

# **SIGNAL INTEGRITY VALIDATION IN MODELLING AND SIMULATION**

**Trainer: Kelvin Teh  
(By Oriontrain Sdn. Bhd.)  
Training Provider: PSDC**

# Engagement Model

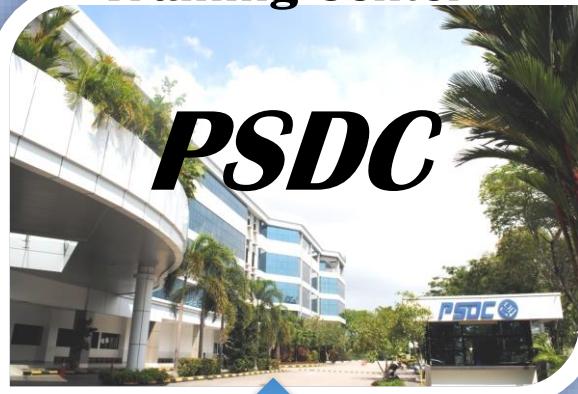
## Factories All industries



## Training

*Customised Trainings  
According to Demands  
and Requirements*

## Training Center



## Training Solutions

## Industrial Solutions

*Customised Solutions  
According to  
Requirements*

Technical  
Consultant Panel

## ORIONTRAIN

- Design training outline
- Design training content
- Conduct training session

Technical Solution  
Consultant & Provider

## ORIONPLEX

- Solution Consultant
- Solution Designer
- Total Solution Provider

# Introduction to Modelling and Simulation Tools

# Introduction to Modelling and Simulation Tools

- Signal integrity validation: requires lots of theory and calculation to model and predict behaviour of high frequency electronic components accurately
- Familiarity with various fields such as circuit theory, EM theory, vector calculus, etc. required to perform manual calculations
- Calculations are time consuming and error prone

# Introduction to Modelling and Simulation Tools

- Making changes to parameters and determining how it affects the system requires time and effort
- Might need to research equations, algorithms and libraries to use
- Visualisation needs to be performed manually by plotting on graph paper or generating the equations in Excel or programming language

# Introduction to Modelling and Simulation Tools

- Modelling and simulation tools allows equations and algorithms to be simulated before implementation
- Contains built-in electronic components, equations and algorithms for various electronics and signal integrity applications
- Parameters can be tuned easily and results observed quickly

# Introduction to Modelling and Simulation Tools

- Design typically modelled and simulated using modelling and simulation tools first before actual implementation on PCB board
- Allows easy experimentation with various electronic models and parameter values without requiring actual hardware
- Model behaviour at different frequencies can be easily simulated as most tools allow frequency sweeping

# Introduction to Modelling and Simulation Tools

- Advantages of modelling and simulation tool
  - Can model behaviour of electronic systems fast without requiring hardware
  - Easy to debug and tune parameters
  - Normally contains rich library of pre-defined electronic components
  - Various visualisation methods (time domain, frequency domain, line charts, eye diagram, etc.) can be used
  - Many of these tools allow automatic generation of PCB layout from schematic

# Introduction to Modelling and Simulation Tools

- Disadvantages of modelling and simulation tool
  - For complex models, the simulation might take time to run, requiring relatively fast computers
  - Not all system behaviour can be accurately modelled/simulated
  - Some modelling and simulation tools are not free, and can be expensive
  - There can sometimes be slight differences in behaviour of model and implemented system, due to application exceeding model bandwidth, component tolerances, algorithm approximations, etc. used by the modelling system

# Common Modelling and Simulation Tools for Machine Vision and AI

- Examples of modelling/simulation tools commonly used for electronic and signal integrity modelling/simulation:
  - MATLAB
  - Proteus
  - EasyEDA
  - ANSYS HFSS
  - Pathwave ADS
  - Sigrity SI

# SIGNAL INTEGRITY VALIDATION IN MODELLING AND SIMULATION

Module 1: Pathwave ADS Basics

# Overview of ADS

# Workspace, Library and Cell

## Workspaces

Different than a project, a workspace give you access to libraries that contain cells, where the cells contain designs.

## Libraries

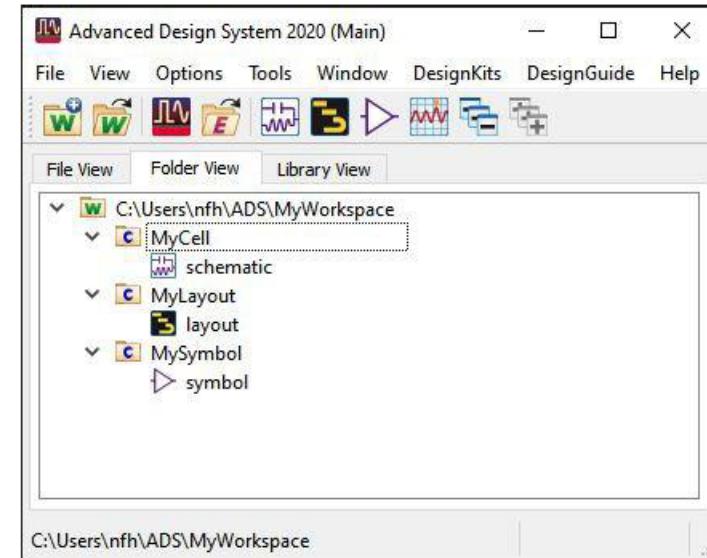
In a workspace, libraries are a collection of cells. But libraries can also be Process Design Kits (PDKs) or separate folders outside of the workspace.

## Cells

Cells are folders that replace design files in the old networks directory. Cells are in libraries and usually contain different views of a design - this means layouts, schematics, and a symbol.

## Symbols

The symbol represents all views in the cell. Usually one symbol is all you need for a cell.

