90 Degree Hybrid Coupler

By

James Corsini

Josué Malaver

Sokol Lushllari

Date: April 25, 2013

Major Qualifying Project submitted to the Faculty of

WORCESTER POLYTECHNIC INSTITUTE

In partial fulfillment of the requirements for the degree of

Bachelor of Science

Approved:

Professor Reinhold Ludwig

Professor John McNeill

David Whitefield

Abstract

In this Major Qualifying Project we were tasked by our project sponsor, Skyworks Solutions Inc., to redesign a 90-hybrid coupler with the center frequency of 1.9GHz. The sponsor's requirements for the new design were to increase the bandwidth and decrease the device area. Both of these requirements were met by developing a theoretical model and were then validated by simulations in Agilent's ADS. Additionally Ansys' HFSS was used to model the new design in a 3-D environment where the electric and magnetic radiation fields can be studied. This was a necessary step in order to develop a model accounting for interference originating from the device. Step by step tutorials were created so that the reader can better understand how to implement the developed models in these simulators. The final design yielded a bandwidth increase of 150% with an area reduction of 63%.

Acknowledgements

We would like to thank Dr. Whitefield and the Skyworks Inc. team, without whom this project would not exist. Specifically, we would like to thank Eric Austin for his knowledge and guidance with the measurements taken at the Skyworks laboratory. Special thanks to our advisor Professor Reinhold Ludwig and our co-advisor Professor John McNeill for their constant guidance and motivation. Also, thanks to Jeffery Elloian who was extremely helpful with the HFSS simulations.

Table of Contents

Table of Figures

Table of Tables

Acronym List

MQP	Major Qualifying Project
IEEE	Institute of Electrical and Electronics Engineers
RF	Radio Frequency
MW	Microwaves
LNA	Low noise amplifier
MMIC	Monolithic microwave integrated circuit
ADS	Advanced Design System
HFSS	High Frequency Structural Simulator
S-parameter	Scattering Parameters
FEM	Finite Element Method
TL	Transmission Lines

1 Introduction

Skyworks Inc. has sponsored a Major Qualifying Project (MQP) to investigate and redesign its current 90-degree hybrid coupler. It is one of the world's leading innovators in high performance analog and mixed signal semiconductors and offers a wide variety of products. These products are used in military, medical, automotive, and handset applications around the world [1]. Specifically, the 90 degree hybrid coupler is utilized in the handset infrastructure because it offers low-loss, high isolation, and exceptional phase and amplitude balance.

The demand for a smaller 90-degree hybrid coupler with a broader bandwidth is proliferating at a quick pace in order to keep up with a rapidly advancing communication industry. Skyworks has deemed the 90-degree hybrid coupler as a product of extreme interest for further development, and its microelectronic implementation has become the focus of this MQP. The goal of this project is to create a new design that will improve upon the existing model in terms of frequency capability while maintaining or decreasing the device size.

2 Background

2.1 Organization of MQP report

The first step taken in this project was to understand the physical performance of the 90-degree hybrid coupler. Throughout this process IEEE journals and several other reputable articles have been studied to gain a better understanding of this device. Additionally, different topologies and currently implemented solutions have been investigated to understand what is state-of-the-art in industry. The current design implemented by the project sponsor Skyworks was thoroughly analyzed through mathematical analysis, software simulations, and on-site measurements. Fully understanding different topologies and the current problems of the existing coupler have provided direction towards the new design approach.

2.2 What are 90 degree hybrid couplers?

Hybrid couplers are four-port devices that split the incident power signal into two output ports. The signals at the outputs are attenuated by three decibels (3dB) and have a 90 degree phase difference with respect to each other. Three decibel attenuation means that 50% of the input power is lost [2]. In addition, reflections due to mismatches are sent to the isolation port preventing any power from reflecting back to the input port. In addition to splitting a signal they can also be used to combine power signals with a high degree of isolation between the ports. A block diagram of this functionality is illustrated in Figure 1.

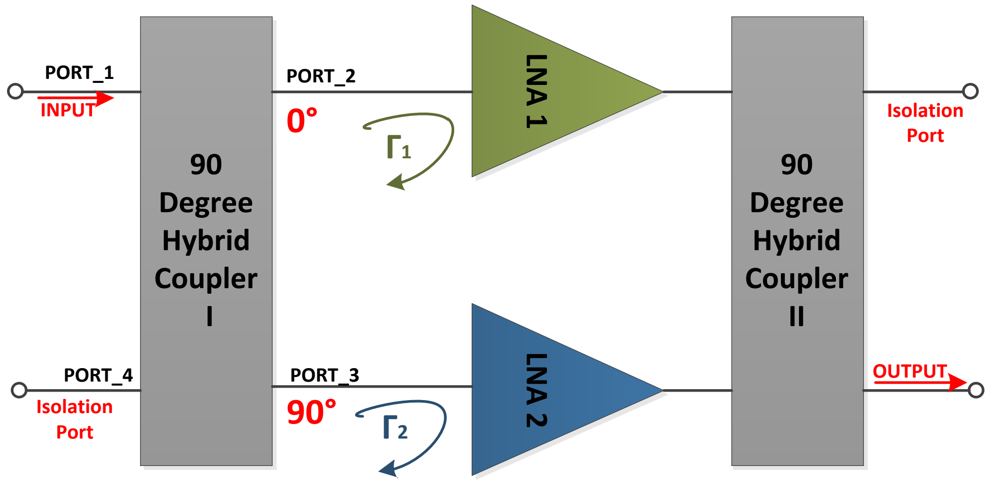


Figure 1: Block diagram of 90 Degree hybrid coupler feeding into LNAs

As seen in Figure 1 we have two hybrid couplers attached to two different low noise amplifiers (LNAs). This is an example of a coupler performing both of its main functions: signal splitting and combining, while the LNAs do their main job of amplifying the signal. Starting at port one a power wave is sent and split at the output ports. From there the divided power is fed to the two LNAs and the amplified outputs are combined together with the second coupler. By placing a hybrid coupler in front of the LNAs, electrical protection for the components located before port one is guaranteed because of the hybrid coupler's ability to significantly reduce the amount of reflected power at the input port. Since there is no reflected power at the input, the hybrid coupler can also be called a balanced structure circuit. In addition to the aforementioned functions these devices are vital in communication systems as circuit protectors, perhaps one of the largest global markets. They can also be used in microwave (MW) phase shifters, antenna feed systems, driver circuit protection, and in-phase/quadrature (I-Q) modulation / demodulation.

90-degree hybrid couplers are often called branch-line couplers. As the name implies power is equally divided between the output ports and are therefore electrically and mechanically symmetrical. These branch-line couplers are built using transmission lines and their size is proportional to the wavelength of the designated center frequency, which can be meters long. This becomes a significant drawback in applications where a small footprint is required.

Hybrid couplers can also be built by using lumped components, which are resistors, inductors, and capacitors with an ideal (lossless) connection. The lumped component design is promising because it provides low insertion loss, wider bandwidth, and a smaller size circuit, making it a good fit for a monolithic microwave integrated circuit (MMIC) [3].

The best way to describe hybrid couplers is through the use of scattering parameters, better known as S-parameters. Specifically, these S-parameters describe the signal in terms of incident and reflected power at the ports of the network. Using the graphs and matrices of the S-parameters (as seen in section 2.7), it is easy to show the 3dB attenuation of the input signal at the operating frequency as well as the bandwidth of the hybrid coupler. Agilent's Advanced Design Simulator (ADS) provides a method to extract the S-parameters of a device under test. ADS has been adopted by a vast majority of companies including the project sponsor Skyworks; it is considered to be an industry standard. For mathematical analysis of the 90-degree hybrid coupler, a combination of even-odd mode decomposition, S-parameters, and ABCD parameters is used, which will be discussed in further detail later on.

2.3 Design topologies

There are many design topologies for hybrid couplers [4]. The most common designs are implemented using transmission lines and lumped elements. In this section, a few topologies are examined to provide a thorough understanding of the limitations and abilities of each.

2.3.1 Distributed elements

A distributed element system is one where the electrical characteristics (resistance, inductance, capacitance, conductance) of each element in the circuit produce unwanted effects. To account for these unwanted effects, also known as parasitics, it is required to describe each element in terms of unit-length. An example of a distributed system is the four port branch line hybrid coupler with transmission line elements shown in Figure 2.

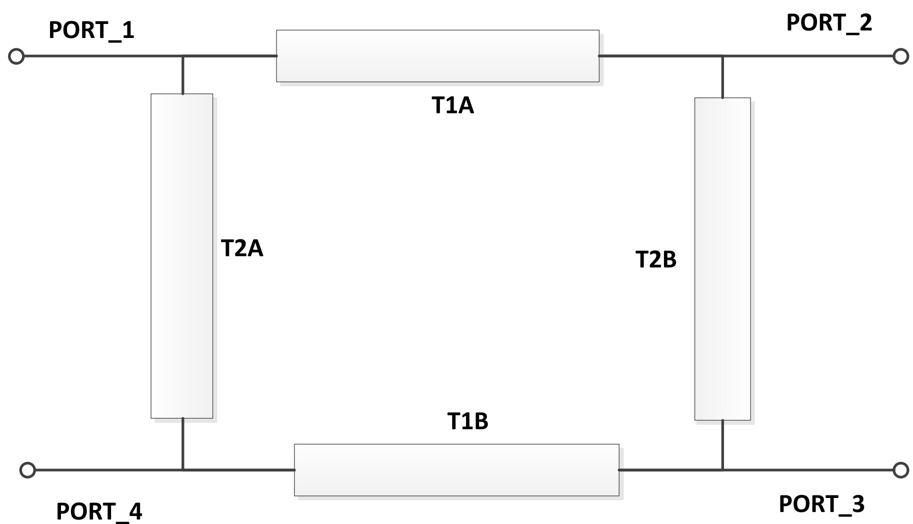


Figure 2: Branch line hybrid coupler

Each transmission line is designed based on the principle of the lambda quarter transformer, which yields perfect matching between ports (refer to Appendix for complete derivation/explanation of the lambda quarter transformer). The length of each transmission line is dependent upon the center frequency, which is determined by the application. The characteristic impedance of the parallel transmission lines in the circuit is the same. Characteristic impedance "is the ratio of the amplitudes of voltage and current of a single wave propagating along the line" [5] (see section 5). In this scenario, there are two characteristic impedances, 50Ω and 35.35Ω. Transmission lines T1A and T1B have the same characteristic impedance, 35.35Ω. Similarly, transmission lines T2A and T2B have the same characteristic impedance of 50Ω. As previously mentioned, all of the transmission lines have approximately the same length because of the lambda quarter transformation behavior.

A second topology using transmission lines is frequently implemented. In this design two transmission lines, again using the lambda-quarter transformer, are placed in parallel at very close proximity in order to create a coupling (or capacitive) effect. An example is shown in Figure 3 where port 1 is the input port, ports 2 and 3 are the output ports, and port 4 is known as the isolation port. The coupling effect permits the signal from port 1 in the first transmission line to be coupled into ports 3 and 4. Without this effect there would be no connection and the entire signal would go to port 2.

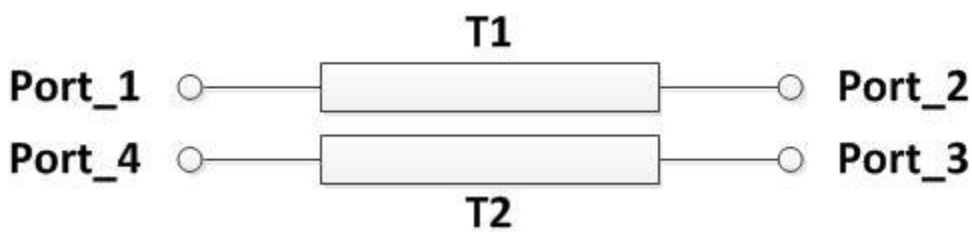


Figure 3: Coupled transmission tine hybrid coupler

2.3.2 Lumped elements

The other topology for 90 degree hybrid couplers is using lumped components. Lumped component systems are especially useful because each element is considered to be independent and the connecting wires are assumed to be ideal. This design is developed by implementing the aforementioned branch line coupler and then implementing a TL to lumped elements transformation. Specifically, each transmission line component is transformed into its so-called Pi or T network representation; illustrated in Figure 4. Section 5 provides a complete derivation of the pi network of the transmission line applicable to the project.

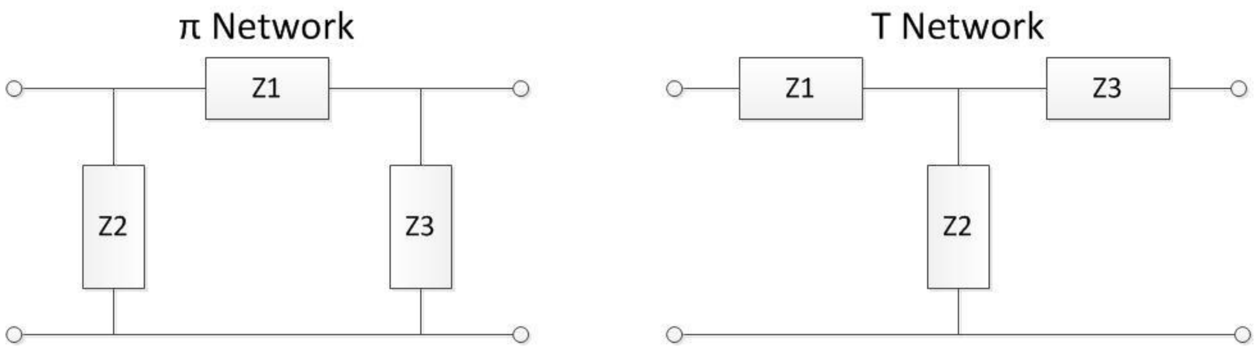


Figure 4: Pi and T network configurations

The design shown in Figure 5 is the current lumped equivalent design used by Skyworks and was given to investigate its circuit behavior and understand its physical implementation. This design builds on the coupled transmission line model previously mentioned. The TL to lumped element transformation yields four inductors and five capacitors. Note that capacitor C3 is the only signal path between ports 1 and 2 to ports 3 and 4. This replaces the coupling effect that occurs when the two transmission lines are placed in close proximity to one another.

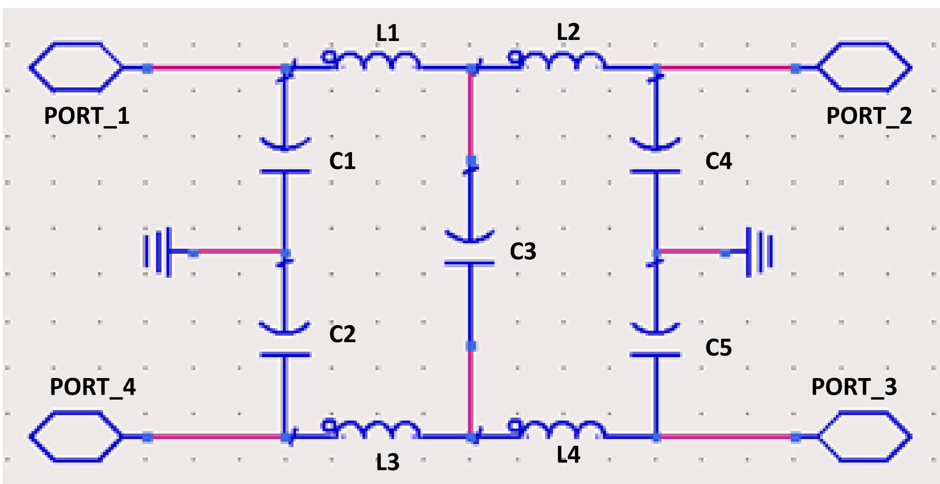


Figure 5: Current design at Skyworks for 90 degree hybrid coupler

2.4 Current problems of 90 degree hybrid couplers in microelectronic circuits

A hybrid coupler's size, cost, interference and bandwidth are often the biggest limitations in its implementation. Keeping these limitations in mind, manufacturers must provide a diverse set of designs to satisfy different frequency bands. Noting the overwhelmingly large spectrum of communication systems, it would be ideal for manufacturers to provide a solution which satisfies needs over a significant range of these frequencies. A case in which this issue is clearly seen is in cell phone applications.

Solutions from manufacturers for cell phone applications currently provide bandwidths between 7% to an ideal of 20% [6]. These bandwidths are quite small, and trying to implement the same device in other applications with different center frequencies would prove to be non-functional. It is important to note that 90 degree hybrid couplers are tuned to a specific frequency, as will be the case even with any improved design. However, if there is a significant improvement of the bandwidth that can be achieved by these devices, it would prove to be quite advantageous for both manufacturers and consumers, reducing the cost while increasing manufacturing efficiency. The main goal is to have one device broad enough to satisfy a wide range of frequency applications.

2.5 S-Parameters, ABCD-Matrix Description

The following sections describe the definition of S-parameters as well as ABCD parameters. S-parameters and ABCD parameters both describe the input to output relationship of a linear network system. However, the ABCD parameters are strictly used for two port networks, in this case they are used to convert a single transmission line to its lumped component equivalent. The S-parameters, on the other hand, are used to describe any multi-port network.

2.5.1 Scattering parameters

A major method of analysis in RF design is S-parameters. They are "power wave descriptors that permit us to define the input-output relations of a network in terms of incident and reflected power waves." [7] S-parameters are used, as opposed to ABCD, because they "do not use open or short circuit conditions to characterize a linear electrical network; instead, matched loads are used [at all ports]. These terminations are much easier to use at high signal frequencies than open-circuit and short-circuit terminations." [8]

When it comes to analyzing S-parameters there are two main variables that need to be accounted for. The first one is $a_{n,}$ or more simply the incident power wave of port n, which is the waveform that goes into the system. The other variable is b_n, also known as the reflected power wave at port n, which represents the reflected signal due to any impedance mismatch in the system. Figure 6 displays these variables in an example of a two port network.

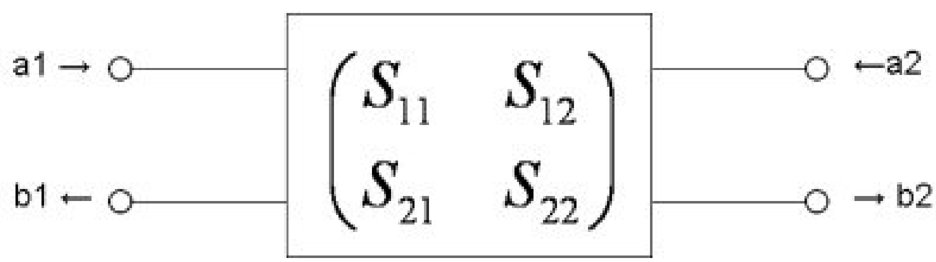


Figure 6: Two port S-parameter network

The equations listed below represent an incident normalized power wave (2.1), and a reflected normalized power wave (2.2). They take into account Z_0, which is the characteristic impedance of the line as well as the total voltage and current at port n. Index n denotes the respective port.

$$a_n = \frac{1}{2*(\sqrt{Z_0})}(V_n + Z_0 * I_n) \tag{2.1}$$

$$b_n = \frac{1}{2*(\sqrt{Z_0})}(V_n - Z_0 * I_n) \tag{2.2}$$

Equations 2.1 and 2.2 can be combined to form the S-parameter matrix for the two port system in the form found in equation 2.3.

$$\begin{pmatrix} b_1 \\ b_2 \end{pmatrix} = \begin{bmatrix} S_{11} & S_{12} \\ S_{21} & S_{22} \end{bmatrix} \begin{pmatrix} a_1 \\ a_2 \end{pmatrix} \tag{2.3}$$

From this matrix the S-parameters are defined as shown in equations 2.4a-d.

$$S_{11} = \frac{b_1}{a_1} = \left.\frac{reflected\ wave}{incident\ wave}\right|_{a_2=0} \tag{2.4a}$$

$$S_{12} = \frac{b_1}{a_2} = \left.\frac{reflected\ wave}{incident\ wave}\right|_{a_1=0} \tag{2.4b}$$

$$S_{21} = \frac{b_2}{a_1} = \left.\frac{reflected\ wave}{incident\ wave}\right|_{a_2=0} \tag{2.4c}$$

$$S_{22} = \frac{b_2}{a_2} = \left.\frac{reflected\ wave}{incident\ wave}\right|_{a_1=0} \tag{2.4d}$$

Each of these parameters represents a specific network characteristic: S_{11} is the input reflection coefficient, S_{12} is the reverse voltage gain, S_{21} is the forward voltage gain, and S_{22} is the output reflection coefficient.

It is important to note that in the S_{ij} notation, i represents the port being examined, whereas j represents the port taken as reference (i.e.; voltage gain at port i with respect to j). In this manner, the gain or loss for any port with respect to any other port can be observed. Setting i and j to the same port yields the reflection coefficient for that port. For the 90 degree hybrid coupler, a four port extension of the S-parameters is employed, which yields a 4x4 matrix.

$$\begin{bmatrix} 0 & S12 & S13 & S14 \\ S21 & 0 & 0 & S24 \\ S31 & 0 & 0 & S34 \\ 0 & S42 & S43 & S44 \end{bmatrix}$$

Figure 7: 4x4 S-parameter matrix

In Figure 7 a 4x4 s-parameter matrix is described. This is an idealized matrix, where the reflection coefficients for each port have been set equal to zero, implying impedance matching at all ports. This matrix was used when analyzing the device under test. In this case, port 1 was assigned to be the input port. Therefore parameter S_{21} refers to the forward voltage gain at port 2 with respect to port 1. Similarly, S_{31} is the forward gain found in port 3 with respect to port 1. Hence these ports will be called the output ports and will be used to verify the functionality of the coupler. Specifically, to assure that the 3dB attenuation of the input signal is being output at port 2 and port 3.

Overall, the S-parameters are an important tool for analyzing RF systems and will be extremely useful when it comes to understanding the simulated and interpreted data.

2.5.2 ABCD Parameters

The ABCD parameters are used to analyze a two port network with respect to the total voltages and currents, as shown in Figure 8. ABCD matrices are mostly used for cascading networks and to convert to and from different network configurations using the conversion table provided in Appendix D.

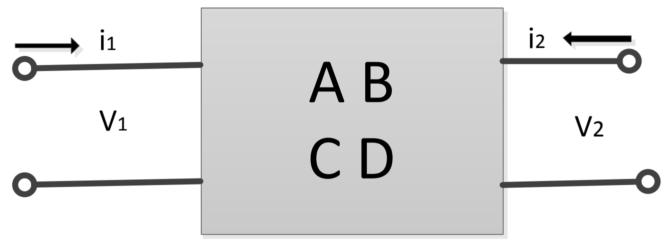


Figure 8: Two-Port network, analyzed with ABCD parameters

The ABCD parameters connect terminal voltages and currents according to

$$\begin{pmatrix} V_1 \\ i_1 \end{pmatrix} = \begin{bmatrix} A & B \\ C & D \end{bmatrix} \begin{pmatrix} V_2 \\ -i_2 \end{pmatrix} \tag{2.5}$$

Where

$$A = \left.\frac{V_1}{V_2}\right|_{i_2=0} \qquad B = \left.\frac{V_1}{-i_2}\right|_{V_2=0} \tag{2.6a/2.6b}$$

$$C = \left.\frac{i_1}{V_2}\right|_{i_2=0} \qquad D = \left.\frac{i_1}{-i_2}\right|_{V_2=0} \tag{2.6c/2.6d}$$

To find the overall ABCD parameters of a cascaded network, as shown in Figure 9, we can multiply the matrices for each network as stated in equation 2.7.

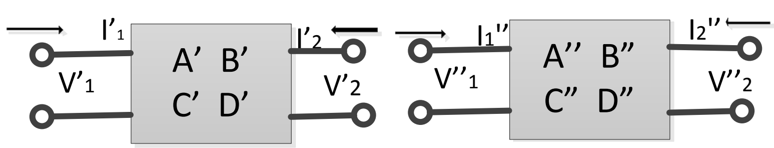


Figure 9: Cascading two networks

$$\begin{bmatrix} A & B \\ C & D \end{bmatrix} = \begin{bmatrix} A' & B' \\ C' & D' \end{bmatrix} \begin{bmatrix} A'' & B'' \\ C'' & D'' \end{bmatrix} \tag{2.7}$$

Specifically the ABCD parameters will be used to calculate values for the pi networks.

2.6 Even and Odd Mode Analysis

Even and odd mode analysis is a technique used to extract the even and odd-mode impedances of a circuit. It is employed in horizontally or vertically symmetric circuits. This technique is based on two principles: symmetry of the circuit and superposition. An example of a hybrid coupler is shown in Figure 10. The coupler is properly terminated (matched) so that the reflected power at the input port is zero.

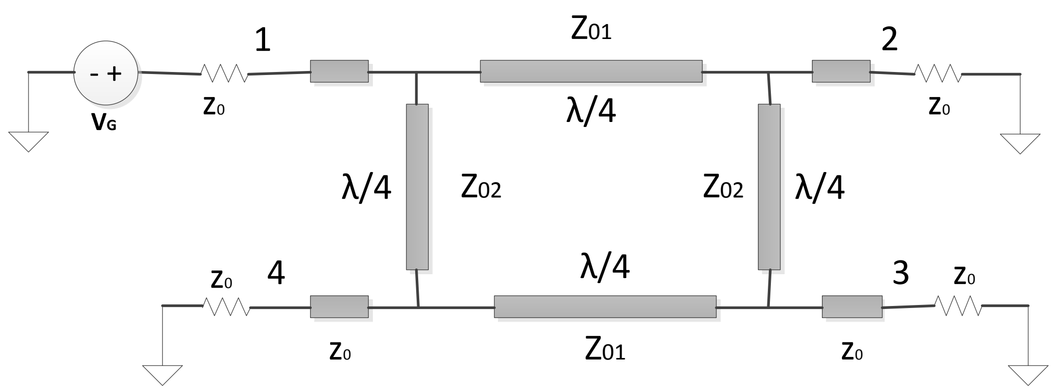


Figure 10: Quad Hybrid connected with a supply voltage

As shown in Figure 11 the hybrid coupler becomes symmetric when a line splits the circuit horizontally. The voltage supply is reconfigured to ensure zero voltage at port 4 and V_G at port 1.

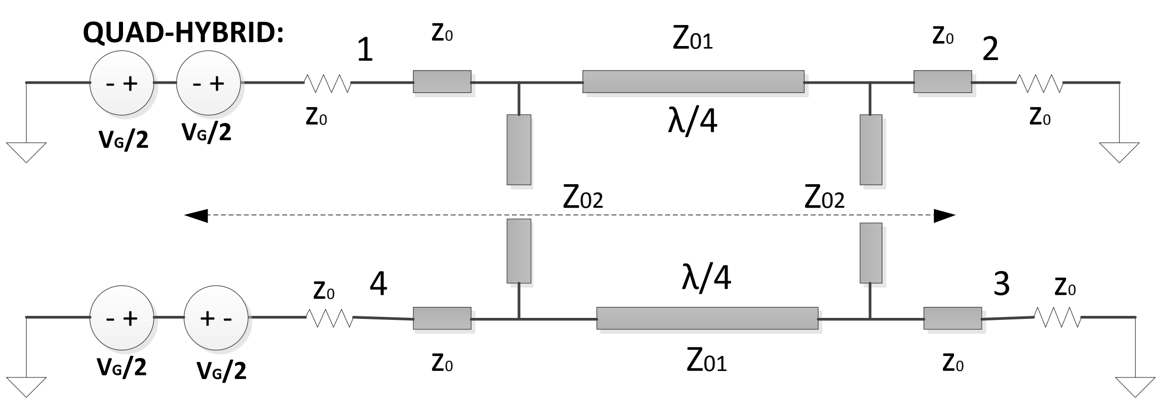


Figure 11: Quad-Hybrid symmetrically split

For even mode analysis the TL is cut in half and set as an open circuit. This is shown in Figure 12 the voltage supply in port 1 and 2 of the quad hybrid has the same polarity.

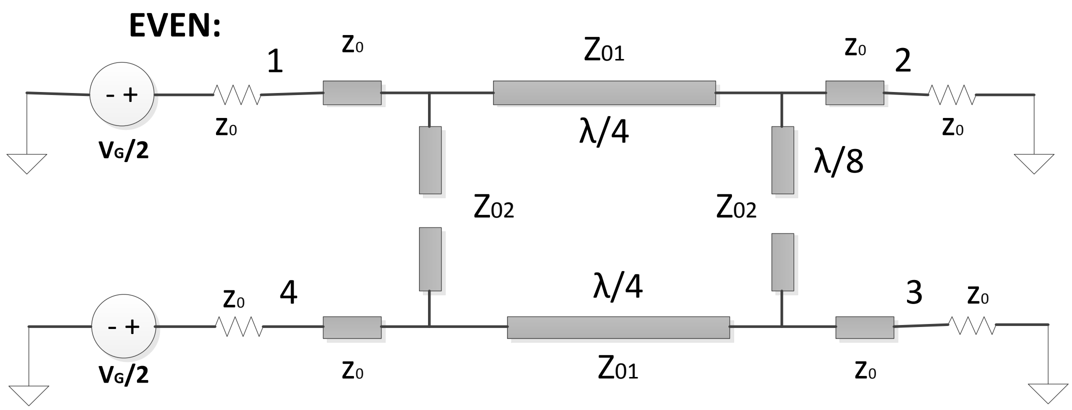


Figure 12: EVEN mode configuration, open TL

For the odd mode analysis the TL is cut in half and grounded along the symmetry line, as is shown in Figure 13. The voltage supply of ports 1 and 2 of the quad hybrid has the opposite polarity. Since the hybrid coupler is a 4 port device, it is analyzed based on the 4x4 matrix representation of the S-parameters. See section 4 for a more in depth analysis.

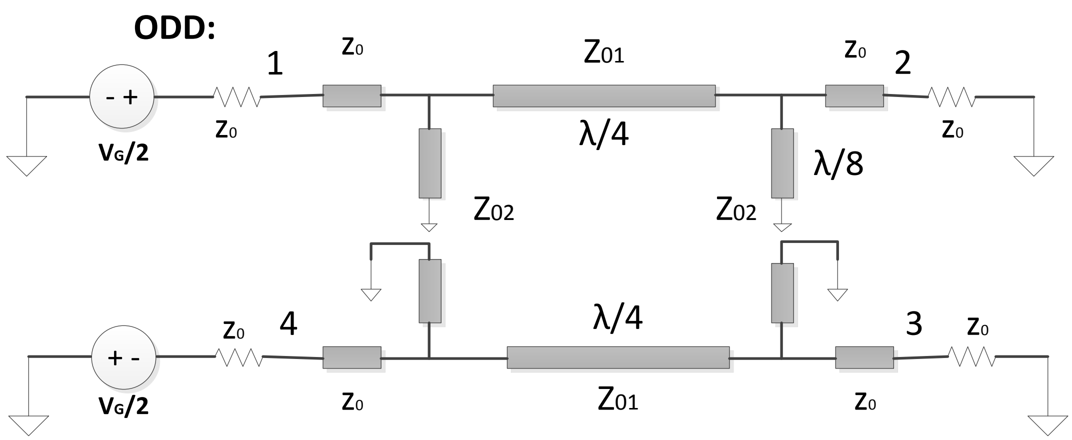


Figure 13: ODD mode, grounded TL

2.7 Network Analyzer Measurements

In the process of developing a new design it is helpful to know how the baseline model functions. Measurements were planned and executed at the Skyworks Inc. laboratory with a network analyzer to find the S-Parameters of the existing coupler.

Taking measurements on a network analyzer requires the user to calibrate the system beforehand. When the measurements for this project were taken at Skyworks, the analyzer was calibrated using two methods. The first method utilized an automatic electronic calibration. The second method was to attach a strip of quarter wavelength transmission line between the ports of the analyzer. A graph of this transmission line (also called thru line) with the electronic calibration can be seen in Figure 14.

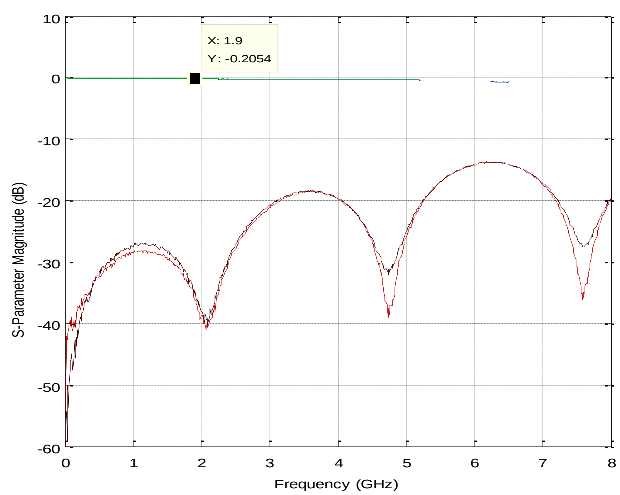


Figure 14: Thru TL using electronic calibration

The difference, as shown by the marker in Figure 14, shows how the transmission line calibration differed from the electronic calibration. At 1.9 GHz the difference in magnitude was -.2054dB, which was not a very significant change (two forms of calibration measurements were taken of the baseline model (Skyworks' original) of the hybrid coupler.) Throughout the measurement process a total of four 90 degree hybrid couplers were used to ensure measuring accuracy.

Figure 15 through Figure 17 were obtained for device 1 using the electronic calibration. They show the phase and amplitude difference as well as the important S-parameters, S_{21} and S_{31}. Because the two calibration types yielded similar results, the graphs using the transmission line calibration, as well as the graphs for the electronic calibration were omitted but can be found in Appendix A.

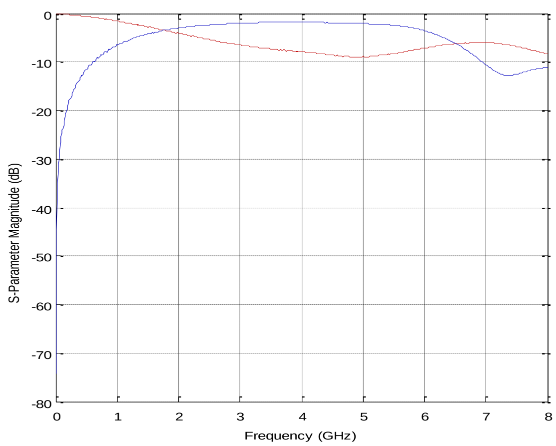


Figure 15: S21 and S31 magnitude using electronic calibration

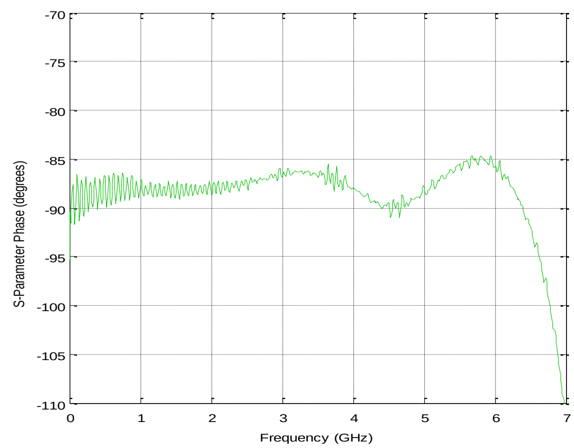


Figure 16: Phase difference using electronic calibration

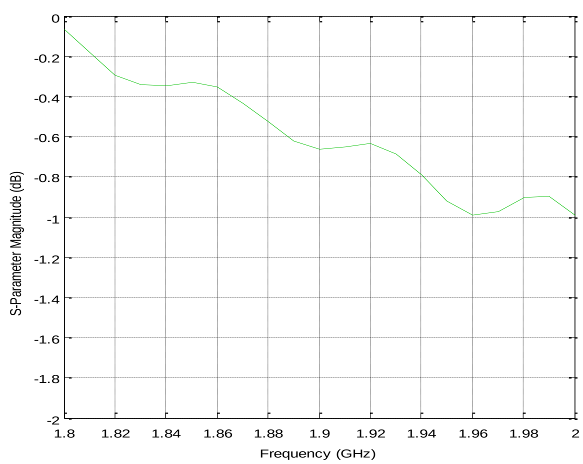


Figure 17: Amplitude difference using electronic calibration

3 Goal of Project

The goal of this project is to redesign a 90 degree hybrid coupler. The new design needs to have at least the same electric performance of the current coupler but with an overall smaller footprint and a larger bandwidth. Using a center frequency of 1.9GHz the goal was to increase the bandwidth of the target frequency. Once a new model is established a combination of Advanced Design System (ADS) and High Frequency Structural Simulator (HFSS) software tools will be used to generate accurate simulations of the circuit. Based on the circuit simulations, a layout will be created and submitted to our sponsor for tapeout and ultimate implementation in gallium arsenide. Ultimately, a prototype of the circuit could be produced so that comparative measurements can be taken on a network analyzer.

4 Theoretical Analysis

To determine the characteristic impedances of transmission lines that is used to build the quad hybrid shown in Figure 10 odd and even mode analysis is applied. As shown in Figure 11-Figure 13 the quad hybrid is cut in half symmetrically and it is configured for each mode. The total reflection coefficient at each port is determined by adding the reflection coefficients (determined from the odd and even modes) using the superposition principle as shown in equations 4.1a-d. These equations show the relationship between the S-parameters and the reflected and incident voltages.

$$V_{01}^{-} = \frac{1}{2}S_{11}^{e}V_{01}^{+} + \frac{1}{2}S_{11}^{o}V_{01}^{+} = \left[\frac{1}{2}S_{11}^{e} + \frac{1}{2}S_{11}^{o}\right]V_{01}^{+} \quad (4.1a)$$

$$V_{02}^{-} = \frac{1}{2}S_{21}^{e}V_{01}^{+} + \frac{1}{2}S_{21}^{o}V_{01}^{+} = \left[\frac{1}{2}S_{21}^{e} + \frac{1}{2}S_{21}^{o}\right]V_{01}^{+} \quad (4.1b)$$

$$V_{03}^{-} = \frac{1}{2}S_{31}^{e}V_{01}^{+} - \frac{1}{2}S_{31}^{o}V_{01}^{+} = \left[\frac{1}{2}S_{31}^{e} - \frac{1}{2}S_{31}^{o}\right]V_{01}^{+} \quad (4.1c)$$

$$V_{04}^{-} = \frac{1}{2}S_{41}^{e}V_{01}^{+} - \frac{1}{2}S_{41}^{o}V_{01}^{+} = \left[\frac{1}{2}S_{41}^{e} - \frac{1}{2}S_{41}^{o}\right]V_{01}^{+} \quad (4.1d)$$

After the TL in the middle of the hybrid coupler is divided in half it is first set as a short circuit then it is set as an open circuit. This open and short circuit is represented as a capacitor. Equation 4.2a represents the capacitive impedance for open circuit in even mode and the equation 4.2b represents the capacitive impedance for short circuit in odd mode.

