Microwave Substrate Filter Implementation for a 5G Advance Antenna System

LEONARDO PADIAL TORÁN
MASTER´S THESIS
DEPARTMENT OF ELECTRICAL AND INFORMATION TECHNOLOGY
FACULTY OF ENGINEERING | LTH | LUND UNIVERSITY

Microwave Substrate Filter Implementation for a 5G Advance Antenna System

Leonardo Padial Torán
leonardo.padial@ericsson.com

Department of Electrical and Information Technology
Lund University

Supervisor: Markus Törmänen (EIT), Alexander Bondarik (Ericsson AB), James H. Bedu-Addo (Ericsson AB)

Examiner: Pietro Andreani (EIT)

September 6, 2018

Abstract

The following text is the Master Thesis at Ericsson AB from the months of February 2018 to August 2018 on radio frequency filter design. The research and the simulations were done with M Bashar Arhayem.

The filter for advanced antenna systems in 5G that we had to design was for 28GHz operating band. The figure 0.1 shows the specifications given. The bandwidth is defined, in our case, based on our Return Losses from B to C, 27.5 GHz to 28.35 GHz, not lower than 15 dB. The lower band, below 26.5 GHz, and the upper bands, from 31 GHz, are attenuated. The insertion losses can be up to 2 dB in the operating band. Also from E to F, from 55 GHz to 56.7 GHz, 2xRF, has to be attenuated by 23 dB. The requirement is made to remove the 2nd harmonic. That will require special treatment, solved by Stepped Impedance Resonators.

The reader can find in the first chapter the theory and basic knowledge to understand filter theory and distributed elements, both microstrip and stripline. Also in the first chapter is mentioned the software used, both ADS and HFSS. In the second chapter we work with uniform impedance resonators. We focus in the transfer function and the resonance points for three different filter configurations. In the third chapter the reader can find the steps to design a stepped impedance resonator based filter. The reason of this improvement is to shift the second resonance of the filter, not to fall at 2xRF, due to the periodicity of the distributed elements. Finally some conclusions and future work will be explained.
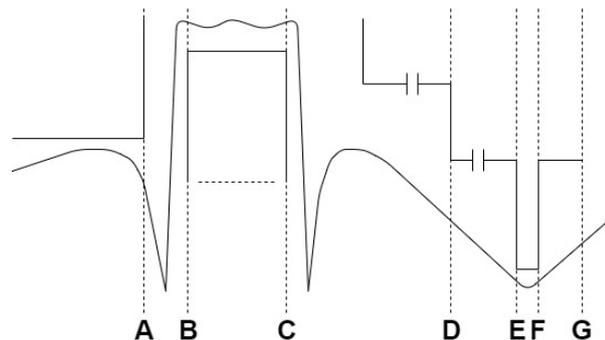
Figure 0.1: Filter specifications.

Acknowledgements

Popular Science Summary

The filter design is a mandatory course for every electrical engineer. The reason why is so important it is because we need a way to discriminate the signals that are important for us.

If you think, for example, when you are talking with someone, the perfect situation would be when there is not noise around you. Then you can listen perfectly the other speaker. If there were more people talking at the same time you need to focus on the information you are interested on. It is easier if the other speakers were talking at different tunes, such lower pitch, or higher pitch, or volume. We naturally filter and distinguish different tunes and volumes in our brain.

That is the purpose of the filter. These days there are a lot of devices connected, mobile phones, medical devices, i.e. pacemaker or sensors for insulin, television broadcasting or connected cars. The different standards are designed to provide reliable connectivity while you keep low the interference created.

With 5G new standards will come. New frequencies will be available for such communications. One of the frequency bands is at 28 GHz.

Table of Contents

List of Figures

List of Tables

Chapter 1

Introduction

Là-bas
Tout est neuf et tout est sauvage
Libre continent sans grillage
Ici, nos rêves sont étroits
C'est pour ça que j'irai là-bas
Là-bas
Faut du cœur et faut du courage
Mais tout est possible à mon âge
Si tu as la force et la foi
L'or est à portée de tes doigts
C'est pour ça que j'irai là-bas
Là-bas, Jean-Jacques Goldman

A filter is a dual port block, with one input port and one output port. The objective of the filter is to let part of the input signal pass through to the output, while the other part of the signal is rejected. In our field, this discrimination is done according to specifications in frequency which translates to the fact that some frequencies experience severe attenuation when passing through (rejection) while others (the desired ones) does not experience almost any loss. It can be digital or analog filter. Even between analog filters, it can be passive or active. By passive means that we don't provide power supply, therefore there is not gain. Digital filter these days are very powerful and accurate, but they consume energy. So we need to get a balance between performance and energy efficiency.

The filter attaches to an antenna is responsible of filtering the signals transmitted and received from the transceiver-device.
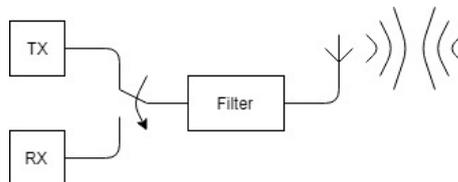
Figure 1.1: The filter designed will be placed between the antenna and the transceiver.

In this chapter the reader can find information about filter synthesis. Then it is described the shape, behaviour and electric parameters of microstrip and stripline. The reader will understand also the reasons why we designed the filter in stripline technology. The software used in this thesis are described with some of the features. Finally Smith Chart is explained.

1.1 Filter synthesis

Filter theory is a complicated field. The main reason is that we cannot achieve ever the ideal situation. The questions we have to keep in mind are:

- Can we achieve a perfect match (zero reflection) over a specific bandwidth?[1]

- If not, how well can we do? What is the trade-off between Γ_m, the maximum allowable reflection in the passband, and the bandwidth?

- How complex must the matching network be for a given specification?

The best performance achievable is given by Bode-Fano criterion.

$$\int_0^\infty ln\frac{1}{|\Gamma(\omega)|}d\omega \leq \frac{\pi}{RC} \tag{1.1}$$

where $\Gamma(\omega)$ is the reflection coefficient seen looking into the arbitrary lossless matching network.
The following conclusions can be read:

- For a given load (a fixed RC product), a broader bandwidth($\triangle\omega$) can be achieved only at te expenses of a higher reflection coefficient in the passband (Γ_m).

- The passband reflection coefficient, Γ_m, cannot be zero unless $\triangle\omega = 0$. Thus a perfect match can be achieve only at a finite number of discrete frequencies.

- As R and/or C increases, the quality of the match($\triangle\omega$ and/or $\frac{1}{\Gamma_m}$) must decreases. Thus, higher-Q circuits are intrinsically harder to match than are lower-Q circuits.

Once this is mentioned then we explain the steps for a quick filter design.

Filter specifications → Low-pass prototype design → Scaling and conversion → Implementation
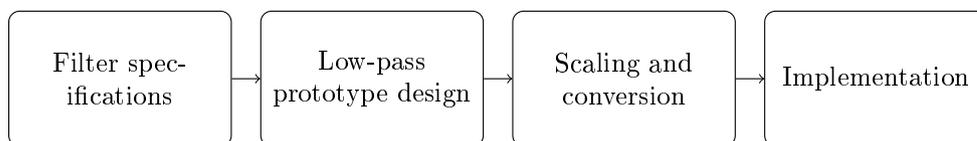
The quickest way is to use a prototype low pass filter. In this filter the impedance and the frequency is normalized to 1. "A perfect filter would have zero insertion loss in the passband, infinite attenuation in the stopband, and a linear phase

---

[1]Information taken from Microwave Engineering David M. Pozar [1].
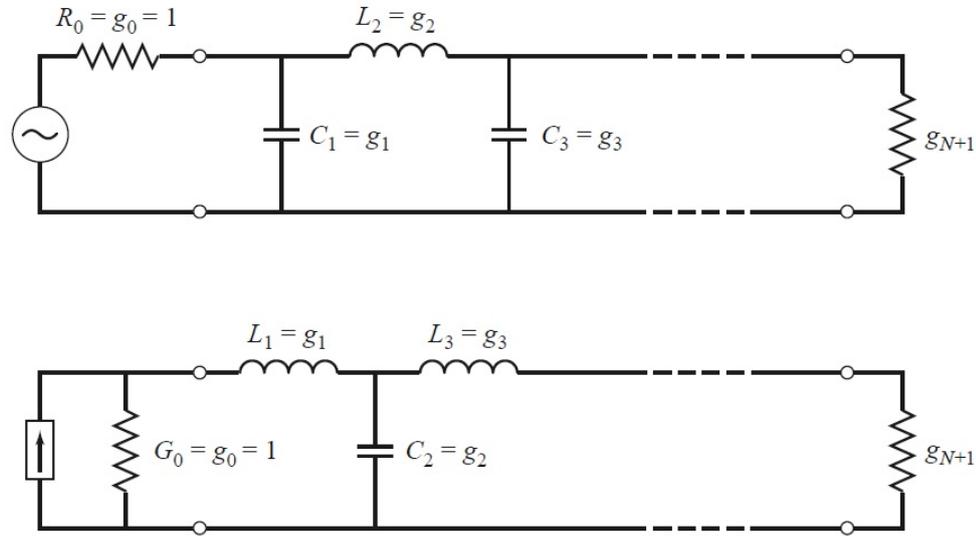
Figure 1.2: Ladder circuits for low-pass filter prototypes and their element definitions. Prototype beginning with a shunt element (top). Prototype beginning with a series element (bottom).

response (to avoid signal distortion) in the passband. Of course, such filters do not exist in practice, so compromises must be made; herein lies the art of filter design."[1] So the question is, how we design a filter. The answer for our problem can be "Filter Design By The Insertion Loss Method" [1]. In the insertion loss method the response is defined by its insertion loss

$$P_{LR} = \frac{\texttt{Power available from source}}{\texttt{Power delivered to load}} = \frac{P_{inc}}{P_{load}} = \frac{1}{1 - |\Gamma(\omega)|^2} \qquad (1.2)$$

The insertion loss (IL) in dB is

$$IL = 10 log P_{LR} \qquad (1.3)$$

We know that $|\Gamma(\omega)|^2$ is an even function of ω; therefore it can be expressed as a polynomial in ω^2.

$$|\Gamma(\omega)|^2 = \frac{M(\omega^2)}{M(\omega^2) + N(\omega^2)} \qquad (1.4)$$

where M and N are real polynomials in ω^2.

$$P_{LR} = 1 + \frac{M(\omega^2)}{N(\omega^2)} \qquad (1.5)$$

When we limit the power loss ratio simultaneously constrains the magnitude of the reflection coefficient, $|\Gamma(\omega)|$.
We can find different filter based on the response

Maximally flat, also called binomial or Butterworth. It has the flattest response possible on the passband.

$$P_{LR} = 1 + k^2 \left(\frac{\omega}{\omega_c} \right)^{2N} \tag{1.6}$$

To determine the order of Butterworth lowpass prototype filter, a specification that is usually the minimum stopband attenuation $L_{As}dB$ at $\Omega = \Omega_s$ for $\Omega > 1$.

$$n \geq \frac{log(10^{0.1L_{As}} - 1)}{2log\Omega_s} \tag{1.7}$$

And the equations to get the g-parameters are

$$g_0 = 1.0 \tag{1.8}$$

$$g_i = 2sin\left(\frac{(2i-1)n}{2n} \right) \quad \text{for i = 1..n} \tag{1.9}$$

$$g_{n+1} = 1.0 \tag{1.10}$$

Chebyshev Type I has ripple in the passband. The ripple actually helps to have later on a more abrupt transition between passband and stopband.

For the required passband ripple $L_{Ar}dB$, the minimum stopband attenuation $L_{As}dB$ at $\Omega = \Omega_s$, the degree of a Chebyshev lowpass prototype, which will meet this specification

$$n \geq \frac{cosh^{-1}\sqrt{\frac{10^{0.1L_{As}}-1}{10^{0.1L_{Ar}}-1}}}{cosh^{-1}\Omega_s} \tag{1.11}$$

And the equations to get the g-parameters are

$$g_0 = 1.0 \tag{1.12}$$

$$g_1 = \frac{2}{\gamma}sin\left(\frac{\pi}{2n} \right) \tag{1.13}$$

$$g_i = \frac{1}{g_{i-1}} \frac{4sin\left[\frac{(2i-1)\pi}{2n} \right]sin\left[\frac{(2i-3)\pi}{2n} \right]}{\gamma^2 + sin^2\left[\frac{(i-1)\pi}{n} \right]} \tag{1.14}$$

$$g_{n+1} = \begin{cases} 1.0 & \text{for n odd} \\ coth^2\left(\frac{\beta}{4} \right) & \text{for n even} \end{cases} \tag{1.15}$$

Sometimes the minimum return loss L_R or the maximum voltage standing wave ratio VSWR in the passband is specified instead of the passband ripple L_{Ar}

$$L_R(\Omega) = 10log[1 - |s_{21}(j\Omega)|^2]dB \tag{1.16}$$

then

$$L_{Ar} = -10log(1 - 10^{0.1L_R})dB \tag{1.17}$$

where L_{Ar} is the passband ripple.

Similarly since the definition of VSWR is $VSWR = \frac{1+|s_{11}|}{1-|s_{11}|}$ we can convert VSWR into L_{Ar} by

$$L_{Ar} = -10log\left[1 - \left(\frac{VSWR-1}{VSWR+1}\right)^2\right]dB \tag{1.18}$$

Chebyshev Type II   has ripple in the stopband but in the bandpass is flat. The bandwidth is defined by the return losses so it might be complicated to get that performance with only one resonance on the bandpass. The overall picture with an elliptic filter might look like an inverse Chebyshev.

Elliptic filter   , also called Cauer[2]. This filter has equalized ripple behaviour in both the passband and the stopband. This configuration allows to have shorter transitions between bandpass and stopband.

1.1.1   Filter Transformation

The different transformations are:

1. Impedance. It is generally annoying to work with high values or extremely low values of capacitance, inductance and impedance, so we normally normalize to the impedance of the network, i.e. 50Ω.

$$\begin{aligned} L' &= R_0 L \\ C' &= \frac{C}{R_0} \\ R'_S &= R_0 \\ R'_L &= R_0 R_L \end{aligned} \tag{1.19}$$

2. Frequency scaling. To translate your prototype filter from unity to ω_c then you need the follow transformation

$$\omega \leftarrow \frac{\omega}{\omega_c}$$

Low-pass to high-pass transformation

$$\omega \leftarrow -\frac{\omega_c}{\omega}$$

Low-pass to band-pass transformation

$$\omega \leftarrow \frac{\omega_0}{\omega_2 - \omega_1}\left(\frac{\omega}{\omega_0} - \frac{\omega_0}{\omega}\right) = \frac{1}{\Delta}\left(\frac{\omega}{\omega_0 - \frac{\omega_0}{\omega}}\right)$$

---

[2]The filter was named after Wilhelm Cauer. His body was located after the end of the 2nd World War in a mass grave of victims of Russian executions.

where

$$\Delta = \frac{\omega_2 - \omega_1}{\omega_0}$$

is the fractional bandwidth of the bandpass. ω_0 can be chosen as

$$\omega_0 = \sqrt{\omega_1 \omega_2}$$
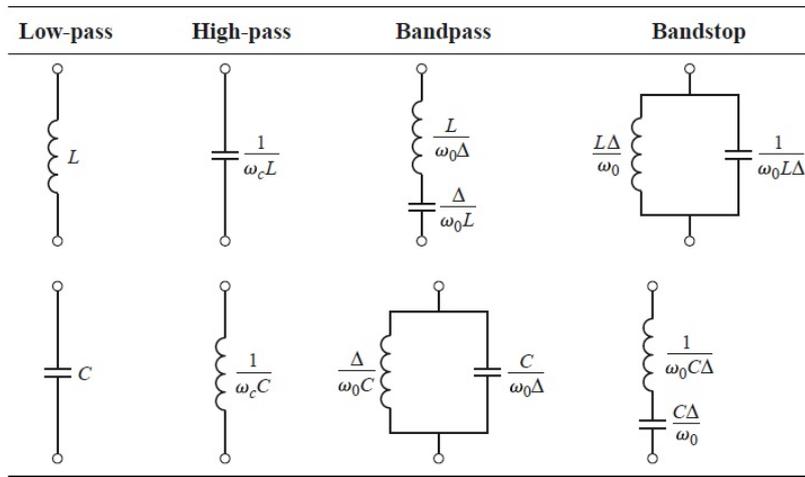
Figure 1.3: Summary of Prototype Filter Transformations $\left(\Delta = \frac{\omega_2 - \omega_1}{\omega_0}.\right)$

Once we know which filter we are going to implement, from now on in this document we are only going to consider band pass filter, we look for the values of g-parameters between the tables from the different practical filters, Butterworth, Chebeshev, inverse Chebyshev, Elliptic. Once we got the g-parameters, we proceed with impedance scaling and frequency scaling.