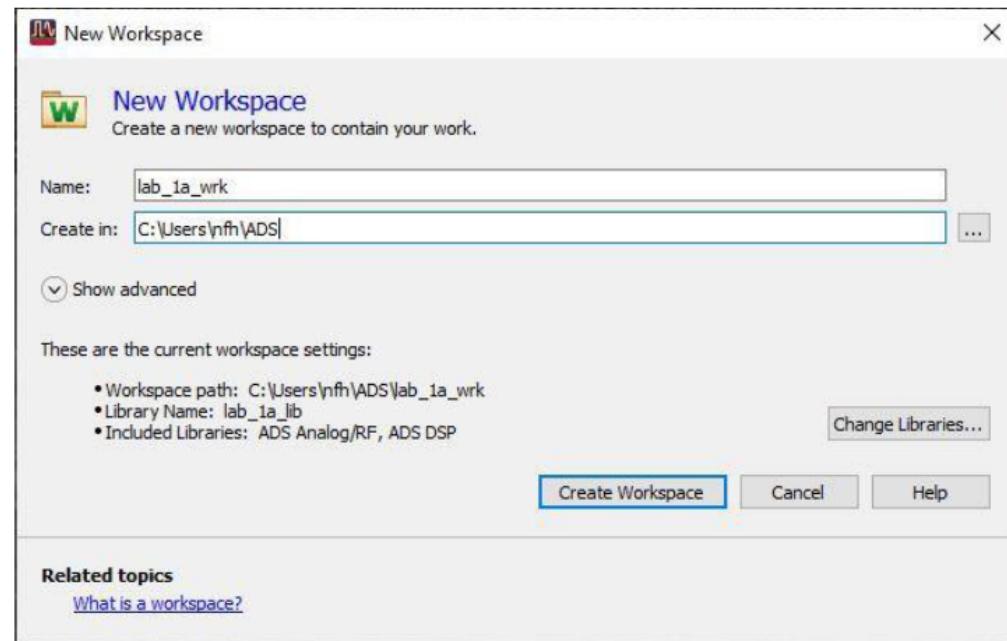


Workspace-Library Hierarchy

# Modelling in ADS: Creating a Workspace

## Step 1 – Create a New Workspace

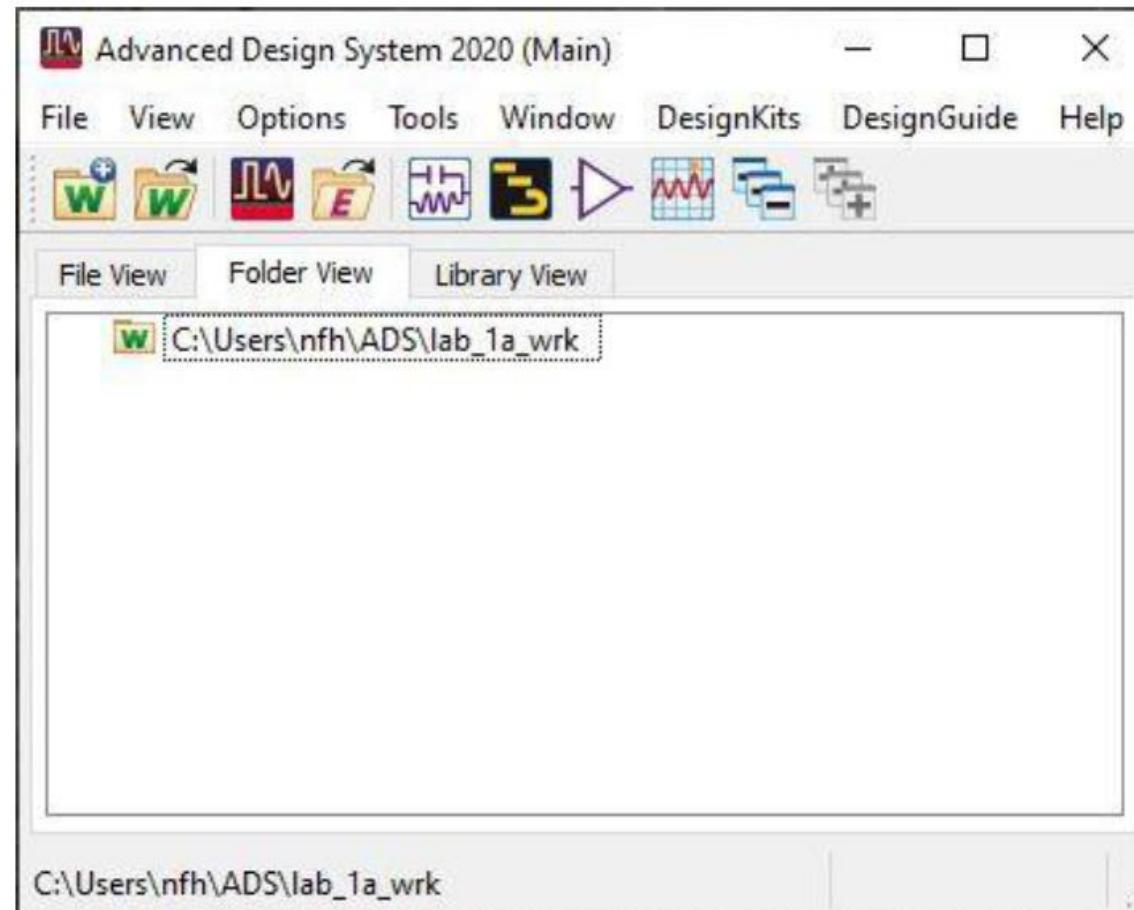
1. Launch PathWave ADS. In the main window select **File > New > Workspace**. Enter workspace name as desired in the dialog box shown in Figure 2. Please note that the workspace name and path to the workspace location should not contain any spaces. If you want to make changes to your PathWave ADS libraries, click on the **Change Libraries...** button; the ADS Analog/RF and ADS DSP libraries are included by default. Once you are satisfied with your workspace name, path, and libraries, click **Create Workspace**.



New Workspace Dialog Box

# Modelling in ADS: Creating a Workspace

- Once you complete the setup wizard, you will see the Main window. This is where your PathWave ADS journey begins.