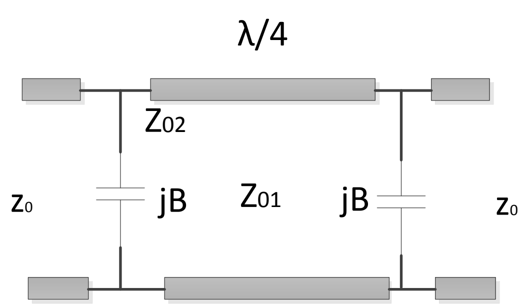


Figure 18: TL stubs

$$jB_e = \frac{1}{Z_e} = \frac{1}{-j\cot(\frac{\beta d}{2})Z_{02}} = j\,Y_{02}\tan\frac{\beta d}{2} \tag{4.2a}$$

$$jB_o = \frac{1}{Z_o} = \frac{1}{j\tan(\frac{\beta d}{2})Z_{02}} = -j\,Y_{02}cot\,\frac{\beta d}{2} \tag{4.2b}$$

$$B_s = B_e + B_o \tag{4.3}$$

From here the total capacitive impedance can be calculated by using superposition principle by simply adding the impedances from the odd and even modes as shown in equation 4.3. β is the phase constant and $\frac{d}{2}$ is the length of the sub section of micro strip line. By building the ABCD matrix all the stages of the network can be cascaded. Equation 4.4 describes this cascaded network.

Stub-1 TL (ABCD-parameters) Stub-2

↓ ↓ ↓

$$\begin{Bmatrix} V_1 \\ I_1 \end{Bmatrix} = \begin{bmatrix} 1 & 0 \\ jB_s & 1 \end{bmatrix} \begin{bmatrix} \cos\beta d & jZ_{01}\sin\beta d \\ jY_{01}sin\beta d & cos\beta d \end{bmatrix} \begin{bmatrix} 1 & 0 \\ jB_s & 1 \end{bmatrix} \begin{Bmatrix} V_2 \\ -I_2 \end{Bmatrix} \tag{4.4}$$

If the last two matrices are multiplied then the matrix found in equation 4.5 is obtained.

$$\begin{bmatrix} \cos\beta d - B_s Z_{01}\sin\beta d & jZ_{01}\sin\beta d \\ jY_{01}sin\beta d + jB_s cos\beta d & cos\beta d \end{bmatrix} \tag{4.5}$$

Next, the first matrix of equation 4.4 and equation 4.5 are multiplied to obtain the ABCD parameters of the network. The final result is given in equation 4.6.

$$\begin{bmatrix} A & B \\ C & D \end{bmatrix} = \begin{bmatrix} \cos\beta d - B_s Z_{01}\sin\beta d & jZ_{01}\sin\beta d \\ jB_s\cos\beta d - j(B_s)^2 Z_{01}\sin\beta d + jY_{01}sin\beta d + jB_s cos\beta d & -B_s Z_{01}\sin\beta d + cos\beta d \end{bmatrix} \tag{4.6}$$

Using the conversion tables, found in Appendix D, S_{11} and S_{12} (in terms of ABCD, characteristic impedance (Z_0), and psi (Ψ)) are obtained as shown in equations 4.7a and 4.7b.

$$S_{11} = \frac{A + \frac{B}{Z_0} - C\,Z_0 - D}{\Psi} \tag{4.7a}$$

$$S_{12} = \frac{2(AD - BC)}{\Psi} \tag{4.7b}$$

Where psi is in terms of ABCD and Z_0 given in equation 4.8.

$$\Psi = A + \frac{B}{Z_0}\, CZ_0 + D \tag{4.8}$$

By multiplying the top and the bottom of the fraction S_{12} and S_{11} by Z_0 a simplified version is obtained.

$$S_{11} = \frac{-C(Z_0)^2 + (A-D)Z_0 + B}{C(Z_0)^2 + (A+B)Z_0 + B} \tag{4.9}$$

$$S_{12} = \frac{2Z_0(AD-BC)}{C(Z_0)^2 + (A+B)Z_0 + B} \tag{4.10}$$

$$S_{11}^{e} = \frac{-C^e(Z_0)^2 + (A^e-D^e)Z_0 + B^e}{C^e(Z_0)^2 + (A^e+B^e)Z_0 + B^e} \tag{4.11}$$

$$S_{11}^{o} = \frac{-C^o(Z_0)^2 + (A^o-D^o)Z_0 + B^o}{C^e(Z_0)^2 + (A^o+B^o)Z_0 + B^o} \tag{4.12}$$

$$S_{12}^{e} = \frac{2Z_0(A^eD^e - B^eC^e)}{C^e(Z_0)^2 + (A^e+B^e)Z_0 + B^e} \tag{4.13}$$

$$S_{12}^{o} = \frac{2Z_0(A^oD^o - B^oC^o)}{C^o(Z_0)^2 + (A^o+B^o)Z_0 + B^o} \tag{4.14}$$

Equations 4.11 and 4.12 represent S_{11} for even (S_{11}^{e}) and odd (S_{11}^{o}) mode while equations 4.13 and 4.14 represent S_{12} for even (S_{12}^{e}) and odd mode (S_{12}^{o}). The phase constant equals $\frac{2\pi}{\lambda}$ and d is the distance which is equal to lambda quarter. When they are multiplied together, the phase of the wave is determined in terms of center frequency (fixed) and variable frequency. This relationship is given in equation 4.15. The center frequency (f_0) is determined by phase velocity and distance as shown in equation 4.16. Phase velocity depends on the effective dielectric of the material and the speed of light. Equation 4.18 determines the phase of the stubs.

$$\beta d = \frac{2\pi}{\lambda}\frac{\lambda_0}{4} = \frac{\pi}{2}\frac{f}{f_0} \tag{4.15}$$

$$f_0 = \frac{v_p}{d} = \frac{c}{(\sqrt{\epsilon_{eff}})d} \tag{4.16}$$

$$d = \frac{\lambda}{4} \tag{4.17}$$

$$\frac{\beta d}{2} = \frac{\pi}{4}\frac{f}{f_0} \tag{4.18}$$

One special case is when $f = f_0$ and the 90 degree phase is obtained as shown in equation 4.19. In equation 4.20 it is shown that tangent equals one which simplifies the equations 4.2a and 4.2b and the characteristic impedance Z_{02} is determined to be equal to Z_0, 50 ohms.

$$\beta d = \frac{\pi}{2} \tag{4.19}$$

$$\tan\left(\frac{\beta d}{2}\right) = \tan\left(\frac{\pi}{4}\right) = 1 \tag{4.20}$$

$$B_e = \frac{1}{Z_o} = Y_{02} = \frac{1}{Z_{02}} \tag{4.21}$$

$$B_O = \frac{1}{Z_o} = -Y_{02} = -\frac{1}{Z_{02}} \tag{4.22}$$

$$Z_{02} = Z_0 \tag{4.23}$$

By using the fact that the $\cos\left(\frac{\pi}{2}\right) = 0$ and the $\sin\left(\frac{\pi}{2}\right) = 1$ the ABCD matrix of equation 4.6 is simplified as shown in equation 4.24.

$$\begin{bmatrix} A & B \\ C & D \end{bmatrix} = \begin{bmatrix} -B_s Z_{01} & jZ_{01} \\ -j(B_s)^2 Z_{01} + jY_{01} & -B_s Z_{01} \end{bmatrix} \tag{4.24}$$

Since $B_s = B_e + B_o$ the ABCD matrix for even and odd mode can be written as shown in equations 4.25 and 4.26.

$$\begin{bmatrix} A^e & B^e \\ C^e & D^e \end{bmatrix} = \begin{bmatrix} -B_e Z_{01} & jZ_{01} \\ -j(B_e)^2 Z_{01} + jY_{01} & -B_e Z_{01} \end{bmatrix} \tag{4.25}$$

$$\begin{bmatrix} A^o & B^o \\ C^o & D^o \end{bmatrix} = \begin{bmatrix} -B_o Z_{01} & jZ_{01} \\ -j(B_o)^2 Z_{01} + jY_{01} & -B_o Z_{01} \end{bmatrix} \tag{4.26}$$

By substituting $B_e = \frac{1}{Z_{02}} = \frac{1}{Z_0}$, $B_o = -\frac{1}{Z_{02}} = -\frac{1}{Z_0}$, and $Y_{01} = \frac{1}{Z_{01}}$ into the above equations a simplified version of even and odd ABCD matrix shown in equations 4.27 and 4.28 is derived.

$$\begin{bmatrix} A^e & B^e \\ C^e & D^e \end{bmatrix} = \begin{bmatrix} -\frac{Z_{01}}{Z_0} & jZ_{01} \\ -\frac{jZ_{01}}{{Z_0}^2} + j\frac{1}{Z_{01}} & -\frac{Z_{01}}{Z_0} \end{bmatrix} \tag{4.27}$$

$$\begin{bmatrix} A^o & B^o \\ C^o & D^o \end{bmatrix} = \begin{bmatrix} \frac{Z_{01}}{Z_0} & jZ_{01} \\ \frac{jZ_{01}}{{Z_0}^2} + j\frac{1}{Z_{01}} & \frac{Z_{01}}{Z_0} \end{bmatrix} \quad (4.28)$$

Now, by substituting each ABCD parameter from the even mode into the S-parameter even mode and setting S_{11}^e equal to zero, assuming zero reflected power, Z_{01} characteristic impedance of TL is determined in terms of the Z_0 characteristic impedance shown in equation 4.30.

$$S_{11}^e = \frac{-C^e(Z_0)^2 + (A^e - D^e)Z_0 + B^e}{C^e(Z_0)^2 + (A^e + B^e)Z_0 + B^e} = 0 \quad (4.29)$$

$$-C^e(Z_0)^2 + (A^e - D^e)Z_0 + B^e = 0 \quad (4.29a)$$

$$(-\frac{jZ_{01}}{{Z_0}^2} + j\frac{1}{Z_{01}})(Z_0)^2 + \left(-\frac{Z_{01}}{Z_0} - \frac{Z_{01}}{Z_0}\right) Z_0 + jZ_{01} = 0 \quad (4.29b)$$

$$-jZ_{01} + j\frac{1}{Z_{01}}(Z_0)^2 - 2Z_{01} + jZ_{01} = 0 \quad (4.29c)$$

$$j(Z_0)^2 = 2\,(Z_{01})^2 \quad (4.29d)$$

$$|j(Z_0)^2| = |2\,(Z_{01})^2| \quad (4.29e)$$

$$(Z_0)^2 = 2\,(Z_{01})^2 \quad (4.29f)$$

The magnitude of the impedances in equation 4.29 determines the characteristic impedance in equation 4.30.

$$Z_{01} = \frac{Z_0}{\sqrt{2}} \quad (4.30)$$

In the odd mode S_{11}^o was set to zero and the same result was obtained. By deriving the characteristic impedances, each transmission line can be designed using the equations of microstrip transmission lines. The other S-parameters are computed using $S_{11}^o, S_{11}^e, S_{12}^e, S_{12}^o$ which are previously defined as shown in equations 4.31 to 4.35.

$$S_{12} = \frac{S_{12}^e}{2} + \frac{S_{12}^o}{2} = S_{21} \quad (4.31)$$

$$S_{13} = \frac{S_{12}^e}{2} - \frac{S_{12}^o}{2} = S_{31} \quad (4.32)$$

$$S_{14} = \frac{S_{11}^{e}}{2} - \frac{S_{11}^{o}}{2} = S_{41} \quad (4.33)$$

$$S_{24} = \frac{S_{12}^{e}}{2} - \frac{S_{12}^{o}}{2} = S_{42} \quad (4.34)$$

$$S_{34} = \frac{S_{12}^{e}}{2} + \frac{S_{12}^{o}}{2} = S_{43} \quad (4.35)$$

5 New design approach

The analytical analysis determined the characteristic impedances of the TLs are shown in Figure 19. This led to the design of the microstrip line to determine the length, width, and height by assuming an infinitesimal TL thickness.

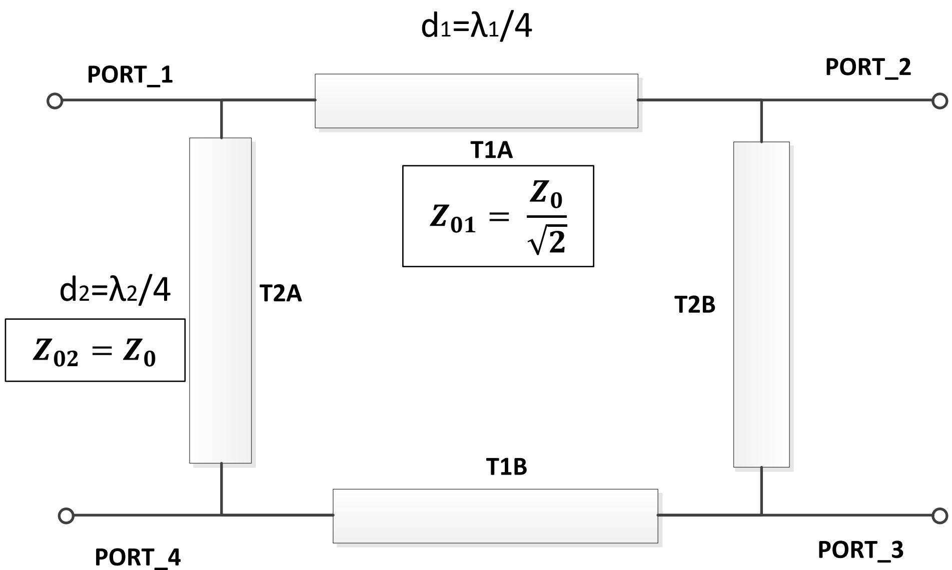


Figure 19: The new design quad hybrid

$$\beta d = \frac{2\pi}{\lambda}\frac{\lambda_0}{4} \tag{5.1a}$$

$$\lambda_0 = \lambda \tag{5.1b}$$

$$\beta d = \frac{\pi}{2} \tag{5.1c}$$

$$d_n = \frac{\lambda_n}{4} \tag{5.1d}$$

$$d_1 = \frac{\lambda_1}{4} = \frac{c}{(\sqrt{\epsilon_{eff1}})f_0} \tag{5.2a}$$

$$d_2 = \frac{\lambda_2}{4} = \frac{c}{(\sqrt{\epsilon_{eff2}})f_0} \tag{5.2b}$$

The characteristic impedance of the transmission T1A is $Z_{02} = Z_0$. Characteristic impedance Z_0 was decided to be set to 50 ohm because this value is the most commonly used in industry. For this process the dielectric constant was set to be $\epsilon_r = 6$. Next, to determine the distance (d) in equation 5.2 the effective dielectric has to be computed using equation 5.3. This equation depends on the height to width ratio which is determined using the graph in Figure 20. The lines are drawn for different values of dielectric constants, in this case 6 is used. As shown in the graph, the width to height ratio is less than 2, this condition determines that the equation 5.3 is used to compute this ratio.

$$\epsilon_{eff2} = \frac{\epsilon_r+1}{2} + \frac{\epsilon_r+1}{2}\left[\left(1+12\frac{h_2}{w_2}\right)^{-1/2} + 0.04\left(1-\frac{w_2}{h_2}\right)^2\right] \qquad (5.3)$$

$$\epsilon_r = 6\,, \qquad Z_f = \sqrt{\frac{\mu_0}{\epsilon_0}} = 376.8\,\Omega\,, \qquad Z_{02} = Z_0 = 50\,\Omega\,, \qquad \frac{w_2}{h_2} \leq 2 \qquad (5.4a/b/c/d)$$

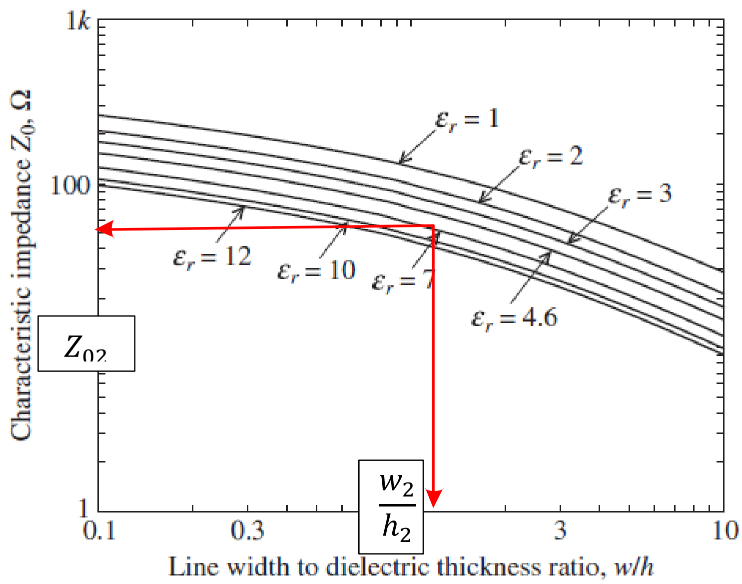


Figure 20: Microstrip Z_0 as function of w/h (Z_{02})

$$\frac{w_2}{h_2} = \frac{8e^A}{e^{2A}-2} \qquad (5.5)$$

In order to solve for the width and height of the second transmission line (T2A &T2B) the factor A must be determined first. This is accomplished by substituting the dielectric constant, effective impedance, and characteristic impedance. The factor A was calculated using equation 5.6.

$$A = 2\pi \frac{Z_0}{Z_f}\sqrt{\frac{\epsilon_r+1}{2}} + \frac{\epsilon_r-1}{\epsilon_r+1}\left(0.23 + \frac{0.11}{\epsilon_r}\right) \tag{5.6}$$

The final results for these transmission lines are shown in equation 5.7.

$$A = 1.737215, \quad \frac{h_2}{w_2} = 1.5021085, \quad h_2 = 0.15\mu m, \quad w_2 = 0.225163\mu m \qquad \text{(5.7a/b/c/d)}$$

The effective dielectric constant of second TL is determined by substituting the dielectric constant and the height to width ratio to equation 5.5. By using the equation 5.2 the length of TL is calculated. The results are shown below in equation 5.8.

$$f_0 = 1.9\ GHz, \quad \epsilon_{eff2} = 4.336012, \quad d_2 = 7.58478\ cm \qquad \text{(5.8a/b/c)}$$

To determine the width to height ratio of the first TL (T1A & T1B) the same procedure is followed. As shown in Figure 21, the approximate value of width to height is determined and since this ratio is greater than two, equation 5.10 is used to compute it.

$$\epsilon_r = 6, \quad Z_f = \sqrt{\frac{\mu_0}{\epsilon_0}} = 376.8\ \Omega, \quad Z_{01} = \frac{Z_0}{\sqrt{2}} = \frac{50}{\sqrt{2}}\Omega = 35.2553\ \Omega, \quad \frac{w_1}{h_1} \geq 2 \qquad \text{(5.9a/b/c/d)}$$

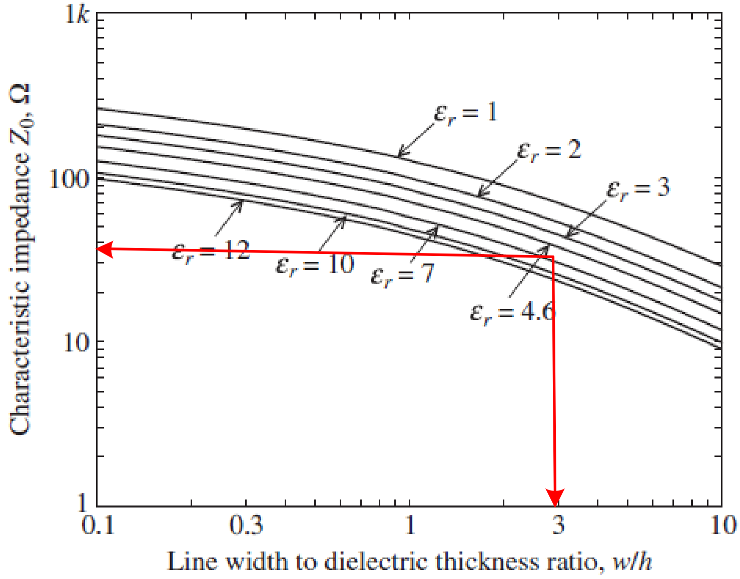


Figure 21: Microstrip Z_0 as function of w/h (Z_{01})

The equation below computes the width to height ratio, but first the B factor in equation 5.11 is calculated and substituted into this formula.

$$\frac{w_1}{h_1} = \frac{2}{\pi}\left\{B - 1 - \ln(2B - 1) + \frac{\epsilon_r - 1}{2\epsilon_r}\left[\ln(B - 1) + 0.39 - \frac{0.61}{\epsilon_r}\right]\right\} \tag{5.10}$$

$$B = \frac{\pi Z_f}{2Z_0\sqrt{\epsilon_r}} \tag{5.11}$$

$$B = 6.834396\,, \quad \frac{h_1}{w_1} = 2.642155\,, \quad h_1 = 0.15\mu m\,, \quad w_1 = 0.39623\mu m \tag{5.12}$$

After the height to width ratio was computed the effective dielectric constant was calculated next. This value and the center frequency are used to compute the length of the TL.

$$\epsilon_{eff1} = 4.561981, \qquad d_1 = 7.3925cm \tag{5.13}$$

All these values lead to a new design which was implemented in ADS as shown in Figure 22.

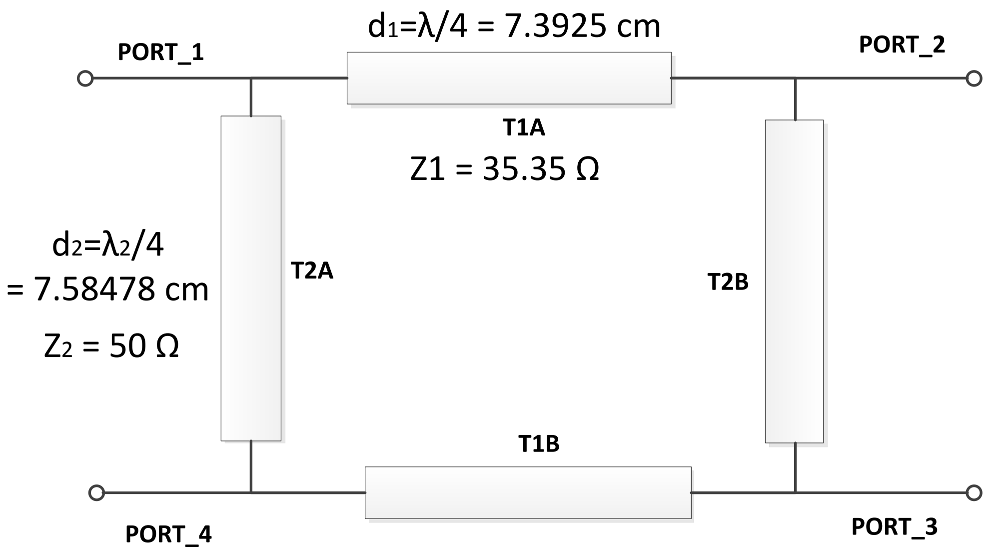


Figure 22: The quad hybrid implemented in ADS

The size of the hybrid coupler in Figure 22 is too large to be implemented as a chip for cell phone applications. Therefore the TL was converted to lumped components by setting the ABCD parameters for the transmission line and Pi network equal to each other as shown in Figure 23.

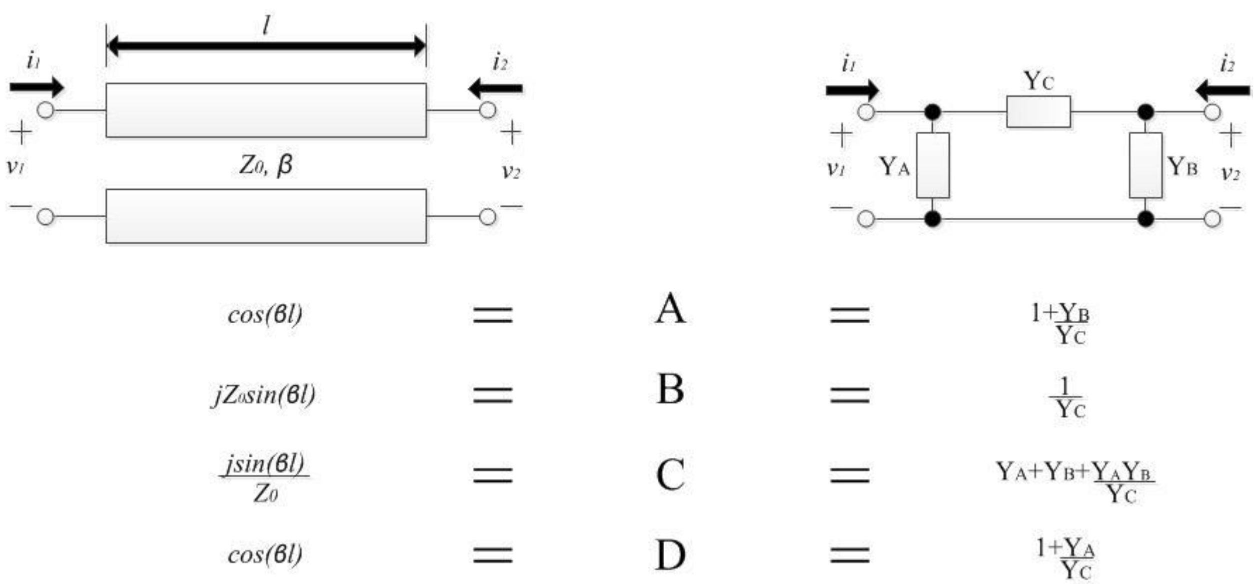


Figure 23: TL to Lumped component conversion

The transmission line model for T1A and T1B is converted to its lumped equivalent as shown in Figure 24. The lumped equivalent circuit is shown in Figure 25. This is a Pi-network with two capacitors and one inductor with values of C=2.393pF and L = 2.961nH.

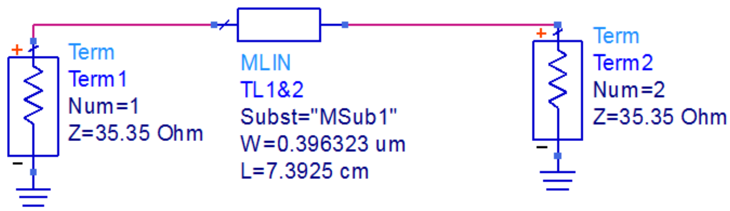


Figure 24: TL representation for T1A & T1B

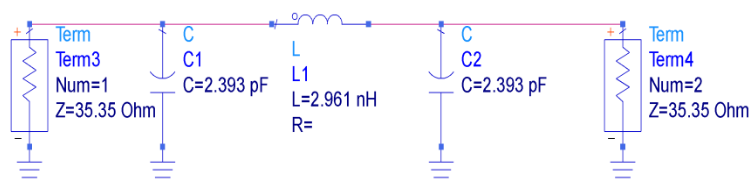


Figure 25: Lumped equivalent for TL T1A & T1B

The transmission line shown in Figure 26 is the representative model for the T2A and T2B. The TL is converted to lumped equivalent using the procedure shown in **Error! Reference source not found.**. The lumped equivalent circuit is shown in Figure 27. Again, this is a Pi-network with two capacitors and one inductor with values of C=1.675pF and L = 4.188nH.

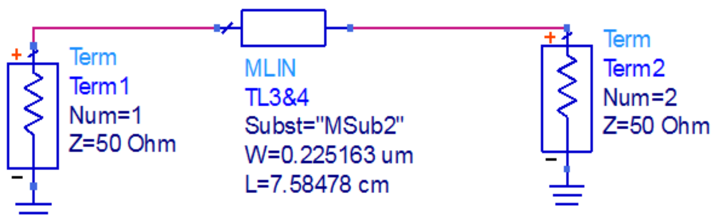


Figure 26: TL representation for T2A & T2B

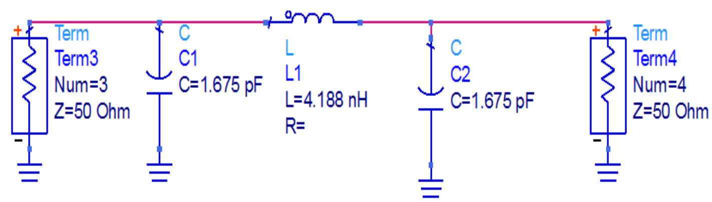


Figure 27: Lumped equivalent for TL T2A & T2B

After the conversion has been done all the Pi-networks are connected together to obtain the circuit shown in Figure 28. This circuit is simplified by combining the parallel corner capacitors.

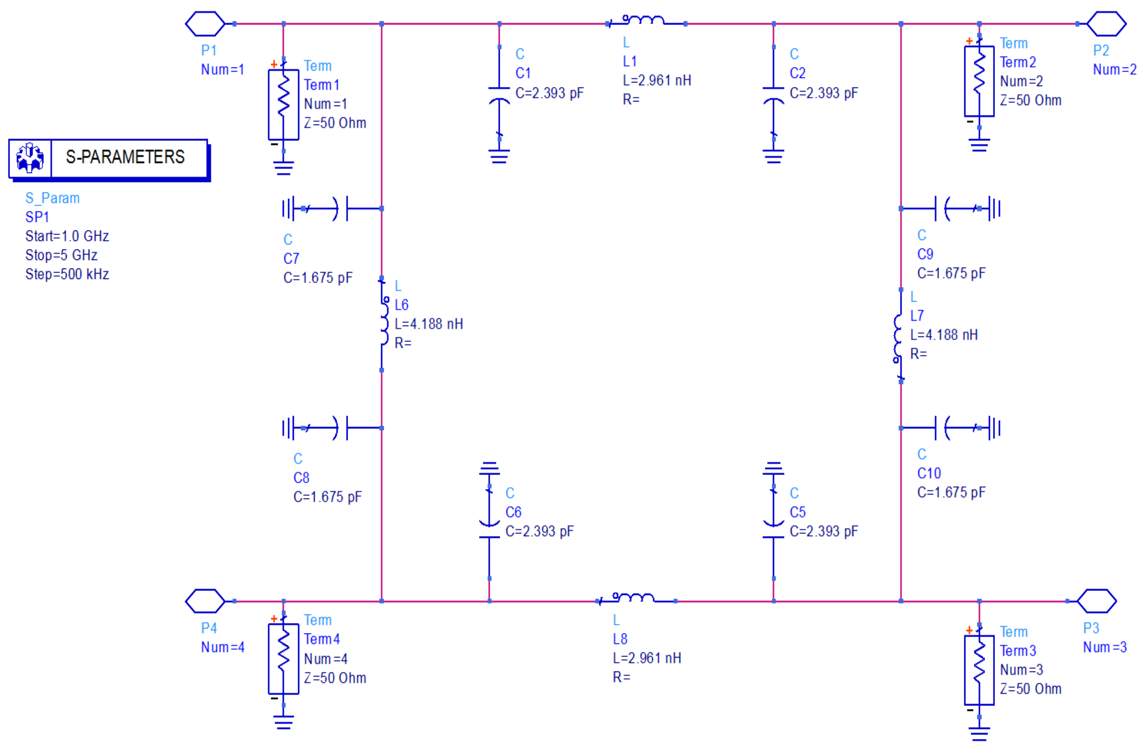


Figure 28: The lumped equivalent circuit of the quad hybrid

The simplified model of the lumped equivalent is shown in Figure 29. This model can be implemented as a chip because its size is smaller and the output results show a greater bandwidth for the 90 degree phase and the 3 dB attenuation at the output ports.

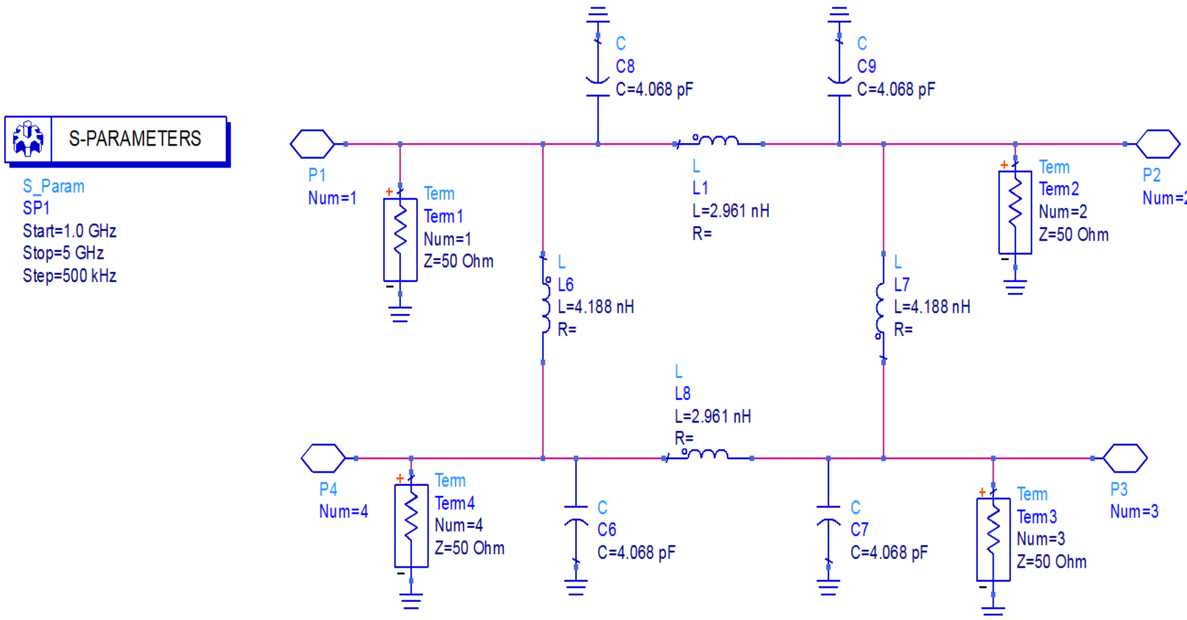


Figure 29: Final Lumped equivalent circuit

6 Simulations

6.1 Simulation and data analysis tools

There are many tools used for analysis including HFSS, ADS, and Matlab. These programs provide the necessary tools that enable models to be generated and simulations to be run on a wide variety of applications.

6.1.1 High Frequency Structural Simulator (HFSS)

"ANSYS HFSS software is the industry-standard simulation tool for 3-D full-wave electromagnetic field simulation and is essential for the design of high-frequency and high-speed component design." [9] It has the ability to utilize either the finite element method (FEM) or the integral equation method to generate meshes as a way of approximating solutions of partial differential equations.

Among its capabilities, is the ability to plot electromagnetic fields (both in the near and far field), many different parameters (S, Z, ABCD), and radiation patterns. HFSS also shares the same mother company (ANSYS) as SolidWorks and is very similar in its modeling capabilities. The ability to import designs from SolidWorks as well as export solutions to a wide variety of programs makes it very versatile. Overall, it is an extremely useful program that provides a lot of pertinent and usable data for high frequency applications.

6.1.2 Advanced Design Systems (ADS)

Another circuit modeler is ADS. ADS is a circuit simulator with abilities ranging from simulation of simple circuits to the ability of arranging complex layouts with many different layers. A great feature found in ADS is the 2.5-D simulator called momentum. Once a layout is created, momentum is used to simulate the design's performance almost like it would on a real die, with the exception of accounting for parasitics. Like HFSS it also uses finite element method to generate meshes and solve circuits.

Solutions from ADS range from simple parameter plots to smith charts and has proven to be another industry standard.

6.1.3 Matlab

Matlab is a mathematical program useful in all sorts of computational analyses. Moreover it “is a high-level language and interactive environment for numerical computation, visualization, and programming.” [10] It can analyze data, plot charts, run programs written in a C based language and do almost anything else mathematical. Algorithms can be generated and specific functions created, and data can be imported and analyzed.

7 Computational Analysis

7.1 ADS Simulations

With the ideal design finalized in section 5, the following step was to simulate the design to verify the theoretical results. The simulation for the ideal design was done in the previously introduced simulator ADS. However, the desired center frequency was set to 1.9GHz to better observe the bandwidth behavior of the device under test. The simulation was done as a frequency sweep ranging from 1.0GHz to 5.0GHz. The final design is demonstrated in Figure 30.

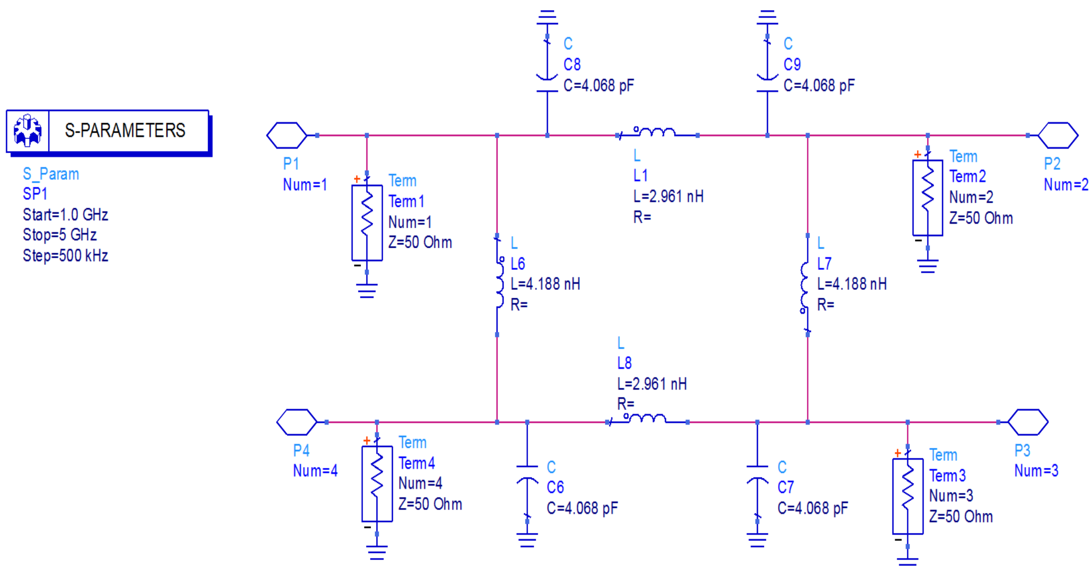


Figure 30: Lumped equivalent circuit

With the simulation complete, ADS can provide the previously discussed S-parameter plots. The first important result examined is the phase difference between port 2 and port 3. This is accomplished by plotting the phase difference between S(2,1), which refers to the output at port 2 with respect to port 1, and S(3,1), which refers to the output at port 3 with respect to port

1. The result of this phase difference is shown in Figure 31 where the plot shows degrees vs. frequency. The plot shows that the 90 degree phase difference is maintained for range of 0.4GHz

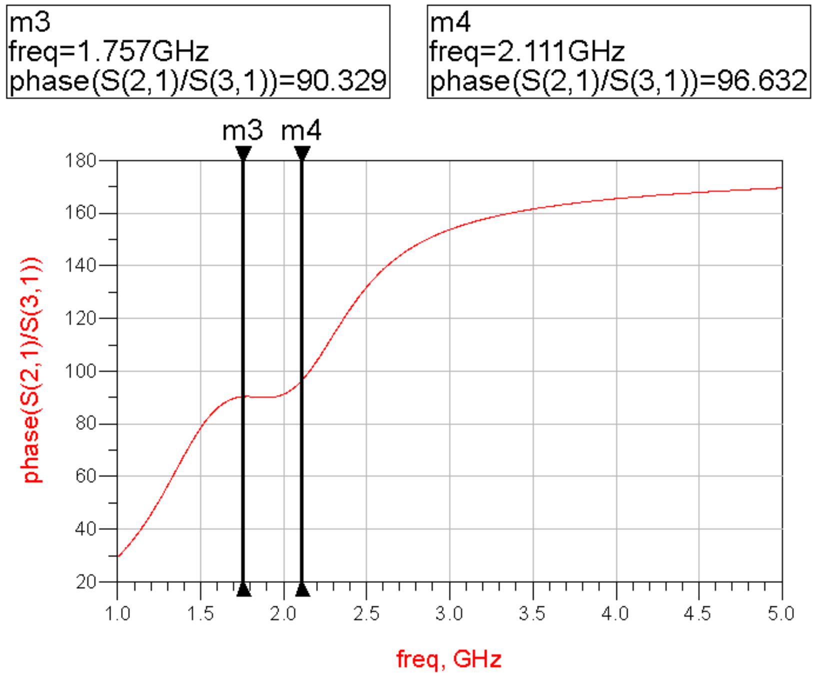


Figure 31: The phase difference of two output ports

Next, the magnitude of the output signal is examined. As previously discussed, half of the power should be observed at port 3 and port 2. This is done by simply plotting the results S(2,1) and S(3,1).