EXAMPLE 1:

Bandpass elliptic filter. $f_c = 28GHz$ and $BW = 3GHz$.

n	Ω_s	$L_{As}dB$	g_1	g_2	g_3	g_4
3	2.5000	30.5161	0.9471	1.0173	0.1205	0.9471

Applying the bandpass transformation previously explained[3] we get the following values.

	g_1	g_2	g_3	g_4
L	2.51226E-09 H	3.00189E-11 H	3.19636E-10 H	2.51226E-09 H
C	1.28976E-14 F	1.07939E-12 F	1.01372E-13 F	1.28976E-14 F
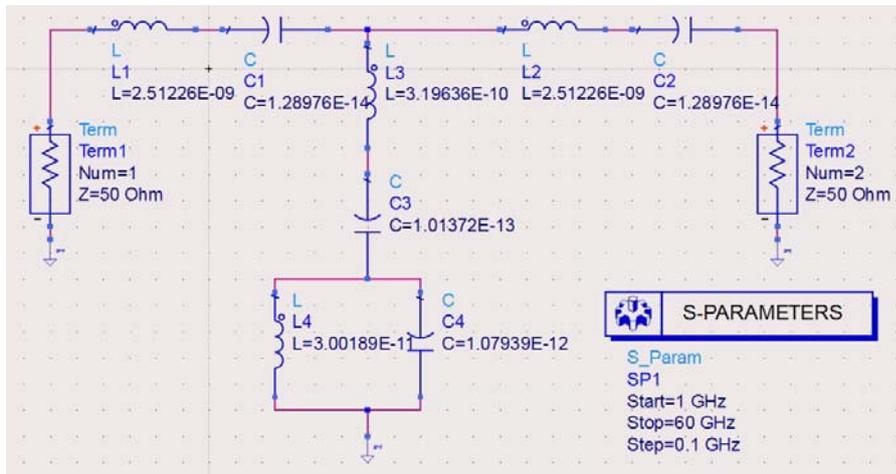
Figure 1.4: Elliptic BPF.
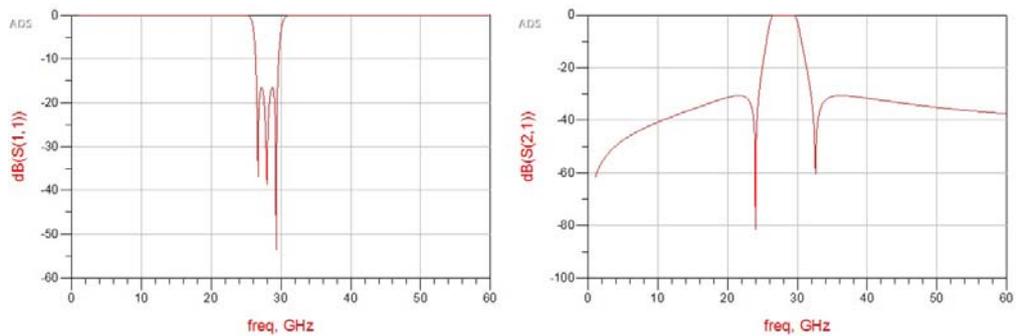
Figure 1.5: Elliptic BPF response. s_{11} and s_{21} parameters.

---

[3]The reader can find more information on Chapter 8: Microwave Filters, Pozar. Bandpass and Bandstop transformations.

We know that lumped components can only be used when the dimensions of the component is $\frac{\lambda}{10}$ or $\frac{\lambda}{20}$ of the wavelength. In the literature one can find that Richard's transformation allows you to translate from lumped component to open- and short-circuited transmission lines stubs. This transformation works well enough around the frequency you are interested on. Then you might find yourself on the situation where you have series open-circuit stub. Then Kuroda identities can solved this problem by placing extra transmission lines in between those open-circuit stubs. Then the Kuroda identities A.1 produce an equivalent open- or short-circuit parallel stubs.
Kuroda identities are not useful for high-pass or bandpass filters. The reader can find more information on the literature.

1.1.2 Immittance Inverters

The immittance structures are used to convert ladder network, which uses both series and shunt immittances, to equivalent forms that use only series or parallel immittances [18].
The immittance inverters can be impedance and admittance inverters. The purpose of these structures is to transform from inductive to conductive and viceversa.
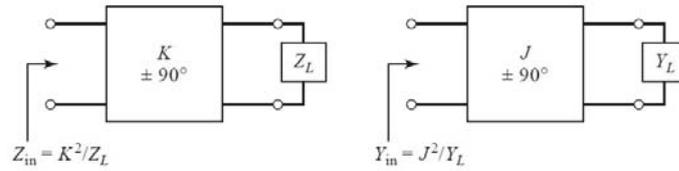
Figure 1.6: Impedance and admitance inverters.

Impedance inverters are known as K-inverters.

$$\begin{bmatrix} A & B \\ C & D \end{bmatrix} = \begin{bmatrix} 0 & \mp jK \\ \pm \frac{1}{jK} & 0 \end{bmatrix} \tag{1.20}$$

Admittance inverters are known as J-inverters.

$$\begin{bmatrix} A & B \\ C & D \end{bmatrix} = \begin{bmatrix} 0 & \pm \frac{1}{jJ} jK \\ \mp jJ & 0 \end{bmatrix} \tag{1.21}$$

We can observe that the values of K and J will modify the impedance or admittance value. In this way we can obtain a more convenient circuit that could be implemented easier.

$$L_{si} = \left(\frac{\Omega_c}{FBW\omega_0} \right) L_{ai}$$

$$C_{si} = \frac{1}{\omega_0^2 L_{si}}$$

Along the thesis, the reader can see the use of these inverters, more concrete, the use of admittance inverters. In the literature is widely use admittance rather than impedance.
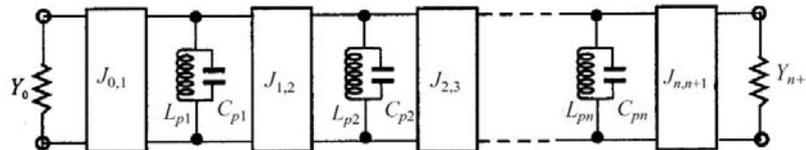
Figure 1.7: Admittance inverters.

$$J_{0,1} = \sqrt{\frac{Y_0 FBW \omega_0 C_{p1}}{\Omega_c g_0 g_1}}$$

$$J_{i,i+1} = \frac{FBW \omega_0}{\Omega_c} \sqrt{\frac{C_{pi} C_{p(i+1)}}{g_i g_{i+1}}}$$

$$J_{n,n+1} = \sqrt{\frac{FBW \omega_0 C_{pn} Y_{n+1}}{\Omega_c g_n g_{n+1}}}$$

$$L_{pi} = \frac{1}{\omega_0^2 C_{pi}} \Big|_{\texttt{i=1 to n}}$$

(1.22)

1.2 Distribuited elements

When lumped components do not suit our design because we are not allowed to use components or, in our case, because the operating band frequencies are high enough, then distributed elements are the solution.

The distributed element model or transmission line model assumes that the attributes of the circuit, such resistance, capacitance or inductance are distributed along the material. So, if we were looking for a capacitance, we would draw a metallic plane over a ground plane with dielectric in between. Or a more narrowed plane would behave as an inductor as many times we have seen during our studies.

We need to understand how waves in the material propagate or transmit from one material to another. We can see the waveguide as a hollow metallic tube of a rectangular or circular shape used to guide electromagnetic waves. [3]

It is possible to propagate several modes of electromagnetic waves within a waveguide. These modes correspond to solutions of Maxwell's equations for particular waveguides. A given waveguide has a definite cutoff frequency for each allowed mode. If the frequency of the impressed signal is above the cutoff frequency for a given mode, the electromagnetic energy can be transmitted through the guide for that particular mode without attenuation. Otherwise the electromagnetic energy with a frequency below the cutoff frequency for that particular mode will be attenuated to a negligible value in a relatively short distance. The dominant mode

in a particular guide is the mode having the lowest cutoff frequency. It is advisable to choose the dimensions of a guide in such a way that, for a given input signal, only the energy of the dominant mode can be transmitted through the guide.
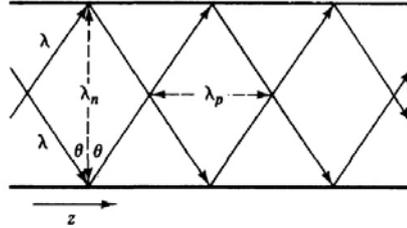
Figure 1.8: Plane wave reflected in a waveguide.

In order to find the cutoff frequency we need to solve the Helmholtz equations. Once we have solved those equations we get the following solutions. The propagation constant is

$$\gamma_g^2 = \gamma^2 + k_c^2 \tag{1.23}$$

where k_c is the cutoff wave number and γ is the intrinsic propagation constant equal to

$$\gamma = \sqrt{j\omega\mu(\sigma + j\omega\epsilon)} = \alpha + j\beta$$

where α represents the attenuation and $j\beta$ the phase.

1.2.1   Microstrip

The fields in the microstrip extend within two media air above and dielectric below so that the structure is inhomogeneous. Due to this inhomogeneous nature, the microstrip does not support a pure TEM wave. This is because a pure TEM wave has only transverse components, and its propagation velocity depends only on the material properties, namely the permittivity ε and the permeability μ. However, with the presence of the two guided-wave media (the dielectric substrate and the air), the waves in a microstrip line will have no vanished longitudinal components of electric and magnetic fields, and their propagation velocities will depend not only on the material properties, but also on the physical dimensions of the microstrip. We want to know what is microstrip based on the dimmensions. And sometimes to know the dimmensions for a certain characteristic impedance.
For the first case

$$Z_0 = \begin{cases} \frac{60}{\sqrt{\epsilon_e}ln\left(\frac{8d}{W} + \frac{W}{4d}\right)} & \text{for } \frac{W}{d} \leq 1 \\ \frac{120\pi}{\sqrt{\epsilon_e}[W/d+1.393+0.667ln(W/d+1.444)]} & \text{for } \frac{W}{d} \geq 1 \end{cases} \tag{1.24}$$

(a) Microstrip line.

$\underline{\hspace{2cm}}$ E

$----$ H
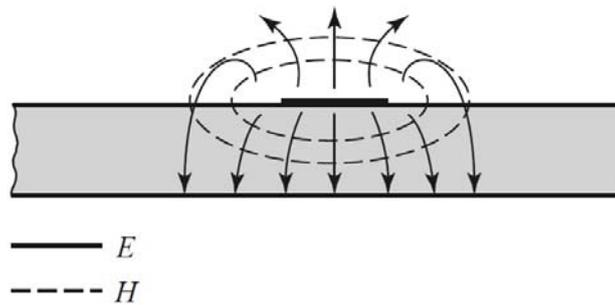
(b) Electric and magnetic fields.

Figure 1.9: Microstrip.

When instead they give us the impedance characteristic and we need to provide some measures

$$\frac{W}{d} = \begin{cases} \frac{8e^A}{e^{2A}-2} & \text{for } \frac{W}{d} < 2 \\ \frac{2}{\pi}\left[B - 1 - ln(2B-1) + \frac{\epsilon_r-1}{2\epsilon_r}\left\{ln(B-1) + 0.39 - \frac{0.61}{\epsilon_r}\right\}\right] & \text{for } \frac{W}{d} > 2 \end{cases}$$
$$(1.25)$$

where

$$A = \frac{Z_0}{60}\sqrt{\frac{\epsilon_r+1}{2}} + \frac{\epsilon_r-1}{\epsilon_r+1}\left(0.23 + \frac{0.11}{\epsilon_r}\right)$$

$$B = \frac{377\pi}{2Z_0\sqrt{\epsilon_r}}$$

The reader might just skip the equations due the length, not complex thou, but long enough to realize that there must be another way to speed up our designs. We have used couple tools along the Thesis that allow us to check at any moment the effective dielectric constant and impedance.

1.2.2 Stripline

In this text, every time a stripline is mentioned, it is meant a symmetric stripline. Which is equivalent to say that the transmission line is half way between both

ground planes. It is possible to find asymmetric stripline and inhomogeneous stripline. The last one consists on two different dielectric materials. When you work with different substrates or heights, you need to keep in mind that the capacitance will be proportional to the dielectric constant and inversely proportional to the heights. When we have two conductor planes, ground planes, and the trans-
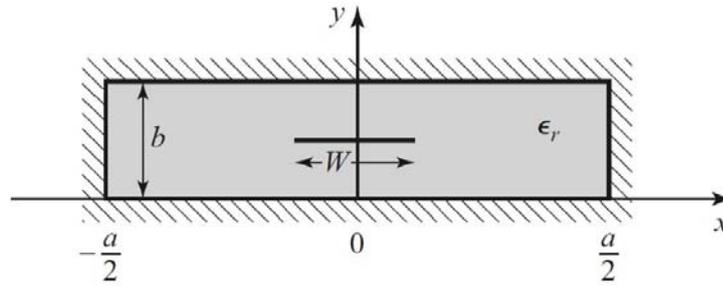
(a) Stripline.
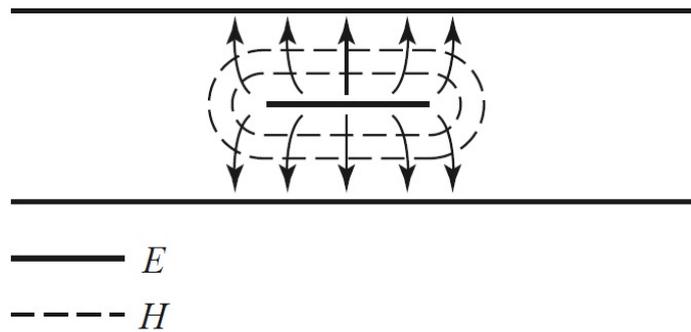
(b) Electric and magnetic fields.

Figure 1.10: Stripline.

mission line, then it supports a TEM[4] waves. It can also transmit multiple modes. Sometimes, regarding a stripline we can find in some literature that next to the stripline we can find vias to limit the modes. In any case, the reader will understand perfectly by the end of the text what it is meant by multiple modes. Most of the filters consider in this Thesis were based on resonators. The physical dimensions of the resonators will allow higher resonate modes.

The phase velocity of a TEM wave is

$$v_p = 1/\sqrt{\mu_0 \epsilon_0 \epsilon_r} \tag{1.26}$$

And the propagation constant of stripline is

$$\beta = \frac{\omega}{v_p} = \omega\sqrt{\mu_0 \epsilon_0 \epsilon_r} = \sqrt{\epsilon_r}k_0 \tag{1.27}$$

---

[4]TEM stands for Transverse electromagnetic.

We write then the characteristic impedance of a transmission line as

$$Z_0 = \sqrt{\frac{L}{C}} = \frac{\sqrt{LC}}{C} = \frac{1}{v_p C} \tag{1.28}$$

where L and C are the inductance and capacitance per unit length of the line.
We canot use the previous formula because we do not know the values of L nor C.
So there is another characteristic impedance equation more convenient for us

$$Z_0 = \frac{30\pi}{\sqrt{\epsilon_r}} \frac{b}{W_e + 0.441b}, \tag{1.29}$$

where W_e is the effective width of the centre conductor given by

$$\frac{W_e}{b} = \frac{W}{b} - \begin{cases} 0 & \text{for } \frac{W}{b} > 0.35 \\ (0.35 - W/b)^2 & \text{for } \frac{W}{b} < 0.35 \end{cases} \tag{1.30}$$

It assumes a strip with zero thickness. The previous equation has 1% deviation over the exact results.
When we design the stripline, we could use the following equation

$$\frac{W}{b} = \begin{cases} x & \text{for } \sqrt{\epsilon_r} Z_0 < 120\Omega \\ 0.85 - \sqrt{0.6 - x} & \text{for } \sqrt{\epsilon_r} Z_0 > 120\Omega \end{cases} \tag{1.31}$$

where

$$x = \frac{30\pi}{\sqrt{\epsilon_r} Z_0} - 0.441 \tag{1.32}$$

1.2.3   Resonator

A resonator is any structure that is able to contain at least one oscillating electromagnetic field. The word "resonator" itself suggest that the conductor will have a resonance, in a similar way than the RLC resonant circuits. It is important then to well understand how a RLC resonant circuit works.
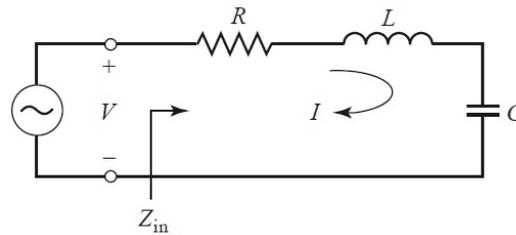
Figure 1.11: A series RLC circuit.

$$Z_{in} = R + j\omega L - j\frac{1}{\omega C} \tag{1.33}$$
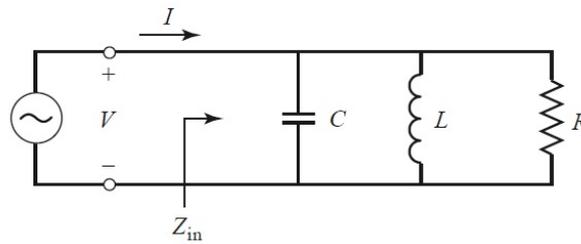
Figure 1.12: A parallel RLC circuit.

$$Z_{in} = \left(\frac{1}{R} + \frac{1}{j\omega L} + j\omega C \right)^{-1} \qquad (1.34)$$

The relation between the bandwidth and the quality factor is

$$BW = \frac{1}{Q_0} \qquad (1.35)$$

The losses of a resonators come from many factors: conductor, dielectric substrate, and radiation. The total unloaded quality factor can be found by adding these losses together, resulting in

$$\frac{1}{Q_u} = \frac{1}{Q_c} + \frac{1}{Q_d} + \frac{1}{Q_r} \qquad (1.36)$$

where Q_c, Q_d and Q_r are the conductor, dielectric, and radiation quality factors, respectively.

Example: Stub component

The stub is a transmission line that is connected only on one end. When a stub
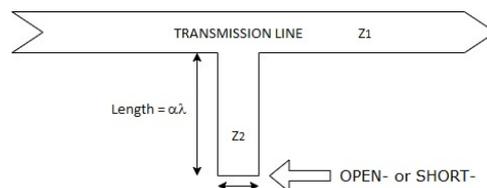
Figure 1.13: Open stub.
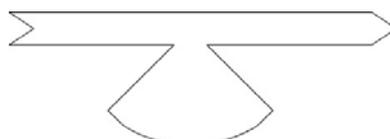
Figure 1.14: Butterfly open stub.

is attached to the main transmission line, the actual behaviour is like attaching an impedance in parallel. But this impedance will resonate depending on the width and the length. Sometime we can see butterfly filters. Those filters have bigger area so, they have different bandwidth because they have different Q-factor. It is easy to see because the capacity created by increasing the surface is bigger. They can be open stub or shorted stub. An easy way to calculate the value or behaviour is starting from the end of the stub. If the stub is open, we could understand it as it has infinity impedance on that side. Then using the Smith Chart, it was explained in previous sections, you start rotating towards the generator as long as your stub length is. If you placed instead a shorted stub, then the impedance on that extreme will be zero, and from there you will turn towards the generator as long as you might need. Wherever you end up, that would be your impedance that you need to add up in parallel to your main line.

Coupling Factor

If we check 1.15a we can analyze the circuit using the current law made by Kirchoff.