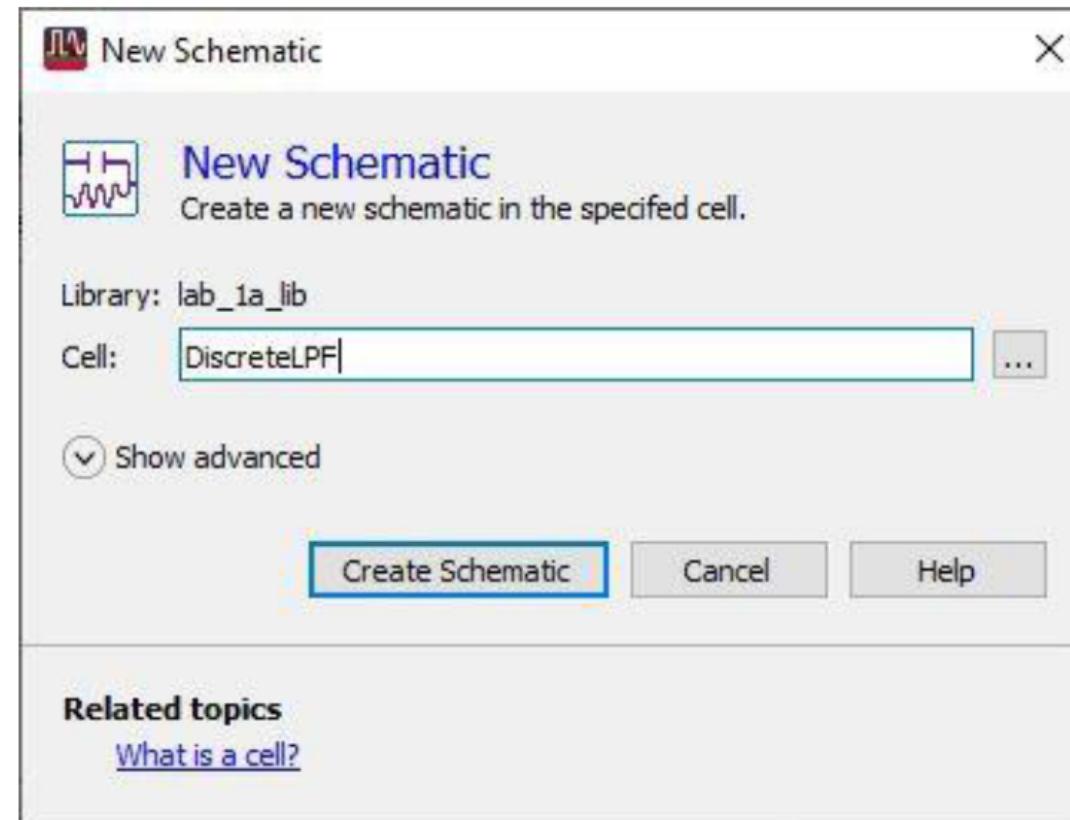
Main PathWave ADS Window

# Modelling in ADS: Creating Schematic Design

## Step 2 – Creating Schematic Design

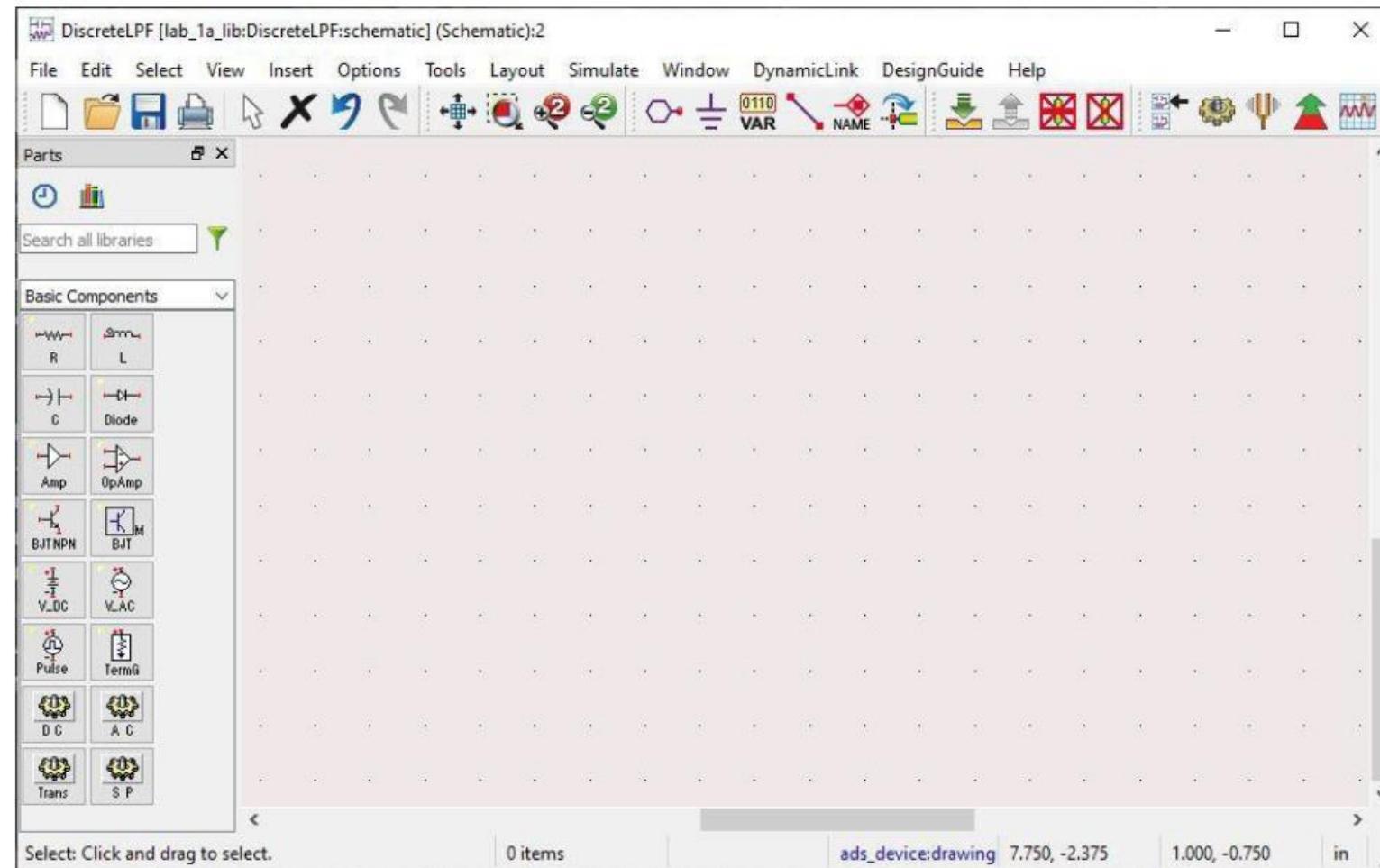
Circuit design typically starts from schematic entry. To start the schematic design, select **File > New > Schematic** or by clicking on the Schematic icon on the main window toolbar.