In Figure 32 the results are shown in terms of magnitude (dB) versus frequency. The plot shows the 3dB attenuation, or half of the power, of the output ports and these are maintained for the same range of frequencies.

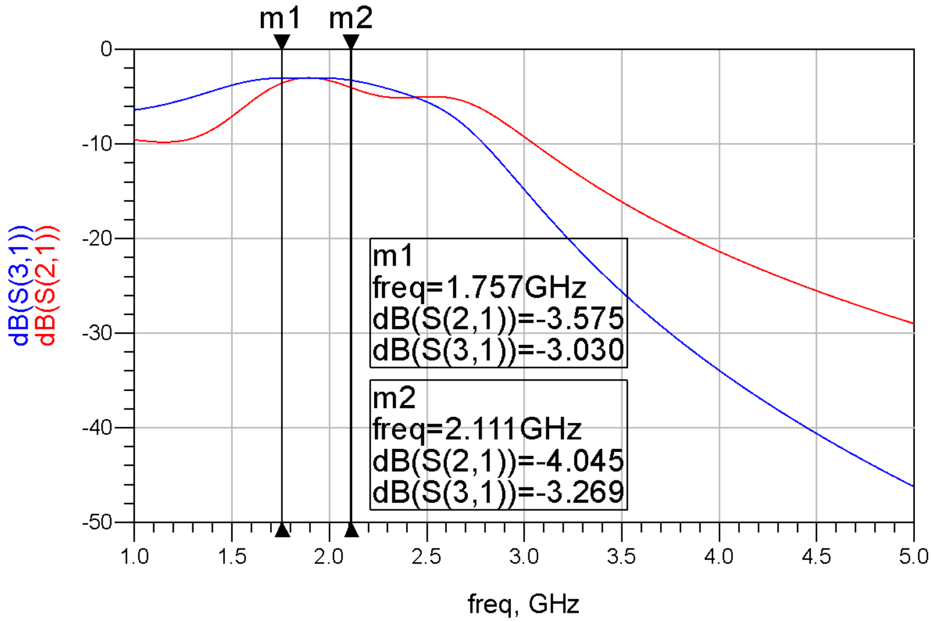


Figure 32: The 3dB attenuation of the two output ports

The 90 degree phase difference and 3dB attenuation are maintained for 0.4GHz. To calculate for the bandwidth we use the equation:

$$BW = \frac{f_u - f_l}{f_c} * 100\% \tag{7.1}$$

Where f_u refers to the upper limit for which the system is functional (in this case 2.111GHz), f_l is the lower limit (in this case 1.757GHz), and finally f_c was the initially determined center frequency of 1.9GHz. Inserting these values into the equation yields a bandwidth of 18.42%. These results confirm the theoretical analysis, in addition to accomplishing one of the project goals.

7.2 Capacitor and spiral inductor calculations

For the schematic in Figure 29 the capacitors and spiral inductors needed to be calculated so that their physical parameters are determined.

In order to find the values for the capacitor of 4.068pF the parallel plate capacitor equation was used [11].

$$C = \frac{\epsilon_0 \epsilon_r A}{d} \tag{7.2}$$

In equation 7.2, C is the capacitance, ϵ_0 is the permittivity of free space (8.85E-12), ϵ_r is the relative dielectric constant, A is the area of the plates, and *d* is the distance between the parallel plates. The relative dielectric constant depends completely upon the material, in the case of this new circuit the relative dielectric was set to 6. Plugging in all of the known values (capacitance, permittivity, dielectric) the only variables left are the area, *A*, and the distance, *d*. In an effort to reduce the total area, the equation above can be analyzed to see that the smallest *d* would minimize the area. Next, according to the layout process used by Skyworks (PHEMT7), the thinnest layer that can be made is one of 0.15um. Using this value for *d* the one variable remaining is *A*.

$$A = \frac{Cd}{\epsilon_0 \epsilon_r} = \frac{(4.068\text{pF})(.15um)}{(8.85*10^{-12})(6.4)} = 1.07733 * 10^{-8} m^2 \tag{7.3}$$

$$\sqrt{1.07733 * 10^{-8} m^2} = 103.795um \tag{7.4}$$

After the area is found in equation 7.3, the square root is taken so that the dimensions of the caps are found in equation 7.4 to be 104um.

Next, the spiral inductors' dimensions were calculated for the values of 4.188nH and 2.961nH. There are three main spiral inductor equations. They are the Modified Wheeler, Current Sheet, and the Monomial Fit [12]. They all have their different merits and usually yield similar results. For these calculations an integrated spiral inductor calculator from Stanford University was used which required some starting values to be known [13]. These values included the number of turns (n), turn spacing (s), turn width (w), and the outer diameter (d_{out}). According to Skywoks' PHEMT7 process the turn spacing and width has certain minimum values that are set in place. In this case the minimum is 6um for both. Instead of choosing the minimum the values, the turn width and the turn spacing were both set to 10um. With regards to the outer diameter, the goal of reducing the total size of the circuit was considered, and because of this the diameter was estimated to be at 300um. The rest was experimenting with the number of turns to see what the optimal number was to obtain the inductance values of 4.188nH and 2.961nH. In order to reduce the total number of spiral inductors on the board, which would have been four, two inductor values were combined into a single one, resulting in a total of two. Then a line feeding into the inductor would split it in two and work in the same way. After tuning the amount of turns to get the desired result it was found that the ideal number was approximately six.

The calculated spiral inductor values, as well as the capacitor values were then utilized in sections 7.3 and 7.4 in order to complete the 3D model.

7.3 Step By Step ADS Layout

This section provides step by step instructions to create the layout for the new design that was discussed in the previous sections. The program used in this section is Agilent's ADS which was also introduced in previous sections for simple 2-D schematics. This section makes use of ADS' layout functionality as well as a 2.5D simulator momentum.

Step 1: The first step is to start the ADS working environment.

Figure 33: Start Agilent's Advanced Design System

Step 2: Once the program fully opens, the first thing to do is create a workspace, where the design will be implemented. To do so, click on **File>New>Workspace** as is shown in the figure below.

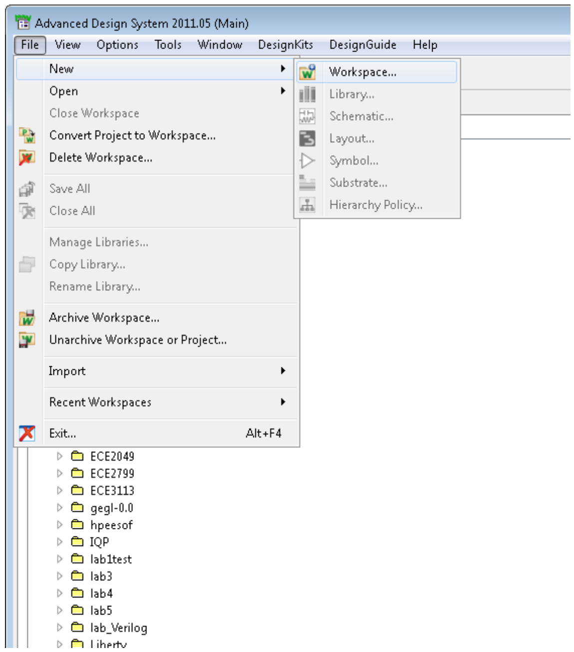


Figure 34: Create a new workspace

Step 3: When creating a new workspace, you will be prompted to specify a name, as well as the location where the file will be saved. If it is not already in the correct location, use the browse button to select the desired location to store all ADS files. Click **next** when done.

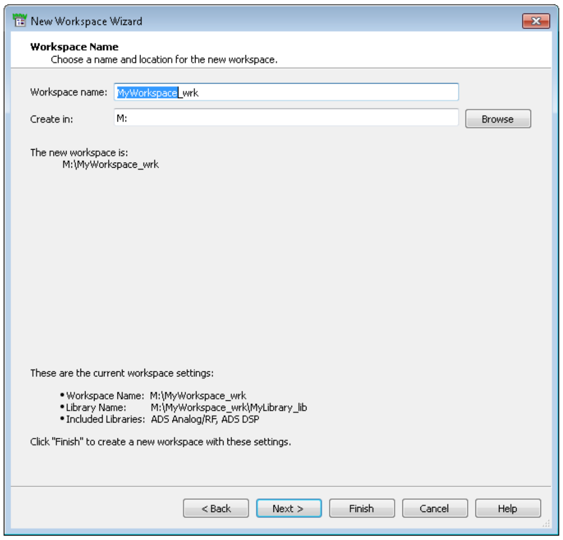


Figure 35: Chose appropriate location to store all files

Step 4: ADS incorporates a very rich library of components and templates for designs. Additional libraries are available on the web, which can be included in the design at this point. Specifically, when transferring designs libraries may be transferred and should be included at this point. Click **next** when done.

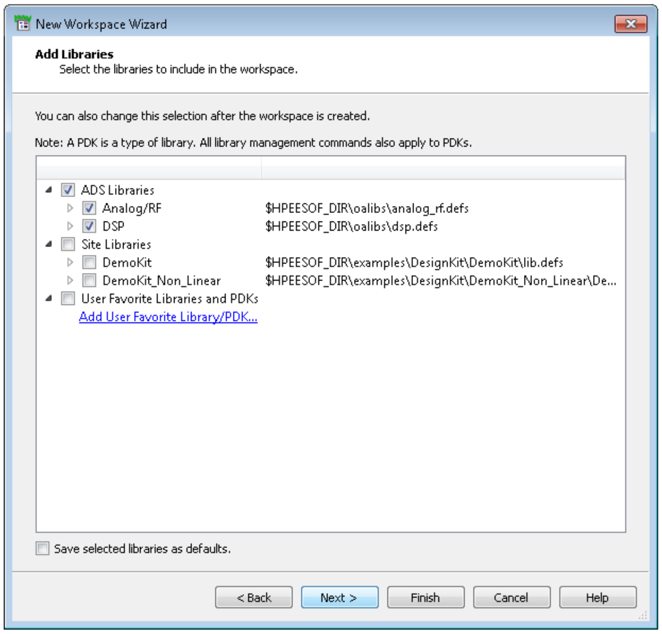


Figure 36: Select the desired libraries

Step 5: ADS stores all designs under libraries. It is important to name the library accordingly to the specific design for ease of later use. This will become important later. Click **next** to proceed.

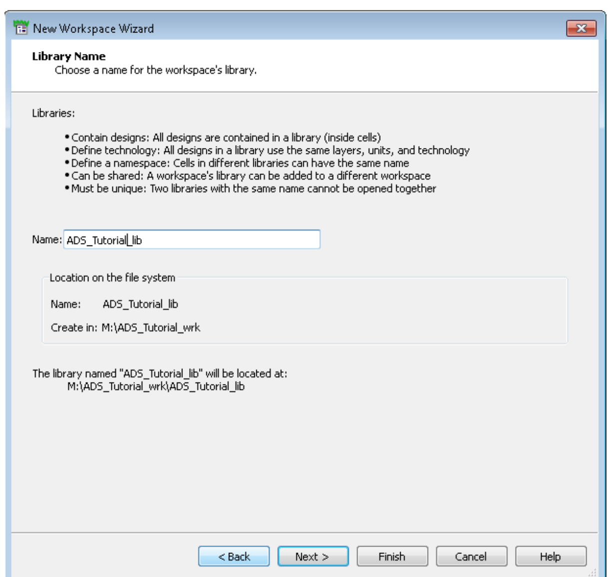


Figure 37: Create your own library

Step 6: In this window, the units are specified for all schematics/layouts created in this workspace. Choose the units of your best liking and click **finished** when done. For this design, microns were the unit of preference as shown below.

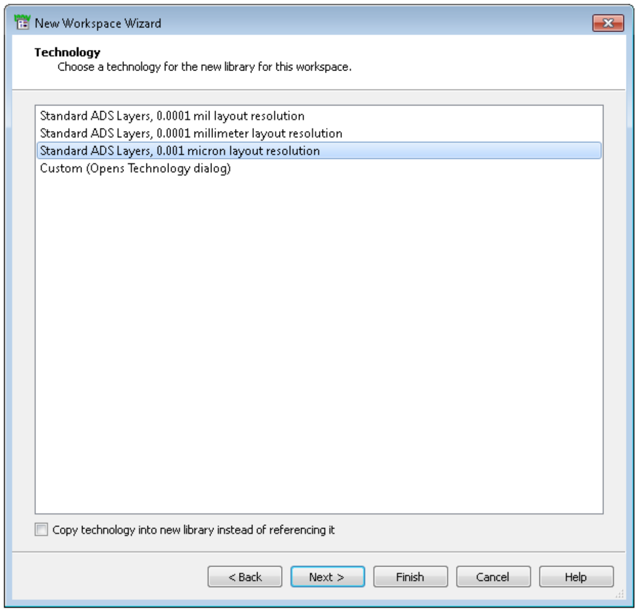


Figure 38: Select default units

Step 7: A new window similar to that of the one on startup will appear with the exception that this will be empty since nothing has been created yet. Go to **File>New>Layout** where you will be prompted to name the layout. A new window will open like the one shown below.

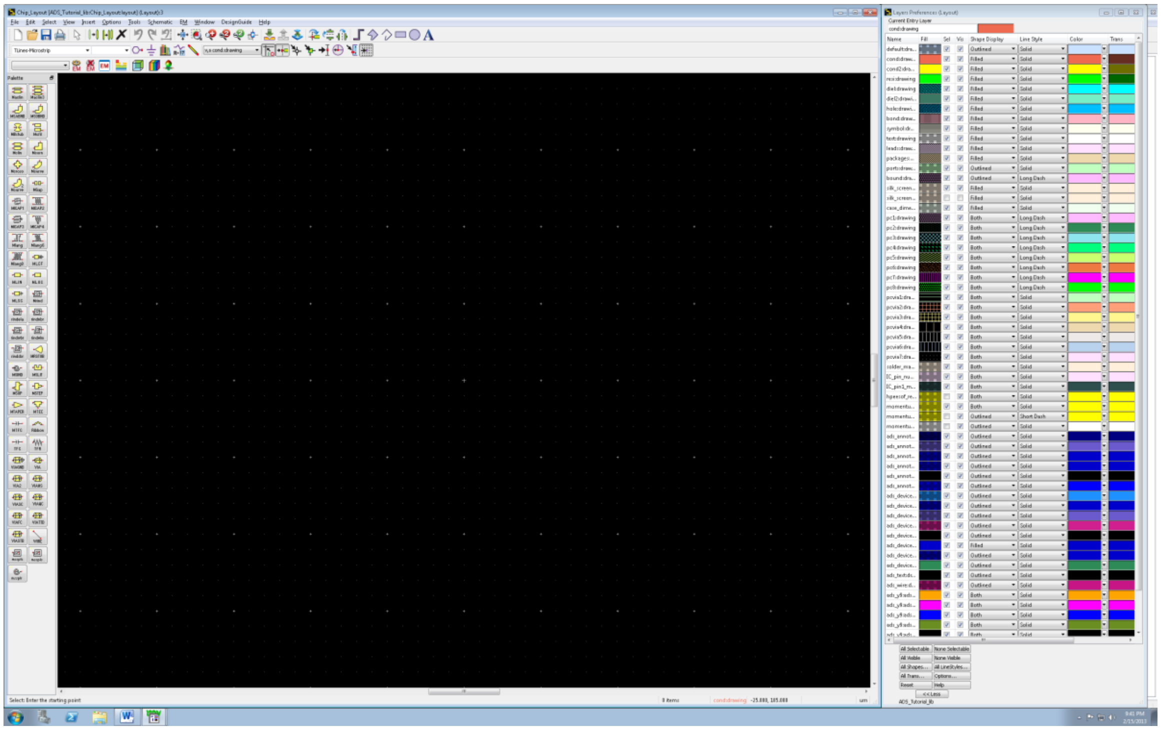


Figure 39: Starting new layout

Step 8: The ADS layout environment works with a systems of layers. Each layer can be considered as conducting metals. Before proceeding, go to **Options>Technology>Layer Properties** and the window shown in Figure 40. Click Add Layer.

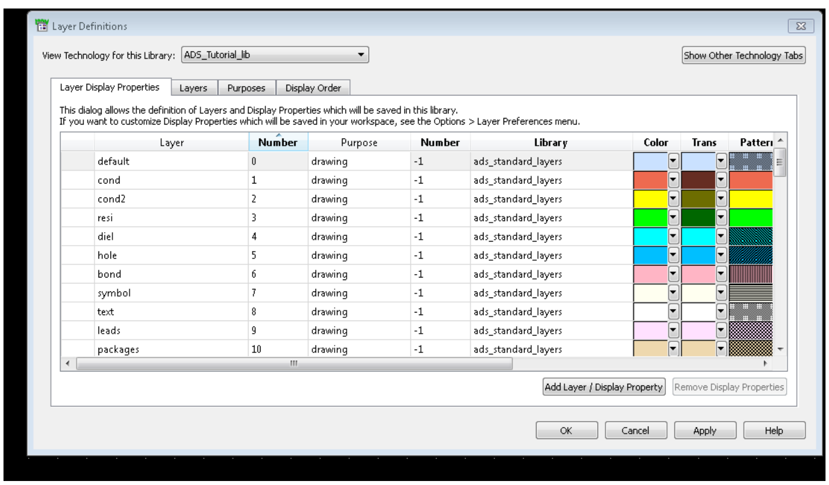


Figure 40: Creating layers

Step 9: The window shown in Figure 41 will appear where the user is prompted to specify the name of a layer along with a variety of different parameters. Each component in the design needs to be different layers (i.e. ports, inductors, capacitors). Specify the name and properties of the layers and press ok when finished. Then select ok again in the previous window. Again, it is important to note that each conductor will need to be mapped as a layer so make sure all the required conductors from the design are specified.

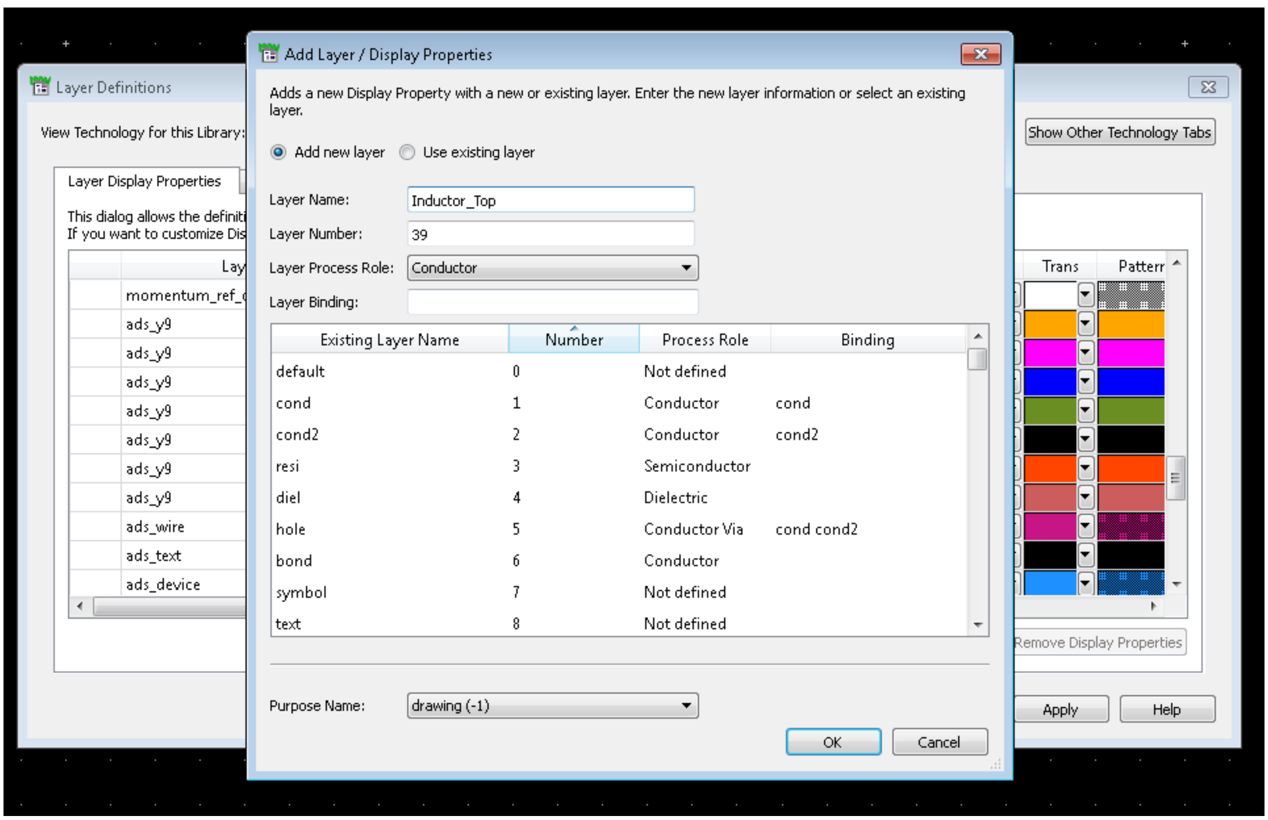


Figure 41: Layer properties

Step 10: It is good practice to specify different colors, transparency and patterns to all the layers. This will be useful especially when creating complex designs, since it will help tell the conducting layers apart one from another. Also, it makes it easy for the designer to follow the design through since no design is created in a single attempt. By going to **Options > Layer Preferences** the window shown in Figure 42 will appear. This shows all the layers that have been defined along with their properties, where the designer can change the preferences of each layer at will.

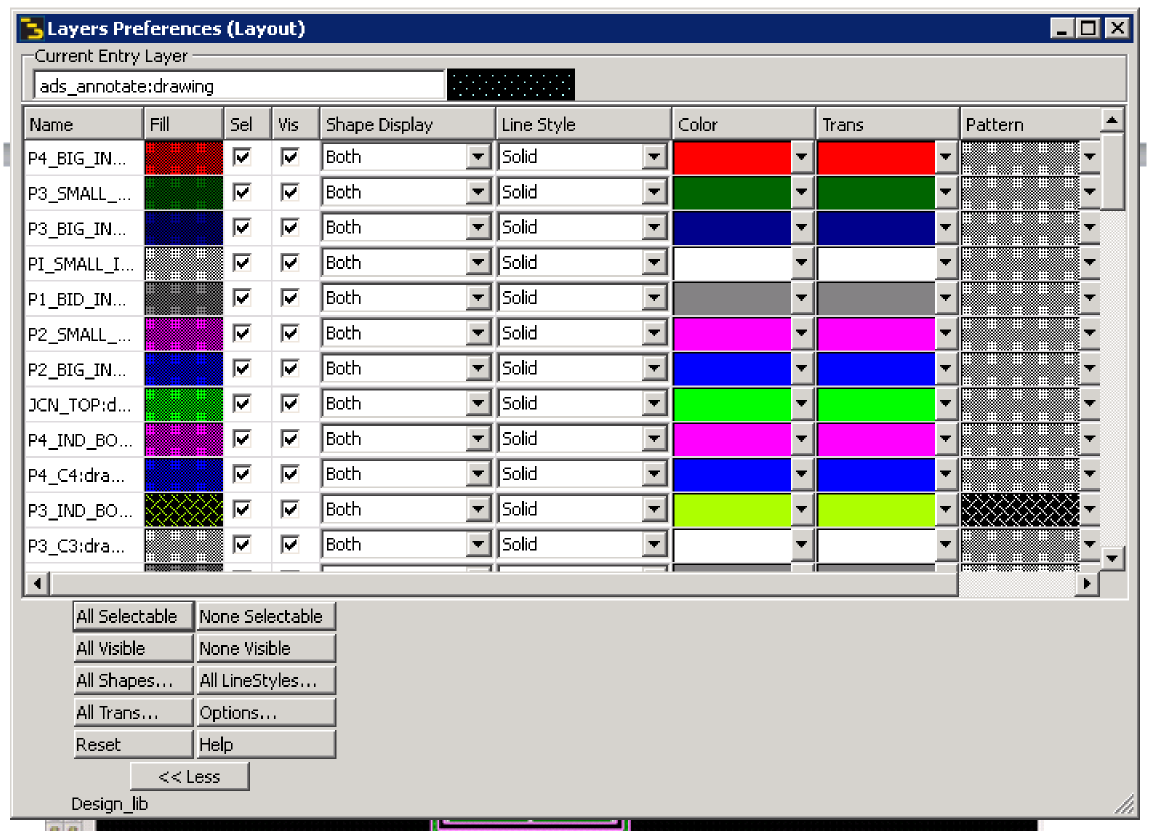


Figure 42: Layer preferences window

Step 11: ADS features a variety of ways to implement a design including a respectable library of components. However, since most printed board designs require custom components this tutorial shows how to design the component using traces. To begin use the “insert path” tool available from the toolbars. Alternatively, it the tool is under **Insert>Path**. To use this tool, click the cursor to set the starting point of the trace. Then, wherever the cursor is moved the trace will follow. After the starting point is specified, the cursor indicates the x and y displacement from that point. You can also specify the width of the trace, whether the turns should be a curve, or squared in the popup window. This is all shown in Figure 43 below.

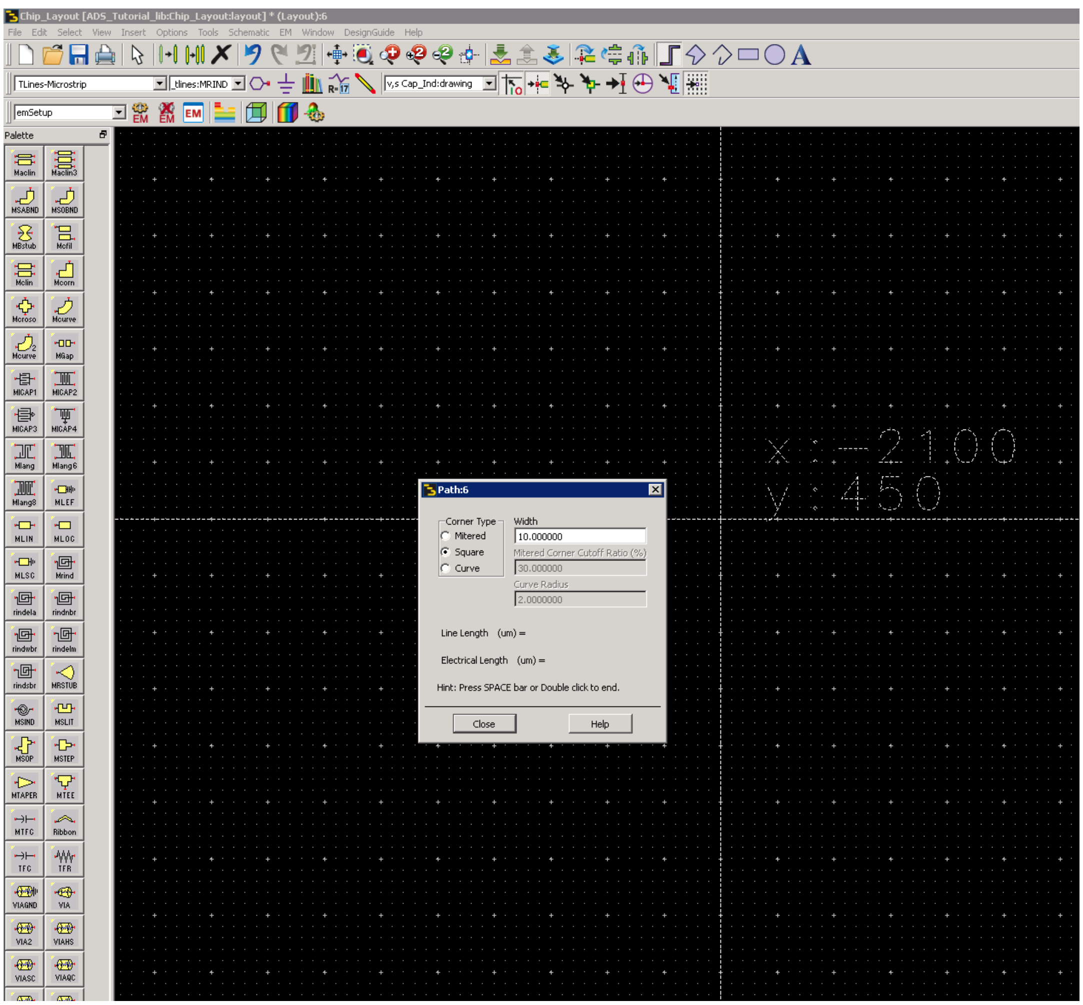


Figure 43: Insert path

Step 12: Create the spiral inductor by creating several turns clicking to indicate every corner. When the final point is reached, simply click twice over the same point to finish the trace. The finalized inductor is shown in Figure 44, where the design of the inductor was determined from the previous sections. When the path has been completed, double click anywhere on the path to get the window shown below. In this window, specify the correct layer for the path. For this design this inductor will be the inductor located at the top of the chip so it belongs to the layer Inductor_Top. The design of this inductor has been previously discussed, which implements two separate inductors required by the design. By using one large spiral inductor and splitting it at a point in between the end points a value of an inductor is obtained, while a connection at the end point of the spiral creates a separate inductor with a different value hence implementing two inductors with a single spiral.

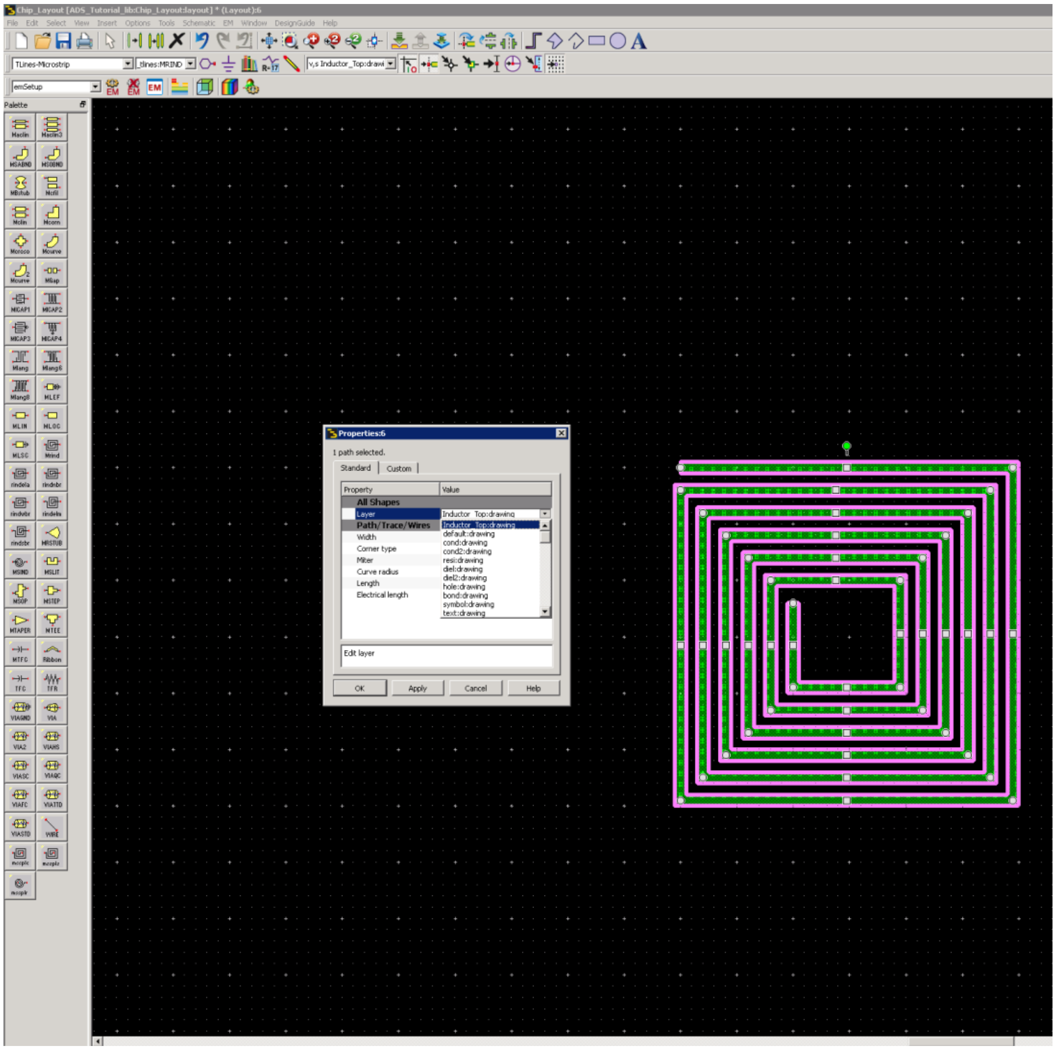


Figure 44: Creating an inductor

Step 13: The capacitors for this circuit will be implemented by using two parallel plates separated over a later specified distance. First the top plate of the capacitor is created using the "Insert Rectangle" tool found again in the toolbars or alternative under **Insert>Rectangle**. Similar to the path, click to indicate the starting point of the rectangle and move the cursor until the right dimensions display on the screen. The dimensions can always be changed through the window shown in Figure 45 which is accessed by double clicking on the layer. Again, remember to specify the correct layer for the top of the capacitor. In this circuit this layer is assigned the name Cap_Top.

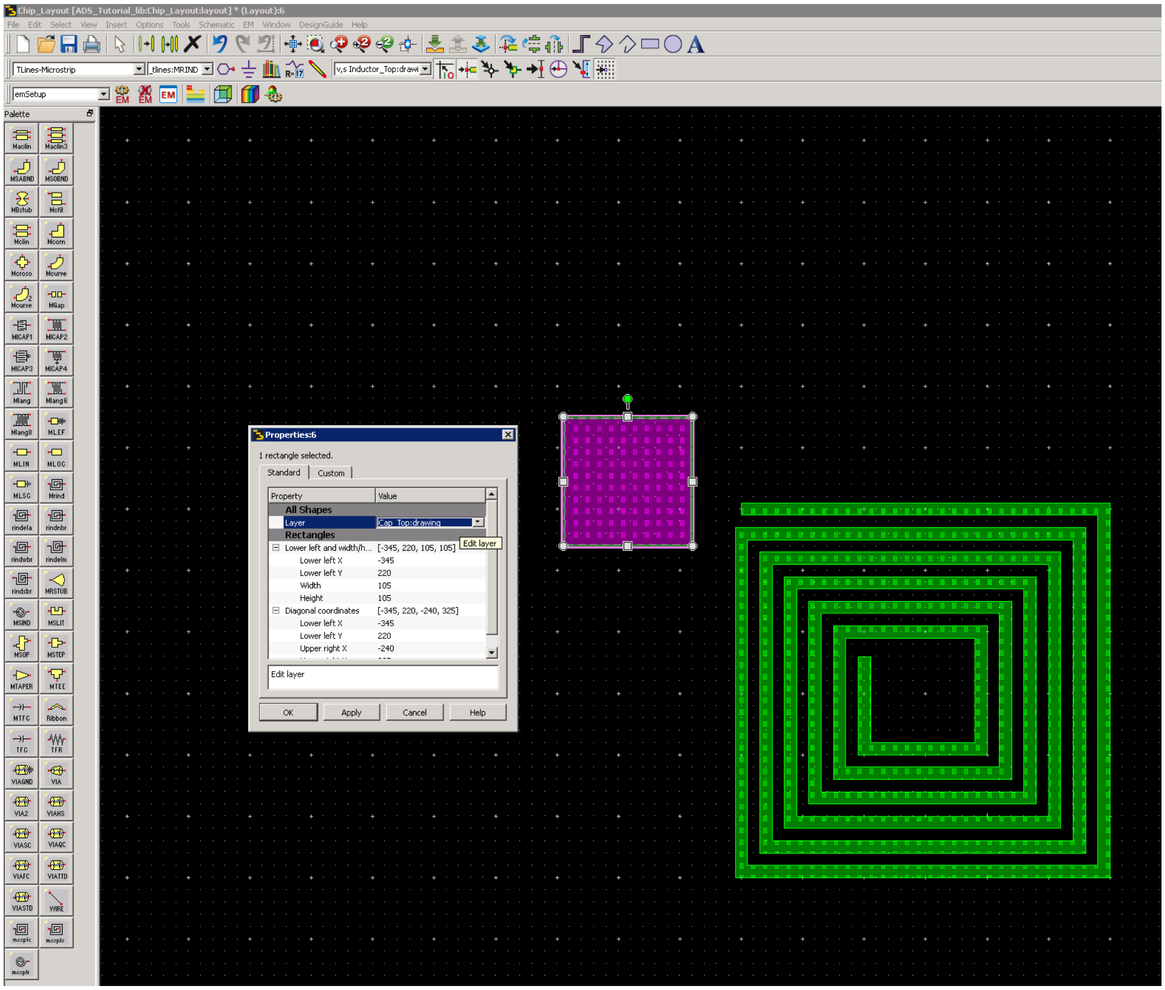


Figure 45: Creating the top plate of the capacitor

Step 14: All the capacitors for this circuit are identical, therefore instead of repeating the previous step three additional times, simply select the capacitor layer. Copy the layer by either pressing CTRL+C or selecting copy from the right mouse click menu. Paste the layer pressing CTRL+V or again using the right mouse click and position each layer where desired. The four top layers have now been created and are shown in Figure 46.

