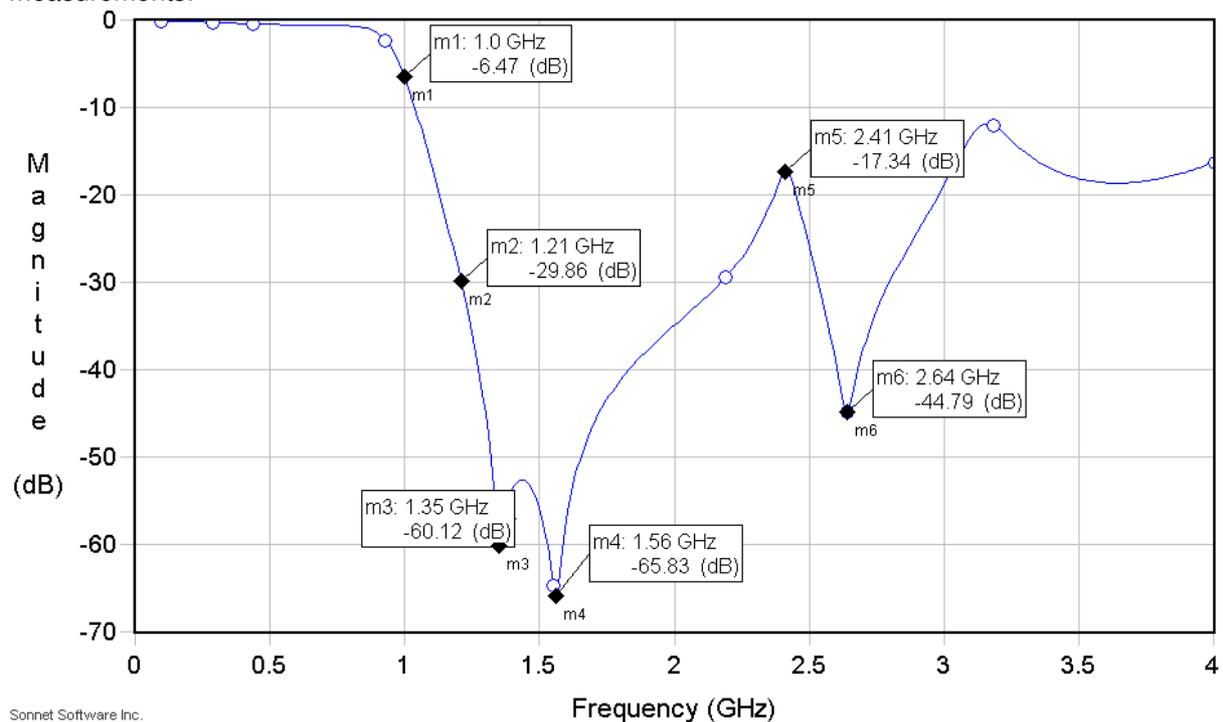


5.2.1 Results for the '1.2mm filter'

The following response was obtained:

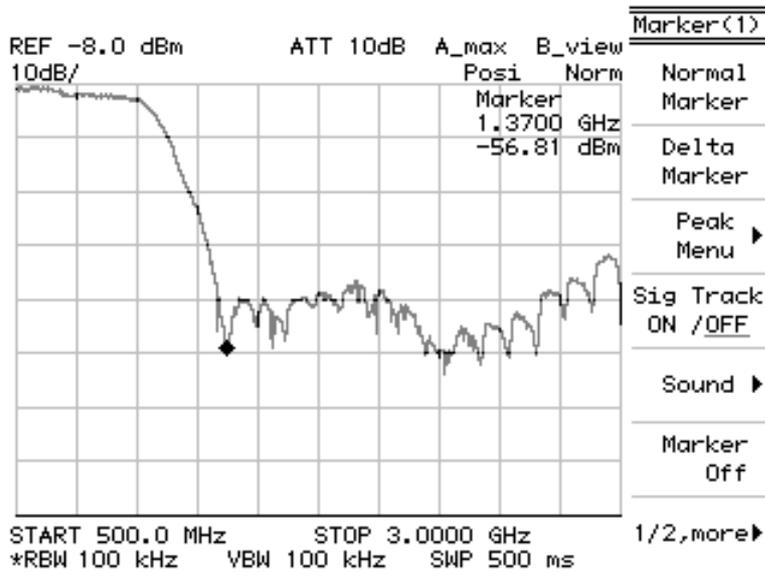


A closer comparison between the simulation and the measurements using several markers shows that the real world filter matches the simulation. Please note that there is a -9dB offset in the measurements.



5.2.2 Results for the '0.2mm filter'

Below the response of the filter:



Below the simulated response with several markers. Please note that there is a -9dB offset in the measurements.