(a) Equivalent circuit of n-coupled resonators for loop-equation formulation.
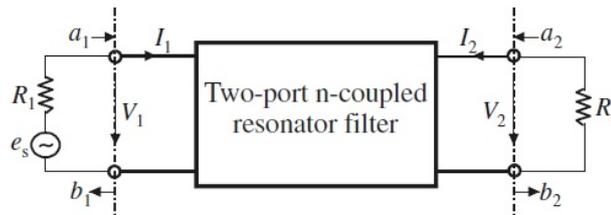
(b) The net representation.

$$\left(R_1 + j\omega L_1 + \frac{1}{j\omega C_1} \right)i_1 - j\omega L_{12}i_2 \cdots - j\omega L_{1n}i_n = e_s$$

$$-j\omega L_{21}i_1 + \left(j\omega L_2 + \frac{1}{j\omega C_2} \right)i_2 \cdots - j\omega L_{2n}i_n = 0$$

$$\vdots$$

$$-j\omega L_{n1}i_1 - j\omega L_{n2}i_2 \cdots + \left(R_n + j\omega_n + \frac{1}{j\omega C_n} \right)i_n = 0$$

(1.37)

in which $L_{ij} = L_{ji}$ represents the mutual inductance between resonators i and j. Another way of writing the equation could be

$$
\begin{bmatrix}
R_1 + j\omega L_1 + \frac{1}{j\omega C_1} & -j\omega L_{12} & \dots & -j\omega L_{1n} \\
-j\omega L_{21} & j\omega L_2 + \frac{1}{j\omega C_2} & \dots & -j\omega L_{2n} \\
\vdots & \vdots & \vdots & \vdots \\
-j\omega L_{n1} & -j\omega L_{n2} & \dots & R_n + j\omega_n + \frac{1}{j\omega C_n}
\end{bmatrix}
\begin{bmatrix} i_1 \\ i_2 \\ \vdots \\ i_n \end{bmatrix}
=
\begin{bmatrix} e_s \\ 0 \\ \vdots \\ 0 \end{bmatrix}
\tag{1.38}
$$

or

$$
[Z] \cdot [i] = [e] \tag{1.39}
$$

All the filters with resonators that we have design were designed using the same resonator, which means they are synchronously tuned. So $\omega_0 = 1/\sqrt{LC}$, where $L = L_1 = L_2 = \dots = L_n$ and $C = C_1 = C_2 = \dots = C_n$.

$$
[Z] = \omega_0 L \cdot FBW \cdot [\bar{Z}] \tag{1.40}
$$

Quick reminder about the Fractional BandWidth, $FBW = \Delta\omega/\omega_0$. $[\bar{Z}]$ is the normalized impedance matrix.

$$
[\bar{Z}] =
\begin{bmatrix}
\frac{R_1}{\omega_0 L \cdot FBW} + p & -j\frac{\omega L_{12}}{\omega_0 L} \cdot \frac{1}{FBW} & \cdots & -j\frac{\omega L_{1n}}{\omega_0 L} \cdot \frac{1}{FBW} \\
-j\frac{\omega L_{21}}{\omega_0 L} \cdot \frac{1}{FBW} & p & \cdots & -j\frac{\omega L_{2n}}{\omega_0 L} \cdot \frac{1}{FBW} \\
\vdots & \vdots & \vdots & \vdots \\
-j\frac{\omega L_{n1}}{\omega_0 L} \cdot \frac{1}{FBW} & -j\frac{\omega L_{n2}}{\omega_0 L} \cdot \frac{1}{FBW} & \cdots & \frac{R_n}{\omega_0 L \cdot FBW} + p
\end{bmatrix}
\tag{1.41}
$$

with

$$
p = j\frac{1}{FBW}\left(\frac{\omega}{\omega_0} - \frac{\omega_0}{\omega}\right)
$$

the complex lowpass frequency variable. It should be noticed that

$$
\frac{R_i}{\omega_0 L} = \frac{1}{Q_{ei}} \quad \text{for } i = 1\dots n \tag{1.42}
$$

Q_{e1} and Q_{en} are the external quality factors of the input and output resonators, respectively. We define now the coupling coefficient as

$$
M_{ij} = \frac{L_{ij}}{L} \tag{1.43}
$$

assuming $\omega/\omega_0 \approx 1$ for a narrow-band approximation, we can say

$$
[\bar{Z}] =
\begin{bmatrix}
\frac{1}{q_{e1}} + p & -jm_{12} & \dots & -jm_{1n} \\
-jm_{21} & p & \dots & -jm_{2n} \\
\vdots & \vdots & \vdots & \vdots \\
-jm_{n1} & -jm_{n2} & \dots & \frac{1}{q_{en}} + p
\end{bmatrix}
\tag{1.44}
$$

where q_{e1} and q_{en} are the scaled external quality factors.

$$
q_{ei} = Q_{ei} \cdot FBW \quad \text{for } i = 1, n \tag{1.45}
$$

and m_{ij} denotes the so-called normalized coupling coefficient

$$
m_{ij} = \frac{M_{ij}}{FBW} \tag{1.46}
$$

1.3 Software

Along these six months, we have used two different software tools. One of them is Advanced Design System, from now on ADS, the second one High-Frequency structure simulator, HFSS. The first one is an integral package, it also includes different assistant tools and also it includes a 3D planar EM simulator. The second one is pure 3D EM simulator. The second one gives more accurate results when we try to simulate the resonators.

1.3.1 HFSS

HFSS stands for High-Frequency structure simulator. It is a software from ANSYS. The simulations are calculated on finite element method. In order to compute our structures we need solve the boundary value problems and the partial differential equations. Those calculations take a large amount of time so the software divides the whole structure into smaller pieces, called finite elements. Then it is easier to solve the equations for such a small elements and add together all the effects.
The HFSS environment is similar to any other CAD. You need to create a project, add a new design and start drawing. A good way of design it is defining the constants and parameters. Our work flow is as follow

First studying and literature review

ADS for quick prototyping and quick understanding of the structure

HFSS for deeper understanding and accurate simulations
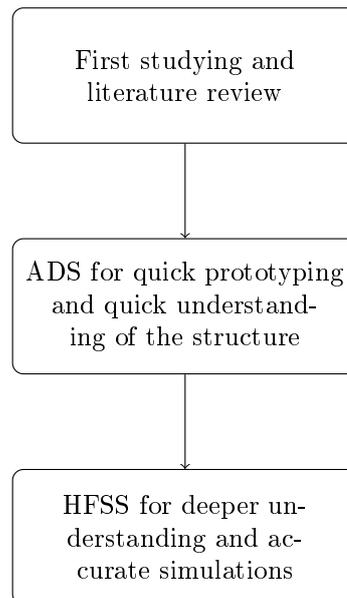
1.3.2 ADS

ADS stands for Advanced Design System, an integral software from Keysight Technologies, which is a spin off from Agilent Technologies, which is also a spin off from Hewlett-Packard. This software includes schematic capture, layout, frequency and time domain simulation and electromagnetic field simulation. While you work in the schematic you can use some tools provided by ADS that helps you on the filter

synthesis, or the LineCalc tools to assists you on the stripline design. Finally, once you have a design that theoretically meet your specifications, then you have the possibility to use Momentum. Momentum is an EM simulator. This will provided more accurate results.

Filter assistant.

1. Select a filter block.
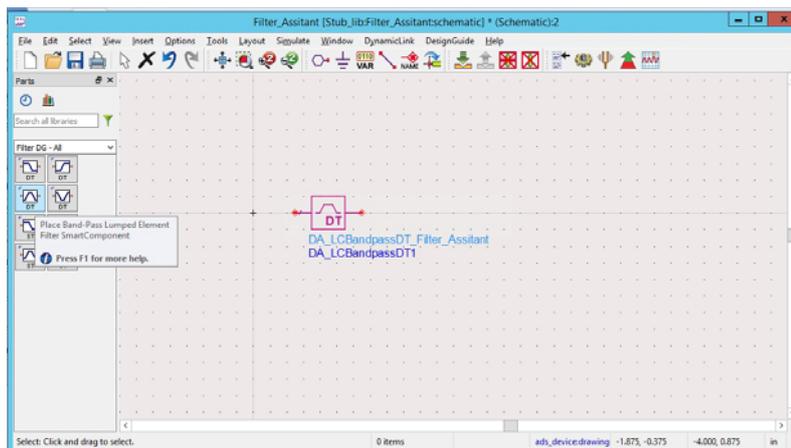
Figure 1.16: Filter from DesignGuide.

2. Select the Filter assistant from DesignGuide.
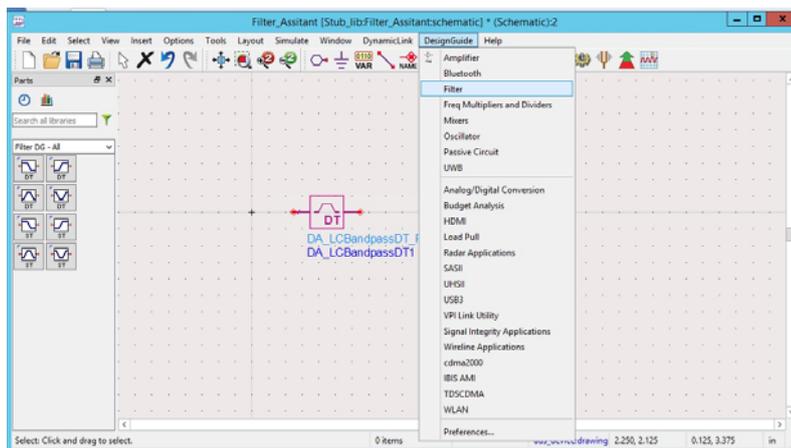
Figure 1.17: Filter from DesignGuide.

3. Select the type of filter, i.e. Elliptic, and the values, low stop band, ripple attenuation, return losses, upper band.
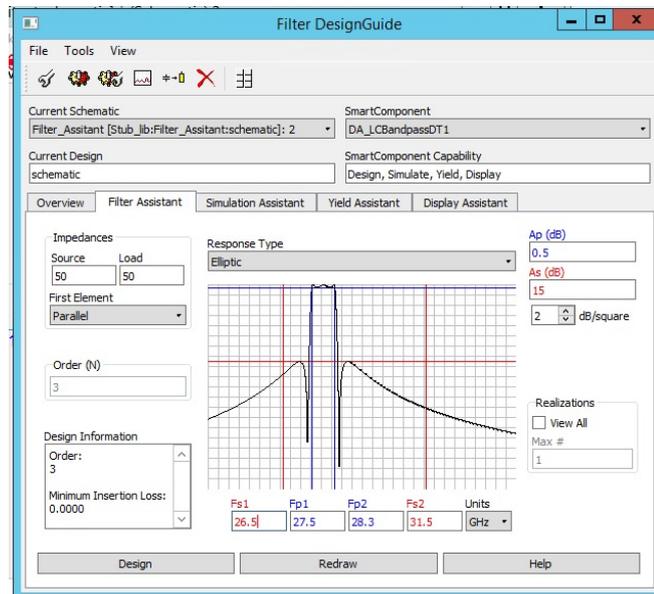
Figure 1.18: Filter from DesignGuide.
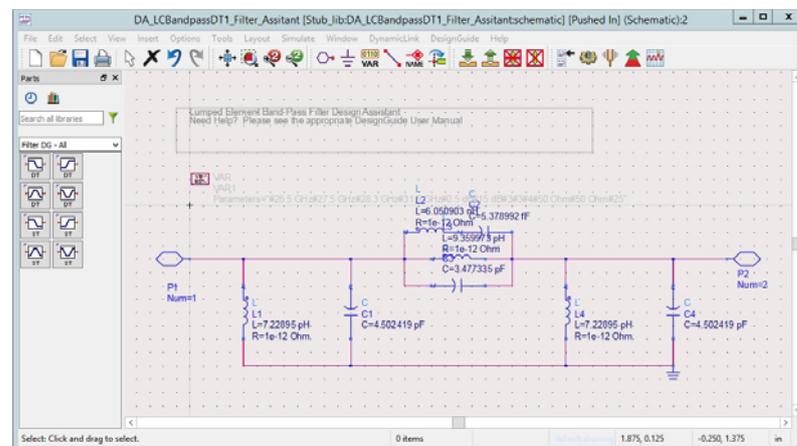
Figure 1.19: Filter from DesignGuide.

4. Draw the result using Lumped Components.

5. Use Transformation Assistant to convert to Distribuited Elements.

LineCalc tool from ADS We have used two software mainly. They are called HFSS[5] and ADS[6]. The use of one or another depends on what you want to achieve. HFSS is more accurate in the results. ADS gives you quicker understanding. One

_____

[5]HFSS stands for High Frequency Structure Simulator. Official Website
[6]ADS stands for Advanced Design System, from Keysight. Official Website
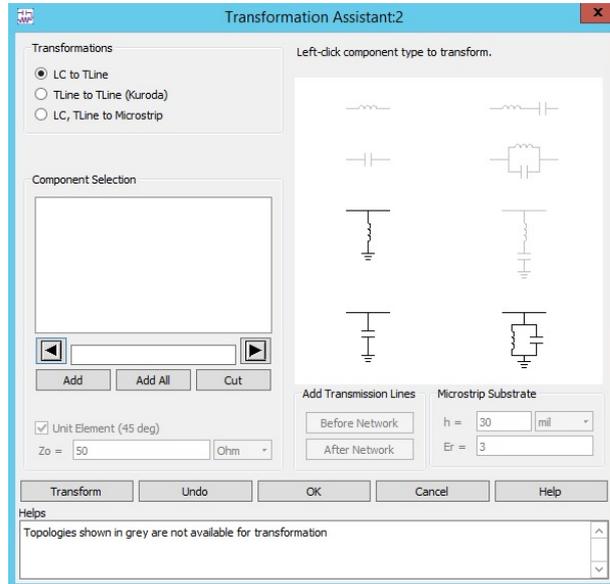
Figure 1.20: Filter from DesignGuide.

of the tools on ADS is called LineCalc. As you can see in the figure 1.21, the
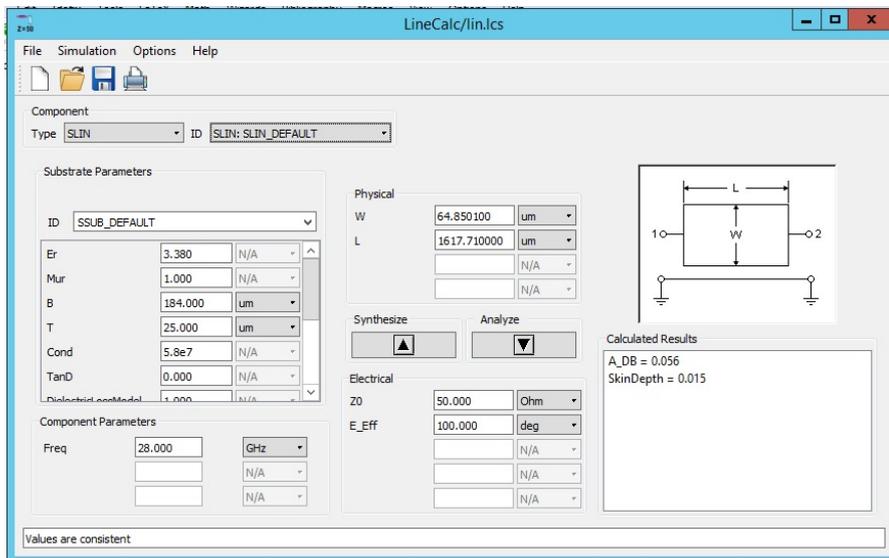
Figure 1.21: LineCalc tool.

tool is divided into different blocks. Top right corner you can find "Component".
Here you pick up the transmission line structure you want to. In the figure 1.21 is
selected "SLIN"A.3, which is referred to Stripline. Then you can find on the centre-
right side "Substrate Parameters". There you can define your substrate. In our

case we have worked mostly with $\epsilon = 3.38$ but eventually we found papers where they work with a substrate with dielectric constant of $\epsilon = 10.8$. You can define as many parameters as you know, i.e. tangent loss, thickness of the substrate, thickness of the conductor plane, etcetera. Lower right side you find "Component Parameters". In our case you can define here the operating frequency. Then you have two options. Either you Synthesize from a "Electrical" parameters, or analyse from a "Physical" parameters.

1.4 Smith Chart

The Smith Chart is a tool created by Phillip H. Smith(1905-1987) to assist in solving problems with transmission lines and matching impedance. The chart has a circular shape. The line crossing the chart represents pure real impedance. Along that line you can find any impedance. On the left side you find 0, and on the right side ∞. When you start working with the Smith Chart you need to normalize the impedance with the impedance of the line or the impedance of the impedance to be matched. Taking an example, using 50Ω transmission line, then you divide by 50Ω and the result is 1Ω. The upper half of the circumference represents positive imaginary impedance. It is equivalent to say that your design at that point has a inductive behaviour. The lower half represents the negative impedance, capacitive behaviour. On our design, we always assume 50Ω on the input and on the output of the filter. So what we would like to see is that we have a resonance at the operation frequency. Our bandwidth is defined on the return losses. Whenever the parameter s_{11} is lower of $15dB$ for example, all along those frequencies.

Chapter 2

Uniform Impedance Resonator

Hay días en que la recuerdo y me pregunto: ¿Qué estará haciendo?
Hay noches en que la extraño y me pregunto: ¿Qué me estoy haciendo?
Travesuras de la niña mala, Mario Vargas Llosa

Along this chapter the reader will discover about the Uniform Impedance Resonators. The difference between these resonators and the Stepped Impedance Resonators is that the spurious harmonics are shifted. The reader might remember that one of the requirements was the second harmonic to be attenuated by 23 dB.

In this chapter the reader will find how to design a coupled line filter and the results regarding a filter at our frequency of interest. When we extend the resonators on the coupled line filter we found out that we can "bend" the resonators without loosing performance. By bending the resonators we also get improvement in terms of surface, so the filter becomes smaller. The shape it takes remind you a hairpin and that is the name it takes in the literature. It is also mentioned about the interdigital and the combline filter. The configuration of these filters make them smaller but the resonators are connected to ground in one of the sides so from manufacturing point of view that might give some problems. Finally based on the research we arrive to the opened square shape resonators. This final configuration meets the requirements provided.