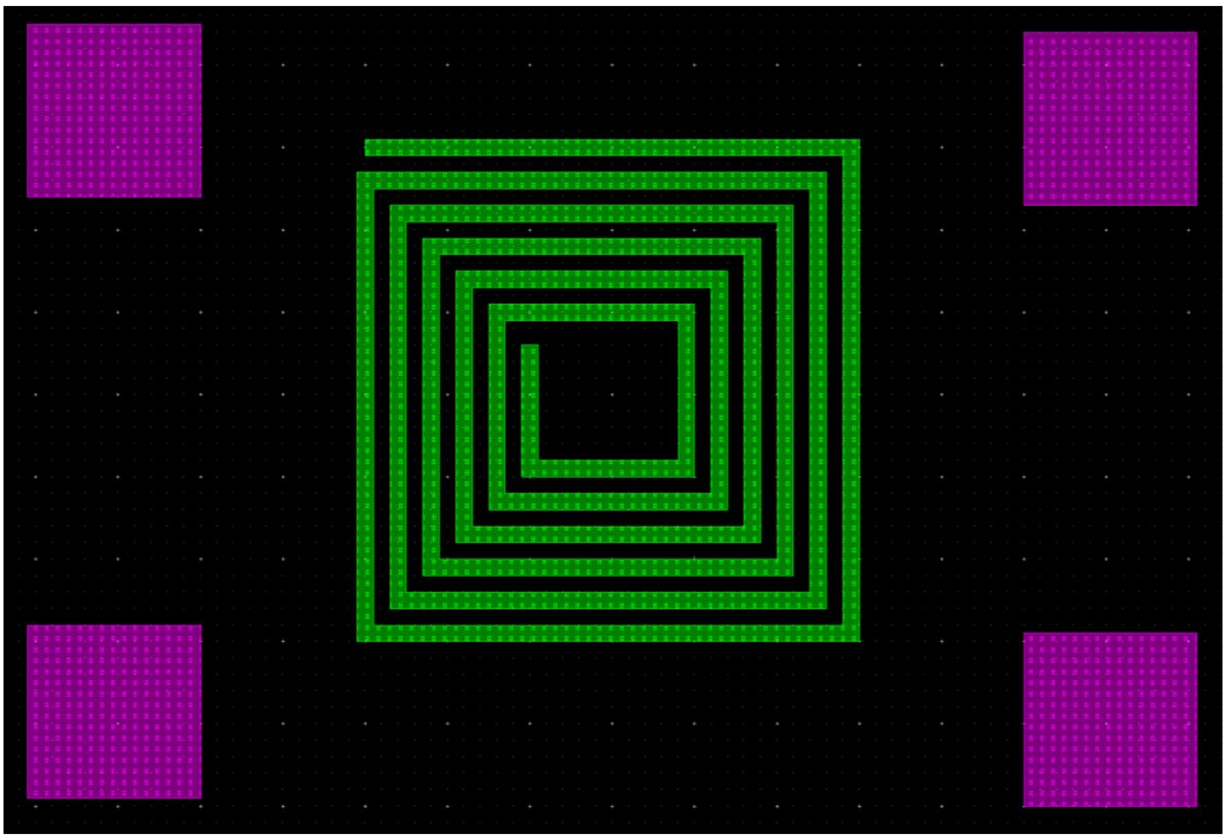


Figure 46: Top plates for capacitors

Step 15: Next, using again the insert rectangle tool create a rectangle that will be used for the ports of the device. Once the rectangle has been created assign the rectangle to the correct layer. In this design they were assigned to the Ports layer as shown in Figure 47. Again, copy the rectangle shape in the same manner described in the previous step and place three additional ones for a total of four ports similar to the layout shown in Figure 47.

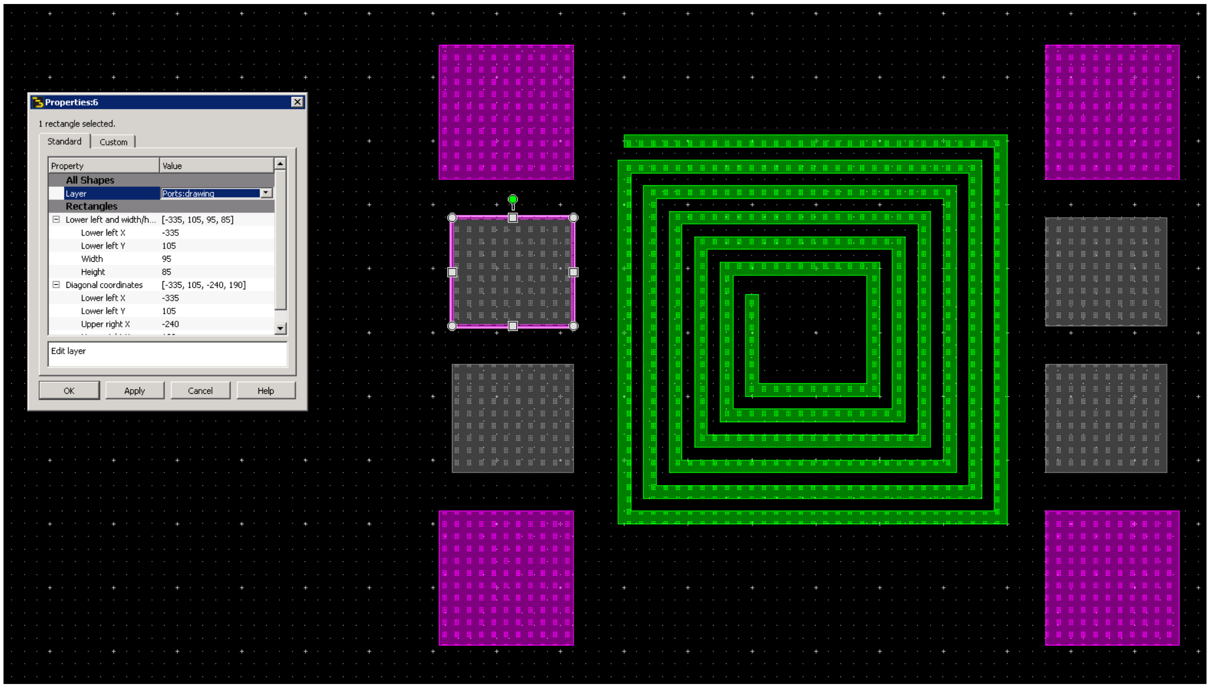


Figure 47: Input pads

Step 16: Next, the bottom plate of the capacitor is created. This is easily done by selecting the four already drawn shapes for the capacitors, copying them, and pasting them into place. Double click on the already selected shapes, and assign the shapes to the correct layer. In this circuit, they were assigned to the layer called Cap_Bot. This is shown in Figure 48 where the layer Cap_top has been altered to make it more transparent in order to see what is behind the layer.

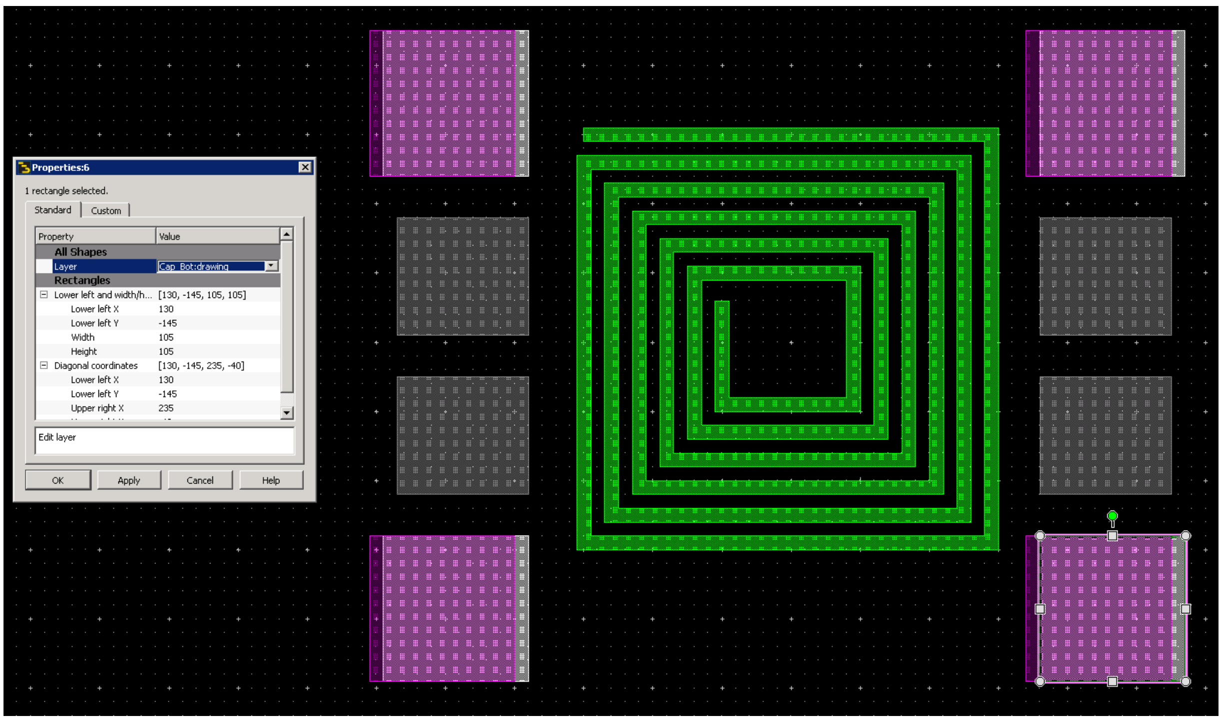


Figure 48: Bottom capacitors

Step 17: To change the transparency of a layer, go to the Layers Preferences window available from Options>Layer Preferences and find the name of the layer that needs to be altered. The name is displayed in the left-most column in Figure 49. Once the layer is found, double click on the column labeled “Trans” (second from the right in the figure below), and new window pops up as shown below. Adjust the slider until satisfied and press ok to continue. The selected layer’s transparency should now have changed, and should be noticed in the layout window.

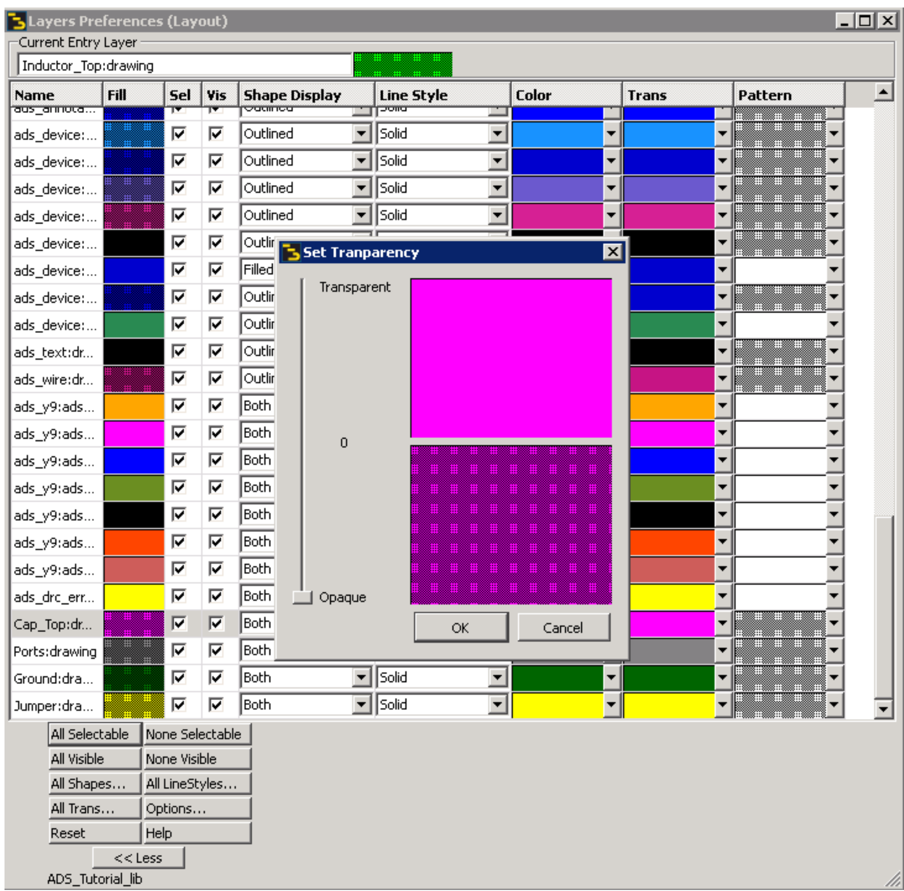


Figure 49: Changing the transparency

Step 18: Symmetry also applies to the inductors of this circuit, therefore to create the bottom inductor simply copy and place it right on the same position as the other. Once the new inductor is placed, assign it to the correct layer as was done with the previous shapes. The bottom inductor was placed in a layer called Inductor_Bot. See Figure 50 for the expected resulting view.

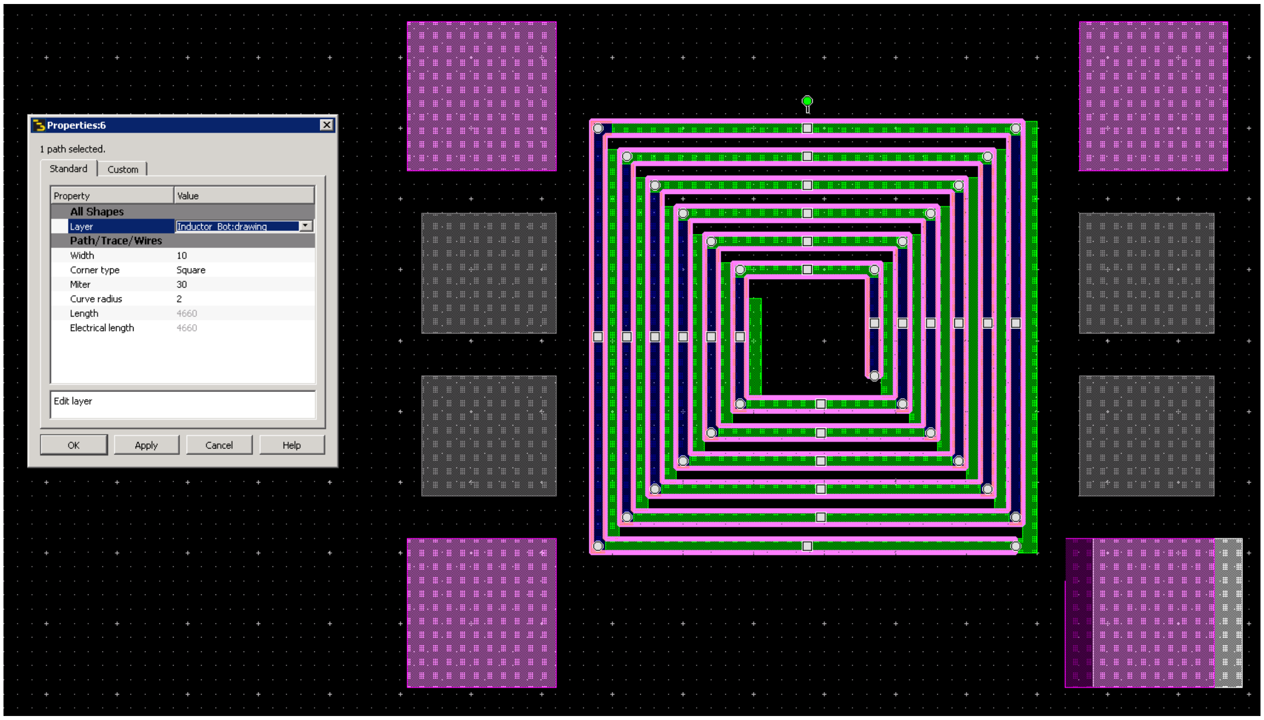


Figure 50: Bottom inductor

Step 19: Next, create the bottom part of the ports. This will act as the common ground, where everything in the circuit will connect to complete the loop. This is simply done by copying the already created layers and placing them in the same place. After placed, select them and assign them to the correct layers as shown in Figure 51. In this picture, the colors of the previous ports appear to have changed. This occurs because the layer that is currently assigned to the bottom part of the port has a higher hierarchy. However, this drawing does not determine the order of the layers but only their general "print" area or the general area where the specified layer should exist. The physical orientation of the layers will be done in later steps, however if this is cause for confusion of the designer the layer hierarchy can be changed by going into the Layer Definitions menu from the **Options>Technology>Layer Definitions.**

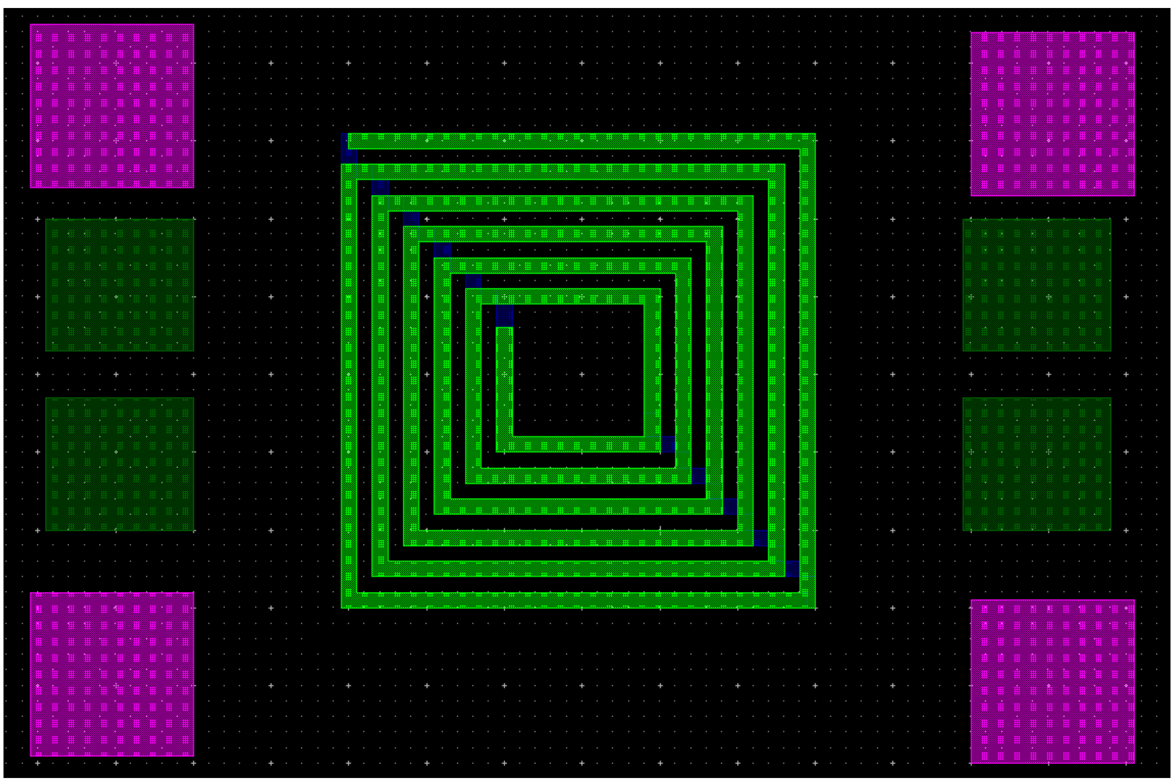


Figure 51: Bottom of the port connectors

Step 20: With the previous step complete, all the main components have been created. The next step is to make the connections between all the components. Always refer to the original schematic as shown in Figure 53 to ensure that the proper connections are being made. In Figure 52 the connection from the end of the large inductor (the end of L3 on the schematic) to port 3 is being made. This is a crucial step because, as can be seen in the figure, the drawing overlaps the inductor layer. Because of this, a single layer must be created which goes from one physical chip layer to one above. From this point the connection can then be run through. To do this, draw a small square at the very end of the inductor. Then start the connection at the port on top of the small square. In total, there should be three layers stacked on top of each other (Inductor end, VIA, path to the port). A similar approach must be taken to make the connection to split the inductor, and connect the bottom inductor as well.

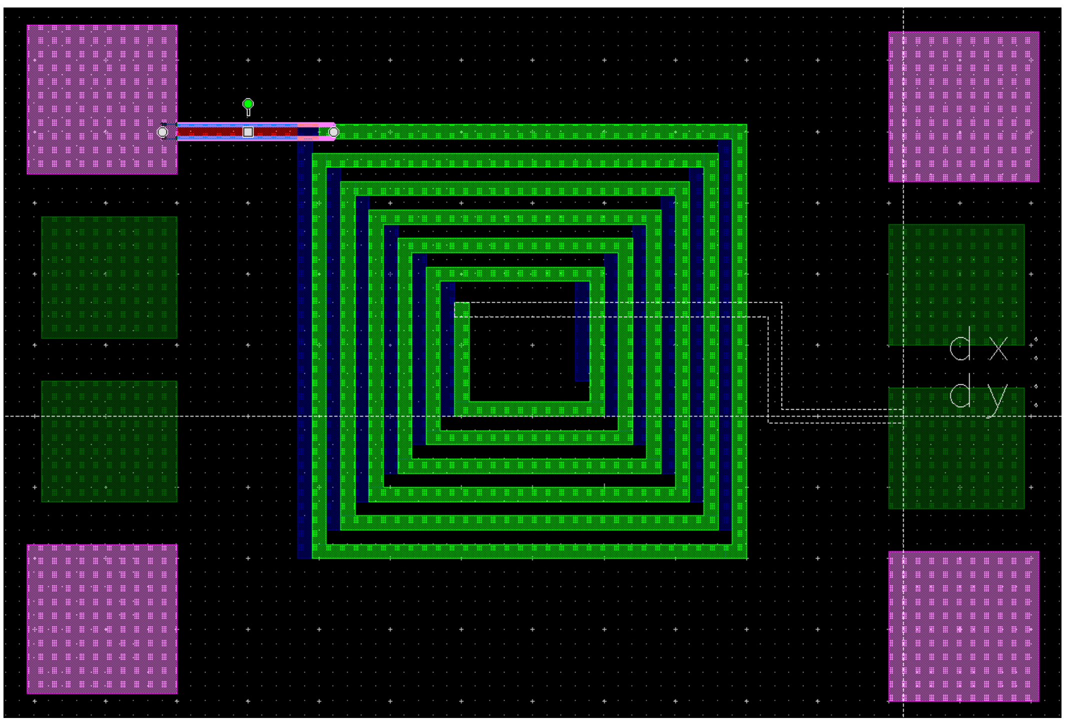


Figure 52: Making connections

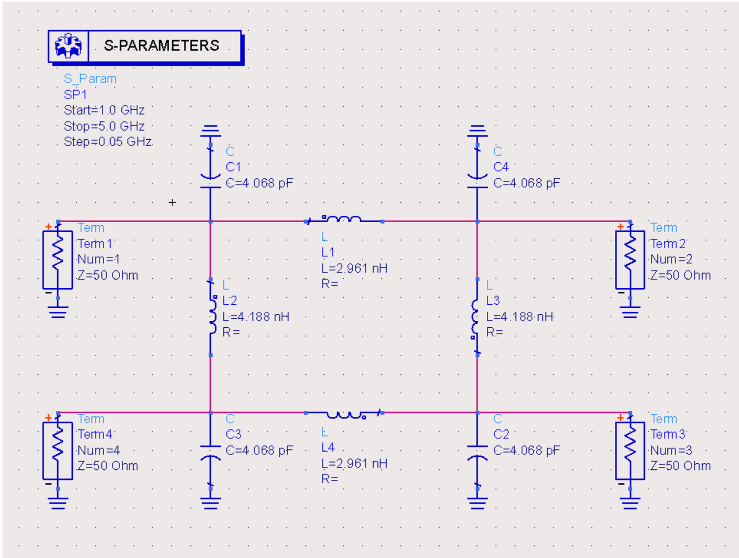


Figure 53: Schematic reference

Step 21: The finalized design is shown in Figure 54. In this design, the layer properties for each component were progressively modified as each connection was implemented in order to easily tell apart all the different layers through different colors. Notice the different colors going from the inductors to the rest of the circuit, the different VIAs and conducting layers are shown in different colors. Once all the proper connections are complete, the last step is to insert the actual ports. To do so, click the insert Pin icon, or **Insert>Pin** and click on the previously placed ports accordingly. The pins will also need to be placed on the right layer so double click on the pin and assign it to the correct layer.

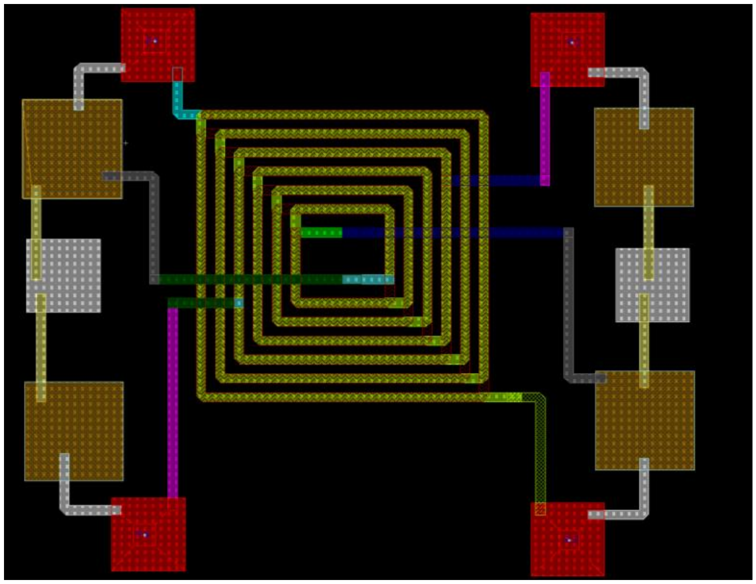


Figure 54: Finalized design

Step 22: With the circuit completed, the last remaining step is to set up the substrate, where all the layers will be mapped and stacked accordingly. To begin, go to **EM>Substrate** and a window will appear asking if a new one substrate is to be created. Press OK, and a window like the one shown in Figure 55 will appear. This step is where all the layers combine to make the entire circuit. The gray are represents the entire conducting plane, where the different layers will be mapped. The very light blue is a bubble of air encapsulating the top of the circuit. The darker blue is the substrate which separates the different conducting layers the substrate is where the VIAs will be mapped.

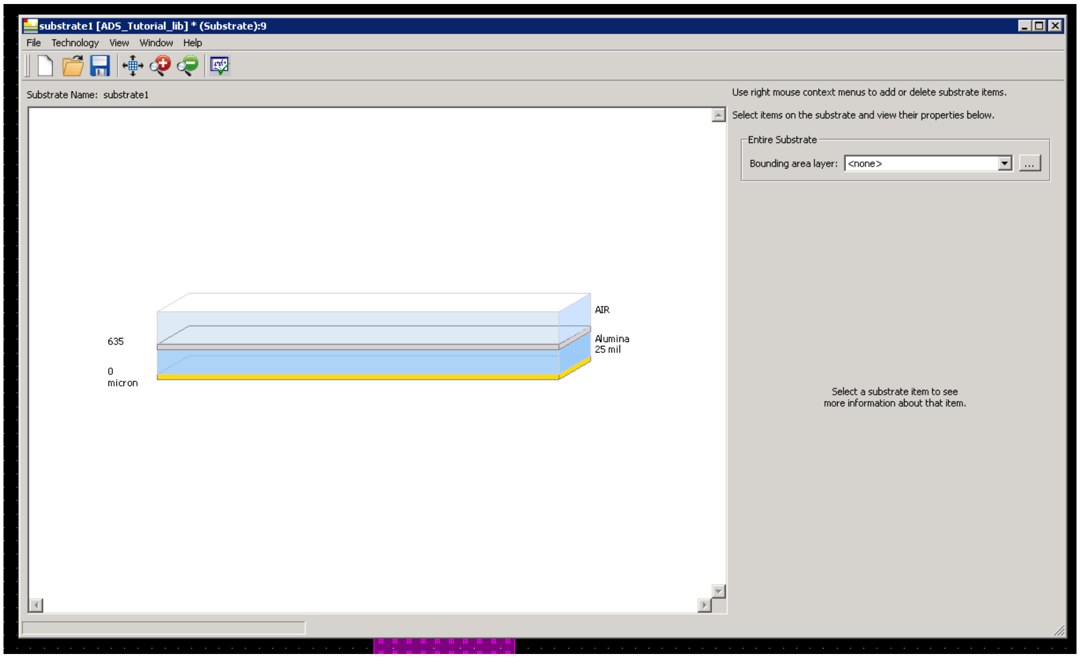


Figure 55: Substrate setup

Step 23: To insert a conducting layer right click on the gray area and select “Map Conductor Layer”. The window will prompt for properties of the conductor layer. First, specify the layer that needs to be placed, in this example first the top inductor is placed. The material of the inductor can be specified, but those parameters depend strictly on the manufacturing process. For this tutorial, it was left as a perfect conductor. Lastly, the thickness of the layer can be specified which here was set to 10 microns. The next step is to keep stacking the layers.

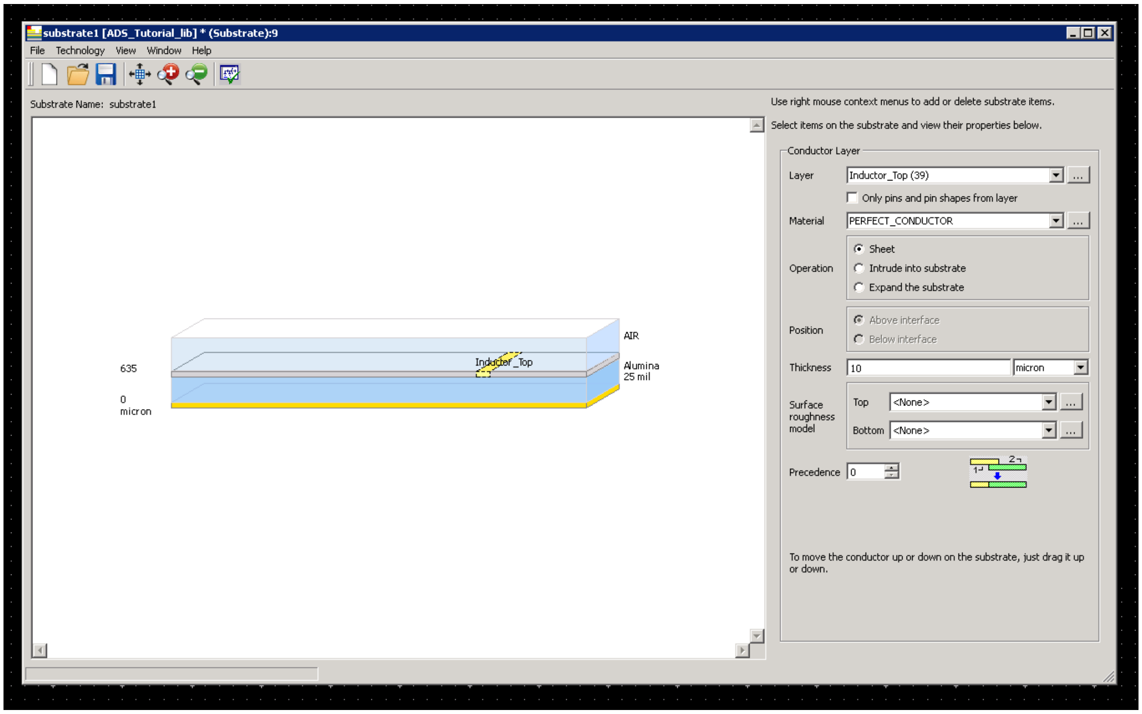


Figure 56: Inserting a conducting layer

Step 24: To keep adding layers on top, right click on the substrate (darker blue) and select "Insert Substrate Layer Above" or below. Different layers can be mapped on top/below or on the side of the already mapped conductor layers. Not that by clicking on the substrate, the material of the substrate can be modified as well as the thickness of the substrate. Change these to match the design requirements. Add all the conducting layers now and make sure to use each layer as much as possible. Note that the VIAs are mapped differently and are discussed on the next step.

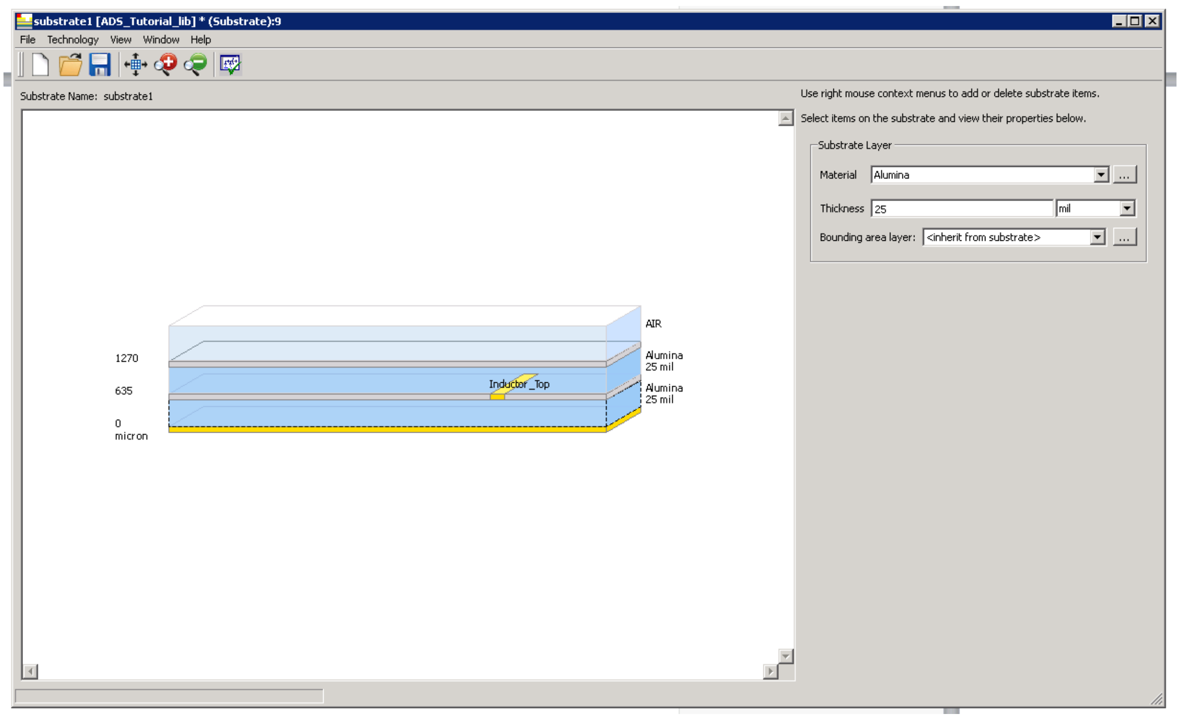


Figure 57: Inserting layers above/below

Step 25: To insert a VIA connecting different layers, right click on the substrate and click "Map Conductor VIA." Similar to the previous conductors, select a layer to assign to the VIA. In Figure 58 the layer jumper connects the inductor to the layer that connects the inductor to the capacitors and the ports, called Cap_Ind. Since the Cap_Ind is on the same layer as the Ports, wherever they overlap in the original layout drawing window will indicate that they are connected. Repeat this procedure until the entire circuit is mapped and ready to function.

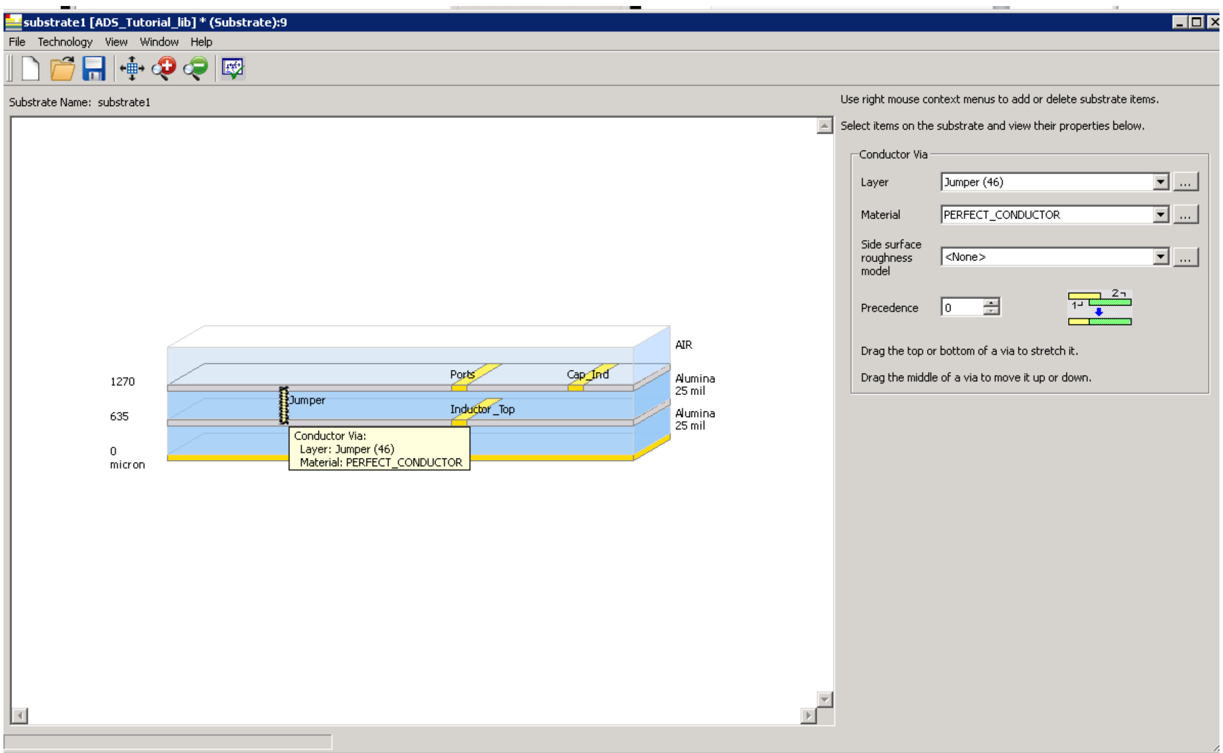


Figure 58: Inserting conductor VIA

Step 26: An example of the finalized substrate is shown in Figure 59. Here all the layers were completely separate to illustrate how exactly the substrate works. Each previously designed layer can be thought of as a conducting layer. Additionally, The VIAs connecting from one layer to the next were all made separate as well. Of course, in a real application, this would be impractical, instead a single VIA would be used to interconnect layers instead of having multiple VIAs from a single layer.

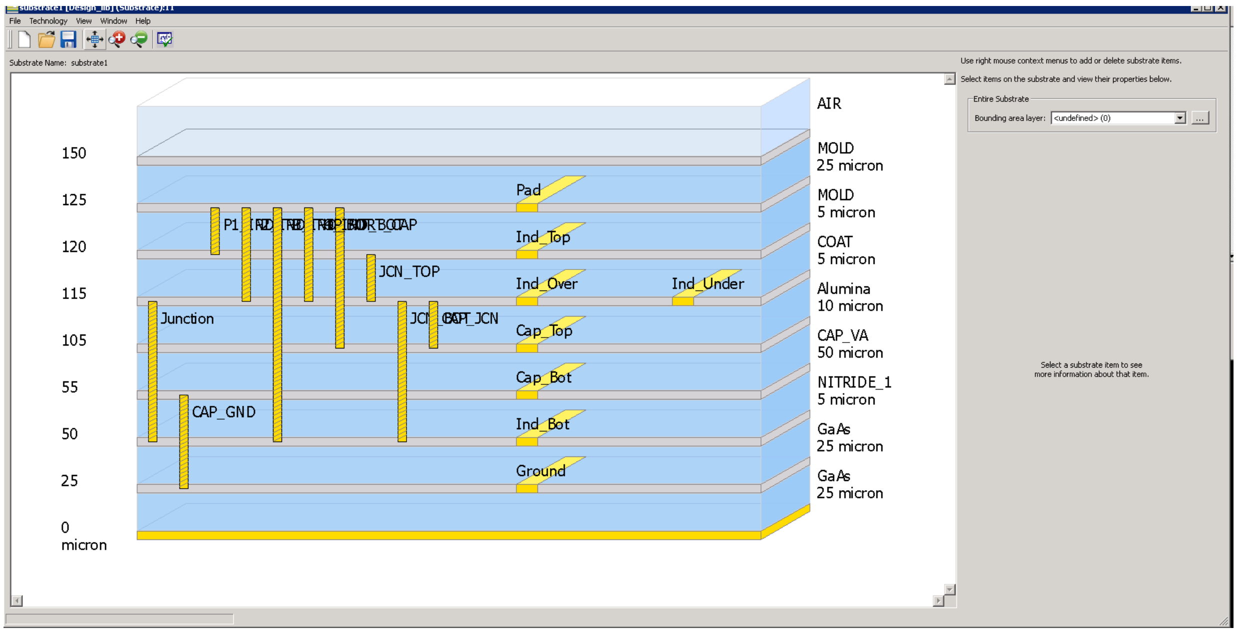


Figure 59: Completed substrate

Step 27: Once the substrate has been completed, a 3D model of the design can be seen by going to **EM>3D EM Preview** an example of which is shown in Figure 60. It is important to note that ADS will not simulate in a 3D environment, it will only display a model of the design. This 3D modeling tool is a good way to ensure that the desired circuit has been implemented, where all the VIAs and layers are shown for full inspection. The 3D EM Preview combines the drawings from the Layout window and the substrate setup to generate the mode.