1. Enter the desired cell name (e.g. DiscreteLPF). Click **Create Schematic**.



# Modelling in ADS: Creating Schematic Design

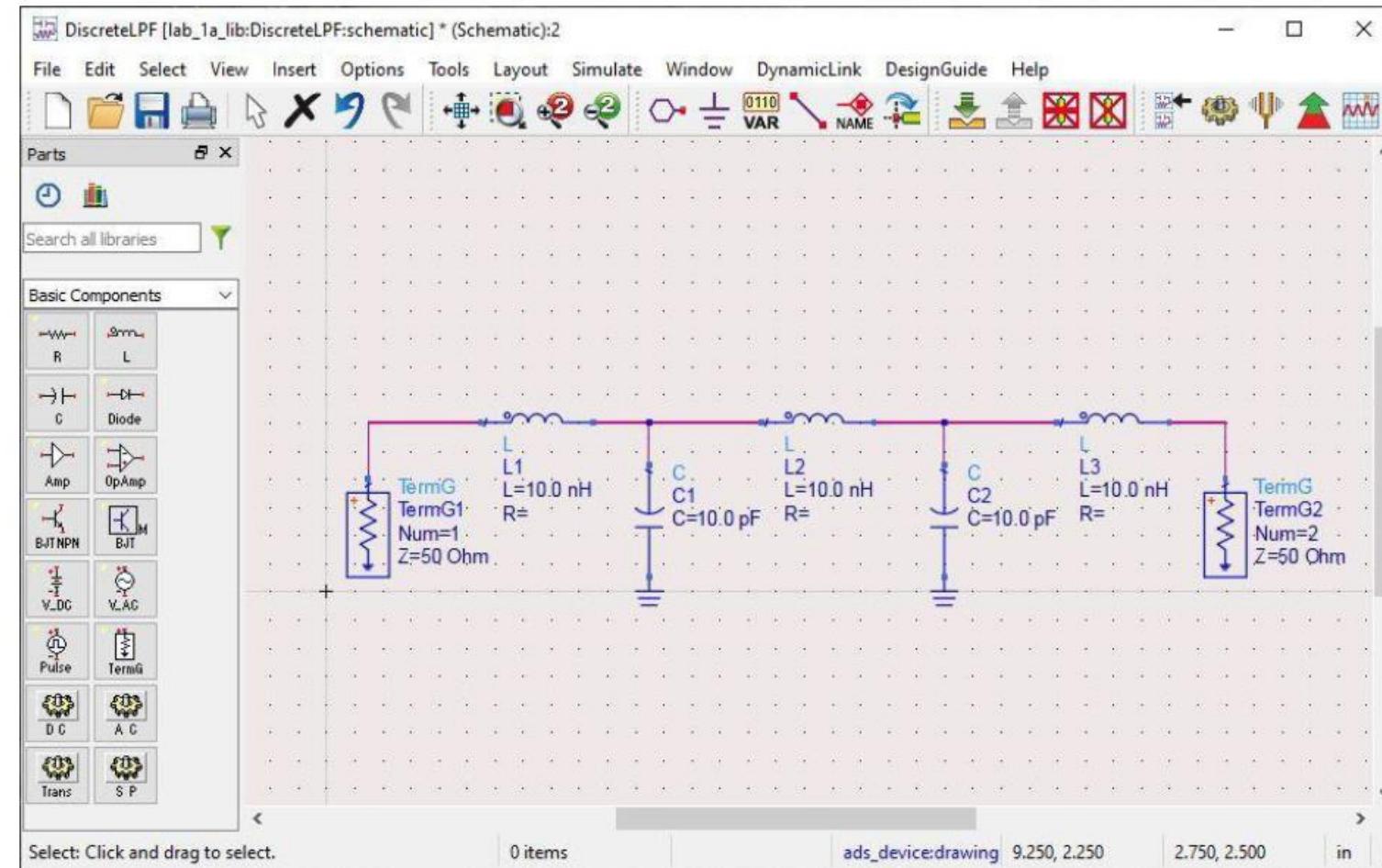
2. A new blank schematic will be shown.



Blank Schematic Window

# Modelling in ADS: Creating Schematic Design

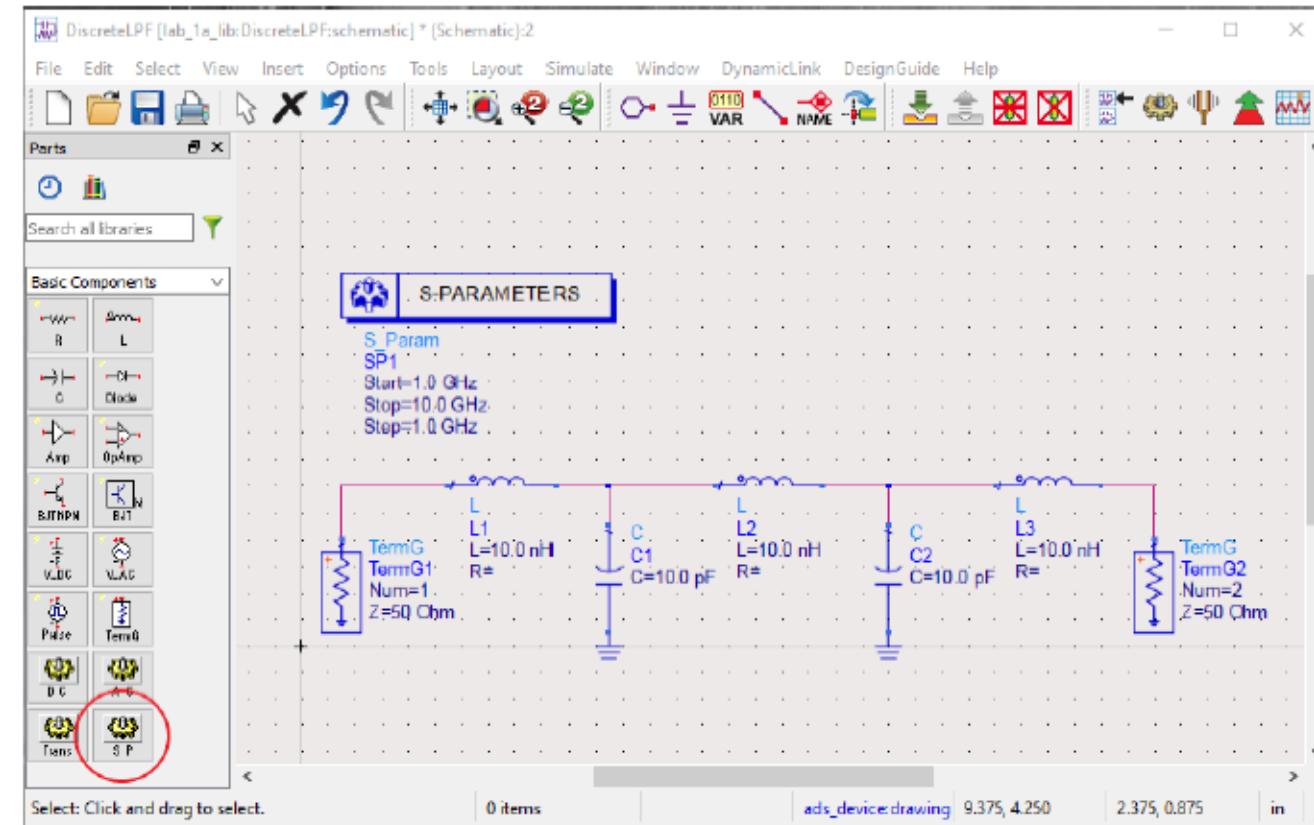
3. Using the Basic Components, draw the schematic shown.



Completed Schematic

# Modelling in ADS: Creating Schematic Design

4. Using the SP block, or S Parameter component in the **Basic Components** menu, add the **S\_Parameters** simulation object to your schematic.