2.1 Coupled Line Filters

When two transmission lines are closed then energy can be coupled from one line to another.
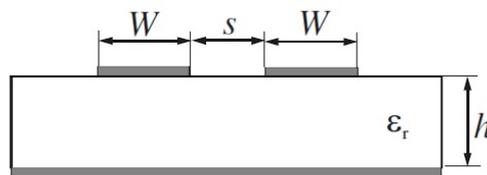
Figure 2.1: Cross section of coupled microstrip lines.

2.1.1 Even- and Odd-Mode Capacitances

The following text is the definition of even- and odd-modes. "Even and odd modes are the two main modes of propagation of the signal through a coupled transmission line pair. Odd mode impedance is defined as impedance of a single transmission line when the two lines in a pair are driven differentially (with signals of the same amplitude and opposite polarity). Even mode impedance is defined as impedance of a single transmission line when the two lines in a pair are driven with a common mode signal (the same amplitude and the same polarity)."[1]
Even-mode capacitance:

$$C_e = C_p + C_f + C'_f \tag{2.1}$$

Odd-mode capacitance:

$$C_o = C_p + C_f + C_{gd} + C_{ga} \tag{2.2}$$

C_p denotes the parallel plate capacitance between the strip and the ground plane:

$$C_p = \varepsilon_0 \varepsilon_r \frac{W}{h} \tag{2.3}$$

C_f is the fringe capacitance as if for an uncoupled single microstrip line. The term C'_f accounts for the modification of fringe capacitance C_f of a single line due the presence of another line. For the odd-mode, C_{ga} and C_{gd} represent, respectively, the fringe capacitances for the air and dielectric regions across the coupling gap.
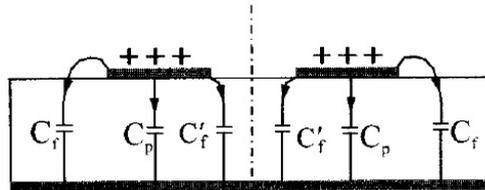
Figure 2.2: Quasi-TEM modes of a pair of coupled microstrip lines: even mode.

The equations:

$$Z_{0e} = Z_0[1 + JZ_0 + (JZ_0)^2] \tag{2.4a}$$

$$Z_{0o} = Z_0[1 - JZ_0 + (JZ_0)^2] \tag{2.4b}$$

where J is the admittance inverter.

$$Z_0 J_1 = \sqrt{\frac{\pi \Delta}{2g_1}} \tag{2.5a}$$

---

[1]Definition taken from https://www.tek.com/support/faqs/what-are-even-and-odd-modes-and-impedances-how-they-related-differential-and-common-mod
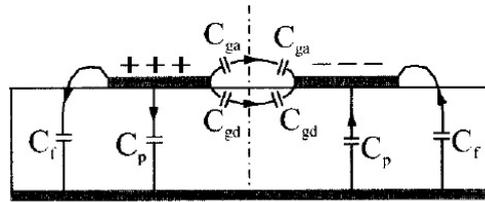
Figure 2.3: Quasi-TEM modes of a pair of coupled microstrip lines: odd mode.

$$Z_0 J_n = \frac{\pi \Delta}{2\sqrt{g_{n-1}g_n}} \quad \texttt{for n = 2, 3, ..., N} \tag{2.5b}$$

$$Z_0 J_{N+1} = \sqrt{\frac{\pi \Delta}{2 g_N g_{N+1}}} \tag{2.5c}$$

Once you found the J_n, you can use the equations 2.4 to calculate the even- and odd- mode.

Bandpass filter coupled edge at 28GHz with a bandwidth of 3GHz.
Using the previous equations we have calculated

Stage	Normalized element, g_k	Admittance inverter, $J_k Z_0$	Even mode impedance, Z_{0e}	Odd mode impedance, Z_{0o}
1	1.5963	0.313691	70.60469	39.23554
2	1.0967	0.118719	56.64063	44.76878
3	1.5963	0.118719	56.64063	44.76878
4	1	0.313691	70.60469	39.23554

The following figures will display the work flow to design the coupled edge filter using ADS. We can see on 2.4 the schematic design using ideal components. The
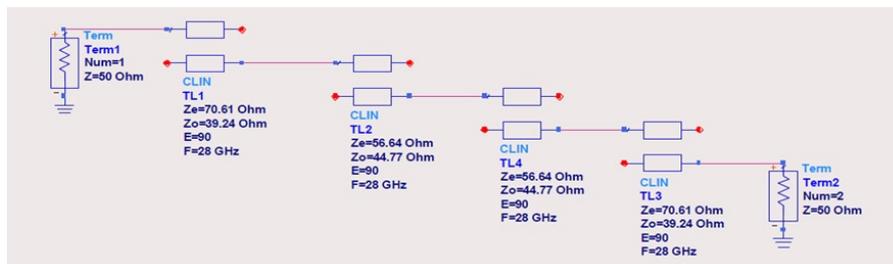
Figure 2.4: Ideal Coupled Transmission Lines at $28GHz$ and $3GHz$.

next two figures, 2.5 and 2.6, show how you use LineCalc tool to get the physical dimensions. Once you have the physical dimensions The following figure 2.7, is the result using ADS. The next figure 2.8 is the results using HFSS.
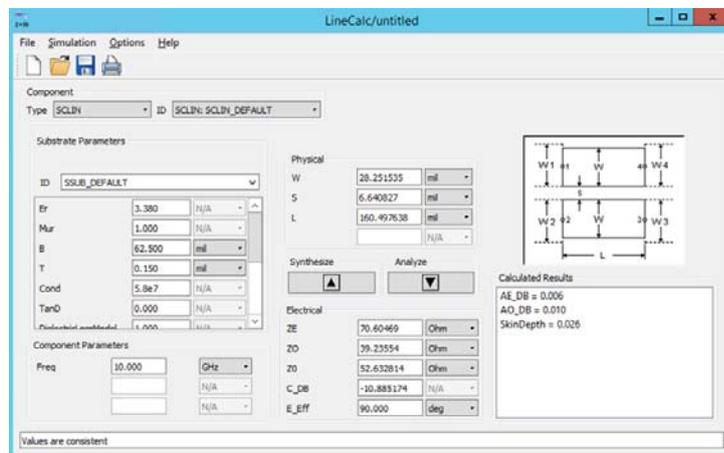
Figure 2.5: LineCalc tool to get the physical dimmensions of the transmission lines and distances for a stripline.
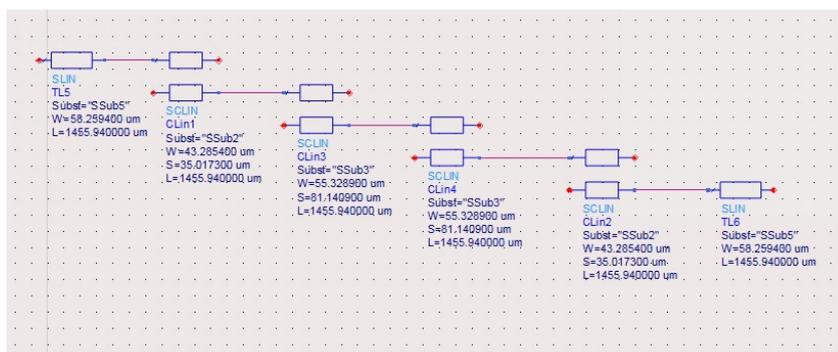
Figure 2.6: Coupled Transmission Lines at $28GHz$ and $3GHz$ using the given numbers from LineCalc.

2.2 Hairpin filter

Once we had understood how the Coupled Edge Filter configuration works, we realized that one of the main issues with that filter is the size. The Hairpin configuration could be the solution. The idea behind that is that you extend the resonators and then you bend them.

External quality factor. The band pass parameters can be calculated by

$$Q_{e_1} = \frac{g_0 g_1}{FBW} \tag{2.6}$$

$$Q_{e_n} = \frac{g_n g_{n+1}}{FBW} \tag{2.7}$$

The spacing between the resonators can be calculated by using the mutual coupling coefficient(M). Mutual coupling coefficients between adjacent resonators is
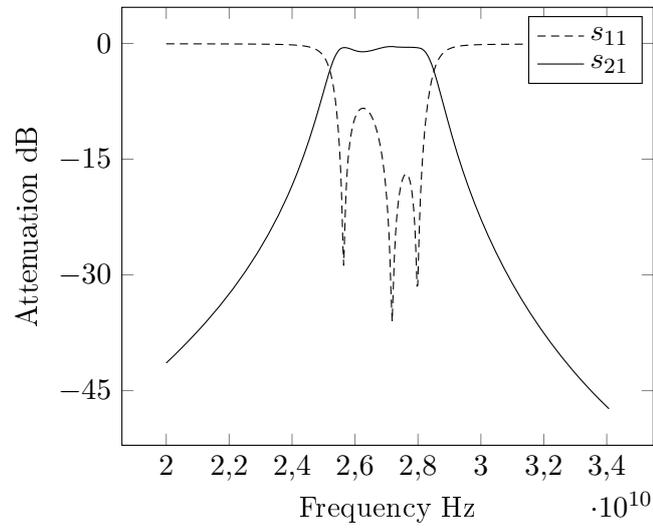
Figure 2.7: Results of the implemented filter from ADS.
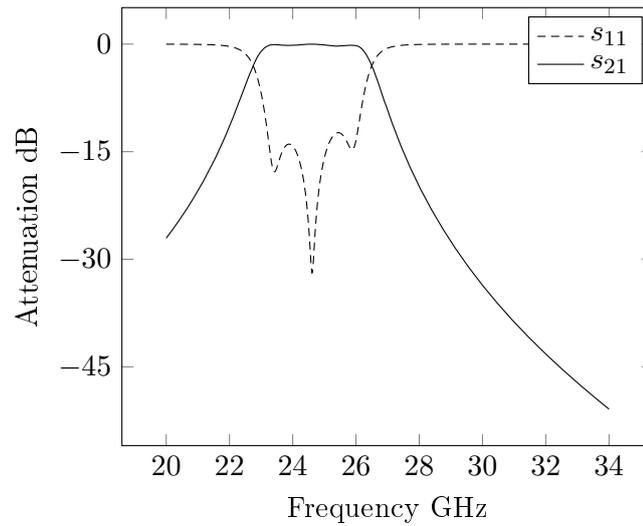
Figure 2.8: Results of the implemented filter from HFSS.

calculated by

$$M_{i,i+1} = \frac{FBW}{\sqrt{g_i g_{i+1}}} \quad \text{for i = 1 to n-1} \tag{2.8}$$

The equation that describes the tap is given

$$t = \frac{2L}{n} sin^{-1}\left(\sqrt{\frac{n}{2}\frac{Z_0}{Z_r Q_e}}\right) \tag{2.9}$$

in which Z_r is the characteristic impedance of the hairpin line, Z_0 is the terminating impedance, and L is about $\lambda_{g_0}/4$ long.
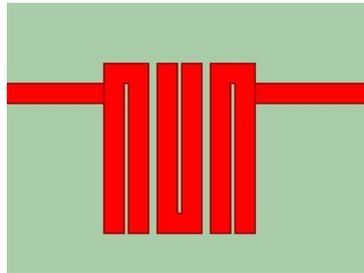
Figure 2.9: Typical configuration of hairpin.

The results after some tuning with ADS.
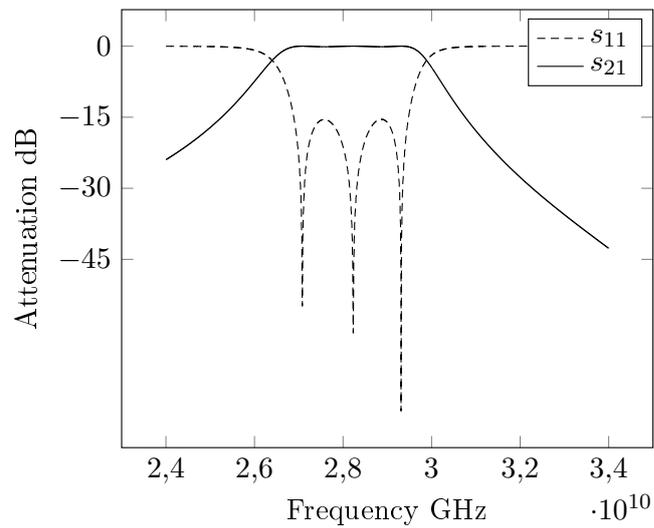
Figure 2.10: Results for three resonators tapped hairpin from ADS.

2.3 Interdigital & Combline filter

The theory of interdigital filter was developed by Gearge L. Matthaei [5]. Typically this filter is implemented on microstrip technology, but as any other filter used along the Thesis, this could be portable to stripline. Each resonator is shorted in one side and opened in the other side. The coupling is between two adjacent resonators spacing $s_{i,i+1}$. The tap can match the source/load characteristic impedance. The combline filter was introduced by Matthaei in 1963
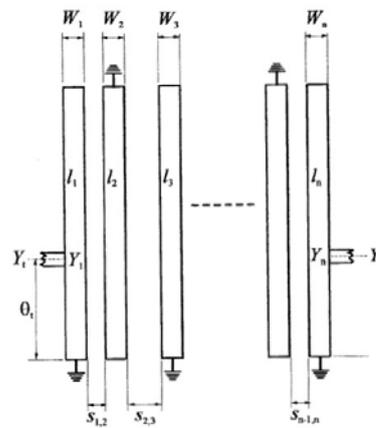
Figure 2.11: General configuration of interdigital.
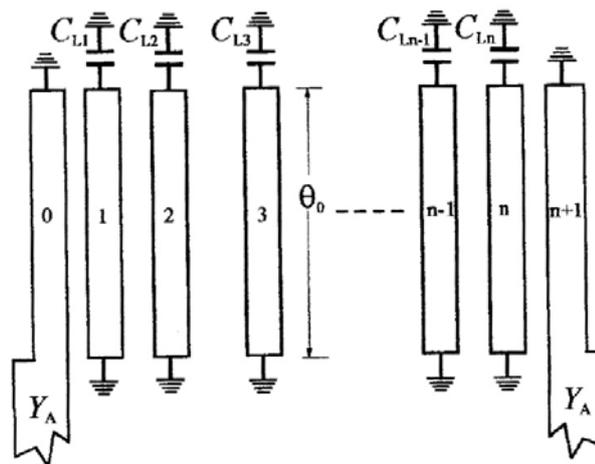
Figure 2.12: General configuration of combline bandpass filter.

2.4   Resonators

After some research we came across a paper titled "Couplings of Microstrip Square Open-Loop Resonators for Cross-Coupled Planar Microwave Filters" [8] and the chapter 8 and 10 or the book "Microstrip filters for RF"[2]. The design parameters of the bandpass filter, i.e., the coupling coefficients and external quality factors, can be determine as follow

$$
\begin{aligned}
Q_{ei} = Q_{eo} &= \frac{g_1}{FBW} \\
M_{i,i+1} = M_{n-1,n-i+1} &= \frac{FBW}{\sqrt{g_i g_{i+1}}} \quad \text{for } i = 1 \text{ to } m - 1 \\
M_{m,m+1} &= \frac{FBW \cdot J_m}{g_m} \\
M_{m-1,m+2} &= \frac{FBW \cdot J_{m-1}}{g_{m-1}}
\end{aligned}
\tag{2.10}
$$

The frequency response can be calculated by

$$
\begin{aligned}
s_{21}(\Omega) &= \frac{Y_o(\Omega) - Y_e(\Omega)}{(1 + Y_e(\Omega)) \cdot (1 + Y_o(\Omega))} \\
s_{11}(\Omega) &= \frac{1 - Y_e(\Omega) Y_o(\Omega)}{(1 + Y_e(\Omega)) \cdot (1 + Y_o(\Omega))}
\end{aligned}
\tag{2.11}
$$

Quick reminder that

$$
\Omega = \frac{1}{FBW} \cdot \left(\frac{\omega}{\omega_0} - \frac{\omega_0}{\omega} \right)
\tag{2.12}
$$

These equations are valid for any type of filter based on coupling resonators.

Chapter 3

Stepped Impedance Resonator

Det finns mitt i skogen en oväntad glänta som bara
kan hittas av den som gått vilse.
Tomas Tranströmer

A stepped impedance resonator (SIR) consists of two or more sections of transmission line, 3.1 shows the basic structures of the two-section stepped impedance resonator and tri-section stepped impedance resonator with different characteristics impedance values. The physical length of these resonators is shorter than the
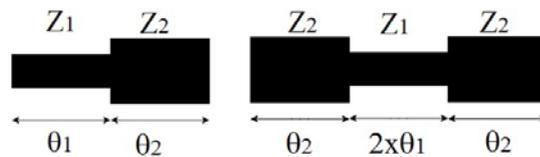
Figure 3.1: Different types of step impedance resonators: (a) two section SIR (b) tri section SIR.

half-wavelength resonator, but they can still resonate at the same frequency as the half-wavelength resonator. Stepped impedance resonators with correct combination, can be designed to reduce the length and push the spurious response of the filter away from the centre frequency of the passband.

The SIR resonators have also good advantages due to the compact size and independent control of the characteristic impedance of the resonator during the design procedure.

Tri section Stepped impedance resonator is a TEM or quasi-TEM mode resonator composed of three transmission lines of different impedances Z_i and electrical lengths θ_i.

The resonator structure to be considered in this study is a non-uniform transmission line resonator with the stripline construction as shown in 3.2. This configuration of SIR is symmetrical and has two different characteristic impedance lines, Z_1 and Z_2, which related to different electrical lengths θ_1, θ_2.