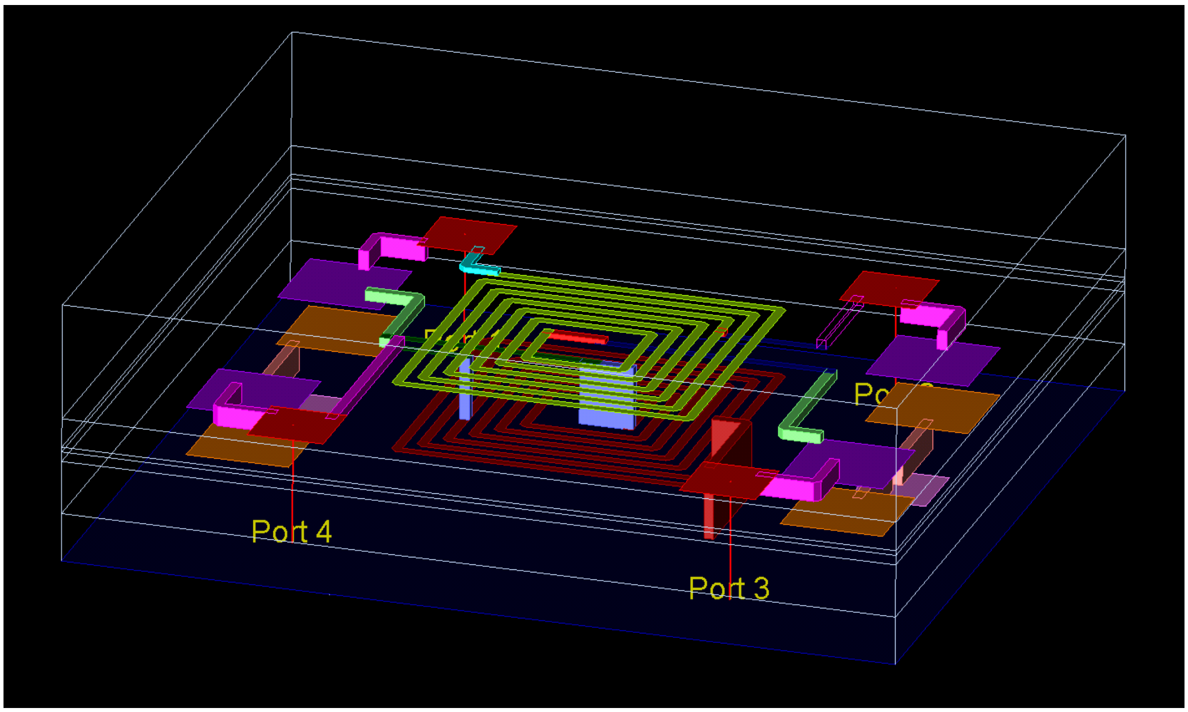


Figure 60: 3-D Model

Step 28: Finally, if everything else has been set correctly, the last step is to simulate. Go to **EM>Simulation Setup** and the window below will appear. If everything has been correctly set, no warnings will display. Otherwise, if a certain area has not been correctly set an exclamation sign will appear next to the area signaling that it requires review. The most important part in the setup is the frequency plan, where the frequency sweep is specified. For this design the center frequency was about 2GHz, therefore the sweep was set to run from 1-5GHz. When done, hit the simulate button in the bottom of the window. These simulations take some time, depending on the computer resources, as well as the complexity of the circuit. For this circuit, using standard computers, the simulations lasted from 1-10 hours again depending on the resources available to the machine.

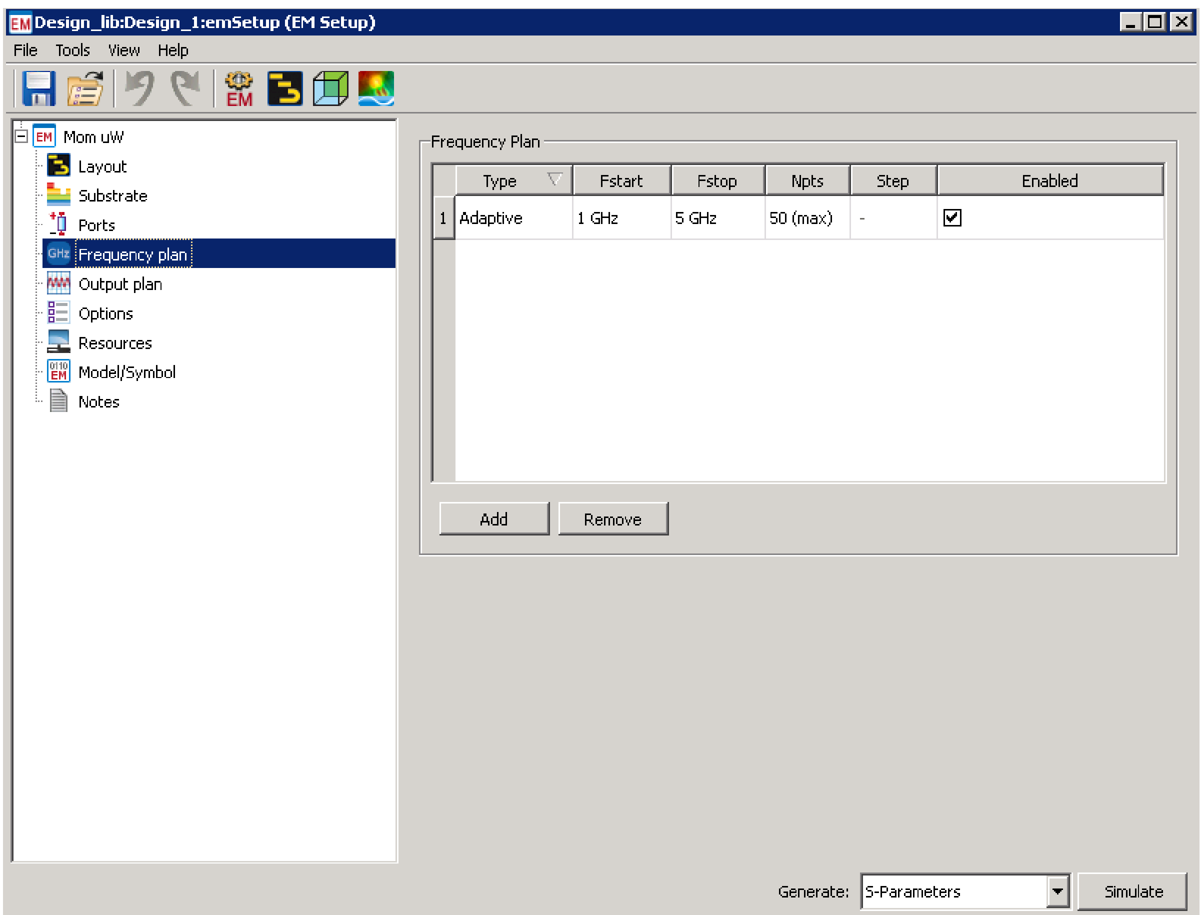


Figure 61: Frequency plan

7.3.1 Momentum Results

Once the layout simulation is complete, a window appears showing all the S-parameters for the device. For this application, the concern is mainly looking at the output, ports 3 and 2, with respect to the input, port 1, the plots of which are shown in the figures below. Although the plot's do not match the expected results from the ideal model, it can be seen that they are somewhat close the predicted values. The parameter S21 approximates the -3dB points but at a higher frequency than expected, S31 appears to not be properly matched and therefore the circuit might need some adjustments. Finally, the most important part which is the phase seems to begin converge around the expected angle, however at a much higher frequency.

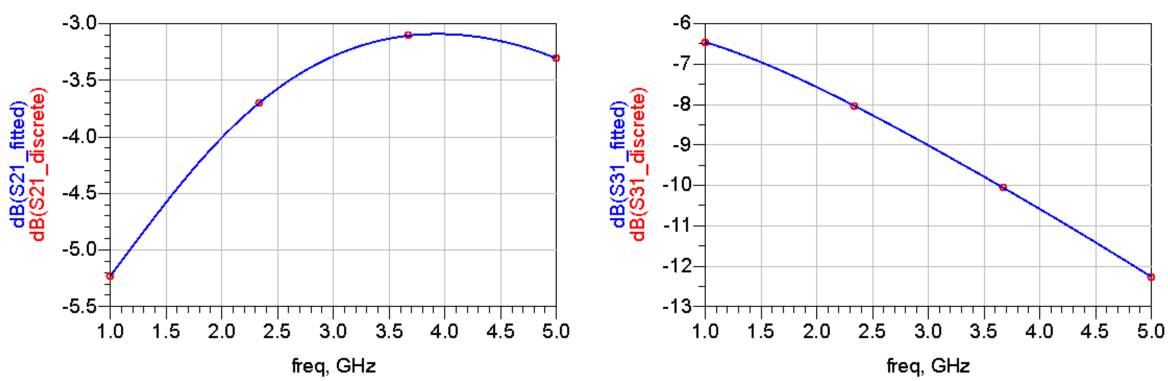


Figure 62: New ADS design results

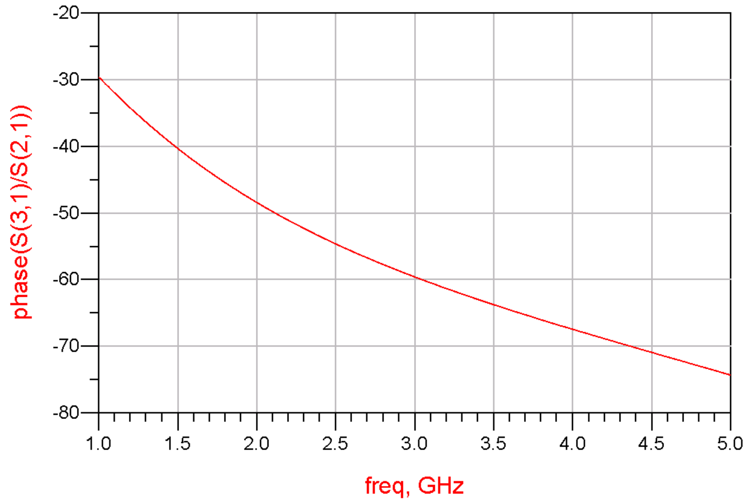


Figure 63: Phase plot for new design

The likely cause of this is the Inductor that was implemented for this design. As previously discussed, an attempt was made to implement two small inductors by creating a large spiral inductor and taping into a point in the middle. This was done to meet one of the requirements of maintaining or reducing the footprint of the device. Calculating for the right Inductor is a very complex task in itself and more so trying to implement this two in one methodology. In addition, it should be noted that although this design reduces length and width dimensions, the height of this circuit is taller than the previous, as shown in Figure 59. This can be significantly reduced by implementing a different layout design which rearranges the order of the layers, i.e. placing the input ports in the middle and breaking off from there, or filling in the gap of the capacitors by placing the inductors in places where the inductors or their connections don't overlap with the capacitors. However, the main goal of the tutorial has been accomplished, to provide the reader with a complete guide on how to create the new layout design.

7.4 Step by step layout instructions for HFSS

When starting a new model in HFSS the most important part is to have a visualization of the design to have a better idea of how to approach the design. For the new design-approach a lumped equivalent model was produced, as shown in Figure 29. In this case the best place to start would be with the spiral inductors, which will both take up the most space of the layout as well as be the best point to base our variables off of. Do not forget to save periodically!

Step 1: After opening up HFSS 14.0 click on the menu project followed by what should be the first choice in the menu, *Insert HFSS Design*.

Step 2: Select draw line in the shortcut menu and draw a random spiral inductor. It does not matter at this point what exactly it looks like so long as the exact number of turns that you want to have in the inductor are there. The inductor that is being created is going to be six turns. When finished right click the screen and select done.

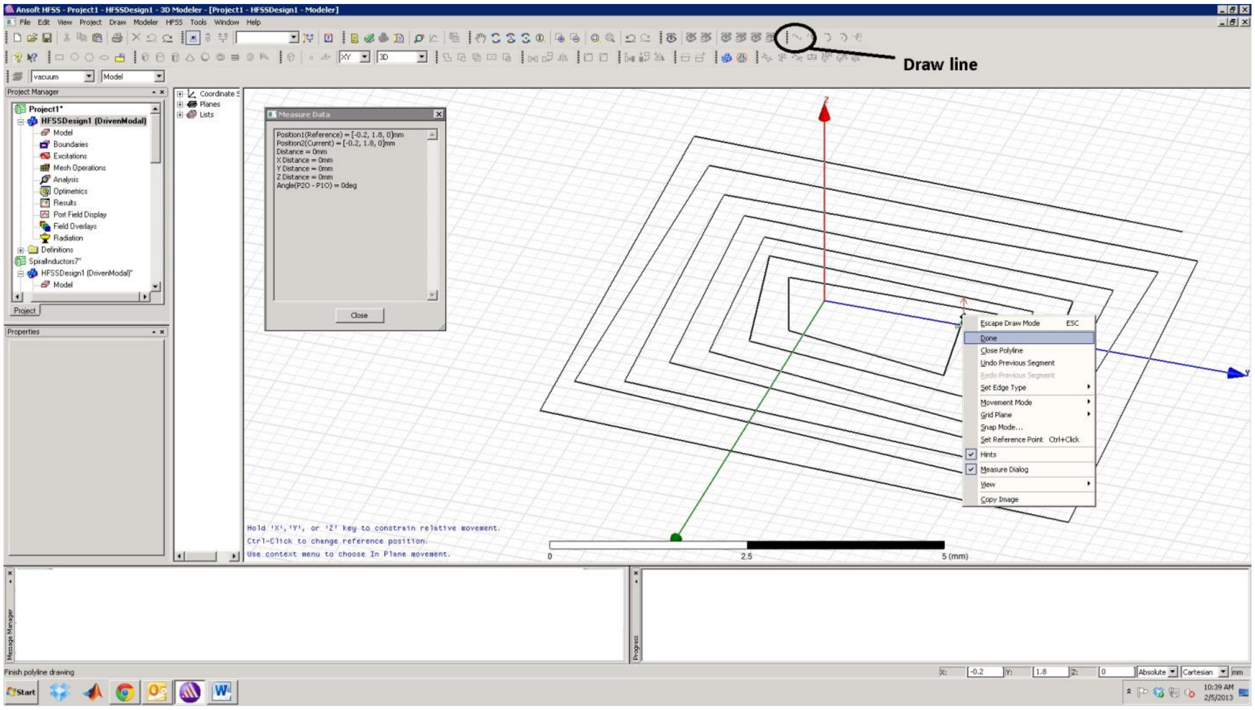


Figure 64: Drawing the spiral inductor

Step 3: Now in the object tree expand **Lines>Polyline 1>Createpolyline** until all of the create lines are shown. Next double click the first line and set the position in the form of a variable so that in the future it can be easily modified. Some good starting coordinates would be [-spiral_diam/2 ,spiral_diam/2 ,0um]. Continuing on the last line, taking into account the spacing between the spirals and the width of the spirals, can be summed up using the coordinates [(-spiral_diam/2+6*(spiral_width+spiral_spacing)),-1*(spiral_diam/2+5*(spiral_width +spiral_spacing)) ,0um]. The rest of these points can be extrapolated using these coordinates. The starting variables should be 400um for spiral diameter, 10um for spiral spacing, and 10um for spiral width.

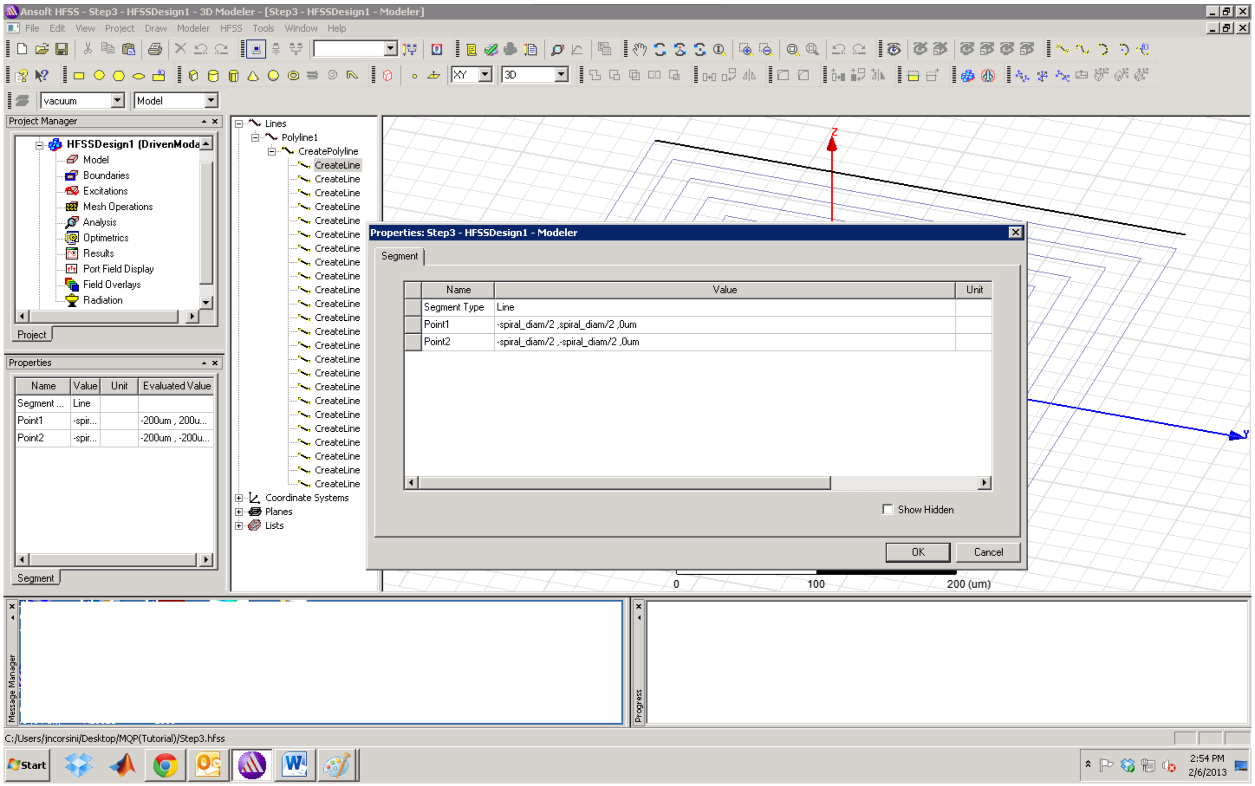


Figure 65: Setting variables to spiral

Step 4: Next double click on create polyline in order to edit the properties. Scroll down to type and change it to rectangle. Then change width/diameter to spiral_width and height to spiral_height. The result should look like the picture below. At this point it would help to start labeling all of the objects that you are going to create. To do this simply double click on the object (Ex. Box5) in the object tree and edit the first field after name.

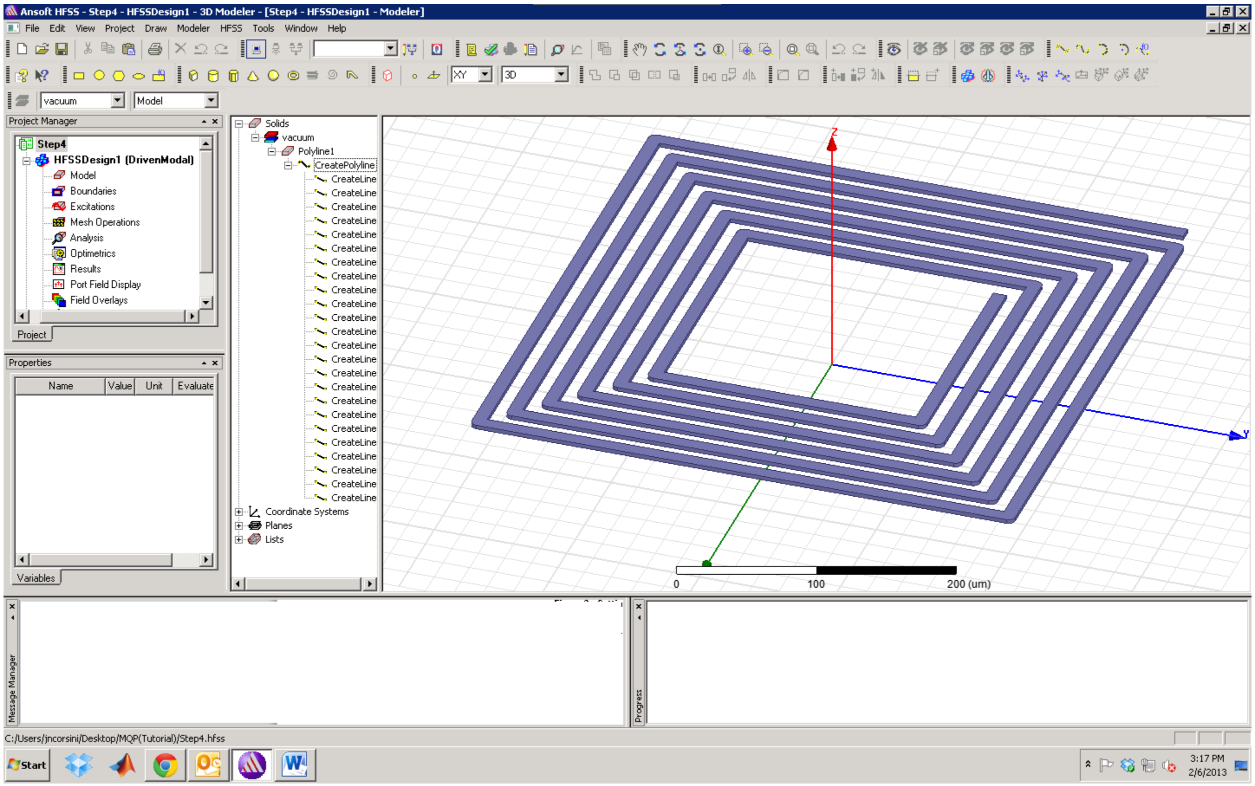


Figure 66: Spiral inductor

Step 5: Now select the spiral inductor that was just made and copy and then paste it. Now it is needed to put them on different layers so go up to **Modeler>CoordinateSystem>Create>RelativeCS>Offset** and create an offset coordinate system of desired distance. Next go to the properties of the second spiral inductor and select the new coordinate system. The next thing that should be done is to right click on the second spiral, go to **Edit>Arrange>Rotate** and rotate it 180 degrees around the z axis. It should now look like it does below. If the spirals are reversed then in the object tree you have to right-click on them and mirror them under **Edit>Arrange->Mirror** option.

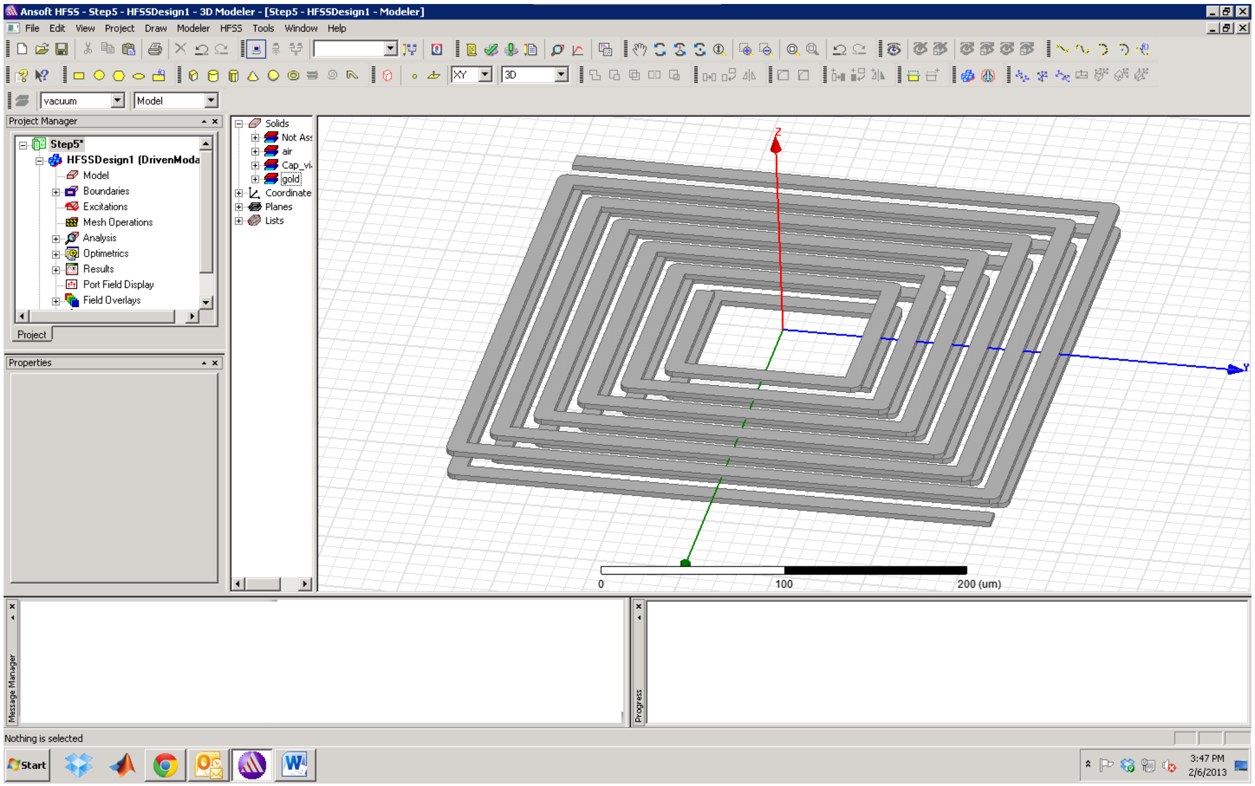


Figure 67: Double spirals

Step 6: Next the bondpads and the capacitors will be created. To start off create a new coordinate system, which will be for the capacitors and bondpads, with the starting point at [0um ,0um ,-spiral_space/2]. Refer to step 5 if help is needed creating one.

Next create two random boxes outside of the spiral inductors. Again, it does not matter what they look like, that will soon be corrected. Also, make sure that the cap and bondpad coordinate system is selected. For a general reference as to where to put them look at Figure 69. Now the difficult part is to get them into the correct position. For the bondpad (orange box) set the starting point at [-bondcap_space/2 ,-spiral_diam/2-cap_size-space_bondcap ,0um] and the xsize equal to –padsize, ysize equal to padsize, and zsize equal to 3um. Padsize, using the PHEMT7 process, can be estimated at 78um. Now, the parallel plate capacitor will be made out of the box drawn, which will act as the top plate. Edit that box and make the starting point [-

(spiral_diam/2)-40.5um ,-(spiral_diam/2)-cap_spiral_space ,cap_via/2, the xsize cap_size, ysize -cap_size, and zsize 2um]. Cap_size was found in section 7.2 and can be rounded to 105um. Cap_via will be utilized soon but for now can be set to .15um. Now click on the top plate of the capacitor and copy and paste it one time. The second box will appear in the object tree to the left. Edit the bottom plate's (the copied box) starting point to be [-(spiral_diam/2)-40.5um ,-(spiral_diam/2)-cap_spiral_space ,-cap_via/2] and zsize to be -2um. There should now be two parallel plates on the screen. Next, make the cap_via by creating a box in between both of the plates. The starting point for the via should be [-(spiral_diam/2)-40.5um ,-(spiral_diam/2)-cap_spiral_space ,-cap_via/2], with xsize equal to cap_size, ysize equal to –cap_size, and zsize equal to cap_via.

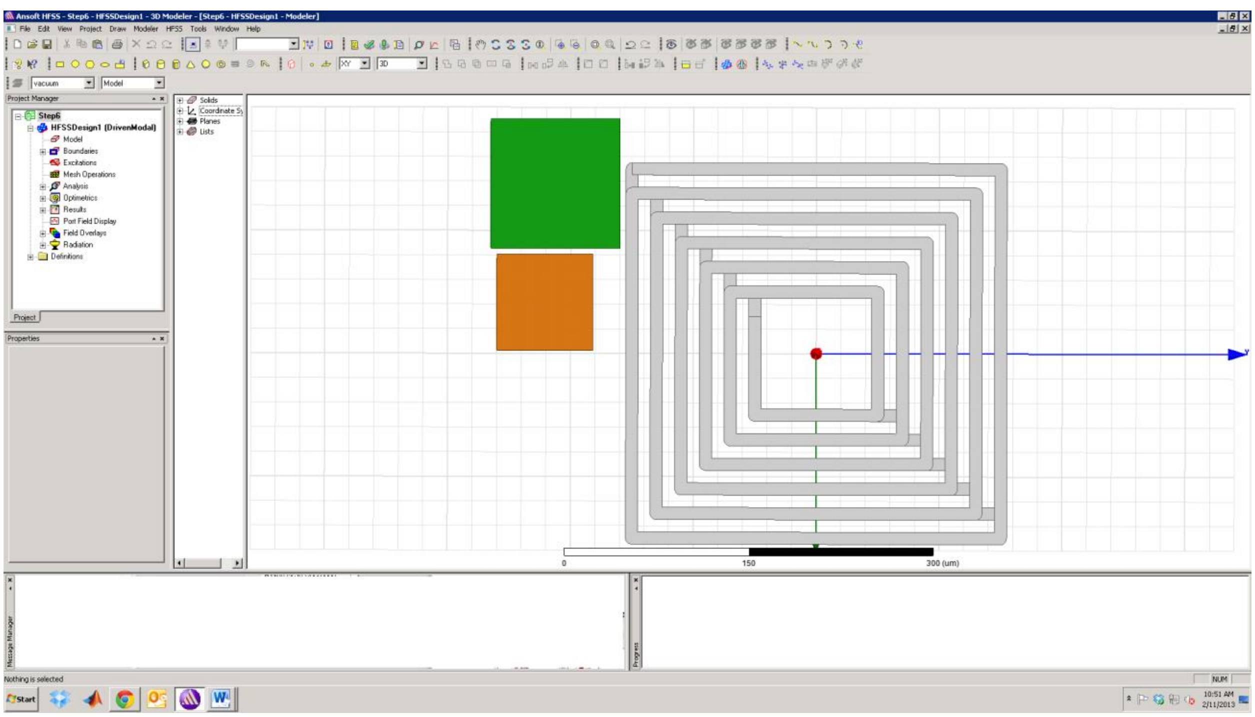


Figure 68: Capacitor and bondpad after correction

The circuit should now look like it does in the above picture. Now, in order to make the three other bondpads and capacitors do the following. First select and copy the first bondpad and then paste it three times. Take the second bondpad and multiply the y starting point quantity by -1 and make ysize negative as well. This will bring it over to the right side of the circuit. Then select the second copy, or third bondpad, and multiply the x and y starting point quantity by -1 and make ysize negative as well make xsize positive. Then select the third copy, or fourth bondpad, and multiply the x starting point quantity by -1 and make xsize positive. Do the same process for the capacitors taking care to copy all three different layers of the cap at the same time. Once finished the circuit should look like the picture in Figure 69. Also, labels are attached to the corresponding caps and bondpads because they will be referenced later on.

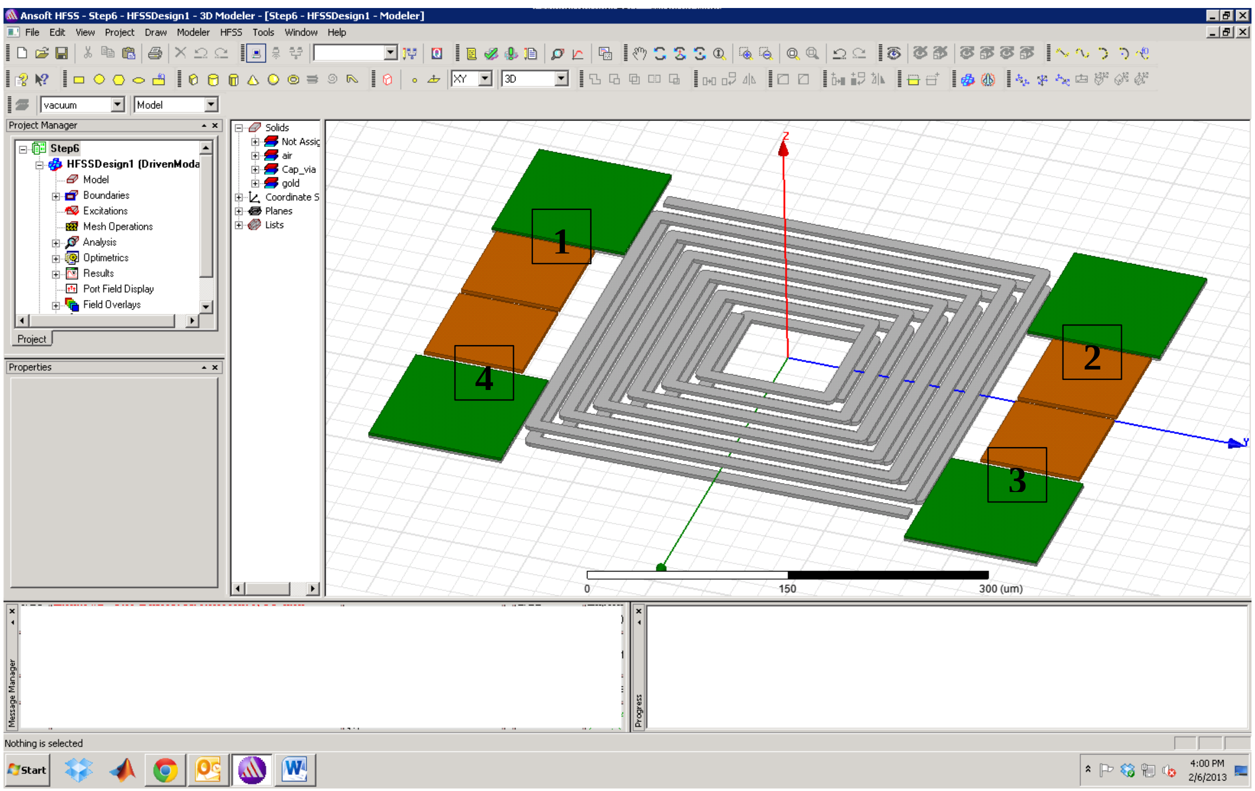


Figure 69: Creating capacitors and bondpads

Step 7: Now the connectors, which create the connections between every piece on the board, will be created. This is the hardest part of the entire build and requires a lot of experimentation to get right. Luckily, it has already been done and the exact coordinates will be given. First start by making a box at the inner point of the spiral inductor, these boxes are so they do not intersect with any unwanted parts. It has a starting point of [-spiral_diam/2+6*(spiral_width+spiral_spacing)+10um) ,(-spiral_diam/2+5*(spiral_width+ spiral_spacing))+5um ,1.5um], xsize and ysize of -10um and zsize of 5um. It will look like it does below in Figure 70.

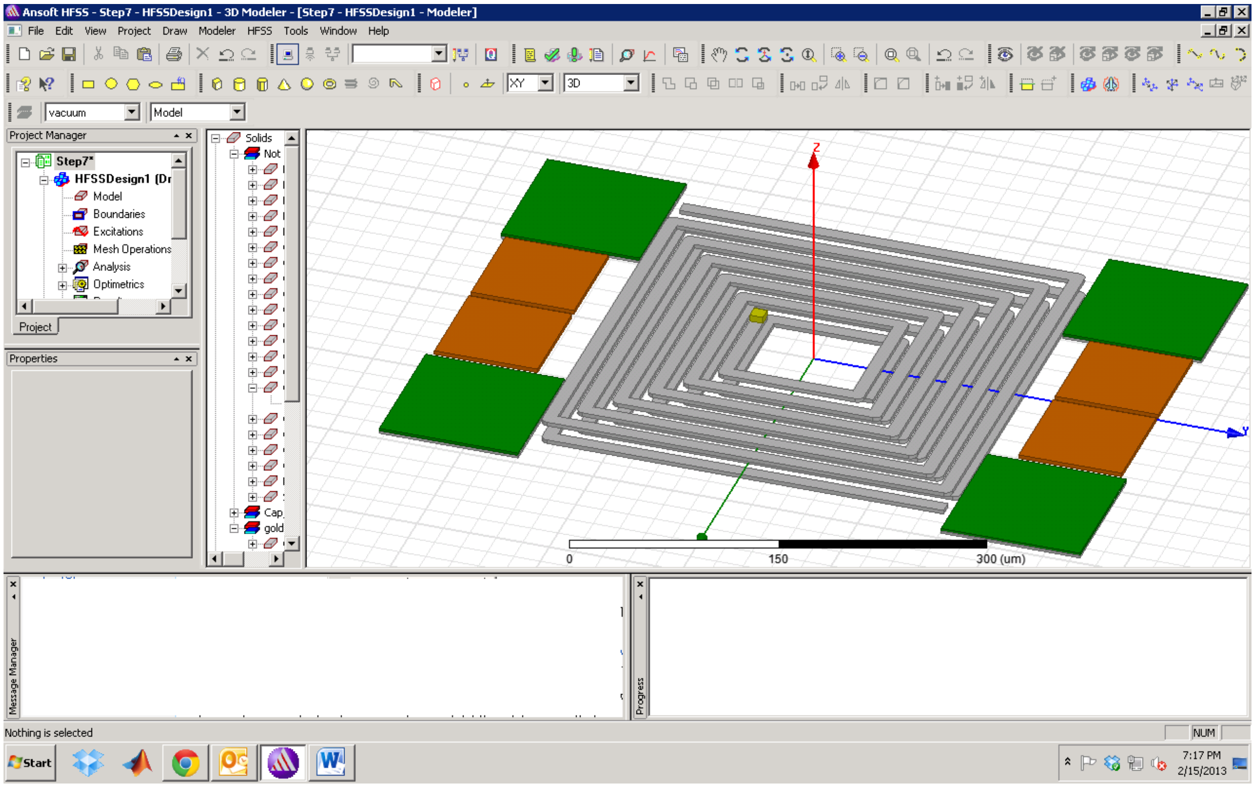


Figure 70: Connector 1 box 1

Next create the matching box on the third bondpad. It will have a starting point of [space_bondcap/2 +pad_size/2 ,spiral_diam/2+cap_size+space_bondcap-pad_size+5um ,3um], xsize and ysize of 10um, and zsize of 11um. It will look like Figure 71.