Schematic with S-Parameters Simulation Object

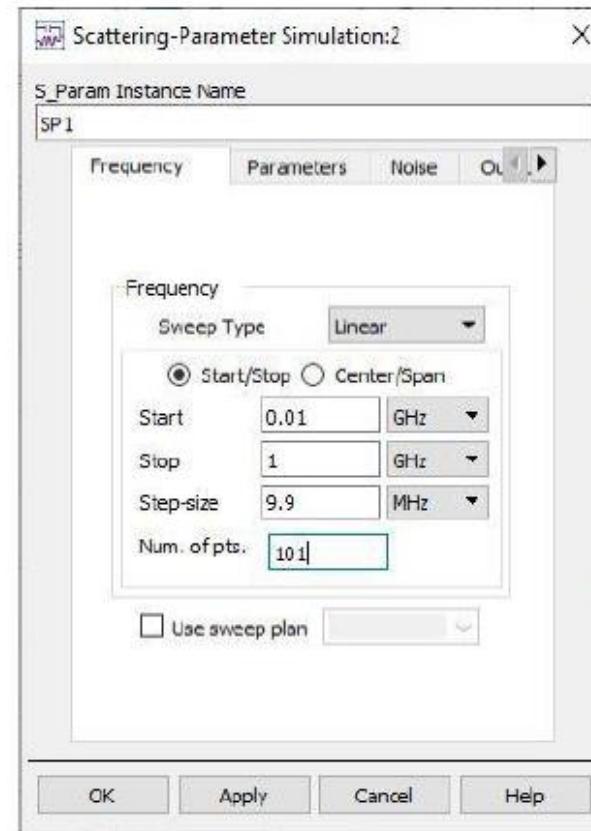
# Modelling in ADS: Creating Schematic Design

- Double click the S\_Parameters component, which will allow you to edit the parameters. The properties window, shown will open. Set the following values:

Start = 0.01 GHz

Stop = 1 GHz

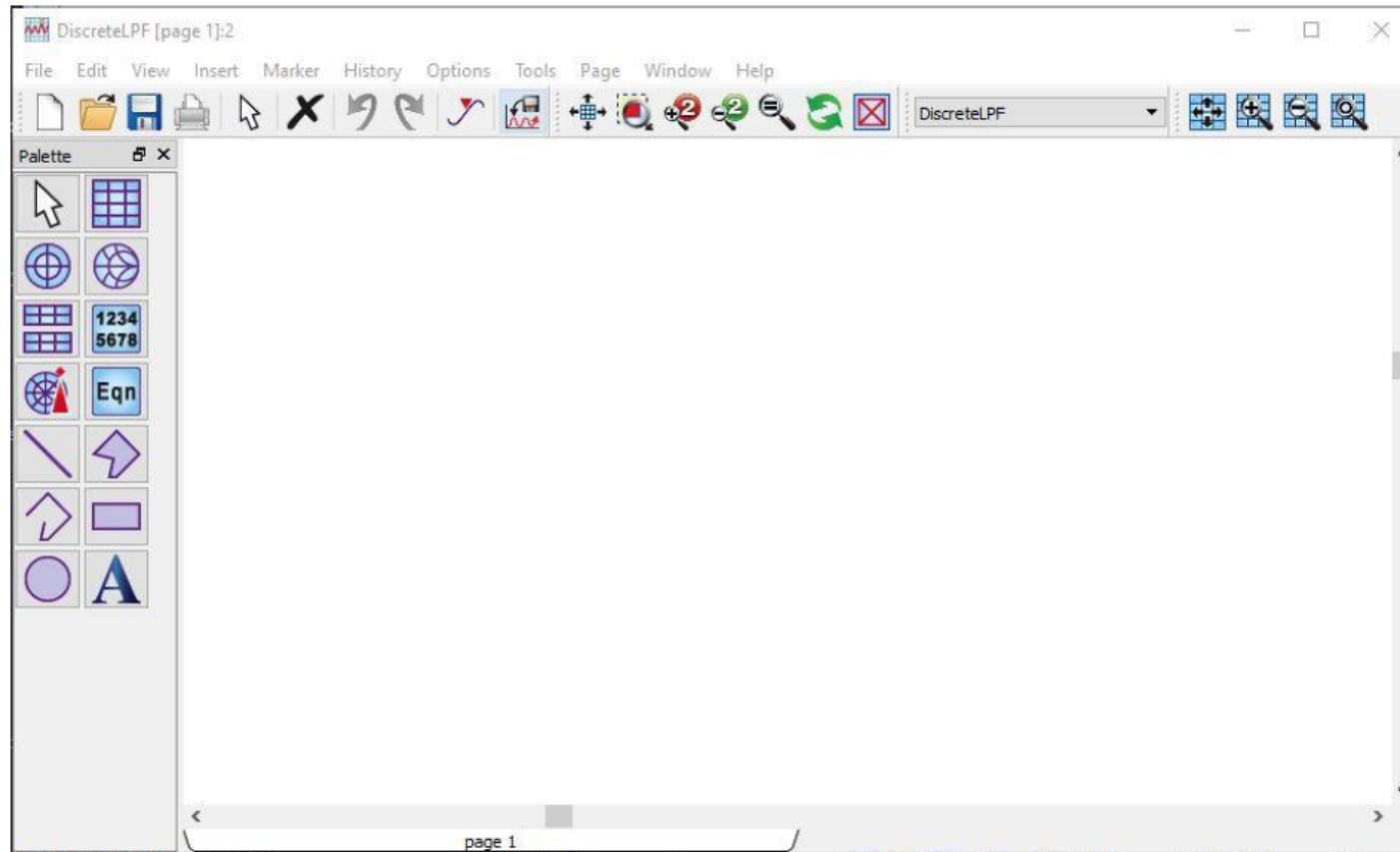
Num. of Points = 101



S-Parameter Simulation

# Modelling in ADS: Creating Schematic Design

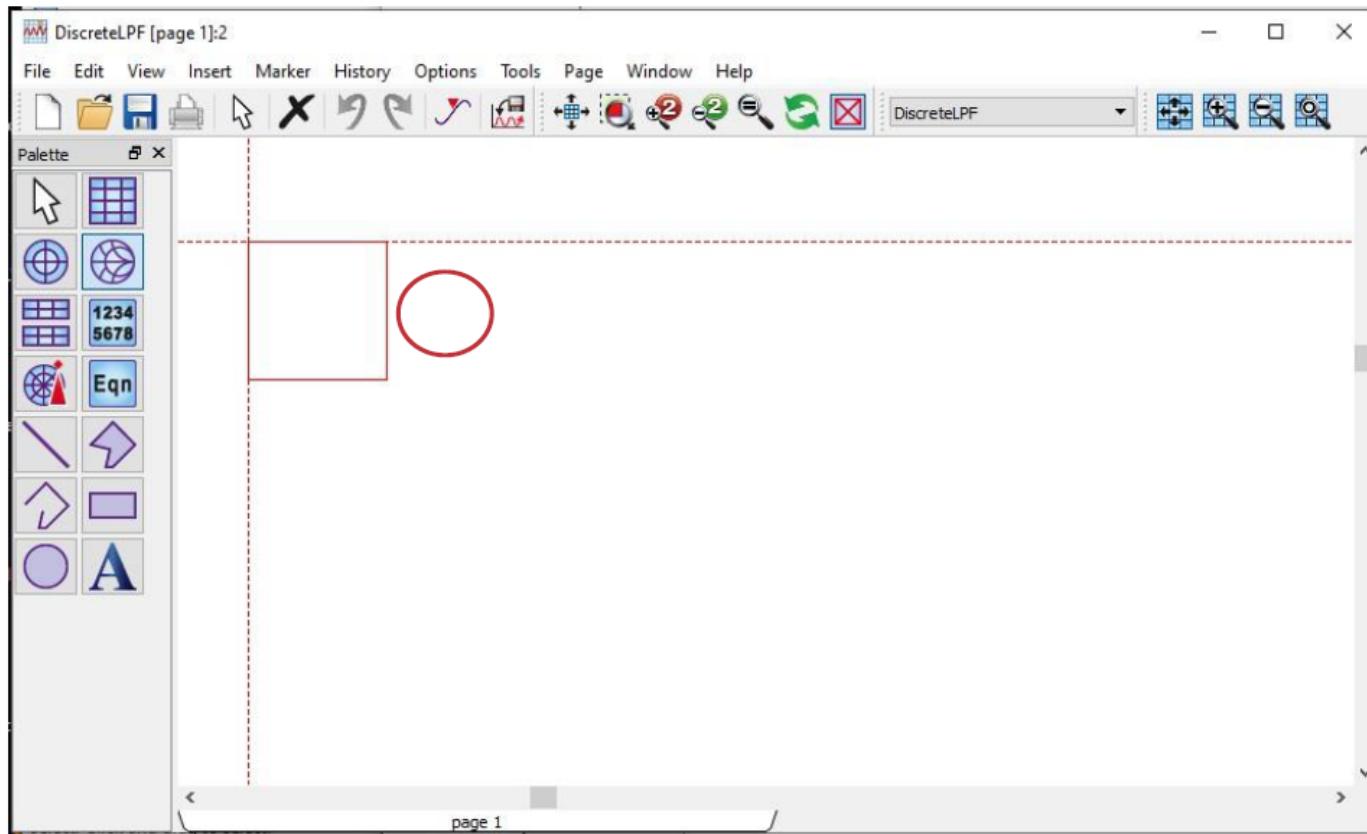
- Once your simulation parameters are setup, go to **Simulate > Simulate**, or press **F7**, to run the simulation. You will be met with a blank simulation results page .



Blank Simulation Results Window

# Modelling in ADS: Creating Schematic Design

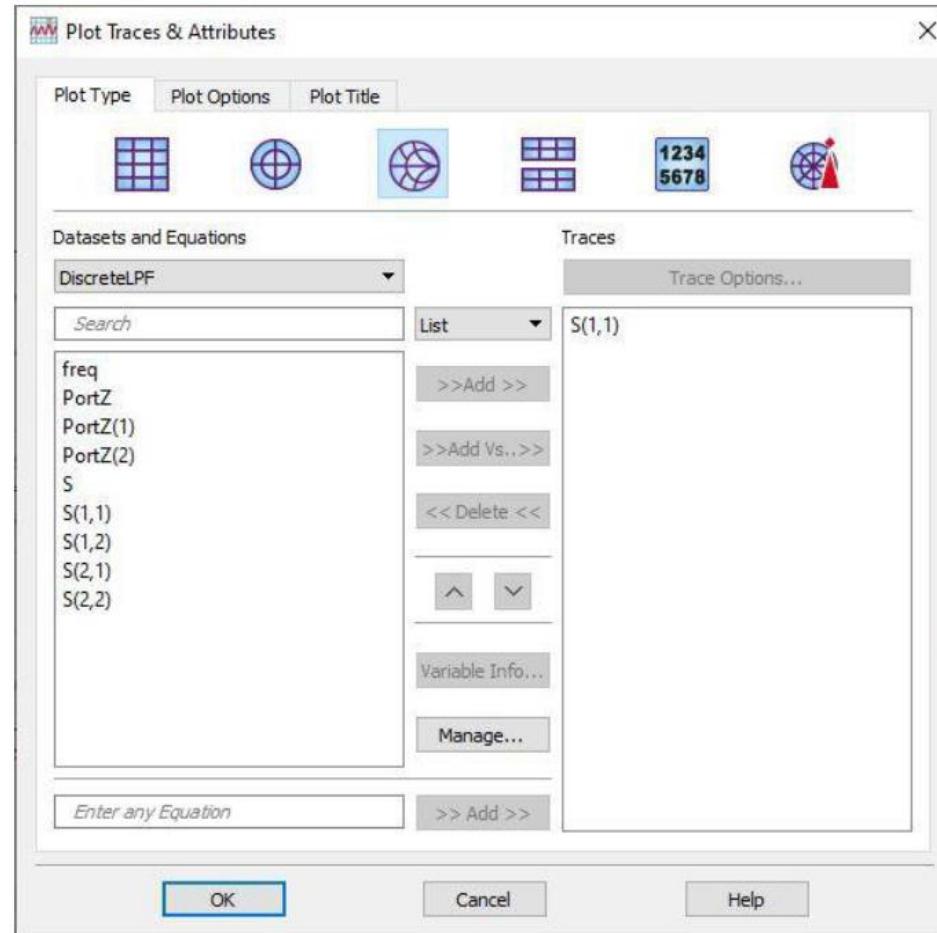
7. Using the buttons in the **Palette Menu** on the left, begin adding the desired graphs to the results window. In this case, we are going to add a Smith Chart for  $S(1,1)$ , as shown. After clicking on the Smith Chart button, click anywhere in the white part of the results window to place the chart.