The input impedance and admittance are defined as Z_{in} and Y_{in} ($=1/Z_{in}$), respectively. The admittance of the resonator from the open end can be expressed
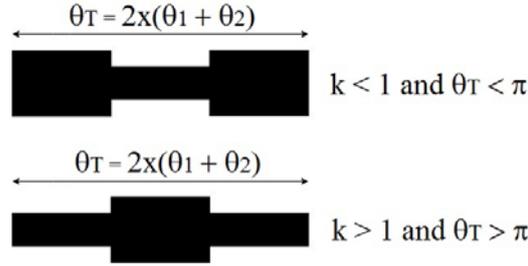
Figure 3.2: (a) Tri section SIR: (a) k<1 (b) k>1.

as [12].

$$Y_i = jY_2 \cdot \frac{2(k \cdot tan\theta_1 + tan\theta_2) \cdot (k - tan\theta_1 \cdot tan\theta_2)}{k(1 - tan^2\theta_1) \cdot (1 - tan^2\theta_2) - 2(1 + k^2) \cdot tan\theta_1 tan\theta_2} \qquad (3.1)$$

Therefore, the fundamental resonance condition, when $Y_{in} = 0$, can be obtained as follows:

$$tan\theta_1 tan\theta_2 = k \qquad (3.2)$$

Where $k = Z2/Z1$ is the impedance ratio, the overall electrical length of the SIR, represented by θ_T, is expressed as:

$$\theta_T = 2 \cdot (\theta_1 + \theta_2) \qquad (3.3)$$

From 3.2 and 3.3

$$tan\frac{\theta_T}{2} = \frac{1}{1-k} \cdot \left(\frac{k}{tan\theta_1} + tan\theta_1 \right) \quad \text{when k} \neq 1 \qquad (3.4)$$

When $k = 1$ that represents equal impedance for each section.
The relationship between θ_T and θ_1 , taking k as parameter is shown in 3.3 It can be seen from 3.3 that the resonator length θ_T has a maximum value when $k \geq 1$ and a minimum value when $k < 1$ (the peaks in the curves). In other words, It can be concluded that for a smallest size (shortest length) stepped impedance resonator, the structure in 3.2-a can be selected where $k < 1$ and $\theta_T < \pi$.
Furthermore, when $\theta_1 = \theta_2$ it could be considered as another special condition for a smallest size (shortest length) SIR [13] , 3.3 shows the longest or shortest resonator length as the fundamental mode frequency is fixed for different values of k. The resonance condition in this case is given by:

$$\theta_1 = tan^{-1}(\sqrt{k}) = \theta_2 \qquad (3.5)$$

The total length of the resonator:

$$\theta_T = 2 \cdot (\theta_1 + \theta_2) = 4 \cdot \theta_1 \qquad (3.6)$$

$$\theta_T = 4 \cdot tan^{-1}(\sqrt{k}) \qquad (3.7)$$
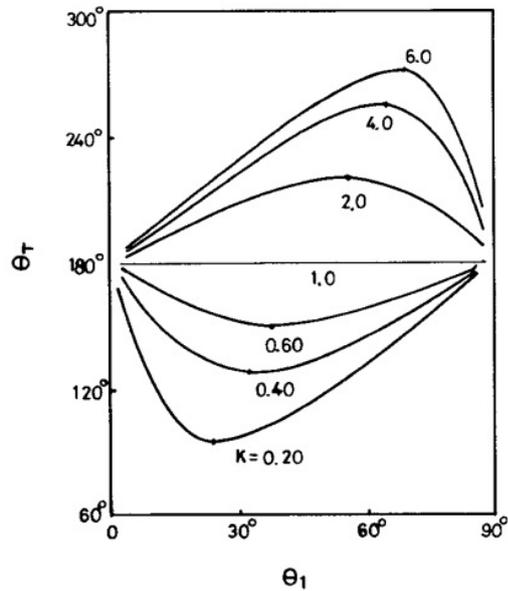
Figure 3.3: Resonance condition of the SIR[12].
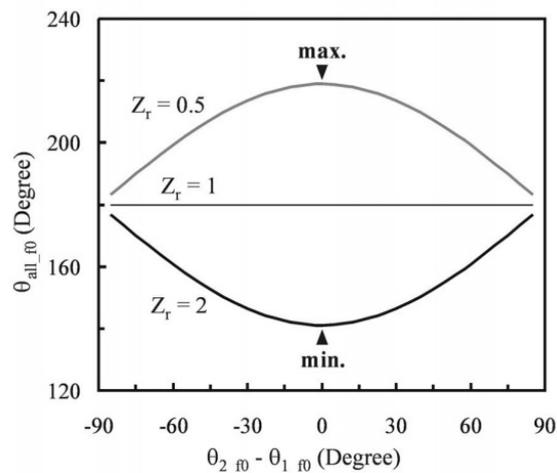
Figure 3.4: Characteristic of the fundamental mode of a stepped impedance resonator with different k[13].

Note that θ_T is the electrical length of SIR, in order to find the physical length, the following equation can be applied:

$$\theta_T = \beta \cdot L = \frac{2\pi}{\lambda} \cdot L = \frac{2\pi f_0}{c} \cdot L \cdot \sqrt{\epsilon} \tag{3.8}$$

The length and resonant frequencies are the main considerations while designing a stepped impedance resonator, for f_0 denoted as the fundamental resonance frequency and f_{s1} , f_{s2} and f_{s3} as the first, second and third spurious resonance frequency respectively, according to [12]:

$$\frac{f_{s1}}{f_0} = \frac{\theta_{s1}}{\theta_0} = \frac{\pi}{2tan^{-1}\sqrt{k}}$$
$$\frac{f_{s1}}{f_0} = \frac{\theta_{s1}}{\theta_0} = 2\left(\frac{f_{s1}}{f_0}\right) - 1 \qquad\qquad (3.9)$$
$$\frac{f_{s3}}{f_0} = \frac{\theta_{s3}}{\theta_0} = 2\left(\frac{f_{s1}}{f_0}\right)$$

From the above equations It can be deduced that the resonator length and corresponding spurious resonance frequencies can be controlled by the impedance ratio k which is illustrated in figure 3.5. In the optimum solution investigation done
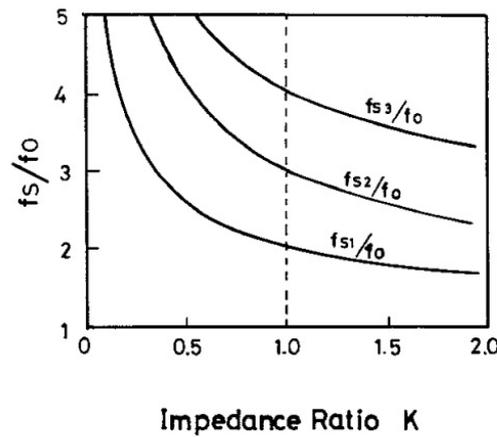
Figure 3.5: Spurious resonance frequency of the SIR[13].

by [13] it is found that the behaviour of the first spurious mode can be totally contrary for the different relation between θ_1 and θ_2 even if the impedance ratio k is fixed, figure 3.6 shows that SIR could have a maximum or minimum spurious when $\theta_2 = 2\theta_1$. Where $n = \frac{\theta_2}{\theta_1}$
From the above figure, the structure in 3.2-b should be avoided where $k > 1$, because the spurious response occurs below twice frequency of the fundamental mode.
Therefore, a better design is to set k to be less than one and as small as possible, and make the trade-off between the length of the resonator and the shifting of the first spurious resonance.
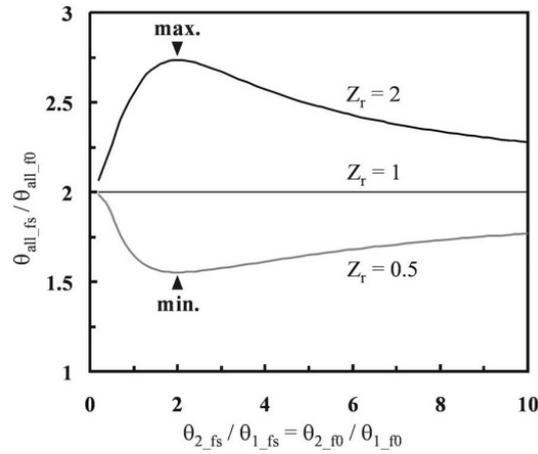
Figure 3.6: Characteristic of θ_1 respect to n and different Z[13].

3.1 Parallel Coupled Filter with SIR Section

In Chapter 2, the synthesis method for designing a band pass filter based on parallel coupled transmission line with quarter-wavelength was described, but the previous synthesis is only valid for quarter-wavelength coupling, considering to $\theta = \pi/2$.

Since, the length θ of step impedance resonator could be designed with different values, where it can set to be $\theta > \pi/2$ or $\theta < \pi/2$, that requires to find a general solution for parallel coupled lines with arbitrary length. The synthesis in [12], has expressed the relationship between even and odd mode impedance and the admittance inverter parameters.

Design Equations for Parallel Coupled SIR Filter This study will follow the same synthesis steps in [12] to design parallel coupled band pass filter with SIR Section:

- Choose value for Z_1 and Z_2 then calculate the impedance ratio $k = Z_2/Z_1$. The values will be limited by the physical dimensions of the stripline. So for instance the highest impedance will be limited by the minimum width of the line.

- Set value for θ_1.
  For $\underline{\theta_1 = \theta_2}$, the length of each section can be determined from $\theta_1 = tan^{-1}(\sqrt{k}) = \theta_2$. The total length

$$\theta_T = 2 \cdot (\theta_1 + \theta_2) = 4 \cdot \theta_1.$$

For $\underline{\theta_1 \neq \theta_2}$, the total length θ_T can be obtained from $tan\frac{\theta_T}{2} = \frac{1}{1-k} \cdot \left(\frac{k}{tan\theta_1} + tan\theta_1 \right)$. The length of θ_2 can be calculated from $\theta_2 = \left(\frac{\theta_T}{2} - \theta_1 \right)$.
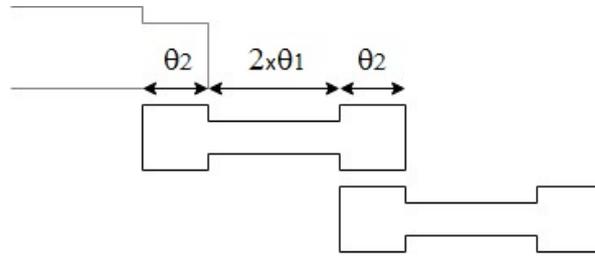
Figure 3.7: Coupled Edge Stepped Impedance filter structure.

- Calculate the slop parameter $b = 2 \cdot \theta \cdot Y_2$ where $\theta = \theta_2$ representing the coupling length.

- Determine the admittance inverter for each section from:

$$J_{01} = \sqrt{\frac{Y_0 b_1 \omega}{g_0 g_1}} = Y_0 \sqrt{\frac{2\omega\theta_0}{g_0 g_1}}$$

$$J_{j,j+1} = \omega\sqrt{\frac{b_j b_{j+1}}{g_j g_{j+1}}} = Y_0 \frac{2\omega\theta_0}{\sqrt{g_j g_{j+1}}} \quad \texttt{j = 1}\sim\texttt{n-1} \qquad (3.10)$$

$$J_{n,n+1} = \sqrt{\frac{Y_0 b_1 \omega}{g_n g_{n+1}}} = Y_0 \sqrt{\frac{2\omega\theta_0}{g_n g_{n+1}}}$$

where ω is the frictional bandwidth and g is the element of low pass filter prototype.

- Finally, the expressions even and odd mode impedance for parallel coupled lines with arbitrary length can be calculated as following:

$$\frac{Z_{oe}}{Z_0} = \frac{1 + (J/Y_0)cosec\theta + (J/Y_0)^2}{1 - (J/Y_0)^2 cot^2\theta}$$

$$\frac{Z_{oo}}{Z_0} = \frac{1 - (J/Y_0)cosec\theta + (J/Y_0)^2}{1 - (J/Y_0)^2 cot^2\theta} \qquad (3.11)$$

Design of 3rd order Parallel Coupled SIR band pass Filter

By considering the same specifications for the desired filter listed in the abstract, where the order N = 3 gives 4 coupled sections, central frequency $f_0 = 28$ GHz, ripple = 0.1 dB and fractional bandwidth = 3%. The same stack up configuration considered in Chapter 2 has been used. The highest of the substrate was chosen to be 179 μm and 383 μm for two design examples of parallel coupled SIR band pass filter, the design follows the previous section procedures. For substrate highest 179 μm, it was noticed that for resonator impedance $Z_1 = 45\Omega$, Z2$= 25\Omega$ and for the characteristic impedance of the system $Z_0 = 50\Omega$, the physical dimensions set below of the limit of our fabrication process and design rules where $w_{1,4} = 41.7$ μm and $S_{1,4} = 24.44$ μm, the width and spacing of the first and last coupling section, respectively.

One of the possible solution is to change the characteristic impedance of the system Z_0 to be:

$$Z_0 = \sqrt{Z_1 Z_2} \approx 35\Omega$$

That improves the width to $w_{1,4} = 101.59$ μm, but on the other hand, the spacing distance will decrease to $S_{1,4} = 17.4$ μm and that contradicts with design rules. Another solution is to set substrate highest to be B= 383 μm. It is possible to control the highest of the substrate by using five layers instead of three. The characteristic impedance and the corresponding TL width will change for B = 383 μm, the table 3.1 gives the width and Z_0 for different substrate heights.

Z (ohm)	W μm
50	62.17 μm
45	79.28 μm
40	100.82 μm
35	128.6 μm
30	165.81 μm
25	217.94 μm
20	296.3 μm
15	427.6 μm

All calculated design parameters are listed below.

Line Impedance:

$Z_1 = 70$ Ω

$Z_0 = 50$ Ω

$Z_2 = 45$ Ω

Impedance Ratio:

k = 0.642

Resonator Length:

$\theta_1 = 38.72°$

$\theta_2 = 38.72°$

$\theta_T = 154.88°$

Stage	Normalized element, g_k	Admittance inverter, $J_k Z_0$	Even mode impedance, Z_{0e}	Odd mode impedance, Z_{0o}
1	1.5963	0.003275	67.23392	39.91696
2	1.0967	0.000582	52.43864	47.77826
3	1.5963	0.000582	52.43864	47.77826
4	1	0.003275	67.23392	39.91696

Using LineCalc of Agilent's Advanced Design System (ADS), the even and odd mode impedance of transmission lines are converted to physical dimensions as listed in Table 3.1.

Stage	Width, $W(\mu m)$	Length, $L(\mu m)$	Spacing, $S(\mu m)$
Input	174.91	100	-
1st coupling section	143.75	626.379	**64.65**
1st uncoupling section	75.23	1252.76	-
2nd coupling section	173.2	626.379	256
2nd uncoupling section	75.23	1252.76	-
3rd coupling section	173.2	626.379	256
3rd uncoupling section	75.23	1252.76	-
4th coupling section	143.75	626.379	**64.65**
Output	174.91	100	-

Simulation results The first simulation results running by momentum ADS shown in the 3.8. The centre frequency of the passband is offset by 400 MHz, which
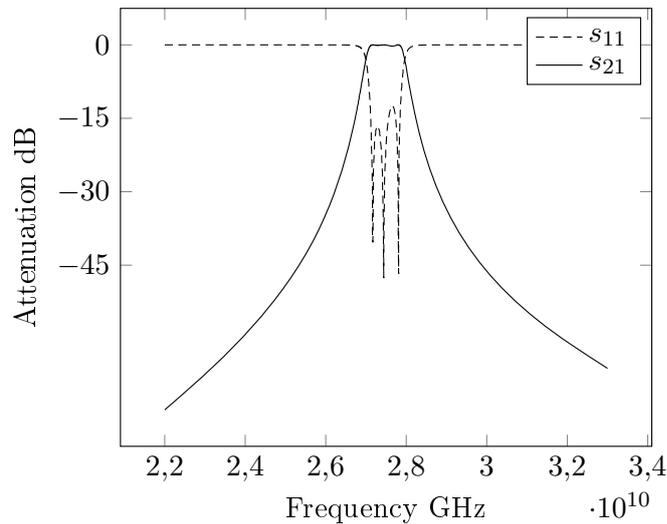
Figure 3.8: Simulation results running by momentum ADS.

could be compensated by shortening the physical lengths of the transmission-line

resonators. It could be concluded that the results obtained from the theoretical calculation are very close to the simulated results. The theoretical equations provide a good estimation for the resonant frequencies and the bandwidth.

The result from fine response of the filter shown in 3.9. The result from broadband
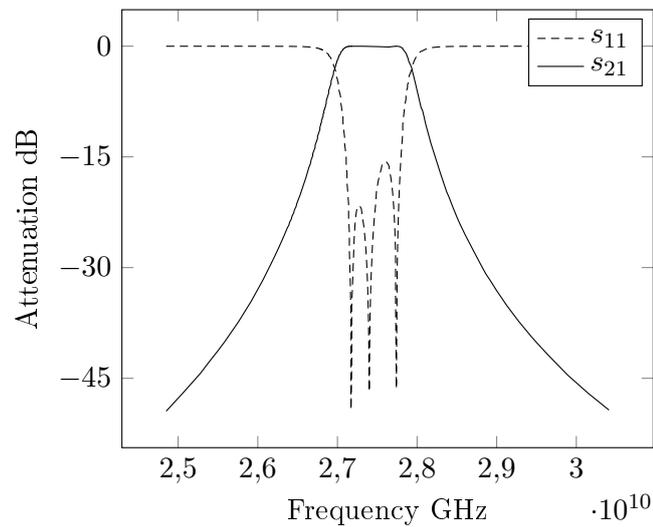
Figure 3.9: Simulation results after fine tuning.

response of the filter shown in 3.10. The filter has the first spurious response at about 62.5 GHz, which is less than expected. Lastly, all the dimensions obtained
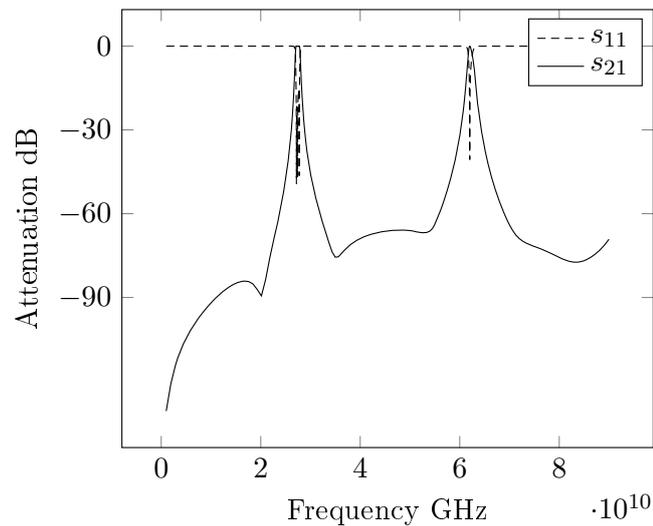
Figure 3.10: Simulation results for broadband response of the filter.

from final tuning are realized in the stripline filter and the layout of the third-order
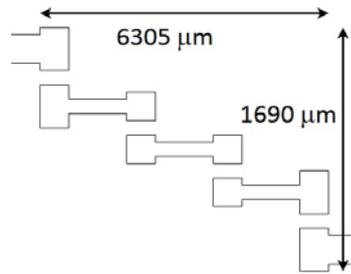
Figure 3.11: The layout 3rd order Parallel Coupled SIR band pass Filter.

bandpass filter with (6.305 mm X 1.69 mm) is shown in 3.11. However, the filter meets the desired specifications for insertion loss and return loss, but it contradicts with the design rule where the spacing of first and last coupling section $S_{1,4} = 64.65$ μm, and the filter size does not meet (5mm X 5 mm).