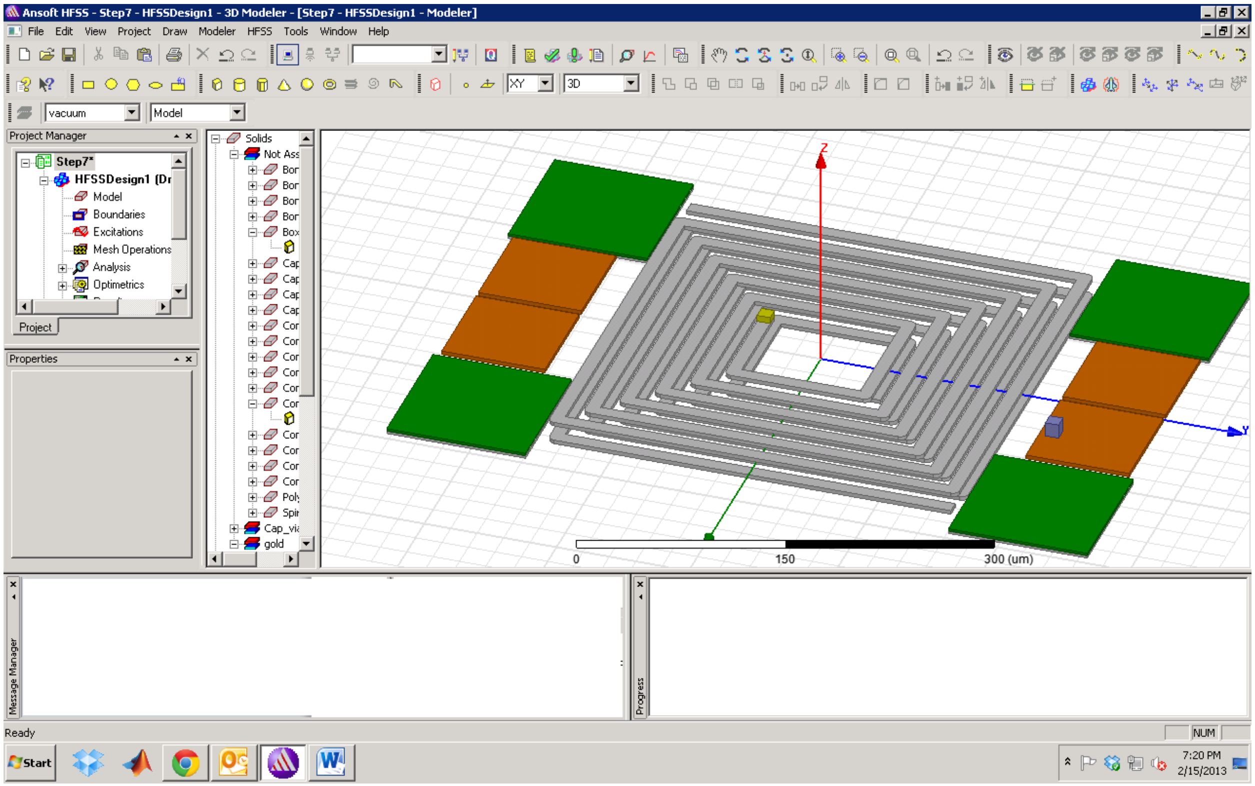


Figure 71: Connector 1 box 2

Next connect the two boxes by selecting polyline in the toolbar and drawing a line from one corner of the box to the other. Edit the polyline to be type rectangle, width 10um, and height 3um. Point 1 of the polyline will be [(-spiral_diam/2+6*(spiral_width+spiral_spacing)) , (-spiral_diam/2+5* (spiral_width+spiral_spacing))-5um ,8um], and point 2 will be [space_bondcap/2 +pad_size/2+5um ,spiral_diam/2+cap_size+space_bondcap-pad_size+15um ,8um]. When all three pieces are created select them all and then right click a selected object and go to **Edit>Boolean>Unite**. This will effectively make all the objects one piece, a necessary part in simulating the program. The result will look like Figure 72.

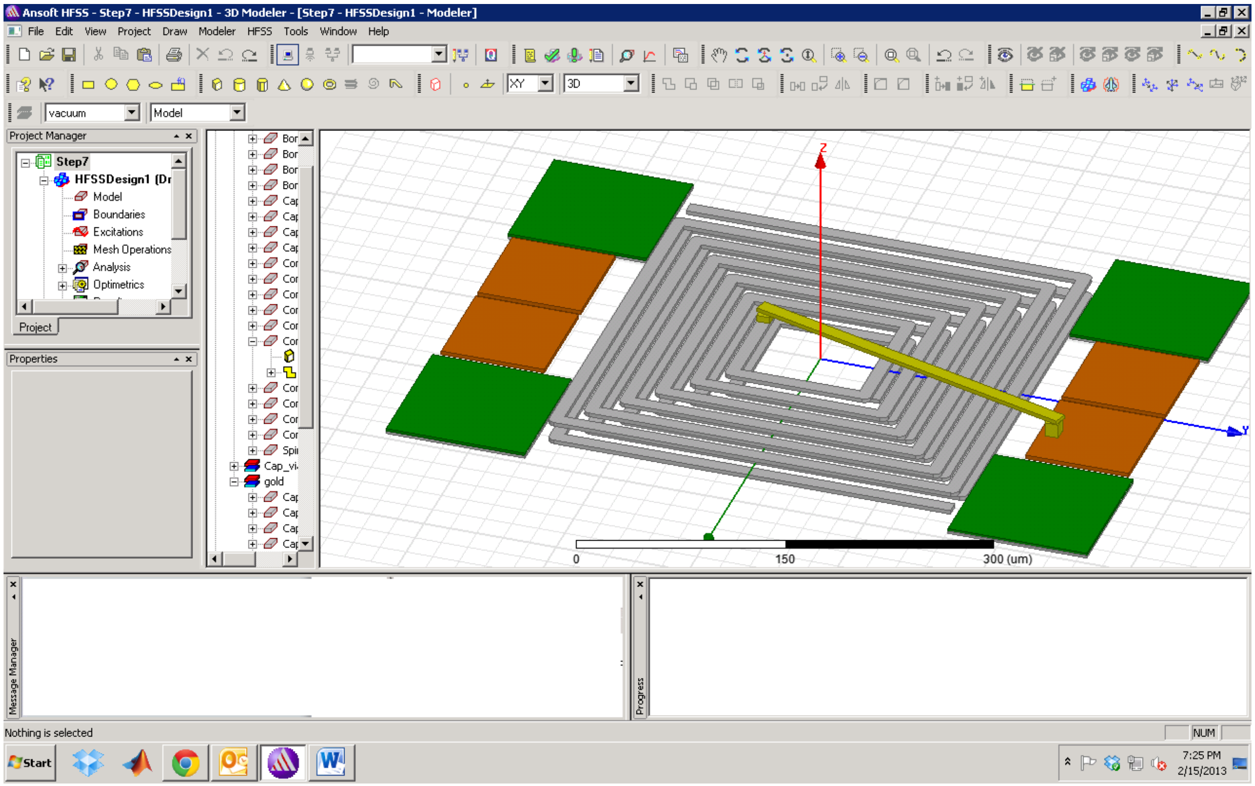


Figure 72: Connector 1 complete

Now, in order to expedite this step, the rest of the connections will be laid out in Table 1 for convenience. Quick note, all polylines should be edited to be rectangles with a width of 10um and height of 3um.

Table 1: List of connection coordinates and sizes

Label	Geometry	Starting Coordinate(s)	Xsize (um)	Ysize (um)	Zsize (um)
Connector 1 A	Box	-1*(spiral_diam/2-cap_size/4)-30um,-1*(spiral_diam/2+ cap_size/4+5um) ,cap_via/2+2um	-10	10	6um-(cap_via/2 +2um)
	Polyline	*Point1*: -1*(spiral_diam/2+cap_size/8)+5um ,-1*(spiral_diam/2+ cap_size/4+5um) ,cap_via/2+6um-(cap_via/2+2um)-spiral_space/2 +3um *Point2*: -spiral_diam/2 ,-spiral_diam/2+10um ,0um			
Connector 1 B	Box	-bondcap_space/2-pad_size/2 ,-spiral_diam/2-cap_size-space_bondcap+pad_size-15um ,0um	-10	35	3
	Box	-bondcap_space/2-pad_size/2 ,-spiral_diam/2-cap_size-space_bondcap+pad_size-15um+25um ,0um	-10	10	-15
	Box	-1*(-spiral_diam/2+6*(spiral_width+spiral_spacing)) ,-1*(-spiral_diam/2+5*(spiral_width+spiral_spacing))-5um ,0um	-10	10	6.5
	Polyline	*Point1*: -1*(-spiral_diam/2+6*(spiral_width+spiral_spacing)) ,-1*(-spiral_diam /2+5*(spiral_width+spiral_spacing))-5um+10um ,-spiral_space/2-6.5um *Point2*: -bondcap_space/2-pad_size/2-10um ,-spiral_diam/2-cap_size-space_bondcap+pad_size-15um+25um ,-16.5um			
Connector 1 C	Box	-1*(space_bondcap/2 +pad_size/2) ,-1*(spiral_diam/2+cap_size+space_bondcap-pad_size+15um+(pad_size*.65)) ,3um	-10	10	8
	Box	-(spiral_diam/2) -40.5um+cap_size*.75 ,-1*((spiral_diam/2)+cap_spiral_space+cap_size*.7) ,cap_via/2+2um	10	-10	(8um-cap_via/2-2um)
	Polyline	*Point1*: -(spiral_diam/2) -40.5um+cap_size*.75 ,-1*((spiral_diam/2) +cap_spiral_space+cap_size*.7)-5um ,cap_via/2+2um+(8um-cap_via/2-2um) Point2: -1*(space_bondcap/2 +pad_size/2) ,-1*(spiral_diam/2 +cap_size+space_bondcap-pad_size+15um+(pad_size*.65))+5um ,11um			

Connector 2 A	Box	(-spiral_diam/2+2*(spiral_width+spiral_spacing))+(spiral_diam/2-2*(spiral_width +spiral_spacing))*.3 ,-1*(-spiral_diam/2+2*(spiral_width+spiral_spacing)) -5um ,1.5um	10	10	5
	Box	-1*(space_bondcap/2 +pad_size/2) ,spiral_diam/2+cap_size+space_bondcap-pad_size+15um ,3um	-10	-10	10
	Polyline	*Point1*: (-spiral_diam/2+2*(spiral_width+spiral_spacing))+(spiral_diam/2-2*(spiral_width+spiral_spacing))*.3 ,-1*(-spiral_diam/2+2*(spiral_width +spiral_spacing)) -5um ,7um *Point2*: -1*(space_bondcap/2 +pad_size/2+5um) ,spiral_diam/2+cap_size+space_bondcap-pad_size+15um ,7um			
Connector 2 B	Box	-1*(space_bondcap/2 +pad_size/2) ,spiral_diam/2+cap_size+space_bondcap-pad_size+15um+(pad_size*.65) ,3um	-10	-10	8
	Box	-(spiral_diam/2) -40.5um+cap_size*.75 ,(spiral_diam/2)+ cap_spiral_space+cap_size*.7 ,cap_via/2+2um	10	10	(8um-cap_via /2-2um)
	Polyline	*Point1*: -(spiral_diam/2) -40.5um+cap_size*.75 ,(spiral_diam/2)+cap_spiral_space+cap_size*.7 +5um ,cap_via/2+1um *Point2*: -1*(space_bondcap/2 +pad_size/2) ,spiral_diam/2+ cap_size+space_bondcap-pad_size+10um+(pad_size*.65) ,5um			
Connector 3 A	Box	(spiral_diam/2) +30.5um ,(spiral_diam/2)+cap_spiral_space+20um ,cap_via/2+2um	-10	-10	5
	Box	spiral_diam/2 -5um ,spiral_diam/2 ,-spiral_height/2	35	-10	spiral_height
	Box	spiral_diam/2 +30um ,spiral_diam/2 ,-spiral_height/2	-10	-10	15
	Polyline	*Point1*: (spiral_diam/2) +25.5um ,(spiral_diam/2)+cap_spiral_space+20um ,cap_via/2+6um *Point2*: spiral_diam/2 +25um ,spiral_diam/2-10um ,-spiral_height/2+15um-spiral_space/2			
Connector 3 B	Box	-1*(-(spiral_diam/2) -40.5um+cap_size*.75) ,(spiral_diam/2)+cap_spiral_space+cap_size*.7 ,cap_via/2+2um	10	10	(8um-cap_via/2-2um)
	Box	(space_bondcap/2 +pad_size/2) ,spiral_diam/2+cap_size+space_bondcap-pad_size+15um+(pad_size*.65) ,3um	-10	-10	8

	Polyline	*Point1:* -1*(-(spiral_diam/2) -40.5um+cap_size*.75)+10um ,(spiral_diam/2)+cap_spiral_space+cap_size*.7 +5um ,cap_via/2+1um+(8um-cap_via/2-2um) *Point2*: (space_bondcap/2 +pad_size/2) -10um ,spiral_diam/2+cap_size+space_bondcap-pad_size+10um+(pad_size*.65) ,11um			
Connector 4 A	Box	-1*(-bondcap_space/2-pad_size/2) ,-spiral_diam/2-cap_size-space_bondcap+pad_size-15um ,0um	-10	35	3
	Box	-1*(-bondcap_space/2-pad_size/2) ,-spiral_diam/2-cap_size-space_bondcap+ pad_size-15um+25um ,0um	-10	10	-15
	Box	spiral_diam/2-(2*spiral_spacing+spiral_width)-(.3*spiral_diam)/2 ,-spiral_diam/2+(2*spiral_spacing+spiral_width)+spiral_spacing/2 ,-1.5um	-10	10	-5
	Polyline	*Point1*: spiral_diam/2-(2*spiral_spacing+spiral_width)-(.3*spiral_diam)/2 ,-spiral_diam/2+(2*spiral_spacing+spiral_width)+spiral_spacing/2+10um ,-spiral_space/2-6.5um *Point2*: -1*(-bondcap_space/2-pad_size/2)-7.5um ,-spiral_diam/2-cap_size-space_bondcap+pad_size-15um+25um ,-15um			
Connector 4 B	Box	(space_bondcap/2 +pad_size/2) ,-1*(spiral_diam/2+cap_size+space_bondcap-pad_size+15um+(pad_size*.65)) ,3um	10	10	8
	Box	-1*(-(spiral_diam/2) -40.5um+cap_size*.75) ,-1*((spiral_diam/2)+cap_spiral_space+cap_size*.7) ,cap_via/2+2um	10	-10	(8um-cap_via/2-2um)
	Polyline	*Point1*: (space_bondcap/2 +pad_size/2) ,-1*(spiral_diam/2+ cap_size+space_bondcap-pad_size+15um+(pad_size*.65))+5um ,11um *Point2*: -1*(-(spiral_diam/2) -40.5um+cap_size*.75)+10um ,-1*((spiral_diam/2)+cap_spiral_space+cap_size*.7) -5um ,cap_via/2+2um+(8um-cap_via/2-2um)			

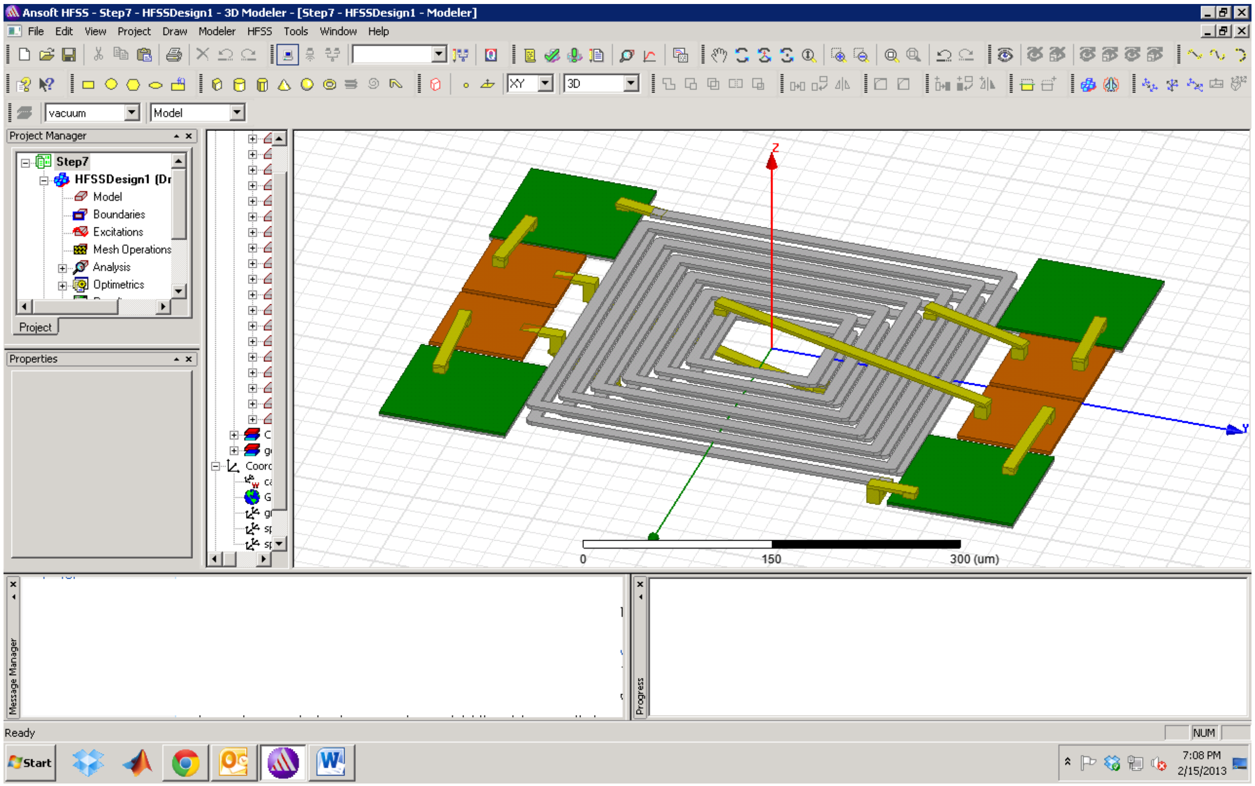


Figure 73: All of the connectors

Once all of the connectors are made the final result should look like it does above in Figure 73.

Now all of the different parts that are touching, and that will be the same material, need to be united. Select all of the objects in the object tree excluding only the bottom capacitor plates and the cap via and unite them using the same method as was used with the connectors.

The final result should be one big united circuit with the capacitor via's and the bottom capacitors left by themselves, as seen in Figure 74.

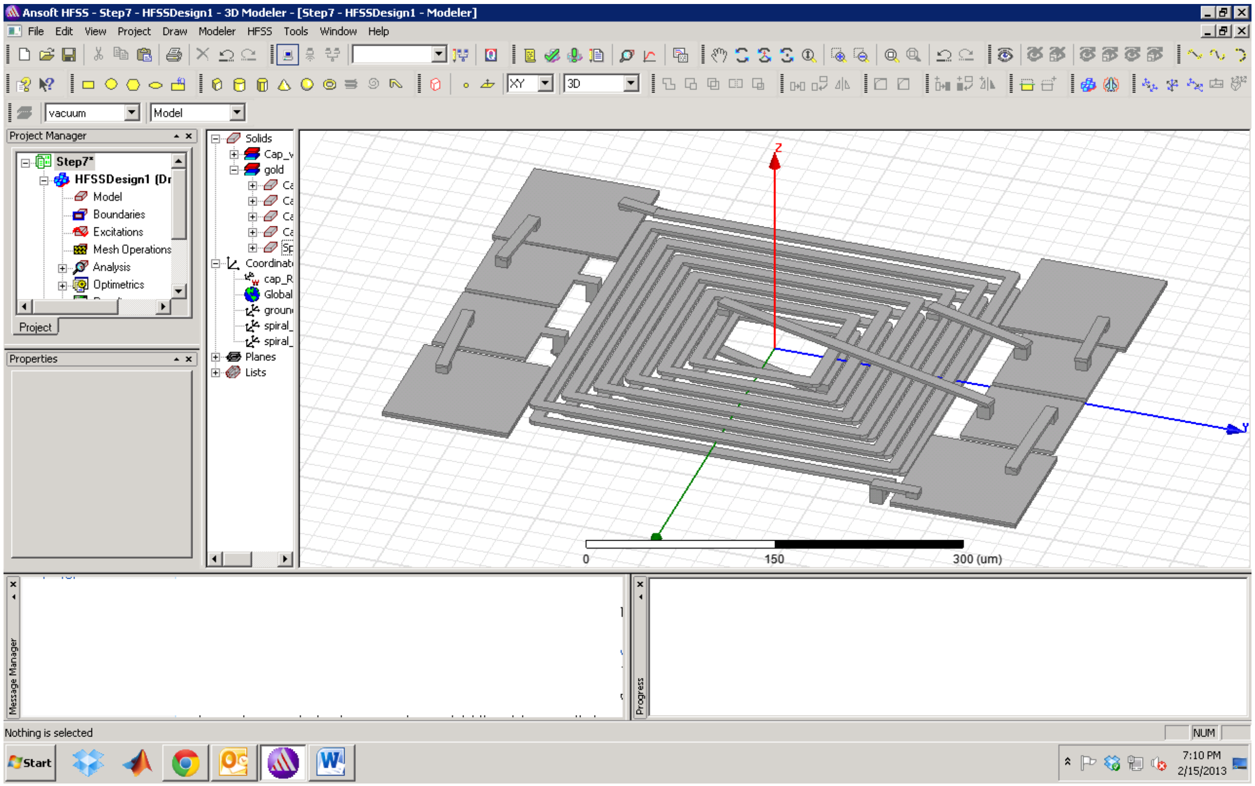


Figure 74: United circuit

Step 8: Next the ground plane will be created. To do this first make sure that the working coordinate system is the default (the one where the first spiral is located). Then create a rectangle of random size and use [-275um ,350um ,-ground_pos] as the starting position, where gound_pos is equal to 300um. Xsize is equal to 550um, ysize is equal to -700um, and zsize is equal to 0. When finished it should look like it does below in Figure 75.

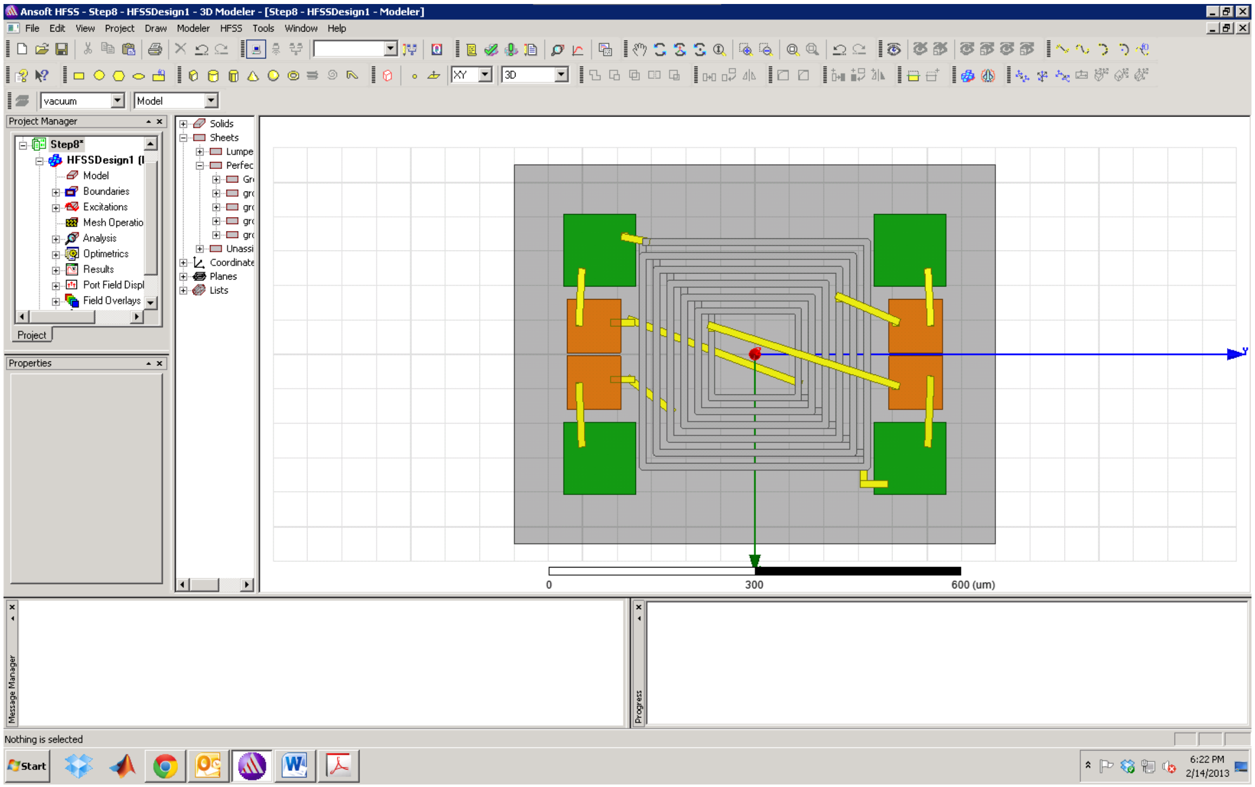


Figure 75: Creating ground plane

After the ground plane is created the stubs need to be made. The ground stubs connect the negative side of the capacitor to ground. In order to create the stubs draw a rectangle the width of the cap to the ground plane and make the starting coordinates in the cap coordinate system [-(spiral_diam/2)-40.5um ,-(spiral_diam/2)-cap_spiral_space -cap_size ,-cap_via/2]. Then make it on the y axis, xsize equal cap_size and zsize equal 290.425um. Then repeat the process used in step 6 when creating the bonpads to make a rectangle for each capacitor. The end result should look like Figure 76.

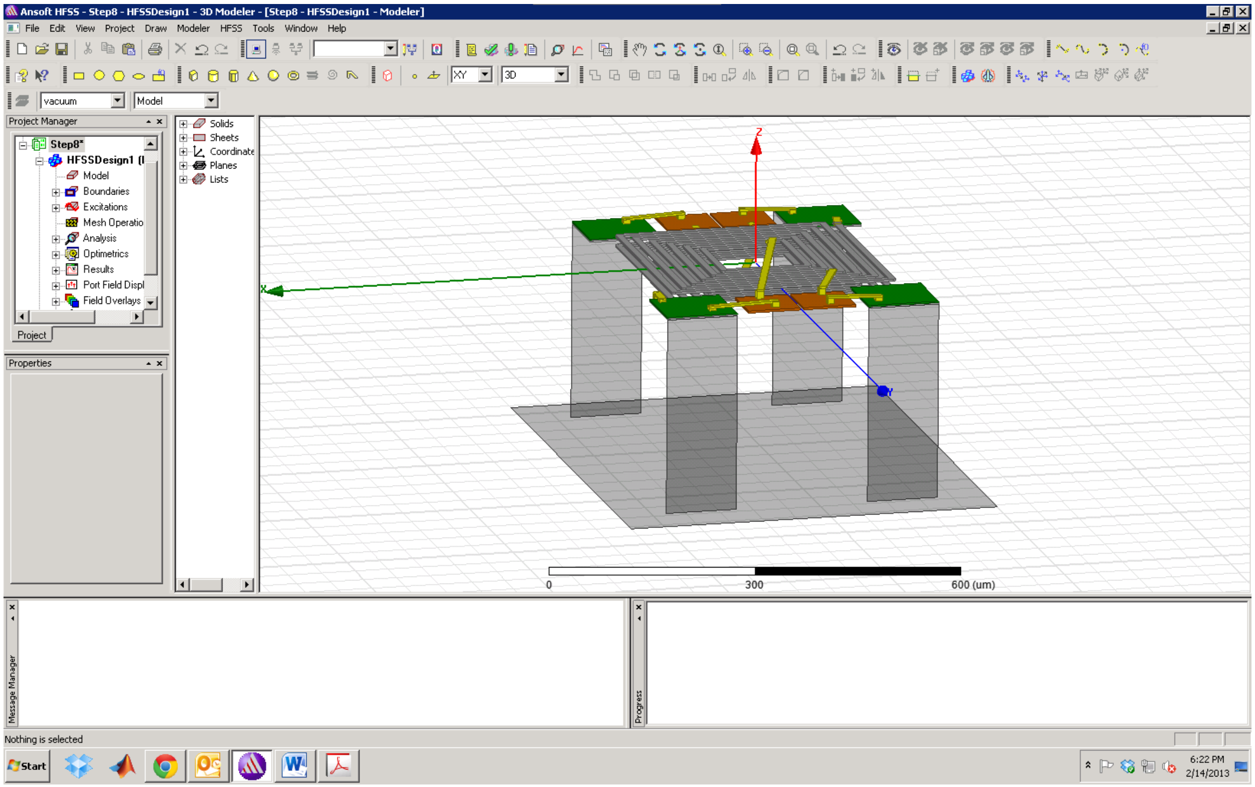


Figure 76: Making ground stubs

After the geometry is correct and the ground stubs are ***actually touching the ground plane*** they need to be assigned officially as ground. To do this hold control and click all of the stubs and the ground plane in the object tree. Once they are all selected right click on one of them and go to boundaries and then select Perfect E.

Step 9: Assigning lumped ports is the next step in this process and is very similar to the way the ground stubs were made. Just like when making the ground stubs create a rectangle from the bondpad to the ground plane. The first one will start at [-space_bondcap/2 ,-spiral_diam/2-cap_size-space_bondcap ,292.5um]. It will run along the y axis, xaxis will be –pad_size, and zsize will be -292.5um. Now use the method used in steps 6 and 8 to create a rectangle on all of the bondpads. The geometry will look like it does in Figure 77.

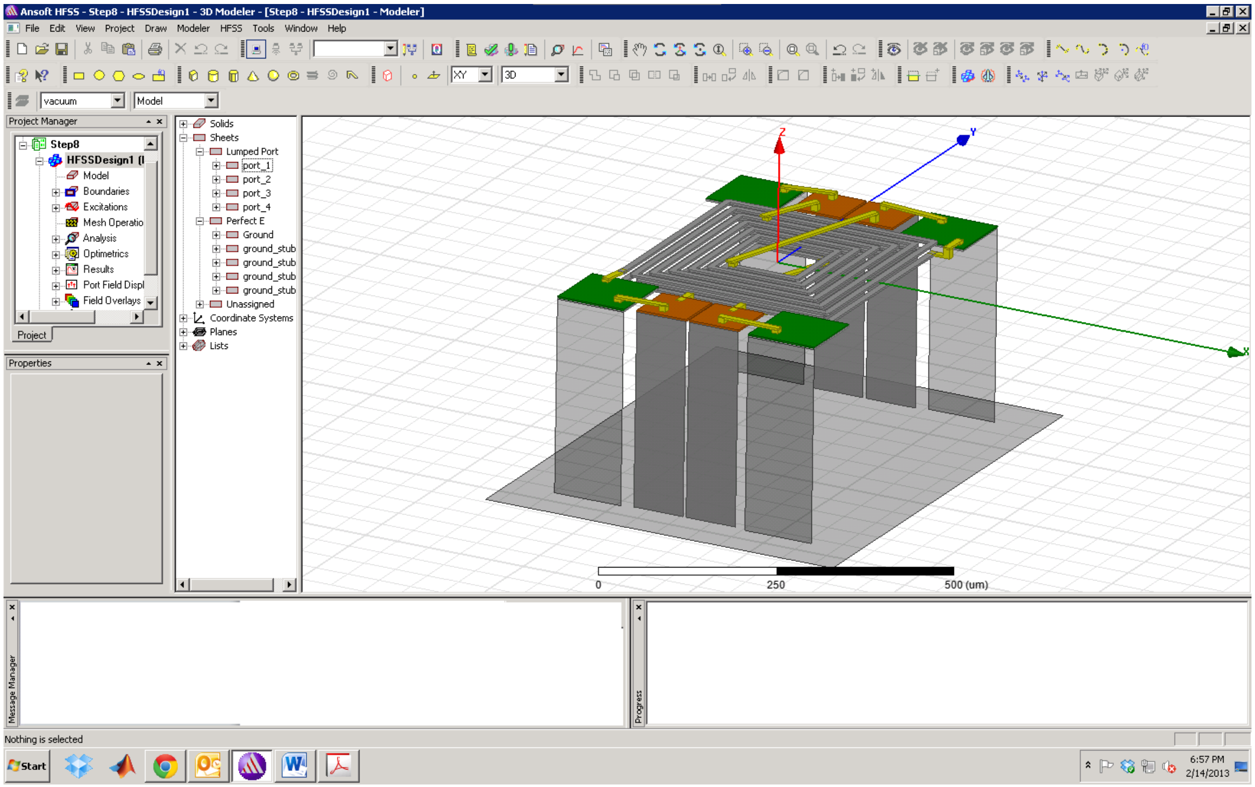


Figure 77: Creating and assigning lumped ports

After the geometry is all set the lumped ports will need to be assigned. To do this right-click on the first plane connecting the bondpad to ground and navigate to assign excitation->lumped port. Once clicked a menu will pop up asking for the full port impedance, leave the resistance at 50Ω and reactance at 0Ω and click next. In this menu there should be one mode with no integration line. Click on none under integration line and select new line. Now click at the top of the rectangle, which is attached to the rest of the circuit, and then at the bottom of the rectangle, which should be touching ground. Then click next, leave the option to renormalize all modes to 50Ω and click finish. When assigning the integration line the port should look like Figure 78.

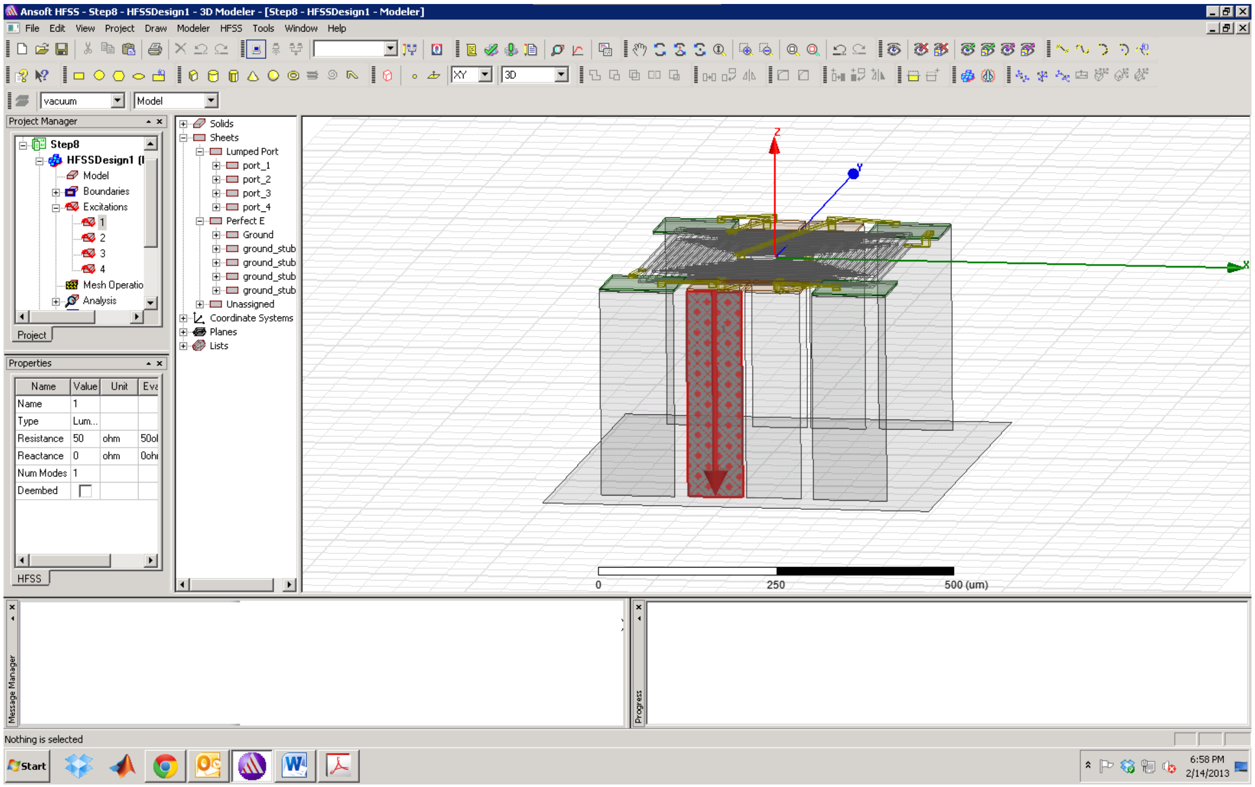


Figure 78: Assigning the integral line for a lumped port

Step 10: Substrate, oxide, and passivation layers are next on the list. The first out of the three is the oxide layer. This layer needs to surround the entire circuit, not including the ground plane, and cannot intersect any part except for planes, which do not count. Start off by making a random box around the circuit. Next set the starting point to [-275 ,-350 ,-25 (um)]. Set xsize to 550um, ysize to 700um, and zsize to 50um. It should now look like Figure 79.

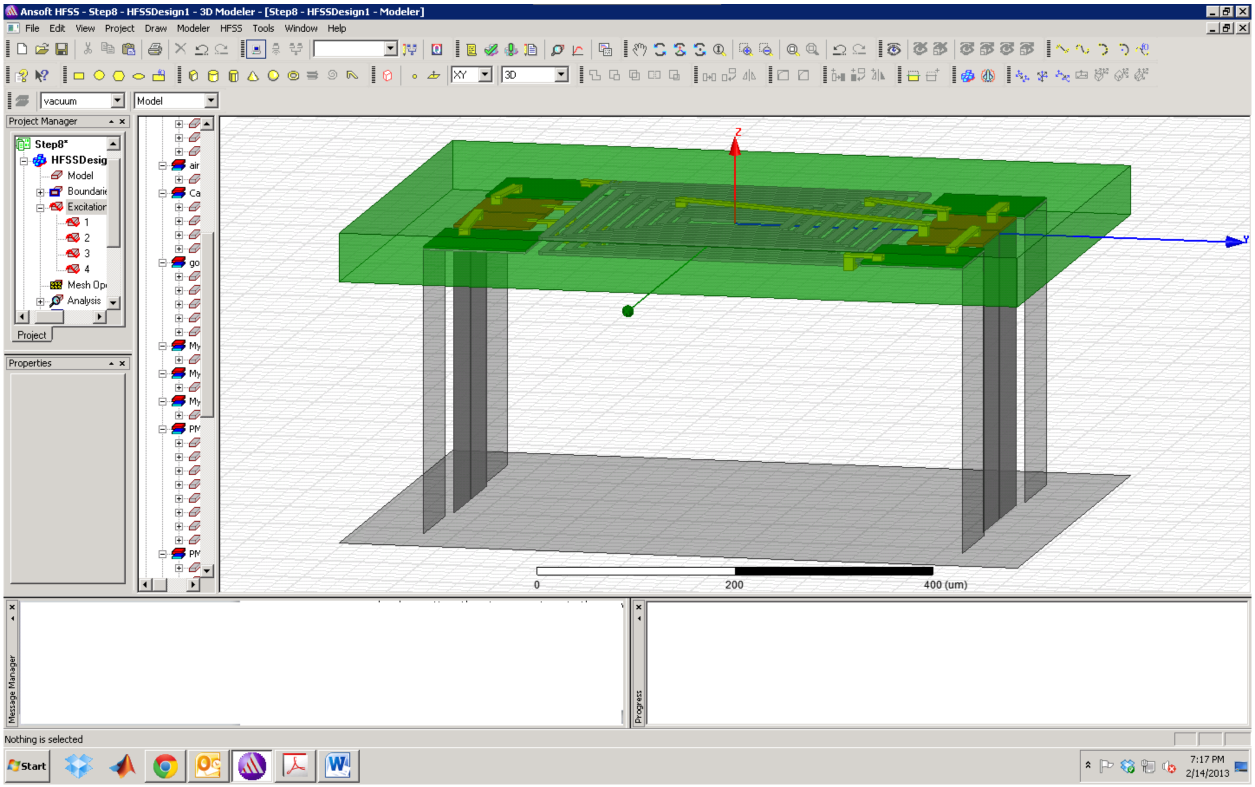


Figure 79: Creating oxide layer

Now to create the passivation layer do the same as with the oxide layer with a starting position of [-275 ,-350 ,25 (um)]. Set xsize to 550um, ysize to 700um, and zsize to 1um. It should now look like Figure 80. The passivation layer is something that forms on the surface when the metal oxidizes [14]. It is an extremely thin layer (only a few atoms thick) that prevents the oxide from flaking off and exposing the circuit [15]. In certain cases it is formed on purpose to gain the benefits of described above.