Inserting Smith Chart for  $S(1,1)$

# Modelling in ADS: Creating Schematic Design

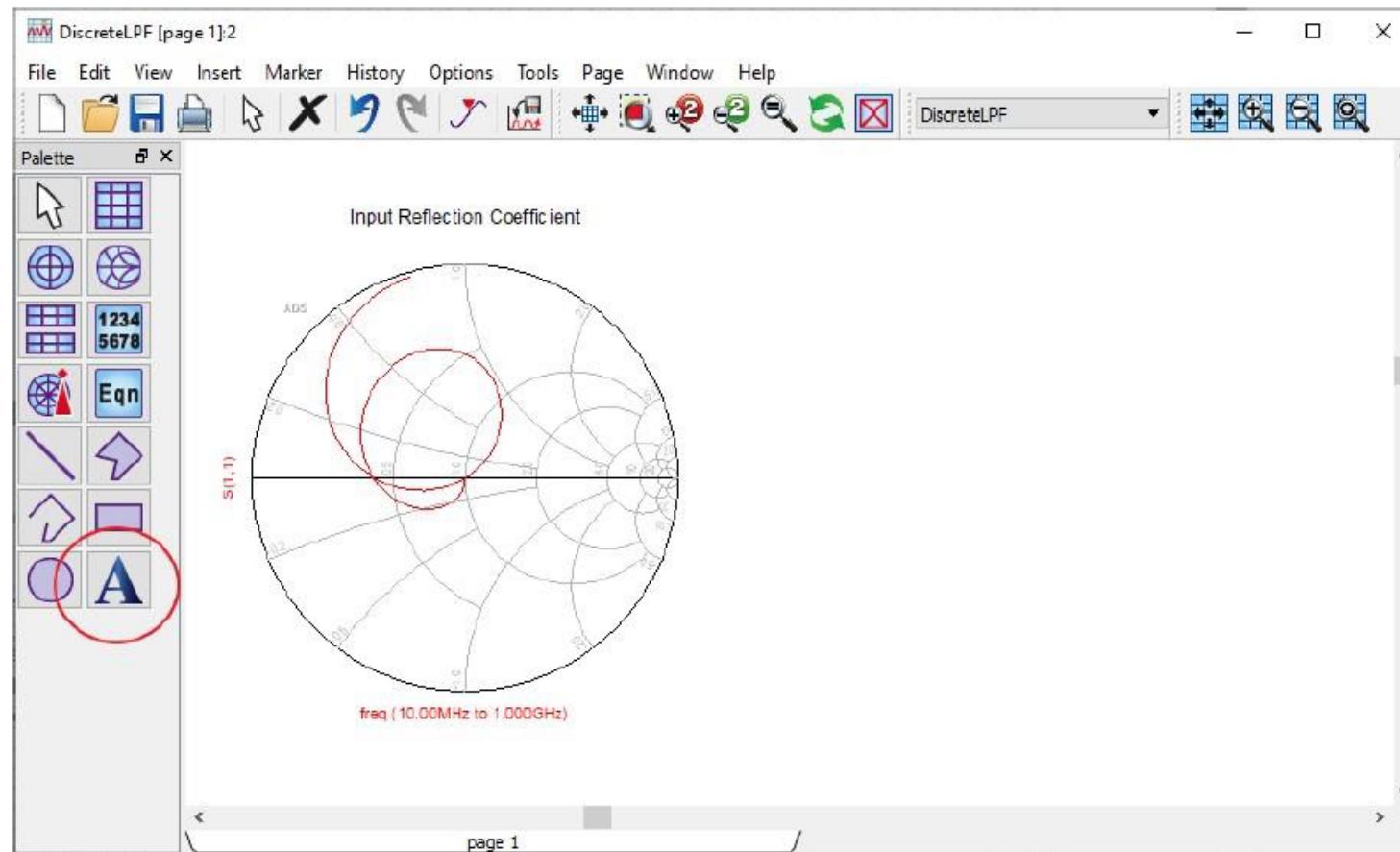
8. A dialog box will open that will allow you to add traces to the plot. Add the S(1,1) trace by selecting S(1,1) from the left side and clicking the >>Add>> button. This window is shown. Click **OK**.



Adding S(1,1) Trace to Smith Chart

# Modelling in ADS: Creating Schematic Design

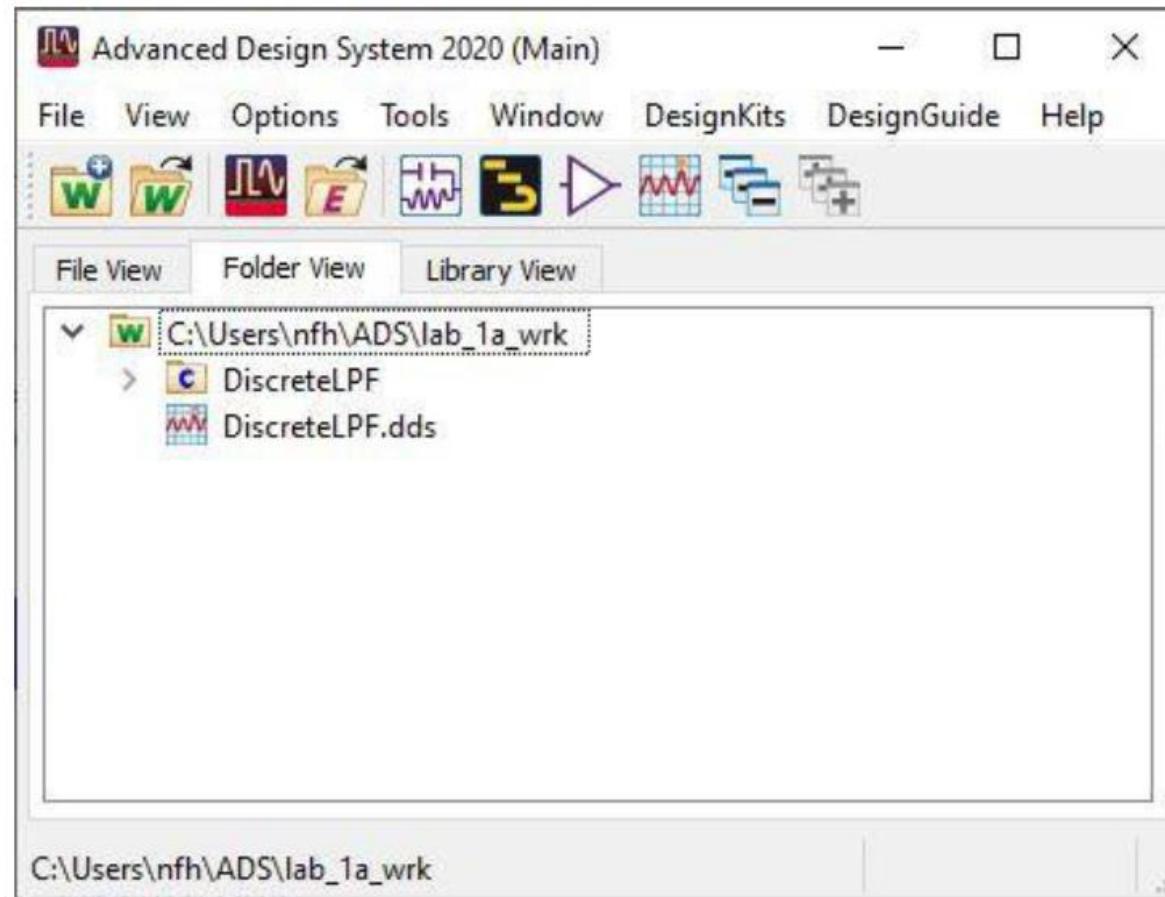
9. Next, using the **Text** tool, add the graph title. Since each results window may contain several graphs, lists, and equations, it is a good practice to label the different objects.