3.2   U shape Stepped Impedance Resonator

The size problem in the previous parallel coupled SIR filter can be compensated by bend the filter into U shape so that space is saved by folding resonator, this idea is similar to the conventional uniform impedance hairpin filter [14] that discussed in chapter 2.

It is important to realize the reduction of the coupled-line lengths, comparing with conventional UIR hairpin, which is expressed only between the lower impedance section this length L2 3.16 is less than $\lambda/2$, which reduces the coupling between resonators.
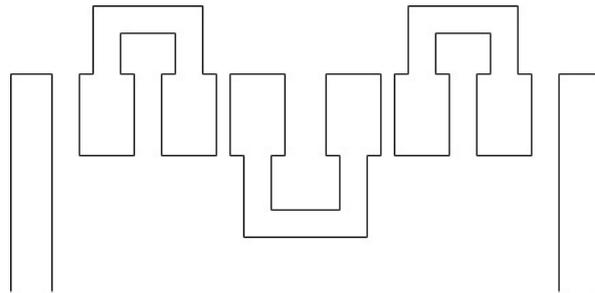
Figure 3.12: Stepped impedance hairpin resonators structure.

3.12 shows the structure of U shape SIR filter, the length L1 of the higher impedance section is bent to L1=2L1'+LG, where LG the length between two arms. The lower impedance section L2 is kept at the same length in order to apply the previous odd- and even-mode analysis. Hence, the input and output couple lines is kept the same. The size using hairpin structure is reduced to 2.89mm x 1.16 mm.

Simulation results The filter is simulated on the same substrate height B= 383 um as the parallel coupled SIR filter. 3.13 shows the simulated output frequency responses, the simulated results are very close to parallel coupled SIR filter in Figure. The result from broadband response of the filter shown in 3.14, the harmonic spurious response happens at 62.5 GHz.
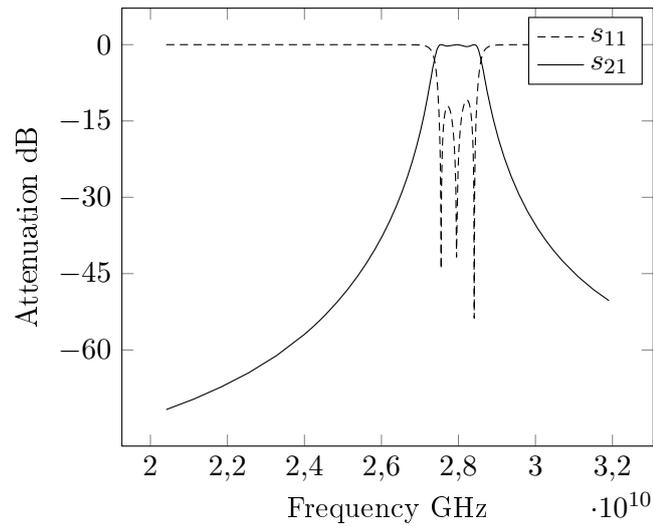
Figure 3.13: Stepped impedance hairpin resonators fine simulation.
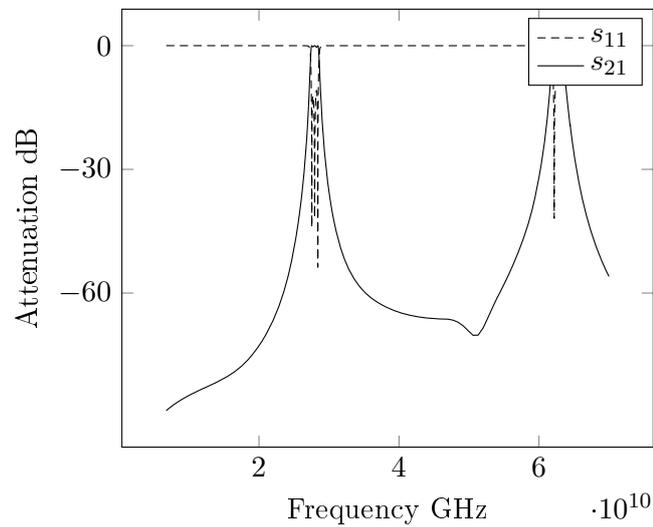
Figure 3.14: Stepped impedance hairpin resonators broad simulation.

It's worth to mention that by changing LG, the length between tow arms, the

filter will change. Comparisons of size changing of different values of LG have been summarized.

Line Impedance:

$$Z_1 = 70 \ \Omega$$
$$Z_0 = 50 \ \Omega$$
$$Z_2 = 45 \ \Omega$$

Impedance Ratio:

$$k = 0.642$$

Resonator Length:

$$\theta_1 = 38.72°$$
$$\theta_2 = 38.72°$$
$$\theta_T = 154.88°$$

It is also noted that the spurious response is shifted according to the value of LG as shown in 3.15, the response at 28 GHz.
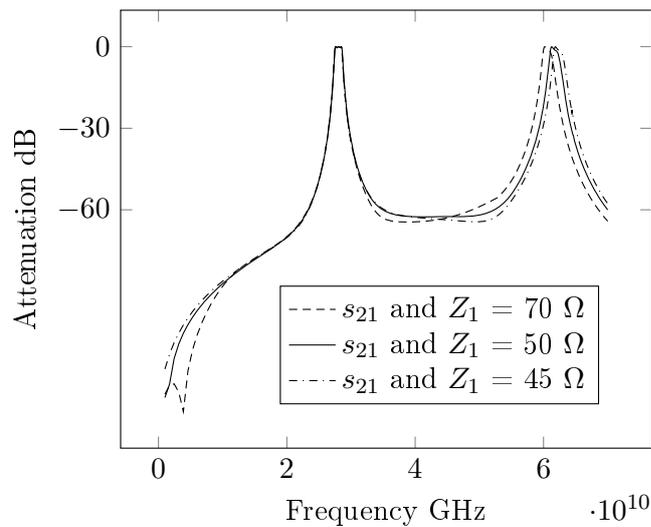
Figure 3.15: Stepped impedance hairpin resonators broad simulation for $Z_1 = 70 \ \Omega$, $Z_1 = 50 \ \Omega$ and $Z_1 = 45 \ \Omega$ using the same θ_1.

3.3   Hairpin SIR filter

Instead of finding the relationship between even and odd mode impedance and the admittance inverter parameters J to make the filter synthesis, an alternative

design method, proposed by Lancaster, [7], is to determine the physical dimensions in terms of coupling coefficients M and external quality Q factors. In the previous design the coupling between two arms of each resonator are neglected, but when two arms of each hairpin resonator are closely spaced, they will act as parallel coupled line and the coupling between them will have impact effect on the resonator response, this will require a different design structure of the step impedance hairpin resonator.

3.3.1 Equivalent circuit model of single hairpin SIR.

The basic structure of one stepped impedance hairpin resonator [15] is shown in 3.16, this structure consists of a single transmission line $L_s = 2 \cdot L_1$ with higher characteristic impedance Z_1 and two symmetric parallel coupled lines $L_C = L_2$ with lower characteristic impedance $Z_2 = \sqrt{Z_{2oe} \cdot Z_{2oo}}$, where Z_{2oe} and Z_{2oo} are the even- and odd-mode impedance of the parallel coupled lines.

As concluded from previous sections, for a smallest size stepped impedance resonator, the impedance ratio k should be less than one:

$$k = \frac{Z_2}{Z_1} = \frac{\sqrt{Z_{2oe} \cdot Z_{2oo}}}{Z_1} < 1$$

$$Z_1 > \sqrt{Z_{2oe} \cdot Z_{2oo}}$$
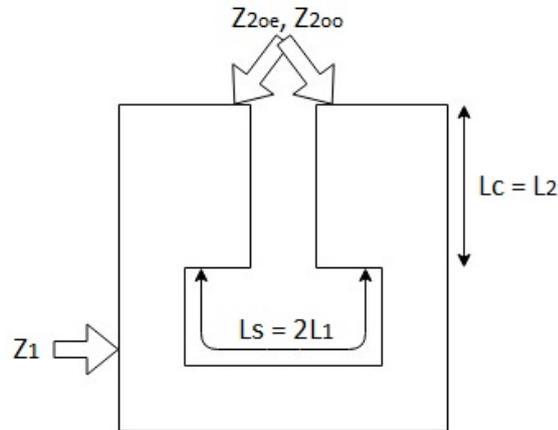
Figure 3.16: Stepped Impedance Resonator.

3.3.2 Design of single hairpin stepped impedance resonator.

A study done Lee and Tsai, ref. [10], for the miniaturized hairpin resonators has shown that the same equations for fundamental resonant and the first spurious resonant can also be applied to tri section step impedance resonators and step

impedance hairpin resonator shown in 3.17. therefore, the equations in section 3.2 can be applied for step impedance hairpin resonator.

Figure 3.17 shows the circuits and parameters of a typical stepped-impedance resonator and a stepped-impedance hairpin resonator.

In order to design a step impedance hairpin resonator as shown in fig, a full-wave EM simulation with the following selected circuit parameters are implemented:

- $Z_1 = 45\ \Omega$ leads to $w_1 = 79\ \mu m$.

- $Z_1 = \sqrt{Z_{2oe}Z_{2oo}} \approx 20\ \Omega$ leads $w_2 = 297\ \mu m$, for g $= 150\ \mu m$ $Z_{2oe} \approx 20.29$ Ω and $Z_{2oo} \approx 19.78\ \Omega$.

- $\theta_1 = \theta_2 = 33.6\circ$ gives $L_c = L_2 = 545\ \mu m$, $L_s = 2 \cdot L_1 = 1090\ \mu m^1$, $L_g = 744\ \mu m$ and $L_a = 130\ \mu m$.
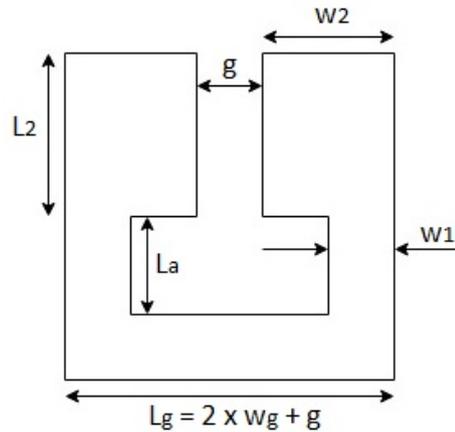
Figure 3.17: Stepped Impedance Resonator.

It is also noted that by changing the gap between two arms g, an improve of the stop-band response could be achieved. In order to explain that, three different structures of the previous resonator with g=100 μm , g=150 μm and g=210 μm were simulated as shown in figure 3.18. In order to meet the design rules , g and La should be greater than 100 μm.

From the simulation results illustrated in figure 3.19, it is observed that when g are closely spaced, a sharp cut-off frequency response happened. This can be explained from the equivalent network, the coupling level between symmetric coupled-line structure related to Z_e and Z_o, and this relationship can be controlled through the load capacitance C_g and parallel-plate capacitances C_p from equation 3.12, furthermore, the spacing g between symmetric coupled-line has direct impact on the value of C_g, In other words, changing g, will affect the load capacitance C_g,

---

[1]Its tunned from 1090 to 1010.
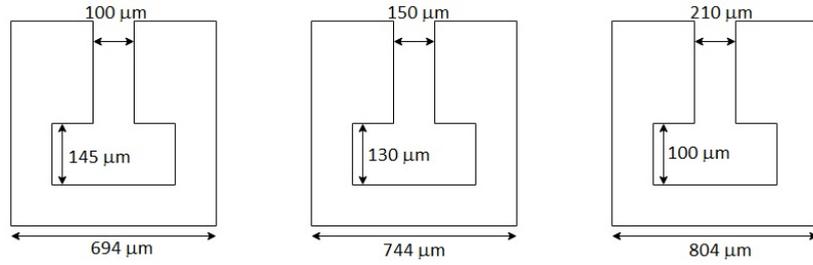
Figure 3.18: Three different stepped impedance to test the stop-band respones.
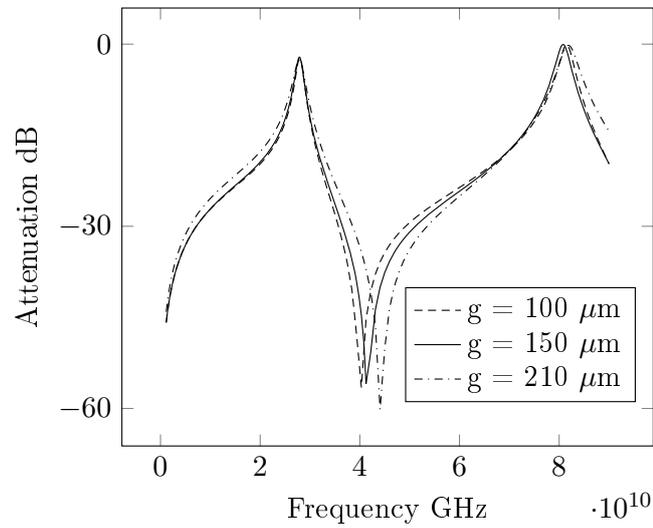
Figure 3.19: Comparison between the results.

and the even and odd mode impedance.

$$C_p = \frac{1}{2\omega Z_{oe} cot(\beta_c L_c)}$$
$$C_g = \frac{Z_{oe} - Z_{oo}}{2\omega Z_{oe} Z_{oo} cot(\beta_c L_c)} \tag{3.12}$$

From figure **??** the inductor L_s in parallel with the load capacitance C_g, can be considered as notch filter and by tuning C_g, the transmission zero produced by the notch will shift away from resonant frequency.

$$L_s = \frac{sin^{-1}(\omega_c L_{st}/Z_s)}{\beta_s}$$
$$L_c = \frac{tan^{-1}[\omega_c Z_{oc}(C_{pst} - C_s - C_\delta)]}{\beta_c} \tag{3.13}$$

The single transmission line: Figure 3.20 shows the equivalent- lumped π-network of single transmission line, by using ABCD matrix, the relationships between lumped and distributed elements can be expressed [15] and [16]:

$$C_s = \frac{1 - cos(\beta_s L_s)}{\omega Z_s sin(\beta_s L_s)}$$
$$L_{st} = \frac{Z_s sin(\beta_s L_s)}{\omega}$$

(3.14)

Where, ω is the angular frequency, L_{st}, and C_s are the equivalent inductance and capacitance of the single transmission line.
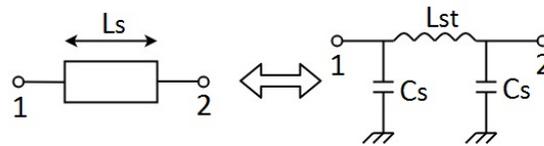
Figure 3.20: The equivalent- lumped π-network of single transmission line.

The symmetric parallel coupled lines: Figure 3.21 shows the equivalent-lumped π-network of symmetric parallel coupled lines, by using ABCD matrix, relationships between lumped and distributed elements can be expressed [15] and [16]:

$$C_g = \frac{Z_{oe} - Z_{oo}}{2\omega Z_{oe} Z_{oo} cot(\beta_c L_c)}$$
$$C_p = \frac{1}{2\omega Z_{oe} cot(\beta_c L_c)}$$

(3.15)
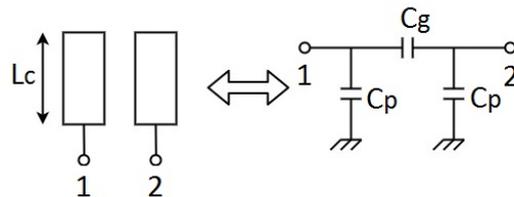
Figure 3.21: The equivalent- lumped π-network of symmetric parallel coupled lines.

Stepped impedance hairpin resonator: The final structure of the Stepped impedance hairpin resonator consists of the single transmission line and the coupled lines combined to each other, figure 3.22 shows the equivalent lumped π-network for this combination.
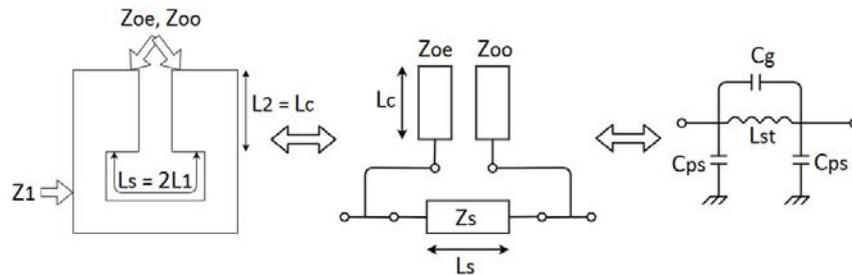
Figure 3.22: The equivalent lumped π-network for stepped impedance hairpin resonator.

Input/output Structure The proper way to design the input and output structure is by using tapped feed lines, two different tapped feed line structures were proposed by Lee and Tsai, ref. [10], in phase input and output feed structures as shown in the figure 3.23 and out phase input and output feed structures as shown in figure 3.25.
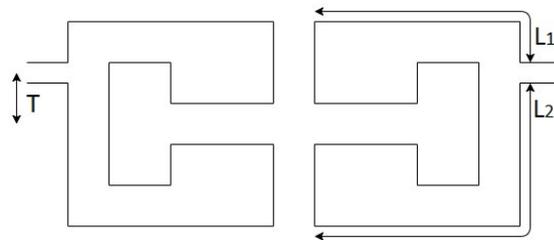
Figure 3.23: 2 stepped impedance resonators in phase.