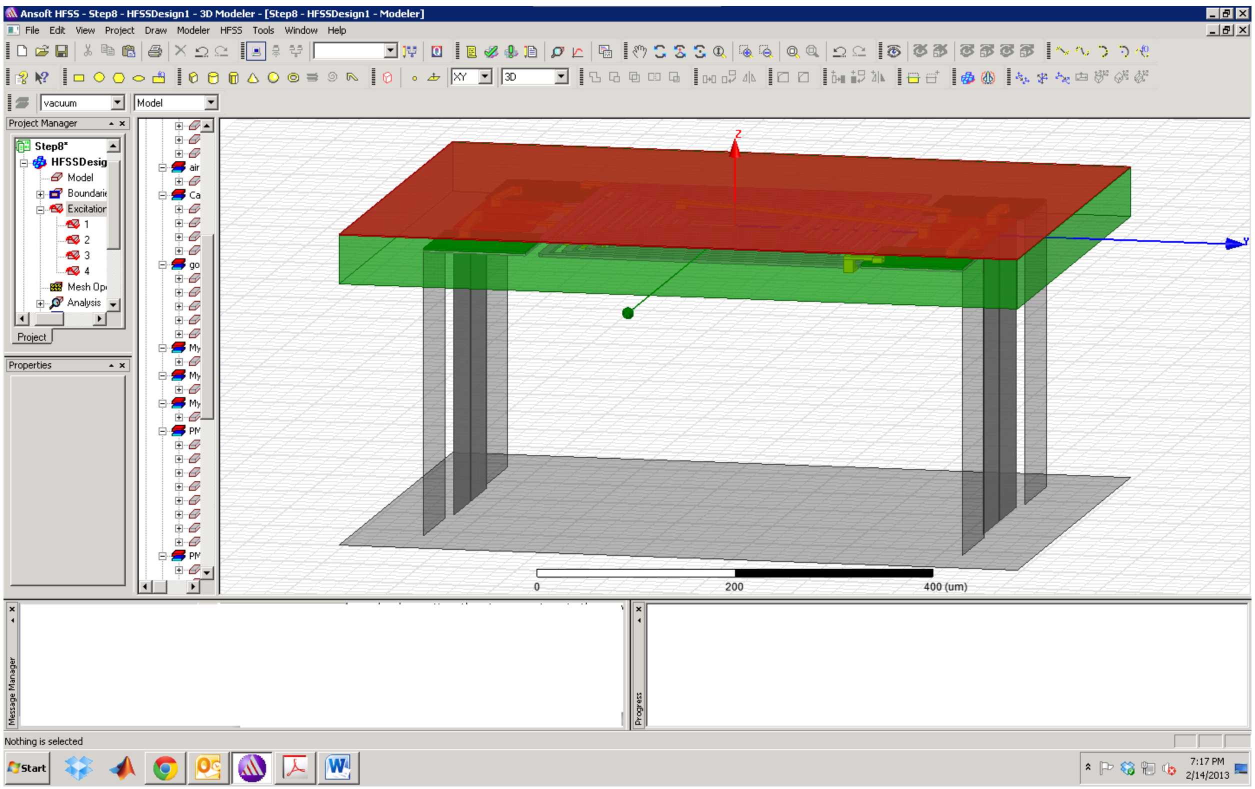


Figure 80: Creating a passivation layer

Now to create the substrate layer do the same as with the previous two layers with a starting position of [275 ,350 ,0 (um)]. Set xsize to -550um, ysize to -700um, and zsize to 267.5um. It should now look like Figure 81.

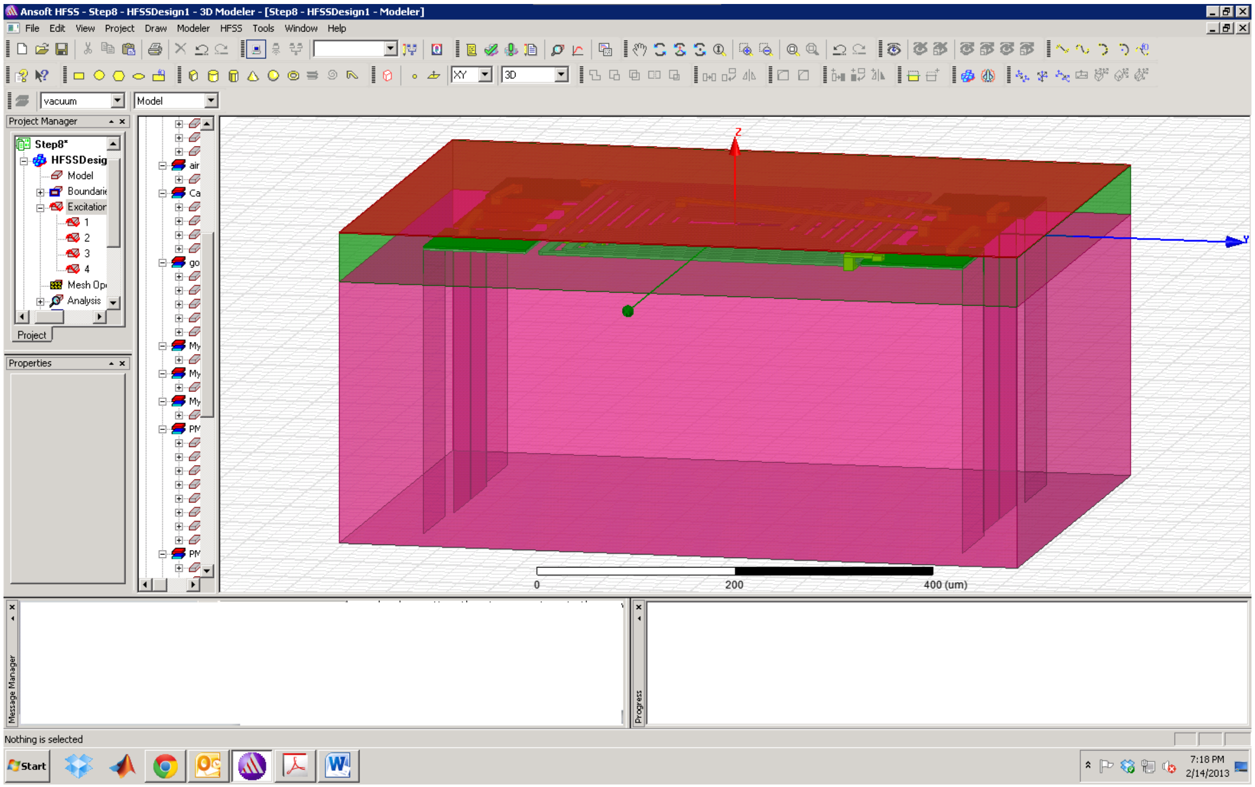


Figure 81: Creating a substrate layer

Step 11: Next a box of air and PML radiation boxes will be created. For the box of air create a big box around everything (even the ground). Set its starting point to [800,-1000 ,872.5 (um)], xsize to -1600um, ysize to 2000um, and zsize to -1200um. This box will encompass everything and is depicted in Figure 82.

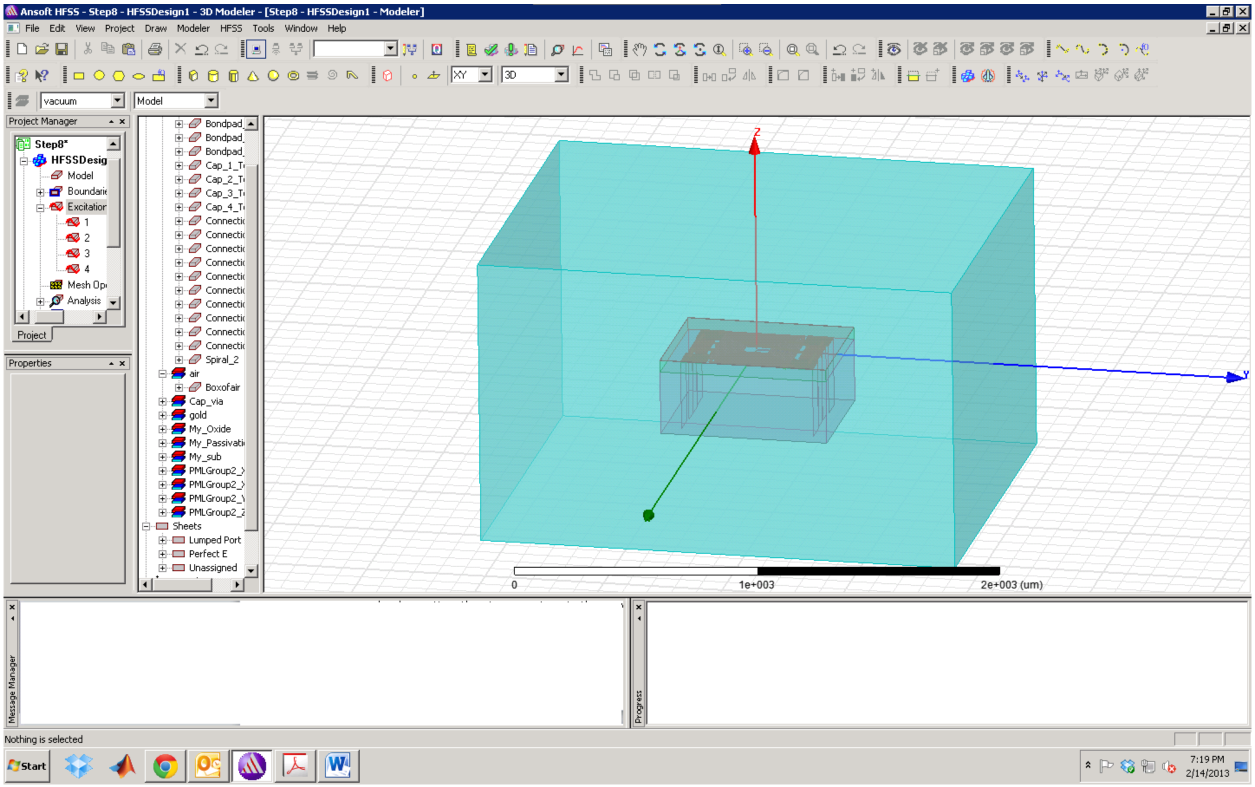


Figure 82: Box of air around everything

Next the radiation boundary needs to be set up. Since it is going to be a permanent boundary the PML wizard is going to be used. PML in this case stands for perfectly matched layer which is a stronger radiation boundary that completely eliminates all radiation from the circuit [16]. In order to use it all the faces of the box of air must be selected. To do this go to Edit->select->by name. Now in the box that pops up find the box of air (this will be easy if it is labeled) and click on it. Then in the right box click all of the faces and press ok. This can be seen in Figure 83.

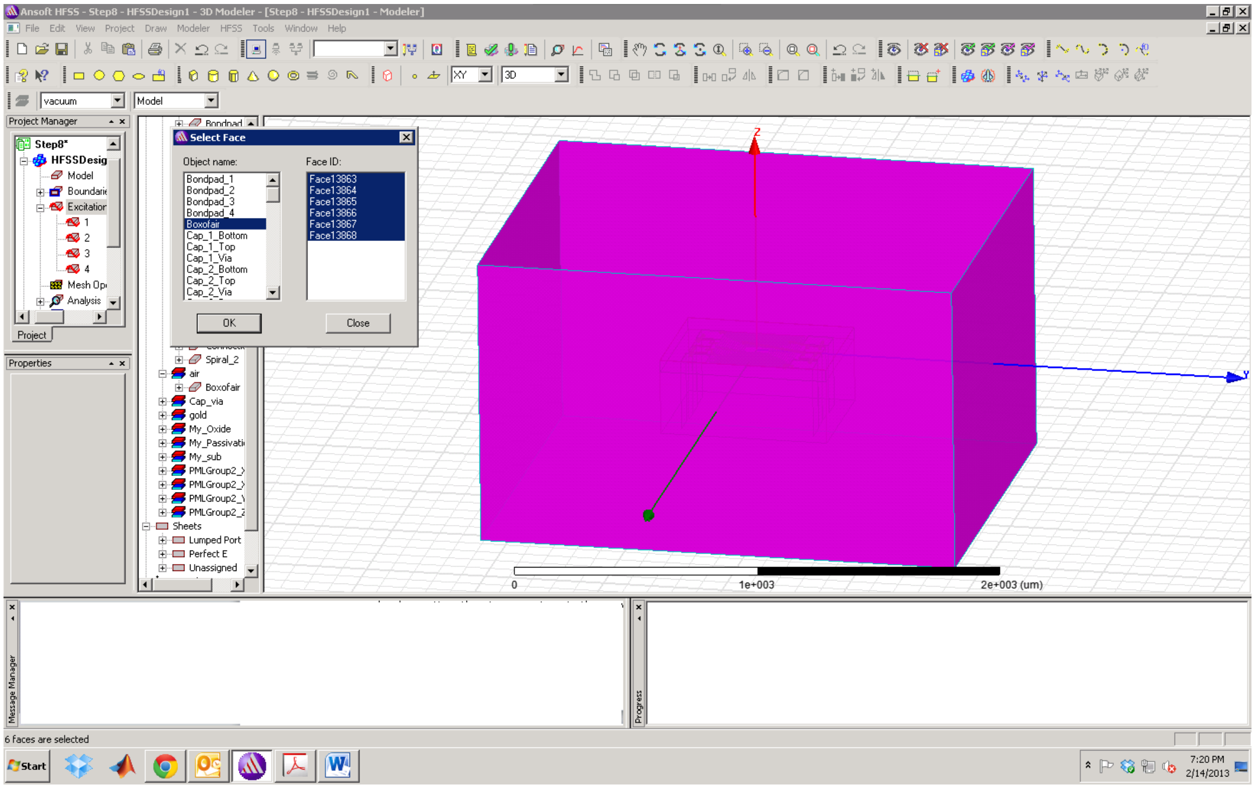


Figure 83: Selecting all face of the box of air

Now go to **HFSS>Boundaries>PML Wizard**. All of the default options will be used so just keep pressing next. Now sit back and watch as boxes form around the box of air, as seen in Figure 84. After they are created click on finish.

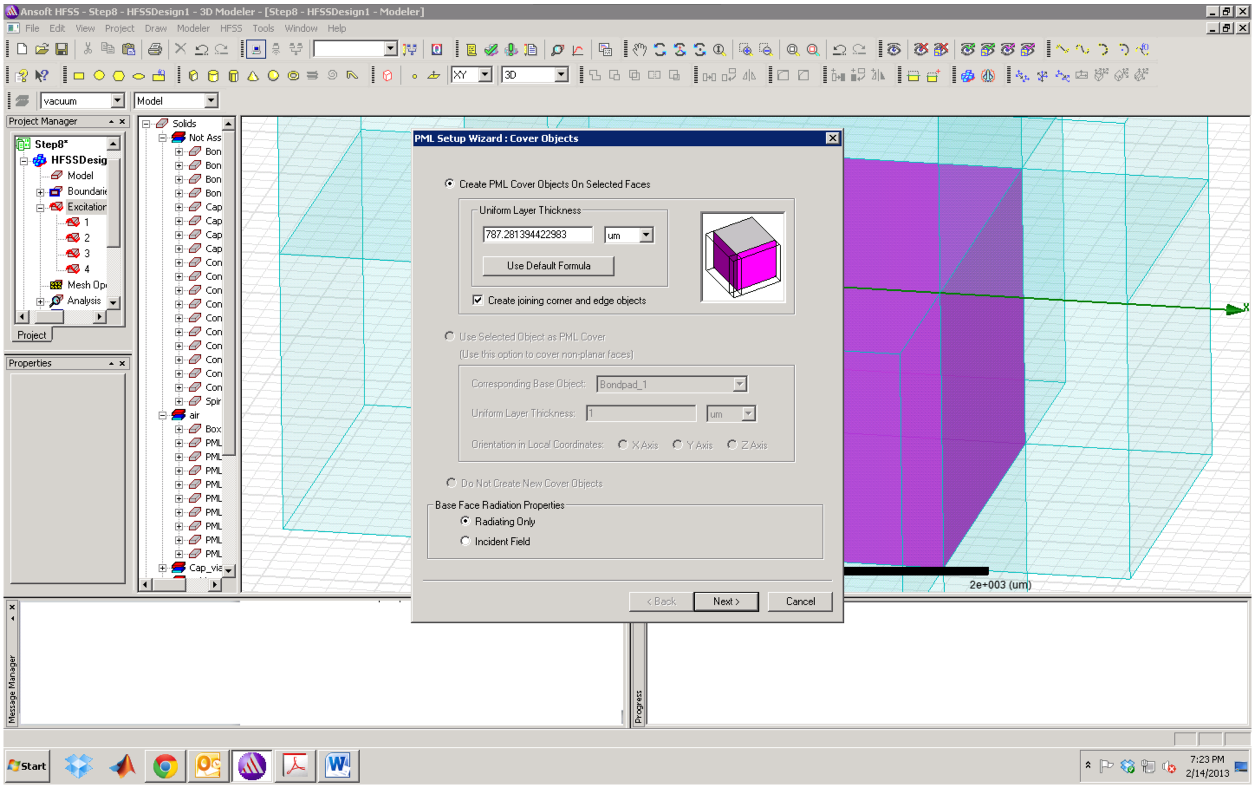


Figure 84: Using PML wizard

Step 12: Next is assigning the material. To do this we select all of the objects that possess the same material. For this circuit first select the piece that was united early with both of the spiral inductors, also select all four of the bottom capacitor plates. Right click one of the selected objects and click on assign material. For these the material gold will be chosen; you can scroll and find it in the list and select it byclicking ok once.

Next select the box of air that the radiation boxes are based off of. Follow the process above to assign the material to air. Once this is complete follow the same process with all of the cap via's which uses a gallium arsenide material. For best effect this material will need to be created. To do this navigate to the assign material menu where a button called create material along the bottom row will be found. Click on this and another menu will pop up. In this change the relative permittivity to 6.4 and leave the rest at their default values, do not forget to give it a name, such as cap_via.

After this select the oxide layer and repeat the same process of creating a new material, this one with relative permittivity at 6.4. For the passivation layer set a relative permittivity to 7.9. For the substrate set the material to silicon. The object tree should now have many different materials listed in it with nothing labeled as vacuum.

Step 13: Next comes setting up the solution. To set up the solution you go to the very left of the screen and into the project manager. If the box is not there simply go to view and select it. In the manager scroll down to analysis and right click, selecting the add solution setup. In this box set the solution frequency to 1.9GHz and under the adaptive passes section set max number of passes to eight and max delta S to .001. Under the advanced tab make sure use radiation boundary on ports is checked and that save fields is checked.

Now that the solution setup is set up right click on it and select add frequency sweep. This will sweep the solution from the max value to the min allowing the ability to create nice graphs. In the frequency sweep options select interpolating and linear step. Start at a frequency of 1GHz and stop at 5GHz with a step size of 0.1GHz. When finish you press ok.

Step 14: Now the design should be ready to simulate. Before this can be done double check that nothing is missing by clicking the validate check mark in the upper toolbars. Once it gives you all check marks close the window and click the exclamation point to start the simulator and generate some results. *Warning: this can take from a few minutes to a few hours depending upon the computers processing power*.

When the simulator is complete go to the project manager and right click on results. The most useful plots for this circuit are the S-parameter ones. To create these go to create modal solution data report ->rectangular plot. Then select S(2,1) and S(3,1) making sure they are in dB, and click on new report at the bottom of the window. Click close to view results. This process can be repeated to give the difference in phase, which is the most important plot of this circuit. To find the difference in phase type ang_deg(S(3,1)/S(2,1)) into the y component and select new report. For different plots experiment with all of the options to get exactly what is required.

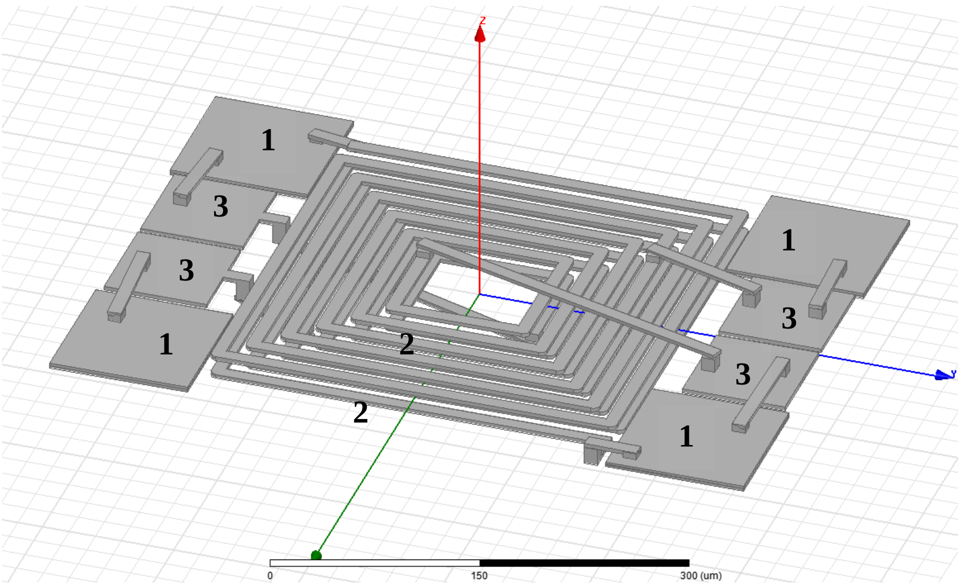
Another very useful result is plotting the electric field onto the circuit itself and seeing the distribution. In order to do this go back to the 3D model by clicking on HFSSDesign in the project manager and select all of the objects. Then go to HFSS->fields->plot fields->E->mag_E. The electric field distribution should now be seen in the 3D modeler window.

All of these results and others can be found in HFSS Results.

Step 15: Other options in HFSS include the ability to set up optimizations. This is an extremely helpful option as it automatically hones in on the answer that is needed by changing designated variables. To designate these variables go to **HFSSD>Design Properties** click on the optimization radio button and select the desired variables. Next right click on optimetrics in the project manager window and select an optimization that is wanted, the normal one is add->optimization. The best optimizer is the quasi newton but this may vary depending upon the circuit. Next setup calculation goals by clicking on setup calculations. The goals of this circuit is to have -3dB (half power) at S(2,1) and S(3,1) and -90degrees at S(3,1)/S(2,1). Then go into the variables tab and be sure to change the min and max values, along with the step size. Beware, the more points used in the optimization the more time it will take, and since it is a geometry change the mesh will have to be recalculated each time which could make this process go on for days at a time. Finally, under the options tab make sure that save fields and mesh is checked, otherwise no results will be saved.

7.4.1 HFSS Results

Implementing the steps as given in the previous section will yield the following results.



1	Capacitors (4.068pF)
2	Spiral Inductors (top and bottom layers)
3	Bondpads

Figure 85: 3D HFSS model

In Figure 85 the three dimensional model of the newly designed hybrid coupler can be seen. It is similar in design to the existing Skyworks coupler except for the existents of a second spiral inductor and no center capacitor.

As explained in step 13 of section 7.4 it was discussed how to plot field results on a model. The result of this can be found in Figure 86 with the electric field.

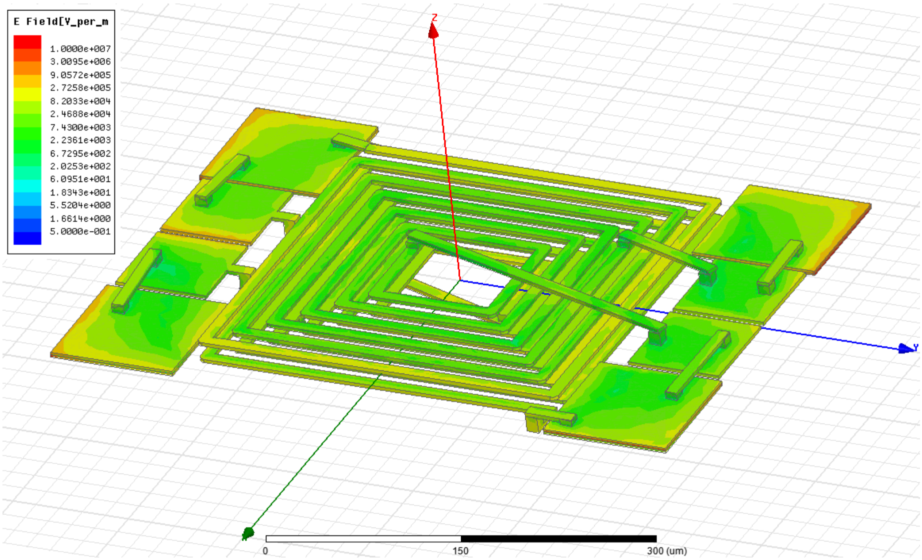


Figure 86: HFSS electric field plot

This plot shows how the skin effect is prevalent throughout the circuit. Another important field representation is the electric field from the cross sectional vantage point. This can be seen in Figure 87.

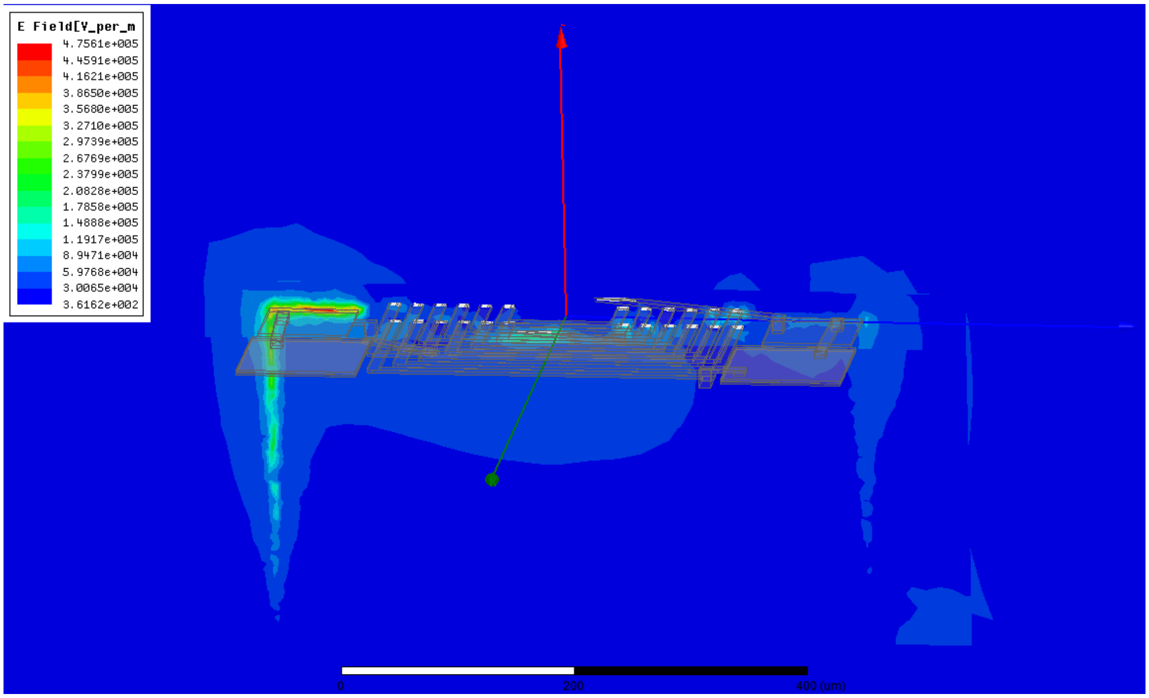


Figure 87: HFSS electric field plot (cross-sectional)

From this view point the build-up of the electric field around ports one and four is prevalent. Next, for the results, are the most important S-parameters, in this case are S_{21} and S_{31}. They can be seen in Figure 88 along with some of the optimized results.

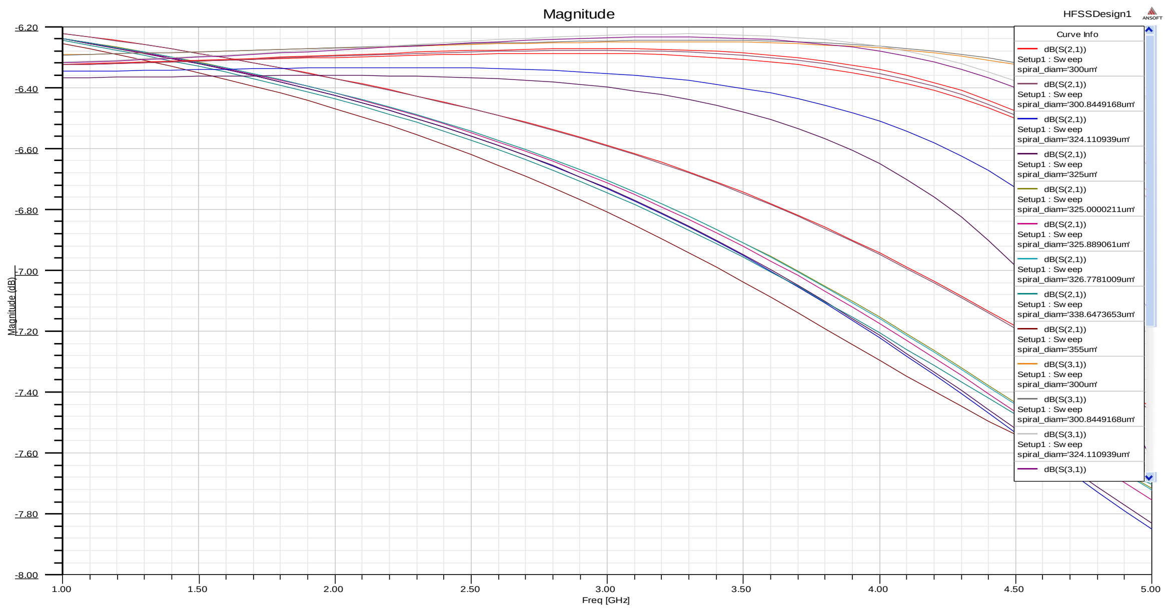


Figure 88: HFSS s-parameter magnitude

From this graph, around the frequency of 1.5GHz, the two parameters cross near -6.3dB. Around the center frequency of 1.9GHz they have already started to diverge but are still within a magnitude of -6.5dB. The goal was to get near 3dB attenuation but unfortunately this circuit did not meet those expectations.

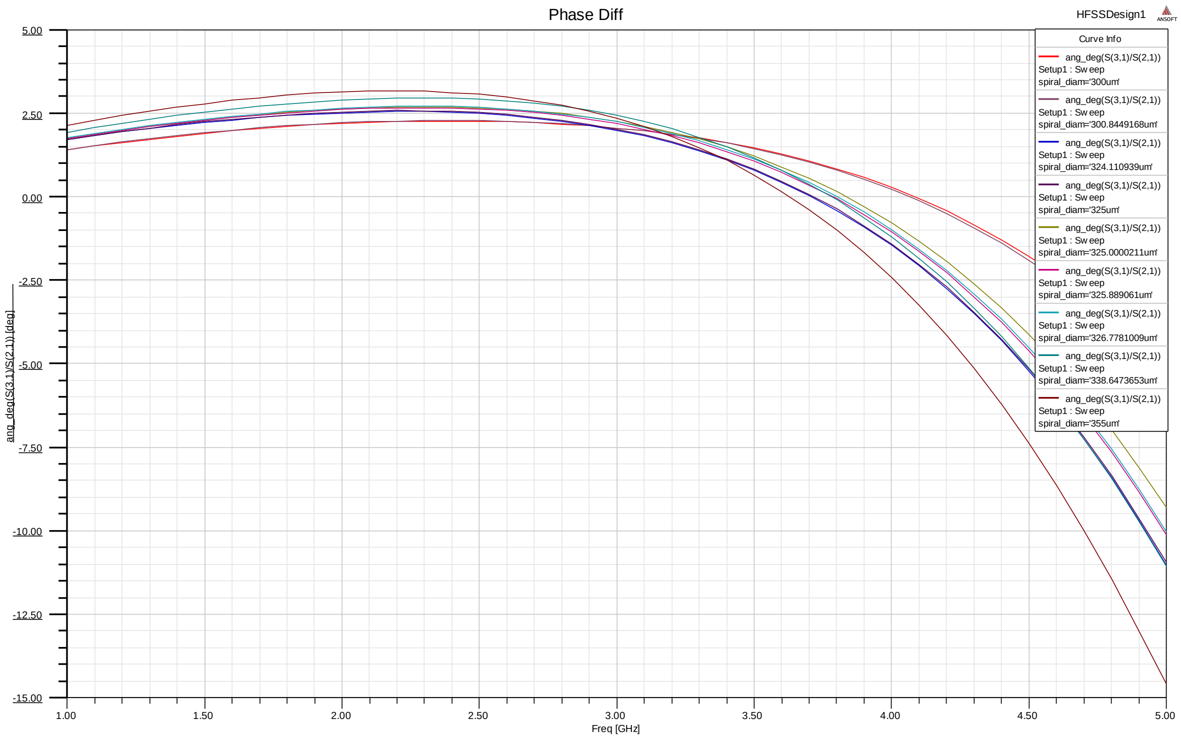


Figure 89: HFSS s-parameter phase difference

The other goal of the project was to reach a 90 degree phase difference between S_{21} and S_{31}. Figure 89 above, plots the difference between the two S-parameters. From 1.0-3.0GHz the phase is 2.50 degrees.

In addition to all of the results shown earlier in this section there is also a way to display many different charts and graphs. There is a wide variety of options to choose from including but not limited to radiation patterns, directivity plots, 3D polar plots, and many more. Some of these options, like the directivity, are not very useful when describing the new hybrid coupler but others, like a 3D polar plot, has some usefulness depending upon what needs to be known. Appendix C contains several more plots.

8 Results

Skyworks' initial design for the coupler with a center frequency of 1.90GHz functioned properly over the 1.85GHz-1.99GHz range yielding a bandwidth of 7.37%. The new design yields an operational range between the frequencies 1.575GHZ-2.111GHz increasing the bandwidth to 18.42%. Additionally, the layout for this new model resulted in having an area of 530x380um, compared to the old 860x640um. Table 2 shows a summary comparing the current Skyworks design to the new one.

Table 2: Comparison of Skyworks model to WPI design

	Skyworks Design	WPI Design
Bandwidth	***7.37%***	***18.42%***
Area	***860x640um***	***530x380um***
Capacitor 1	.375pF (4x)	4.068pF (4x)
Capacitor 2	1.25pF (1x)	N/A
Inductor 1	3nH (2x)	2.961nH (2x)
Inductor 2	3nH (2x)	4.188nH (2x)

A significant drawback that can be observed is the fact that the new capacitor value is significantly larger than the previously implemented capacitors. In implementation this design would call for a larger area. Fortunately, the new design does not require a coupling capacitor to be placed between the inductors, and therefore the design of the spiral inductor was adjusted to make better use of the total area inside the spirals. By removing this capacitor the new design achieves a more stable operation for the required ranges.

9 Conclusion

The main project goals were to increase the bandwidth of the 90-degree hybrid coupler and to attempt to decrease the device area. These goals were successfully achieved as seen in the previous sections with a bandwidth increase of 150%, while also reducing the device area by 63.46%.

Currently, Skyworks offers two separate devices: one at 1.85GHz-1.99GHz operation and another device at 1.71GHz-1.88GHz. With the new developed design, the new design could easily cover the entire spectrum required by these two devices. However, the implementation of this device would require Skyworks to modify its fabrication process. This is because our design has two extra metal layers on top of the existing fabrication process. Changing that process would require a significant time and monetary investment. If Skyworks is willing to do this, they might be able to satisfy the two different frequencies with a single chip.

10 Future Improvements

In ideal lumped components the redesign of the hybrid coupler in this report proved to have excellent results. When those components were translated over to distributed elements, unfortunately the results did not meet expectations. One future improvement that can be pursued is to investigate the distributed circuit implementation side and study needed modifications. Obviously, a distributed element solution represents a more viable solution as it eliminates the parasitic behavior of lumped elements.

Moreover, in order to reduce the area of the circuit topology a few more layers were added, making the whole circuit smaller but at the expense of a taller realization. An improvement would be to take this design and reduce the height so that the number of overall layers is reduced. This would allow the hybrid coupler to be manufactured within the limitations of the Skyworks fabrication process.