It is observed by Lee and Tsai, ref. [10], that the in phase feed structure would introduce a pair of transmission zeros in stop band.
The first zero is created at the frequency when the length of arm (1) of the input resonator is about quarter-wavelength while the other occurs at the frequency when the length of arm (2) is about quarter wavelength. In order to prove this in our study, two step impedance hairpin resonators using different feed structure are designed and simulated. The simulation responses shown in Figure 2 agree with [10] observation.
It is also noted that the transmission zeros shift according to the tap position T as illustrated in figure 3.27 for T = 50 μm, 100 μm and 200 μm.
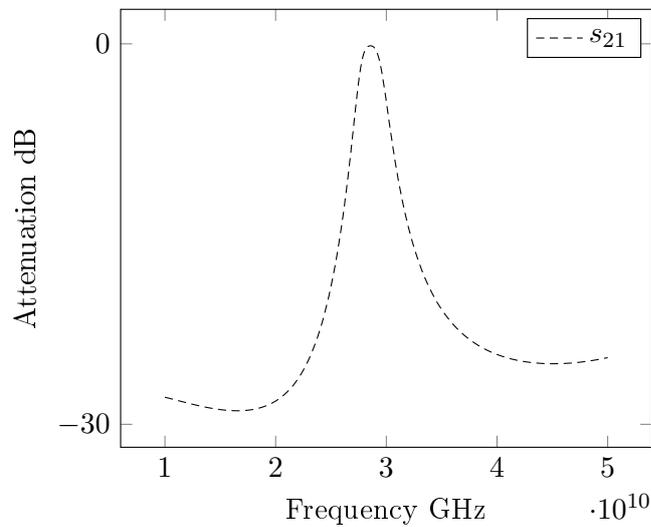
Figure 3.24: Transfer function for 2 resonators with symmetric tap.
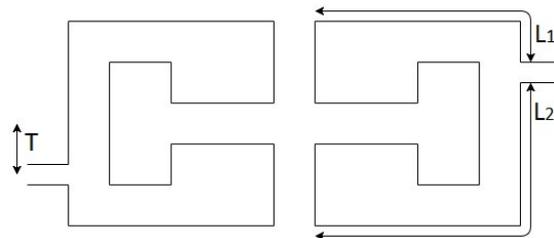
Figure 3.25: 2 stepped impedance resonators out phase.

Stepped impedance filter configuration. The conventional structures of the step impedance hairpin filters only provide Butterworth and Chebyshev characteristics. In order to get elliptic response, filters using cross-coupled are introduced by [10] and [7]. Using cross coupling between non-adjacent resonators will generate a pair of attenuation pole near the resonance frequency.

An elliptic function bandpass filter is constructed by using four step impedance hairpin resonators with input and output tapped feed structures as shown in figure 3.28. Three main coupling structures are used in the design

- Mixed coupling: Coupling structures between, the first and second resonator M12, and the third and fourth resonator M34, are based on mixed coupling.

- Magnetic coupling: Coupling structure between the second and the third resonator M23, is based on Magnetic coupling.

- Electric coupling: Coupling structure between the first and the fourth resonator M14, is based on Electric coupling.

In order to design the filter and calculate the coupling coefficient of the above
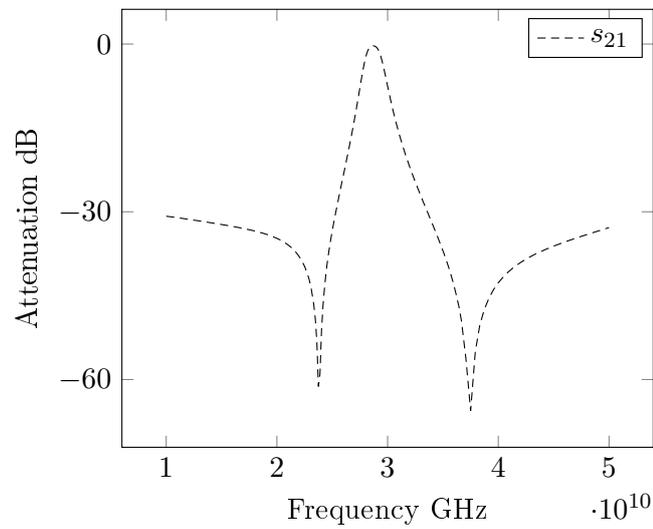
Figure 3.26: Transfer function for 2 resonators with non-symmetric tap.
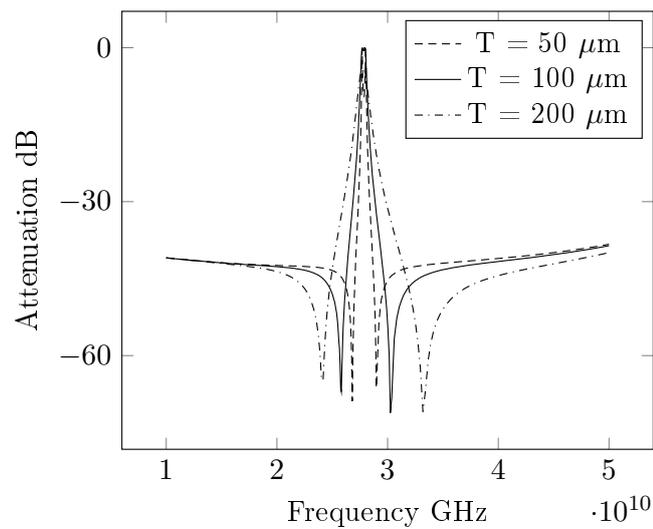
Figure 3.27: Comparison between the results shifting T.

main coupling strictures, full-wave simulator using momentum ADS and ANSOFT HFSS are implemented to design.

Design cross-coupled stepped impedance hairpin filter. Cross-coupled bandpass filter using stepped impedance hairpin resonators has been designed on substrate with relative permittivity ε_r of 3.38 , thickness of 179 μm and centre frequency of 28GHz with a FBW of 2.85%. The resonator in figure 3.28 has been
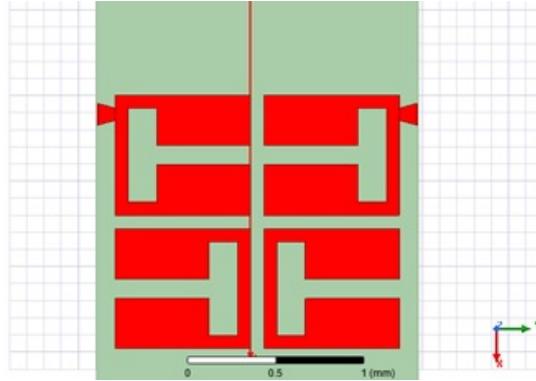
Figure 3.28: Four stepped impedance resonators filter.

selected to give some freedom for (tuning and manufacturing tolerance). A four-
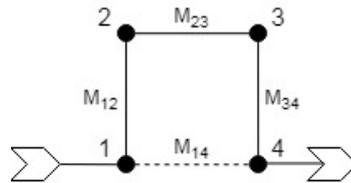
Figure 3.29: Coupling diagram of 4 resonator filter.

pole (n = 4) Elliptic function low-pass prototype with a Ω_a value of 2.00 is chosen, Where Ω_a is the value of the attenuation pole frequency. The element value are found to be: $g_1 = 0.94982$, $g_2 = 1.35473$, $J_1 = 0.12333$, $J_2 = 1.0181$. The coupling coefficients and the external quality factor Q_{exl} of the filter can be calculated as:

$$M_{12} = M_{34} = \frac{FBW}{\sqrt{g_1 g_2}} = 0.0224$$

$$M_{23} = \frac{FBW \cdot J_2}{g_2} = 0.0191$$

$$M_{14} = \frac{FBW \cdot J_1}{g_1} = -0.0033$$

$$Q_{e1} = Q_{e4} = \frac{g_1}{FBW} = 37.39$$

The coupling coefficient can be extracted from EM simulation, for each pair of coupled resonators , the coupling coefficient can be obtained by using the following equations from Hong and Lancaster, ref. [7]:

$$M_{ij} = \frac{f_2^2 - f_1^2}{f_2^2 + f_1^2}$$

Where fl and f2 are lower and higher peak frequencies of the simulated response, respectively.

Also, the external Q value which is the relation between the resonators and input/output can be written as [2]:

$$Q_e = \frac{f_0}{\Delta f_{\pm 90\circ s_{11}}}$$

Where f_0 is the resonant frequency of the resonator and the value of $\pm 90\circ$ is determined from the frequency at which the phase shifts $\pm 90\circ$ with respect to the absolute phase at f_0. By using the design curves of the coupling coefficient and the external Q, the the parameters such as distant t, s_{14}, s_{23} can be found and the all filter dimensions can be realized as shown in fig90.

Results   Momentum ADS and HFSS are used to simulate and optimize the filter. The first simulation is done by Momentum ADS, the simulation results are shown in figure 3.30.
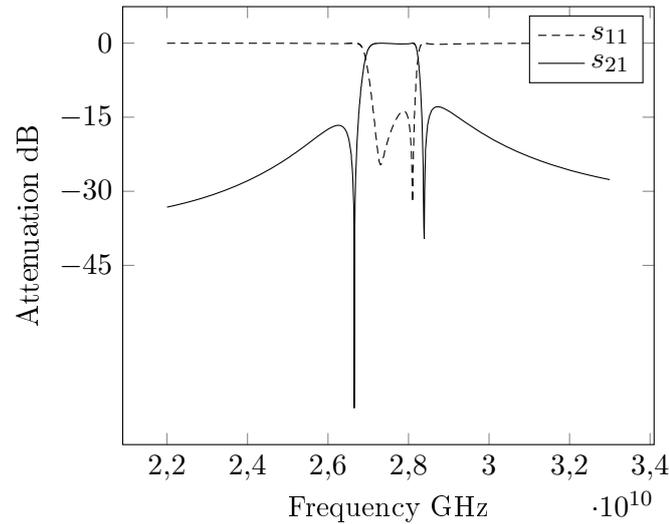
Figure 3.30: Result on HFSS of the 4 resonators tapped filter without dielectric loss.

The optimization is done by HFSS and the following parameters are obtained as: D1 = 1 mm, D2 = 0.9 mm, R1 = 0.73 mm, R2 = 0.43 mm, L = 0.21 mm as show in figure 3.31. The figure 3.32 is the same including a tangent loss of 0.005.
It is observed that the size of overall filter is 4 mm x 7.5 mm x 1.056 mm, which is meet the size requirement. The optimization results from HFSS are shown in figure 3.31. The return loss is better than 14.5 dB within the passband and the insertion loss less than 12dB below 26 GHz and less than 4dB from 3 GHz to 50 GHz.
This design has two transmission zeros at the stopband: at 26.7 GHz and other is at 28.7 GHz.
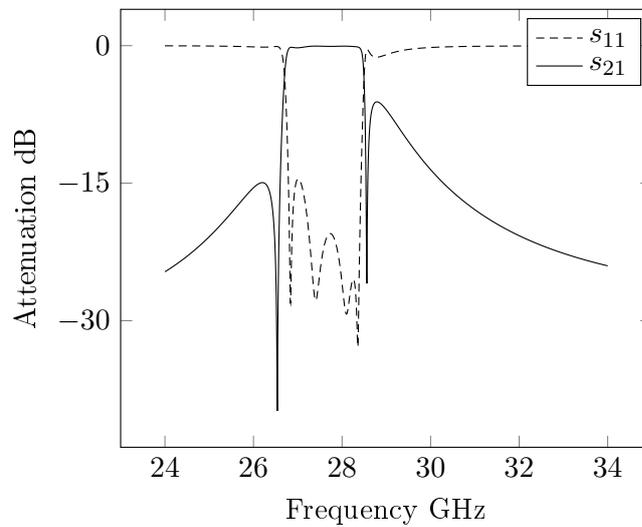
Figure 3.31: Result on HFSS of the 4 resonators tapped filter without dielectric loss.
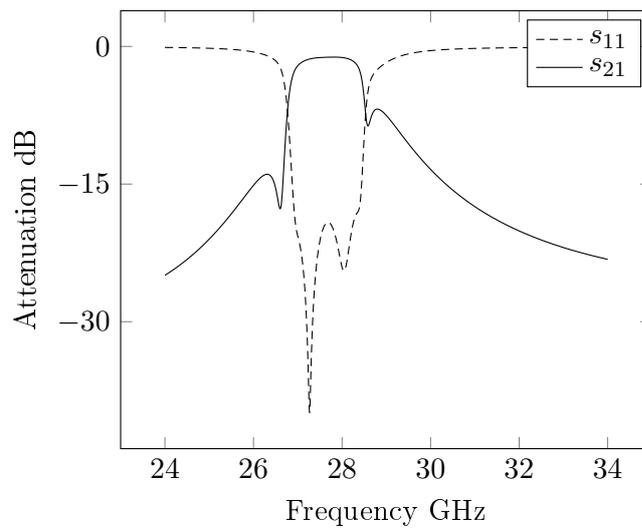
Figure 3.32: Result on HFSS of the 4 resonators tapped filter with 0.005 dielectric loss.

The result from broadband response of the filter shown in figure 3.33. The filter has the first spurious response at about 74.5 GHz, which is consider good shifting. the insertion loss less than 23dB from 55 GHz to 57.7 GHz less than 12 dB from 56.2 GHz to 60 GHz. With different input output feed points where the signals at the input and output are out phase as illustrated I fig, an extra pair of transmission zeros are introduced as show in figure 3.34.
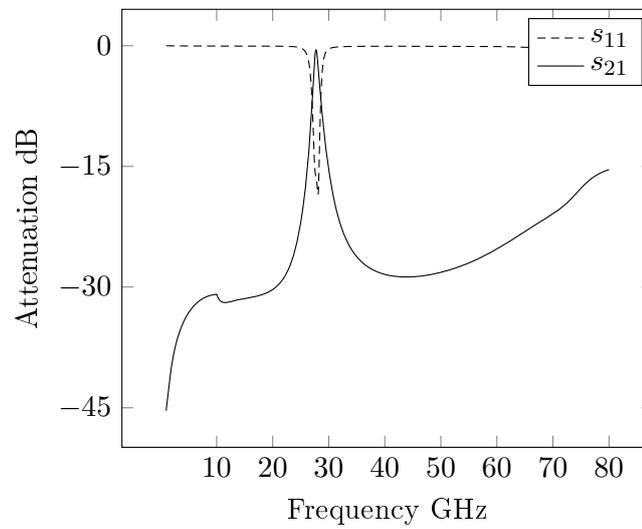
Figure 3.33: Result on HFSS of the 4 resonators tapped filter with 0.005 dielectric loss broad sweep.

Through this design, four transmission zeros are generated at the upper and lower stop band, which give excellent stop band selectivity. Furthermore, this design has similar response comparing to the in phase feed structure design, the layout is illustrated in figure 3.34.
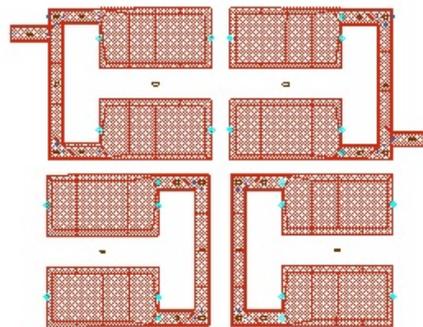
Figure 3.34: Four resonators filter with non-simetrical tap.

The simulation results of the previous configuration are plotted on the figure 3.35.
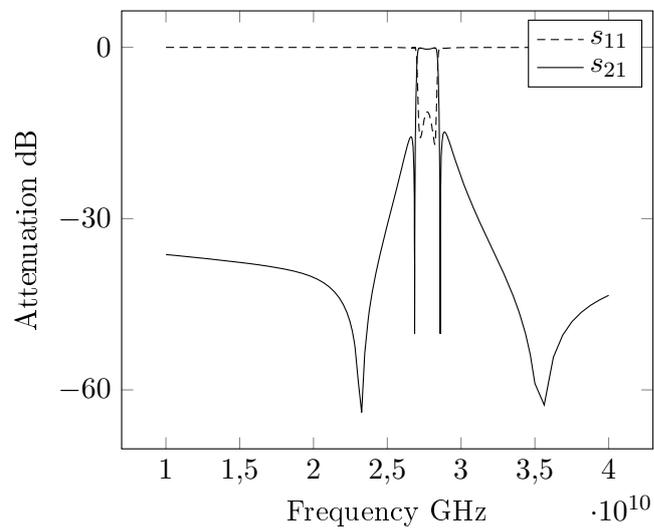
Figure 3.35: Result on ADS of four resonators filter with non-symmetrical tap.

Chapter 4

Conclusions & Future work

In secret we met
In silence I grieve,
That thy heart could forget,
Thy spirit deceive.
If I should meet thee
After long years,
How should I greet thee?
With silence and tears.
Lord Byron

4.1 Conclusions

We try to find a filter that meets all our requirements. For that purpose we have analyse different structures and type of filters. From the beginning we have studied stub, coupled edge, hairpin shape and open-loop resonators based filters. In the first step, we have focused on the fundamental frequency (28 GHz) and for that we used uniform impedance resonators. Under the space constraints we have, the Elliptic response is the only one that allows us to meet the requirements in the band pass and out-of-band attenuations. Therefore we have chosen the open-loop resonators because they allow Elliptic response.

In the next step, we have focused to suppress the harmonics of the local oscillator. The stepped impedance resonators has the possibility to shift the higher order resonance frequencies to remove the spurious harmonics from the local oscillator. The filter is designed at 28 GHz, so we expected a spurious harmonic at 56 GHz. The stepped impedance resonator has shown in the simulation that we have shifted correctly the second resonance removing the spurious harmonics, figure 3.33.