Appendix A

Below is the MATLAB code used to import and interpret the results taken from the network analyzer.

```
% James Corsini
% importing data from network analyzer

clear all; close all; clc;
% change the variable mynum to the case that you want to see
mynum=5;
switch mynum
    case 1
        % Ecal 90 Degree Hybrid Device 1
        directory  =  '\\ece-homes.ece.wpi.edu\jncorsini\MQP\WPI  Data\WPI  Data\ECal\90-DEGREE HYBRID\Device 1';
        filename = 'SMATRIX.s4p';
        head='Ecal 90 Degree Hybrid Device 1';
    case 2
        % Ecal 90 Degree Hybrid Device 2
        directory  =  '\\ece-homes.ece.wpi.edu\jncorsini\MQP\WPI  Data\WPI  Data\ECal\90-DEGREE HYBRID\Device 2';
        filename = 'SMATRIX.s4p';
        head='Ecal 90 Degree Hybrid Device 2';
    case 3
        % Ecal 90 Degree Hybrid Device 3
        directory  =  '\\ece-homes.ece.wpi.edu\jncorsini\MQP\WPI  Data\WPI  Data\ECal\90-DEGREE HYBRID\Device 3';
        filename = 'SMATRIX.s4p';
        head='Ecal 90 Degree Hybrid Device 3';
    case 4
        % Ecal 90 Degree Hybrid Device 4
        directory  =  '\\ece-homes.ece.wpi.edu\jncorsini\MQP\WPI  Data\WPI  Data\ECal\90-DEGREE HYBRID\Device 4';
        filename = 'SMATRIX.s4p';
        head='Ecal 90 Degree Hybrid Device 4';
    case 5
        % Ecal Thru Device 1
        directory = '\\ece-homes.ece.wpi.edu\jncorsini\MQP\WPI Data\WPI Data\ECal\THRU\Device 1';
        filename = 'THRU.s2p';
        head='Ecal Thru Device 1';
    case 6
        % Ecal Thru Device 2
        directory = '\\ece-homes.ece.wpi.edu\jncorsini\MQP\WPI Data\WPI Data\ECal\THRU\Device 2';
        filename = 'THRU.s2p';
        head='Ecal Thru Device 2';
    case 7
        % Ecal Thru Device 3
        directory = '\\ece-homes.ece.wpi.edu\jncorsini\MQP\WPI Data\WPI Data\ECal\THRU\Device 3';
        filename = 'THRU.s2p';
        head='Ecal Thru Device 3';
    case 8
        % Ecal Thru Device 4
        directory = '\\ece-homes.ece.wpi.edu\jncorsini\MQP\WPI Data\WPI Data\ECal\THRU\Device 4';
        filename = 'THRU.s2p';
        head='Ecal Thru Device 4';
    case 9
        % TRL 90 Degree Hybrid Device 1
        directory  =  '\\ece-homes.ece.wpi.edu\jncorsini\MQP\WPI  Data\WPI  Data\TRL\90-DEGREE HYBRID\Device 1';
        filename = 'SMATRIX.s4p';
        head='TRL 90 Degree Hybrid Device 1';
    case 10
```

```
        % TRL 90 Degree Hybrid Device 2
        directory   =   '\\ece-homes.ece.wpi.edu\jncorsini\MQP\WPI   Data\WPI   Data\TRL\90-DEGREE
HYBRID\Device 2';
        filename = 'SMATRIX.s4p';
        head='TRL 90 Degree Hybrid Device 2';
    case 11
        % TRL 90 Degree Hybrid Device 3
        directory   =   '\\ece-homes.ece.wpi.edu\jncorsini\MQP\WPI   Data\WPI   Data\TRL\90-DEGREE
HYBRID\Device 3';
        filename = 'SMATRIX.s4p';
        head='TRL 90 Degree Hybrid Device 3';
    case 12
        % TRL 90 Degree Hybrid Device 4
        directory   =   '\\ece-homes.ece.wpi.edu\jncorsini\MQP\WPI   Data\WPI   Data\TRL\90-DEGREE
HYBRID\Device 4';
        filename = 'SMATRIX.s4p';
        head='TRL 90 Degree Hybrid Device 4';
end


if (mynum==5||mynum==6||mynum==7||mynum==8)
    display('Only two ports')
    s_parameters = import_sNp(directory,filename);

    S11_dB = squeeze(s_parameters.dB(1,1,:));
    S12_dB = squeeze(s_parameters.dB(1,2,:));
    S21_dB = squeeze(s_parameters.dB(2,1,:));
    S22_dB = squeeze(s_parameters.dB(2,2,:));
    freq = s_parameters.freq;

    figure(1);
    plot(freq,S11_dB,'color',[0.75 0 0]);
    title([head,' S11/21 (dB)']);
    xlabel('Frequency (GHz)');
    ylabel('S-Parameter Magnitude (dB)');
    hold on; grid on;
    plot(freq,S12_dB,'color',[0 0 0.75]);
    plot(freq,S21_dB,'color',[0 0.75 0]);
    plot(freq,S22_dB,'color',[0.25 0 0]);
else
    s_parameters = import_sNp(directory,filename);

    phase_S21_unwrap = (180/pi)*unwrap((pi/180)*squeeze(s_parameters.phase(2,1,:)));
    phase_S31_unwrap = (180/pi)*unwrap((pi/180)*squeeze(s_parameters.phase(3,1,:)));
    S21_dB = squeeze(s_parameters.dB(2,1,:));
    S31_dB = squeeze(s_parameters.dB(3,1,:));
    amp_diff = S21_dB - S31_dB;
    phase_diff = phase_S21_unwrap - phase_S31_unwrap;
    freq = s_parameters.freq;
    f_low = 1.8;
    f_high = 2.0;

    figure(1);
    plot(freq,S21_dB,'color',[0.75 0 0]);
    xlabel('Frequency (GHz)');
    ylabel('S-Parameter Magnitude (dB)');
    title([head,' S21/31 (dB)']);
    hold on;
    grid on;
    plot(freq,S31_dB,'color',[0 0 0.75]);
    figure(2);
    plot(freq,amp_diff,'color',[0 0.75 0]);
    xlabel('Frequency (GHz)');
    ylabel('S-Parameter Magnitude (dB)');
    grid on;
    title([head,' Amp Diff']);
    ylim([-2 0]);
    xlim([f_low f_high]);
    figure(3);
    plot(freq,phase_diff,'color',[0 0.75 0]);
```

```
    xlabel('Frequency (GHz)');
    ylabel('S-Parameter Phase (degrees)');
    title([head,' Phase Diff']);
    grid on;
    ylim([-110 -70]);
end
```

Following are the graphs for devices 2-4 for the electronic calibration.

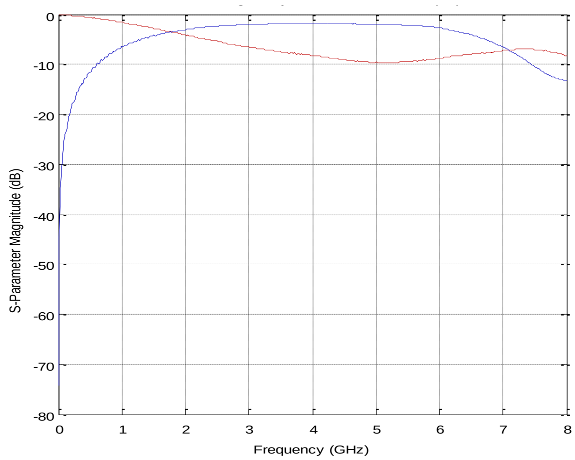


Figure A.1: Hybrid Coupler 2 S_{21} and S_{31} Magnitude (dB)

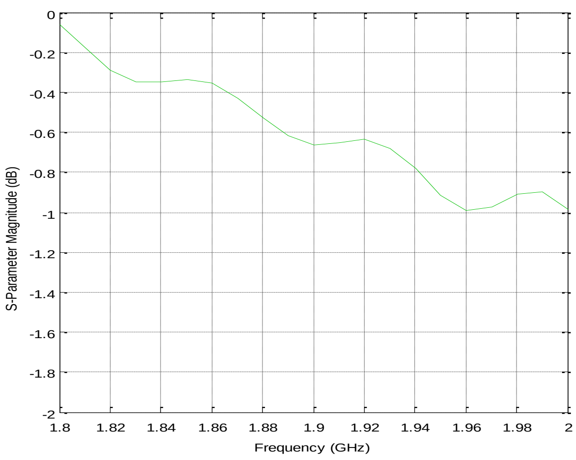


Figure A.2: Hybrid Coupler 2 S_{21} and S_{31} Amplitude Difference

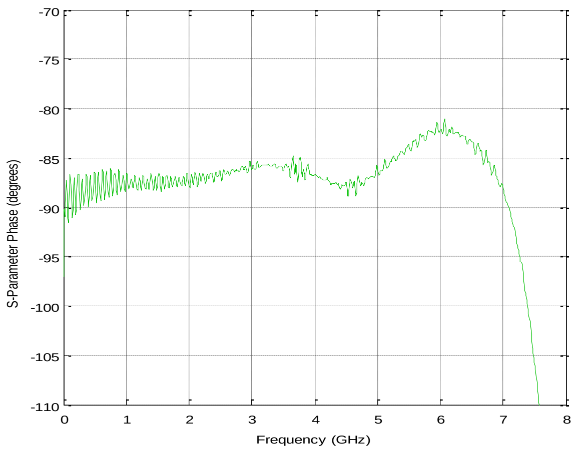


Figure A.3: Hybrid Coupler 2 S_{21} and S_{31} Phase Difference (degrees)

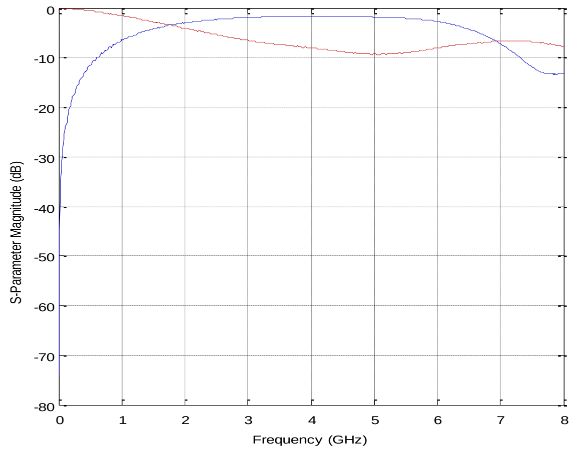


Figure A.4: Hybrid Coupler 3 S_{21} and S_{31} Magnitude (dB)

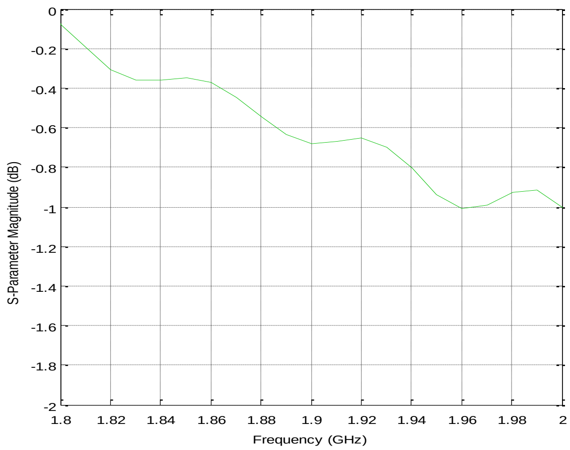


Figure A.5: Hybrid Coupler 3 S_{21} and S_{31} Amplitude Difference

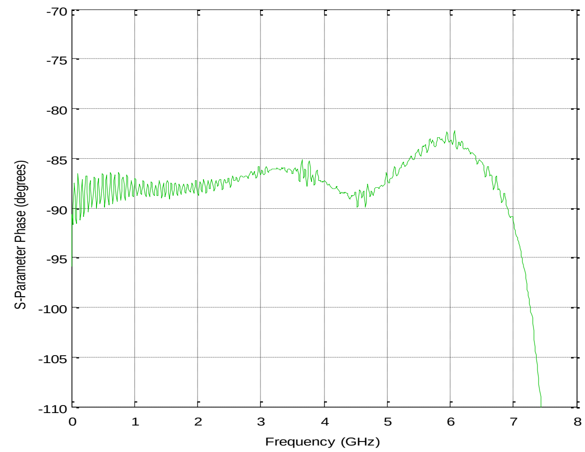


Figure A.6: Hybrid Coupler 3 S_{21} and S_{31} Phase Difference (degrees)

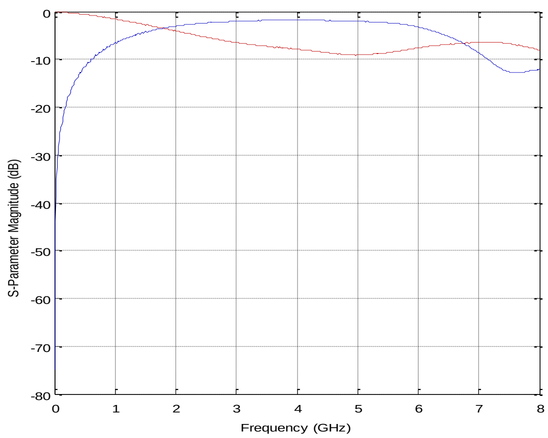


Figure A.7: Hybrid Coupler 3 S_{21} and S_{31} Magnitude (dB)

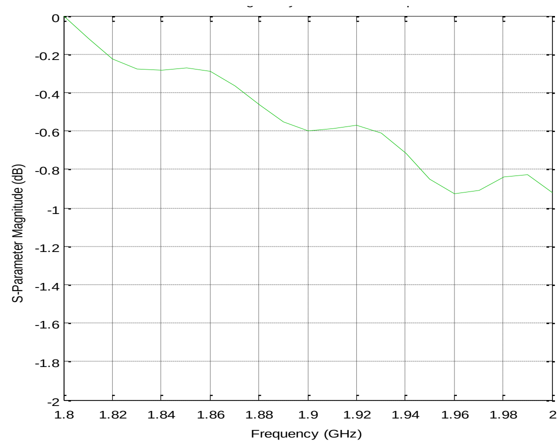


Figure A.8: Hybrid Coupler 4 S_{21} and S_{31} Amplitude Difference

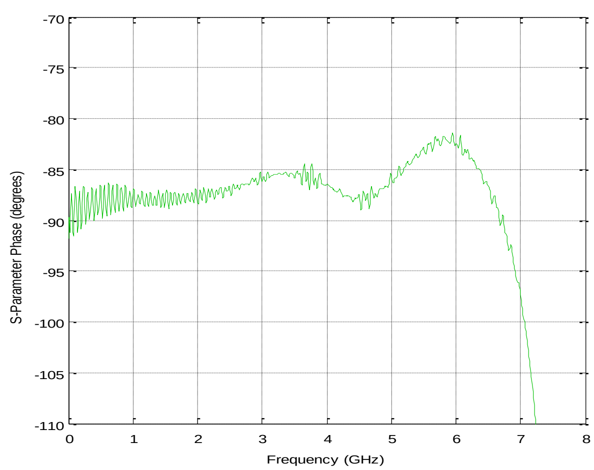


Figure A.9: Hybrid Coupler 4 S_{21} and S_{31} Phase Difference (degrees)

Following are the graphs for devices 1-4 for the transmission line calibration.

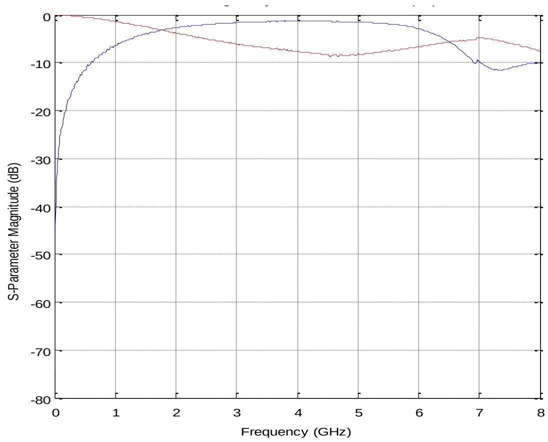


Figure A.10: Hybrid Coupler 1 S_{21} and S_{31} Magnitude (dB)

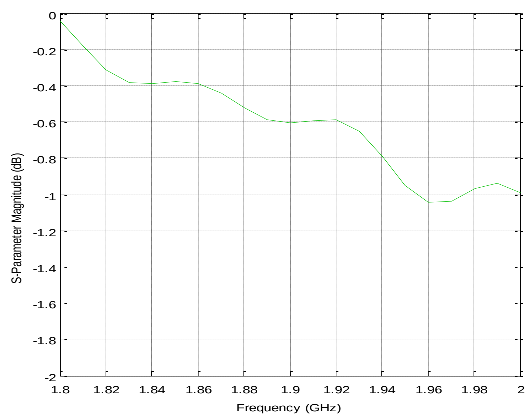


Figure A.11: Hybrid Coupler 1 S_{21} and S_{31} Amplitude Difference

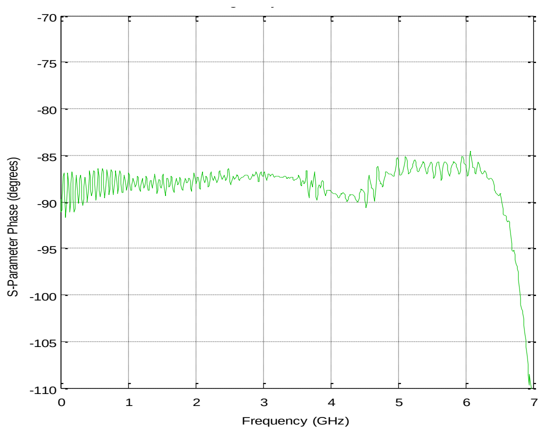


Figure A.12: Hybrid Coupler 1 S_{21} and S_{31} Phase Difference (degrees)

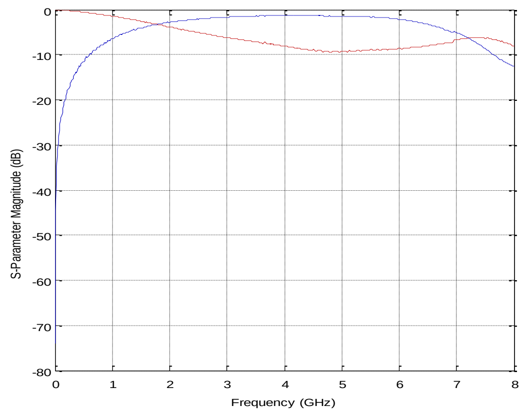


Figure A.13: Hybrid Coupler 2 S_{21} and S_{31} Magnitude (dB)

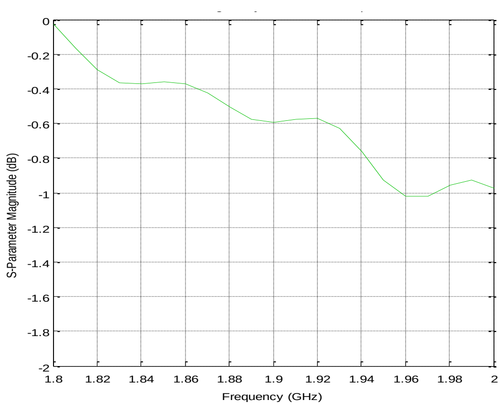


Figure A.14: Hybrid Coupler 2 S_{21} and S_{31} Amplitude Difference

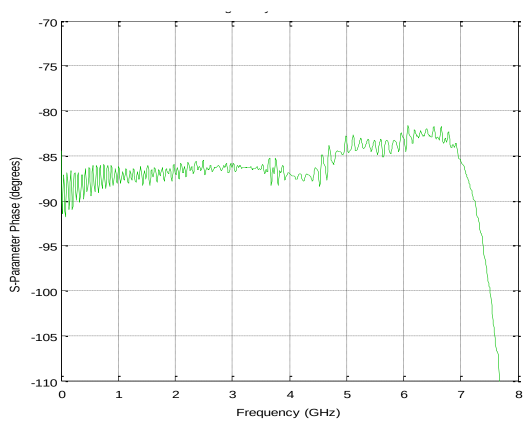


Figure A.15: Hybrid Coupler 2 S_{21} and S_{31} Phase Difference (degrees)

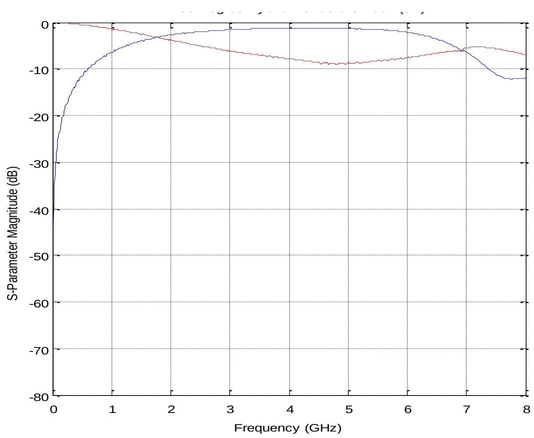


Figure A.16: Hybrid Coupler 3 S_{21} and S_{31} Magnitude (dB)

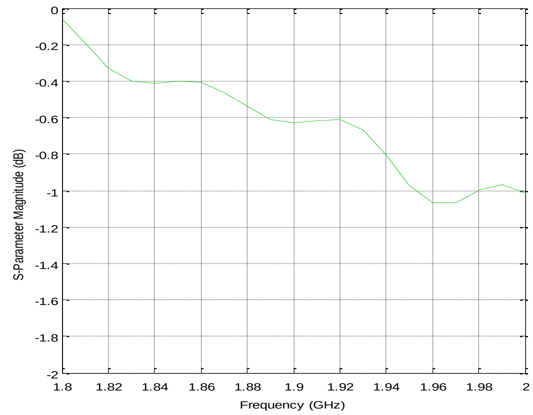


Figure A. 17: Hybrid Coupler 3 S_{21} and S_{31} Amplitude Difference

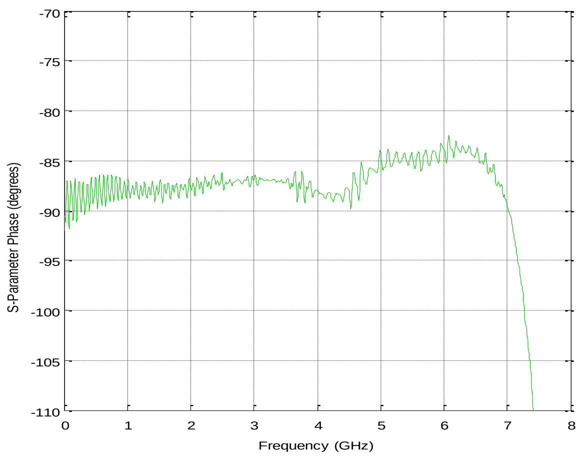


Figure A. 18: Hybrid Coupler 3 S_{21} and S_{31} Phase Difference (degrees)

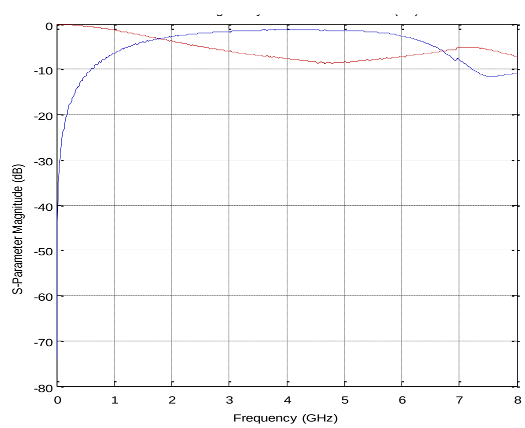


Figure A. 19: Hybrid Coupler 4 S_{21} and S_{31} Magnitude (dB)

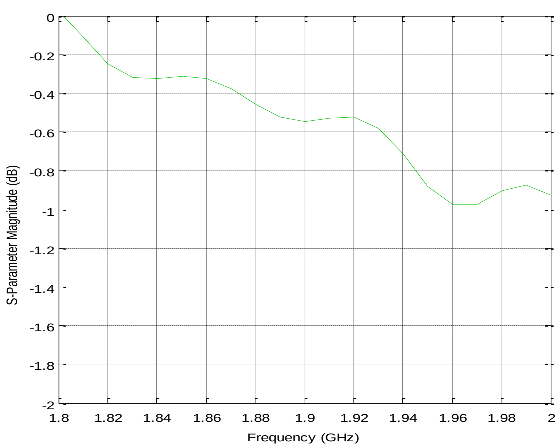


Figure A. 20: Hybrid Coupler 4 S_{21} and S_{31} Amplitude Difference

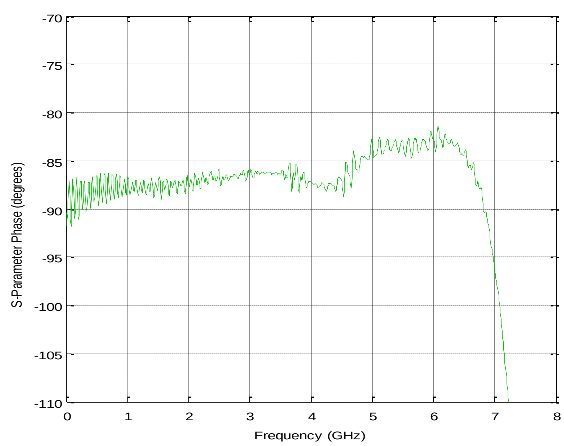


Figure A. 21: Hybrid Coupler 4 S_{21} and S_{31} Phase Difference (degrees)

Appendix B

The voltage reflection coefficient is the ratio of the reflected and incident voltage wave at the load. This ratio is shown in equation 1.

$$-\Gamma = -\frac{V_0^-}{V_0^+} = \frac{I_0^-}{I_0^+} \qquad \text{(B.1)}$$

The voltage coefficient can be represented in terms of characteristic impedance and load as shown in equation 2.

$$\Gamma = \frac{Z_L - Z_0}{Z_L + Z_0} \qquad \text{(B.2)}$$

When the reflection coefficient is equal zero ($Z_L = Z_0$), the load is matched to TL. Most of the time Z_L is complex quantity therefore Γ is complex quantity too. Also, depends on the length of TL. It is described as shown in equation 3.

$$\Gamma_l = \Gamma e^{-j2\beta l} \qquad \text{(B.3)}$$

To find the input impedance at a particular point in TL, simply solve for Z_L from equation two. The result is obtained in equation four.

$$Z_{in} = Z(l) = Z_0 \frac{1+\Gamma_l}{1-\Gamma_l} \qquad \text{(B.4)}$$

Now, substitute the reflection coefficient of equation three into equation four in order to obtain the input impedance in terms of normalized impedance ($z_L = \frac{Z_L}{Z_0}$), characteristic impedance, and phase constant ($\beta = \frac{2\pi}{\lambda}$). The equation five shows the result of this mathematical manipulation.

$$Z_{in} = Z_0 \frac{z_L + j tan(\beta l)}{1 + j z_L tan(\beta l)} \qquad \text{(B.5)}$$

If the length $l = \frac{\lambda}{2}$ then $Z_{in} = Z_L$. So, if half- lambda TL is added there is no loss on the system. In case the length $l = \frac{\lambda}{4}$ then $Z_{in} = \frac{{Z_L}^2}{Z_0}$ and if $Z_L = Z_0$ the system is matched and there is zero reflection coefficient.

Appendix C

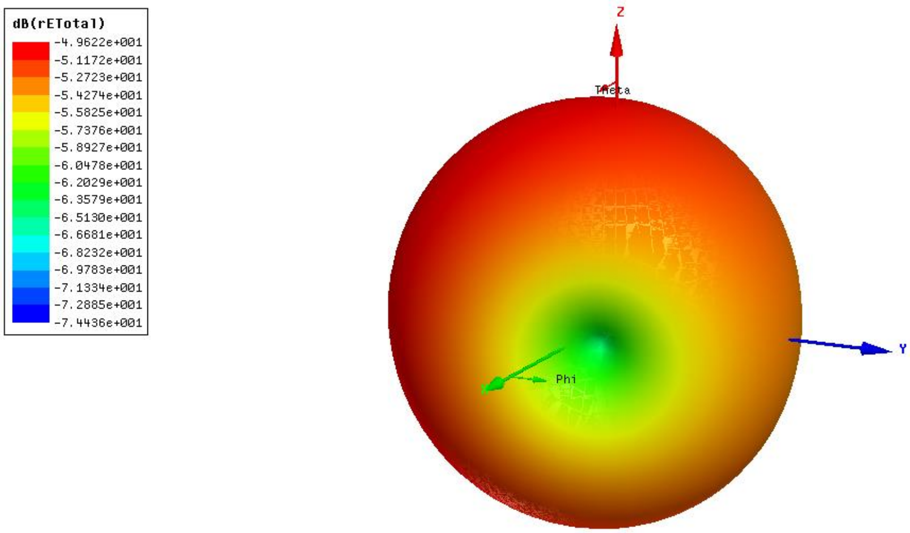


Figure C.1 Appendix: 3D polar plot

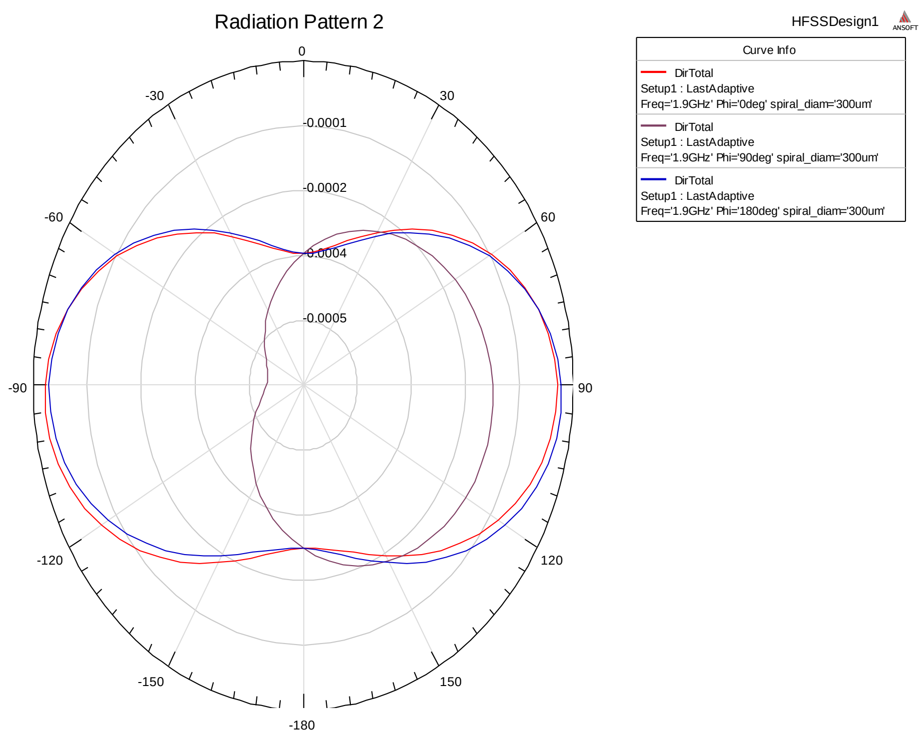


Figure C.2 Appendix: Directivity

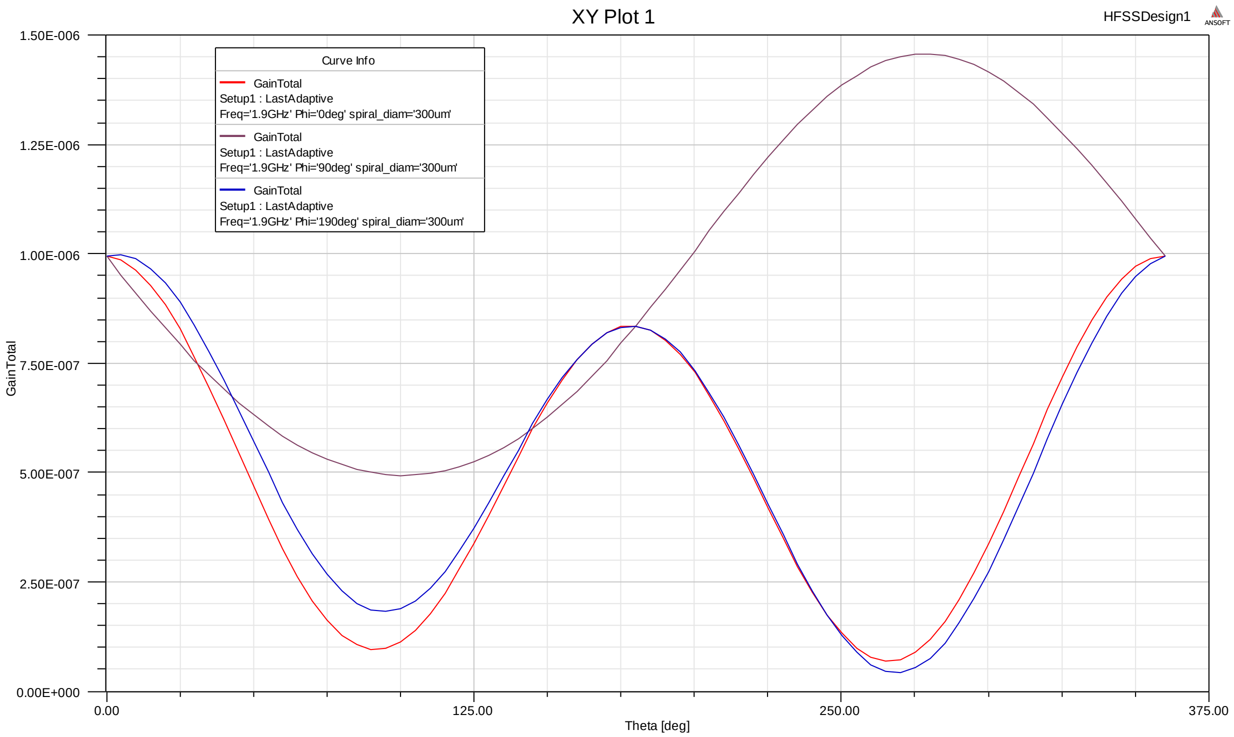


Figure C.3 Appendix: XY gain plot

Appendix D

	[Z]	[Y]	[h]	[ABCD]
[Z]	$\begin{bmatrix} Z_{11} & Z_{12} \\ Z_{21} & Z_{22} \end{bmatrix}$	$\begin{bmatrix} \frac{Z_{22}}{\Delta Z} & -\frac{Z_{12}}{\Delta Z} \\ -\frac{Z_{21}}{\Delta Z} & \frac{Z_{11}}{\Delta Z} \end{bmatrix}$	$\begin{bmatrix} \frac{\Delta Z}{Z_{22}} & \frac{Z_{12}}{Z_{22}} \\ -\frac{Z_{21}}{Z_{22}} & \frac{1}{Z_{22}} \end{bmatrix}$	$\begin{bmatrix} \frac{Z_{11}}{Z_{21}} & \frac{\Delta Z}{Z_{21}} \\ \frac{1}{Z_{21}} & \frac{Z_{22}}{Z_{21}} \end{bmatrix}$
[Y]	$\begin{bmatrix} \frac{Y_{22}}{\Delta Y} & -\frac{Y_{12}}{\Delta Y} \\ -\frac{Y_{21}}{\Delta Y} & \frac{Y_{11}}{\Delta Y} \end{bmatrix}$	$\begin{bmatrix} Y_{11} & Y_{12} \\ Y_{21} & Y_{22} \end{bmatrix}$	$\begin{bmatrix} \frac{1}{Y_{11}} & -\frac{Y_{12}}{Y_{11}} \\ \frac{Y_{21}}{Y_{11}} & \frac{\Delta Y}{Y_{11}} \end{bmatrix}$	$\begin{bmatrix} -\frac{Y_{22}}{Y_{21}} & -\frac{1}{Y_{21}} \\ -\frac{\Delta Y}{Y_{21}} & -\frac{Y_{11}}{Y_{21}} \end{bmatrix}$
[h]	$\begin{bmatrix} \frac{\Delta h}{h_{22}} & \frac{h_{12}}{h_{22}} \\ -\frac{h_{21}}{h_{22}} & \frac{1}{h_{22}} \end{bmatrix}$	$\begin{bmatrix} \frac{1}{h_{11}} & -\frac{h_{12}}{h_{11}} \\ \frac{h_{21}}{h_{11}} & \frac{\Delta h}{h_{11}} \end{bmatrix}$	$\begin{bmatrix} h_{11} & h_{12} \\ h_{21} & h_{22} \end{bmatrix}$	$\begin{bmatrix} -\frac{\Delta h}{h_{21}} & -\frac{h_{11}}{h_{21}} \\ -\frac{h_{22}}{h_{21}} & -\frac{1}{h_{21}} \end{bmatrix}$
[ABCD]	$\begin{bmatrix} \frac{A}{C} & \frac{\Delta ABCD}{C} \\ \frac{1}{C} & \frac{D}{C} \end{bmatrix}$	$\begin{bmatrix} \frac{D}{B} & -\frac{\Delta ABCD}{B} \\ -\frac{1}{B} & \frac{A}{B} \end{bmatrix}$	$\begin{bmatrix} \frac{B}{D} & \frac{\Delta ABCD}{D} \\ -\frac{1}{D} & \frac{C}{D} \end{bmatrix}$	$\begin{bmatrix} A & B \\ C & D \end{bmatrix}$

Figure D.4: Conversion between different network representations [7]

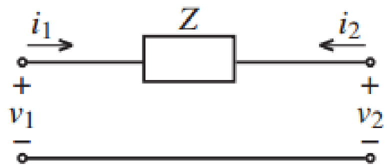
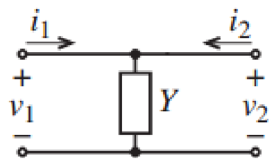
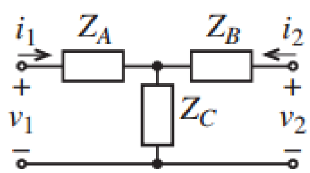
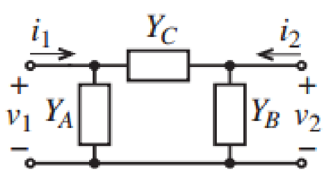
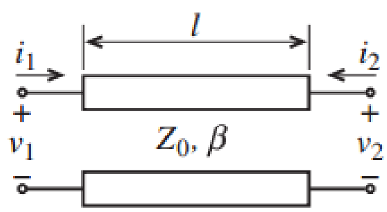
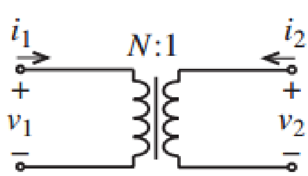
Circuit	ABCD Parameters	
	$A = 1$	$B = Z$
	$C = 0$	$D = 1$
	$A = 1$	$B = 0$
	$C = Y$	$D = 1$
	$A = 1 + \frac{Z_A}{Z_C}$	$B = Z_A + Z_B + \frac{Z_A Z_B}{Z_C}$
	$C = \frac{1}{Z_C}$	$D = 1 + \frac{Z_B}{Z_C}$
	$A = 1 + \frac{Y_B}{Y_C}$	$B = \frac{1}{Y_C}$
	$C = Y_A + Y_B + \frac{Y_A Y_B}{Y_C}$	$D = 1 + \frac{Y_A}{Y_C}$
	$A = \cos(\beta l)$	$B = jZ_0 \sin(\beta l)$
	$C = \frac{j\sin(\beta l)}{Z_0}$	$D = \cos(\beta l)$
	$A = N$	$B = 0$
	$C = 0$	$D = \frac{1}{N}$

Figure D.5: ABCD-parameters of several two-port circuits [7]

Bibliography

[1] "Couplers and Splitters," 29 January 2010. [Online]. Available: http://www.microwaves101.com/encyclopedia/couplers.cfm. [Accessed November 2012].

[2] "The Decibel Defined," 5 August 2008. [Online]. Available: http://sss-mag.com/db.html. [Accessed September 2012].

[3] G. Breed, "Classic Design for Lumped Element and Transmission Line 90-Degree Coupler," *High Frequency Eletronics*, pp. 62-66, September 2007.

[4] F. T. Ulaby, E. Michielssen and U. Ravaioli, Fundamentals of Applied Electromagnetics (6th Edition), Prentice Hall, 2010.

[5] R. Ludwig and G. Bogdanov, RF Circuit Design: Theory & Applications (2nd Edition), Prentice Hall, 2008.

[6] "Scattering Parameters," February 2013. [Online]. Available: http://en.wikipedia.org/wiki/Scattering_parameters. [Accessed November 2012].

[7] Ansys HFSS, "Ansys HFSS," [Online]. Available: http://www.ansys.com/Products/Simulation+Technology/Electromagnetics/High-Performance+Electronic+Design/ANSYS+HFSS. [Accessed January 2013].

[8] MathWorks, "MATLAB," [Online]. Available: http://www.mathworks.com/products/matlab/ . [Accessed January 2013].

[9] HyperPhysics, "Capacitance of Parallel Plates," 2000. [Online]. Available: http://hyperphysics.phy-astr.gsu.edu/hbase/electric/pplate.html . [Accessed September 2012].

[10] S. S. Mohan, M. del Mar Hershenson, S. P. Boyd and T. H. Lee, "Simple Accurate Expressions for Planar Spiral Inductances," *IEEE Journal of Solid-State Circuits*, pp. 1419-1424, 1999.

[11] Skyworks Inc., "Markets," [Online]. Available: http://www.skyworksinc.com/Markets.aspx. [Accessed February 2013].

[12] Skyworks Inc, "HY19-12,HY19-12LF: 90-Degree Hybrid 1.85-1.99GHz," 8 June 2006. [Online]. Available: http://www.skyworksinc.com/uploads/documents/200234B.pdf.

[13] Stanford University, "Integrated Spiral Inductor Calculator," [Online]. Available: http://www-smirc.stanford.edu/spiralCalc.html. [Accessed January 2013].

[14] J. Brubaker, "What Is Metal Oxide?," [Online]. Available: http://www.ehow.com/about_6160418_metal-oxide-passivation_.html. [Accessed February

2013].

[15] NACE International, "The Passivation Layer," [Online]. Available: http://events.nace.org/library/corrosion/Surface/Passivation-layer.asp. [Accessed February 2013].

[16] S. G. Johnson, "Notes on Perfectly Matched Layers (PMLs)," March 2010. [Online]. Available: http://math.mit.edu/~stevenj/18.369/pml.pdf. [Accessed February 2013].