In conclusion, using four step impedance resonators filter gives a good narrow band response, insertion loss < 2 dB, ripple < 0.5 dB and return loss > 15 dB. It also gives the required out-of-band power attenuation. It also keeps the size small. And it moves the second resonance in order to avoid the resonance of the filter matches with the spurious harmonics.

The two software tools, HFSS and ADS, were also learned in the process. A quick prototyping is important in the industry. HFSS and ADS are widely used in companies and research environments. They are not only 3D electromagnetic

simulators, but also they contains tools, as filter assistant which gives you a quick idea of the order of the filter, or LineCalc which gives you an idea about the different limitation in terms of physical dimensions.

4.2   Future work

The Thesis had a limited time. Filter theory is very broad and deep. Along the text the reader could find how complex can be to meet the requirements.
The next step is to perform a tolerance analysis, aiming to estimate the filter performance knowing the dielectric material tolerances, and the fabrication limitations. Then manufacturing the filter, either alone, either with an antenna to analyse. Both cases are interesting. Once it is manufactured, tests need to be done and comparison between theoretical analysis and measurements will be made. Finally implementation and measurements over a commercial device would be taken. Another point here would be the miniaturization of the filter. New techniques based on the learning and papers from last years, can be applied to reduce even more the design without loosing performance.
Finally, an analysis should be done to estimate the maximum transmit and received power. This is very important for the filter implementation.

References

[1] Pozar, David M. (2012). Microwave engineering. Hoboken, NJ :Wiley

[2] J. S. Hong and M. J. Lancaster (2001). Microstrip filters for RF/Microwave applications. New York :Wiley

[3] Liao, Samuel Y. "Microwave Devices and Circuits". NJ :Prentice Hall

[4] K.S.Khandelwal1 and A.K.Kureshi (2014). Realization of Microstrip Band-Pass Filter Design. IJARCET

[5] G. L. Matthaei, "Interdigital band-pass filters," IEEE Trans., MTT-10, 1962, 479-492

[6] S. Caspi and J. Adelman, "Design of combline and interdigital filters with tapped-line input," IEEE Trans., MTT-36, April 1988, 759-763.

[7] Jia-Sheng Hong, Michael J. Lancaster, "Cross-Coupled Microstrip Hairpin-Resonator Filters", IEEE Trans., January 1998

[8] Jia-Sheng Hong, Michael J. Lancaster, "Coupling of Microstrip Square Open-Loop Resonators for Cross-Coupled Planar Microwave Filters", IEEE Trans., December 1996

[9] Edward G. Cristal, Sidney Frankel. "Hairpin-Line and Hybrid Hairpin-Line/Half-Wave Parallel-Coupled-Line Filters", IEEE Trans., November 1972

[10] Sheng-Yuan Lee, Chih-Ming Tsai. "New Cross-Coupled Filter Design Using Improved Hairpin Resonators", IEEE Trans., December 2000

[11] J.-S. Hong, "Couplings of asynchronously tuned coupled microwave resonators," IEE Proc. Microwaves, Antennas and Propagation, 147, Oct. 2000, 354-358

[12] Mitsuo Makimoto, Sadahiko Yamashita, "Bandpass Filters Using Parallel Coupled Stripline Stepped Impedance Resonators", IEEE Trans., December 1980

[13] Sheng-Yuan Lee, "Optimum Resonant Conditions of Stepped Impedance Resonators", IEEE Trans., October 2005

[14] E. G. Cristal and S. Frankel, "Design of hairpin-line and hybrid haripin-parallel-coupled-line filters," IEEE MTT-S, Digest, 1971, 12-13.

[15] Lung-Hwa Hsieh- and Kai Chang, "Compact, Broad-Stopband Elliptic-Function Lowpass Filters Using Microstrip Stepped Impedance Hairpin Resonators" IEEE Trans., June 2003

[16] M. Sagawa, K. Takahashi and M. Makimoto, "Miniaturized hairpin resonator filters and their application to receiver front-end MICs," in IEEE Transactions on Microwave Theory and Techniques, vol. 37, no. 12, pp. 1991-1997, Dec. 1989.

[17] P. R. Geffe, "Simplified Modern Filter Design", New York John F. Rider, 1963, Appendix 4.

[18] Dejan V. Tosic, Milka Potrebic, "Symbolic analysis of immittance inverters", TELFOR 2006.

[19] Smith, P. H.; Transmission Line Calculator; Electronics, Vol. 12, No. 1, pp 29-31, January 1939
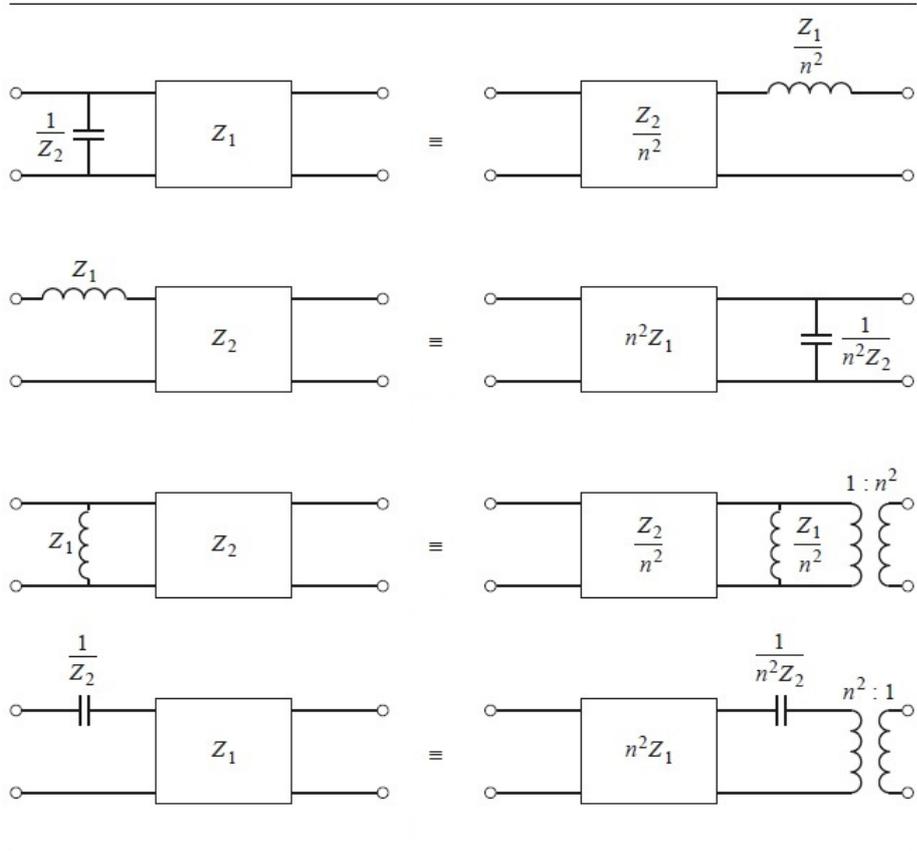
Kuroda identities

Figure A.1: The Four Kuroda identities $\left(n^2 = 1 + Z_2/Z_1\right)$.

TABLE 8.8   Ten Canonical Coupled Line Circuits

Circuit	Image Impedance	Response
	$Z_{i1} = \dfrac{2Z_{0e}Z_{0o}\cos\theta}{\sqrt{(Z_{0e}+Z_{0o})^2\cos^2\theta-(Z_{0e}-Z_{0o})^2}}$ $Z_{i2} = \dfrac{Z_{0e}Z_{0o}}{Z_{i1}}$	Low-pass
	$Z_{i1} = \dfrac{2Z_{0e}Z_{0o}\sin\theta}{\sqrt{(Z_{0e}-Z_{0o})^2-(Z_{0e}+Z_{0o})^2\cos^2\theta}}$	Bandpass
	$Z_{i1} = \dfrac{\sqrt{(Z_{0e}-Z_{0o})^2-(Z_{0e}+Z_{0o})^2\cos^2\theta}}{2\sin\theta}$	Bandpass
	$Z_{i1} = \dfrac{\sqrt{Z_{0e}Z_{0o}}}{(Z_{0e}+Z_{0o})\sin\theta}\,\sqrt{(Z_{0e}-Z_{0o})^2-(Z_{0e}+Z_{0o})^2\cos^2\theta}$ $Z_{i2} = \dfrac{Z_{0e}Z_{0o}}{Z_{i1}}$	Bandpass
	$Z_{i1} = \dfrac{Z_{0e}+Z_{0o}}{2}$	All pass
	$Z_{i1} = \dfrac{2Z_{0e}Z_{0o}}{Z_{0e}+Z_{0o}}$	All pass
	$Z_{i1} = \sqrt{Z_{0e}Z_{0o}}$	All pass
	$Z_{i1} = -j\,\dfrac{2Z_{0e}Z_{0o}}{Z_{0e}+Z_{0o}}\cot\theta$ $Z_{i2} = \dfrac{Z_{0e}Z_{0o}}{Z_{i1}}$	All stop
	$Z_{i1} = j\,\sqrt{Z_{0e}Z_{0o}}\,\tan\theta$	All stop
	$Z_{i1} = -j\,\sqrt{Z_{0e}Z_{0o}}\,\cot\theta$	All stop

The Complete Smith Chart

Black Magic Design

RADIALLY SCALED PARAMETERS
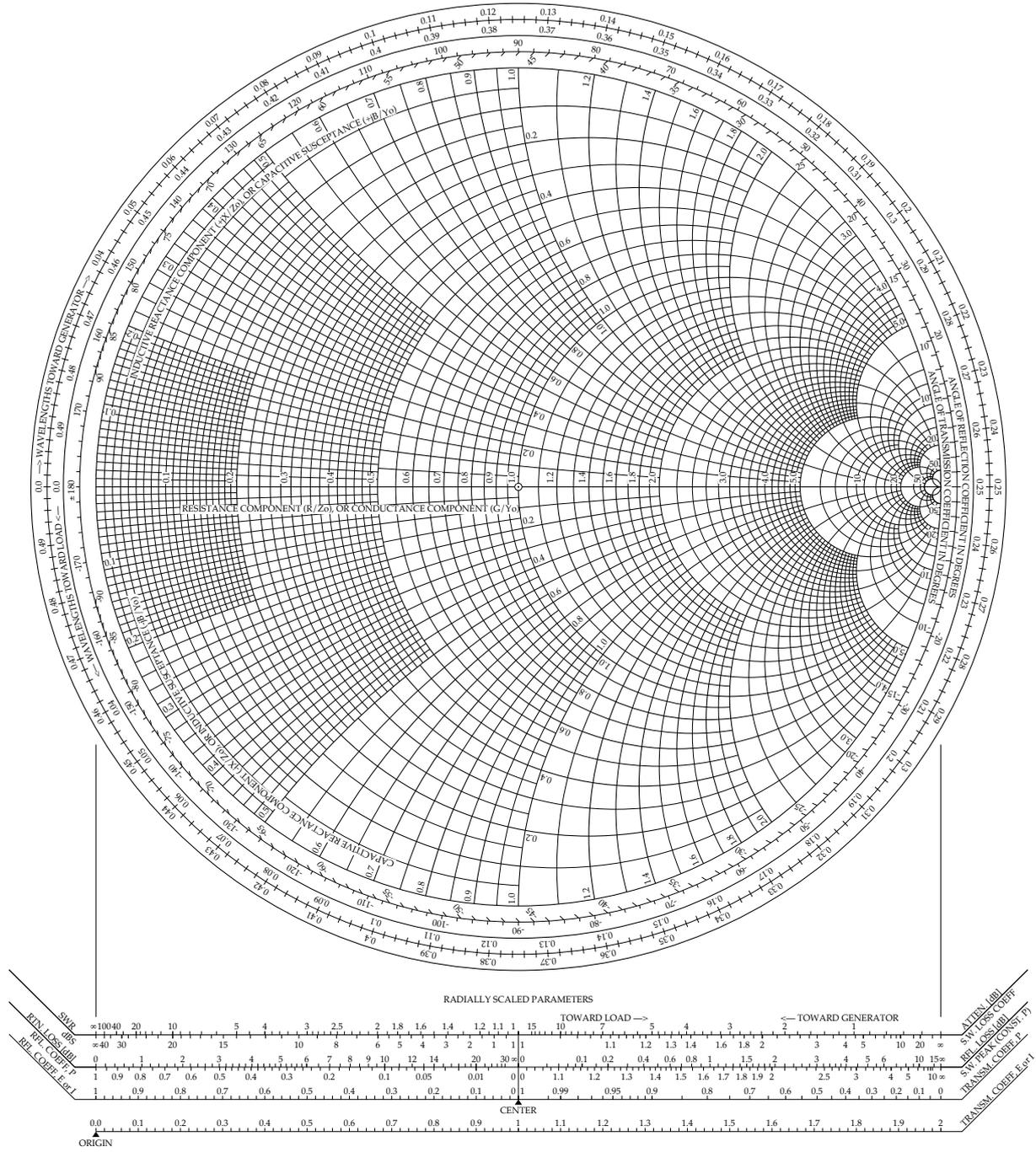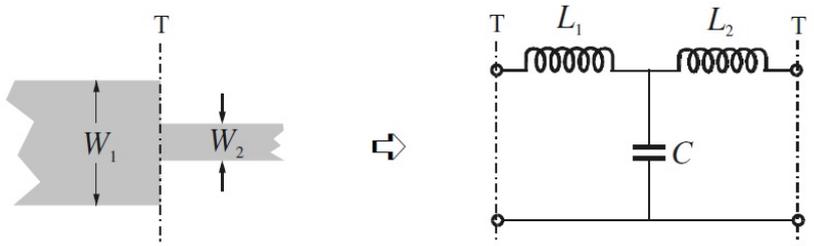
(a) Microstrip discontinuity: step.
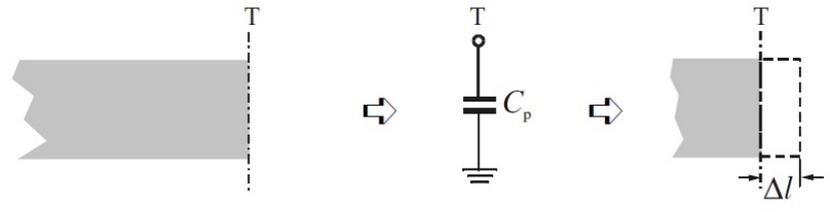
(b) Microstrip discontinuity: open end.
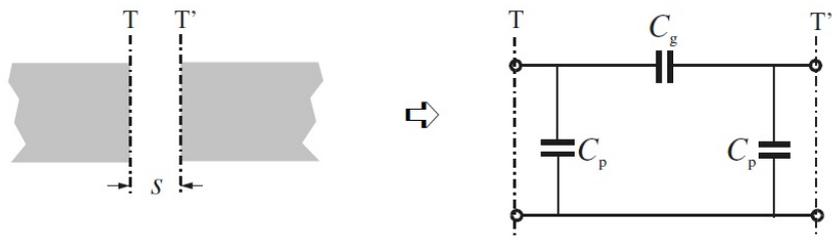
(c) Microstrip discontinuity: gap.
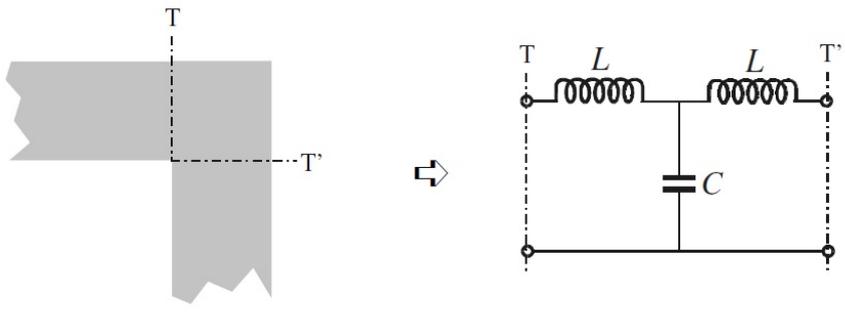
(d) Microstrip discontinuity: bend.

Figure A.2: Discontinuities on transmission lines.

SLIN (Stripline)

Symbol

Illustration
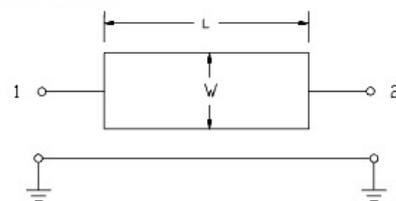
Range of Usage

W > 0 (for T = 0)
W ≥ 0.35 × B (for T > 0)
T ≤ 0.25 × B
where
B = ground plane spacing (from associated SSUB)
T = conductor thickness (from associated SSUB)

Parameters

Name	Description	Units	Default
Subst	Substrate instance name	None	SSub1
W	Line width	mil	25.0
L	Line length	mil	100.0
Temp	Physical temperature	°C	None
Layer	(for Layout option) Conductor layer number: cond1, cond2	None	cond1

- The frequency-domain analytical model is as follows. For centered stripline of negligible thickness (T=0), the characteristic line impedance is calculated from the exact formula derived by Cohn using conformal mapping. For a centered stripline of finite thickness, Wheeler's approximate formula for the characteristic line impedance and attenuation factor are used. The attenuation formula provides a smooth transition from dc resistance to resistance due to skin effect at high frequencies. Dielectric loss is also included in the model.

- For offset stripline, a model developed by William Getsinger for Keysight and based on the formula of Shelton, Cohn and Wheeler is used. For an offset stripline of negligible thickness (T=0), the characteristic line impedance is calculated from the exact formula derived by Shelton using conformal mapping. For an offset stripline of finite thickness, Shelton's exact formula is combined with Cohn's formula for a centered thick stripline to formulate an approximate formula. Additionally, the attenuation formula developed by Wheeler is used. The attenuation formula provides a smooth transition from dc resistance to resistance due to skin effect at high frequencies. Dielectric loss is also included in the model.

- If the Subst parameter refers to an SSUBO whose spacing parameter S has a non-zero value, the component is considered offset for simulation and layout purposes. A reference to SSUBO with its spacing parameter S=0 is equivalent to a reference to the SSUB.

- The "Temp" parameter is only used in noise calculations.

- For noise to be generated, the transmission line must be lossy (loss generates thermal noise).

Figure A.3: Stripline ADS information.

LUND

UNIVERSITY