Results Window with Smith Chart

# Modelling in ADS: Creating Schematic Design

10. Save the results and schematic. After close both windows, the main window should look like



PathWave ADS Main Window After Saving

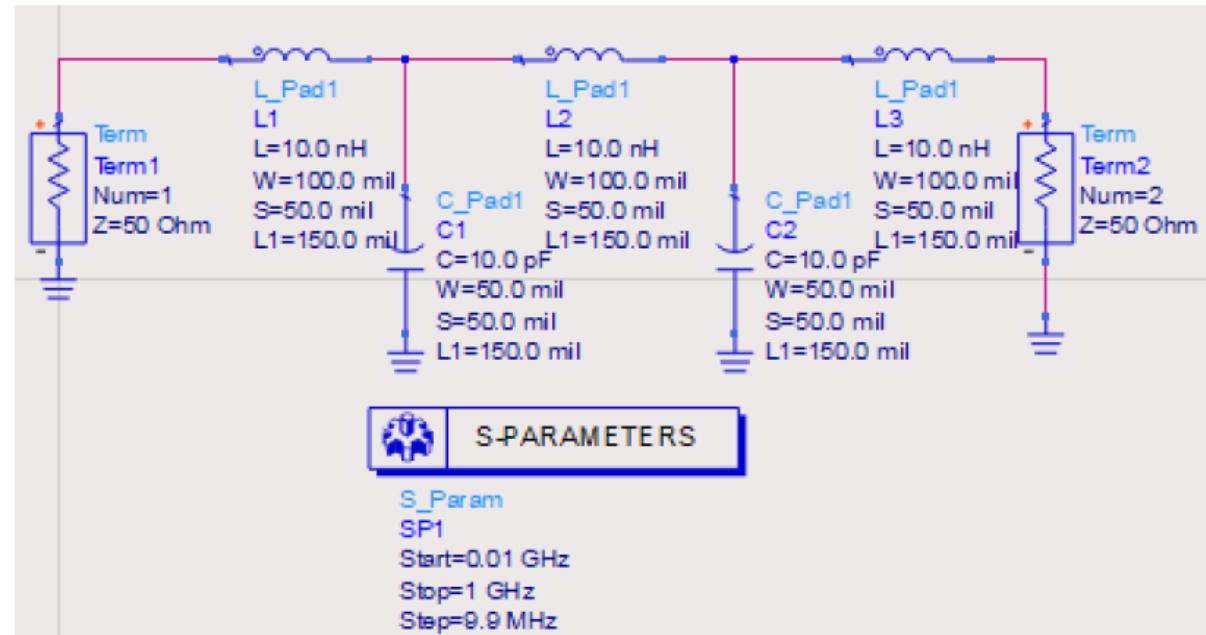
# Tuning and Optimisation

# What is Tuning?

Tuning is a way to change the component values and see the impact on circuit performance. This is a manual way of achieving the required performance from a circuit, which works well in certain cases.

# What is Optimisation?

Optimization is an automated procedure of achieving the desired circuit performance in which PathWave Advance Design System (ADS) can modify the circuit component values in order to meet the specific optimization goals. Care should be taken to select reasonable values while setting the goals. Otherwise, the optimization may not find a solution. Additionally, the optimized component values should be within the practical limits, which is typically decided by designers considering the practical limitations.



Schematic

# Tuning in PathWave ADS

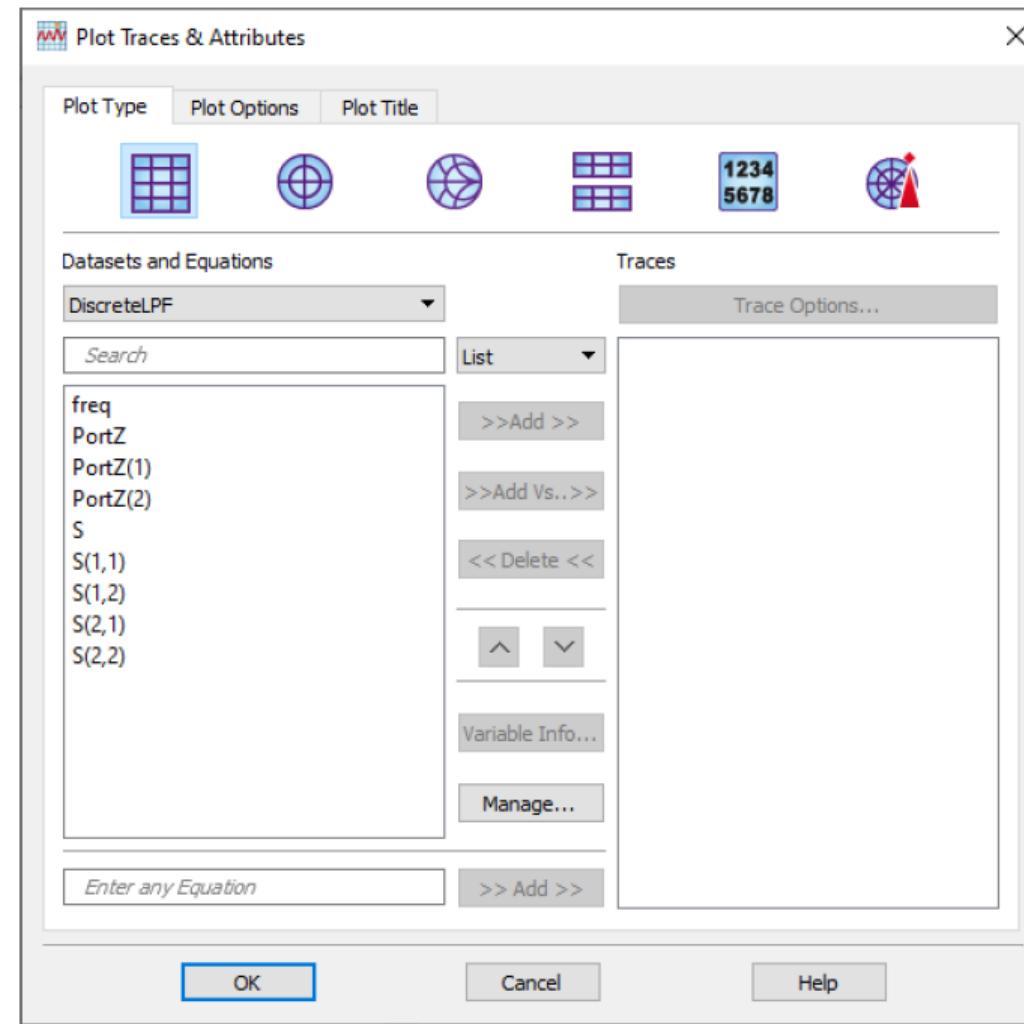
This step introduces the PathWave ADS tuning feature, which allows you to tune parameter values and see the simulation results in real time. To use this feature, first enable tuning and then select the components that will be tuned. For this section, we will use the LPF example from the previous chapter.

1. Open the LPF schematic from “Getting Started with PathWave ADS”. Delete or deactivate  the Display Template component  **DisplayTemplate**. Press Simulate .
2. In the data display, delete all the plots and insert a new rectangular plot on top right of the palette.
3. In the window that opens, select S(1,1) and S(2,1) and click >>Add>>. When prompted, select the units as dB.

# Tuning in PathWave ADS



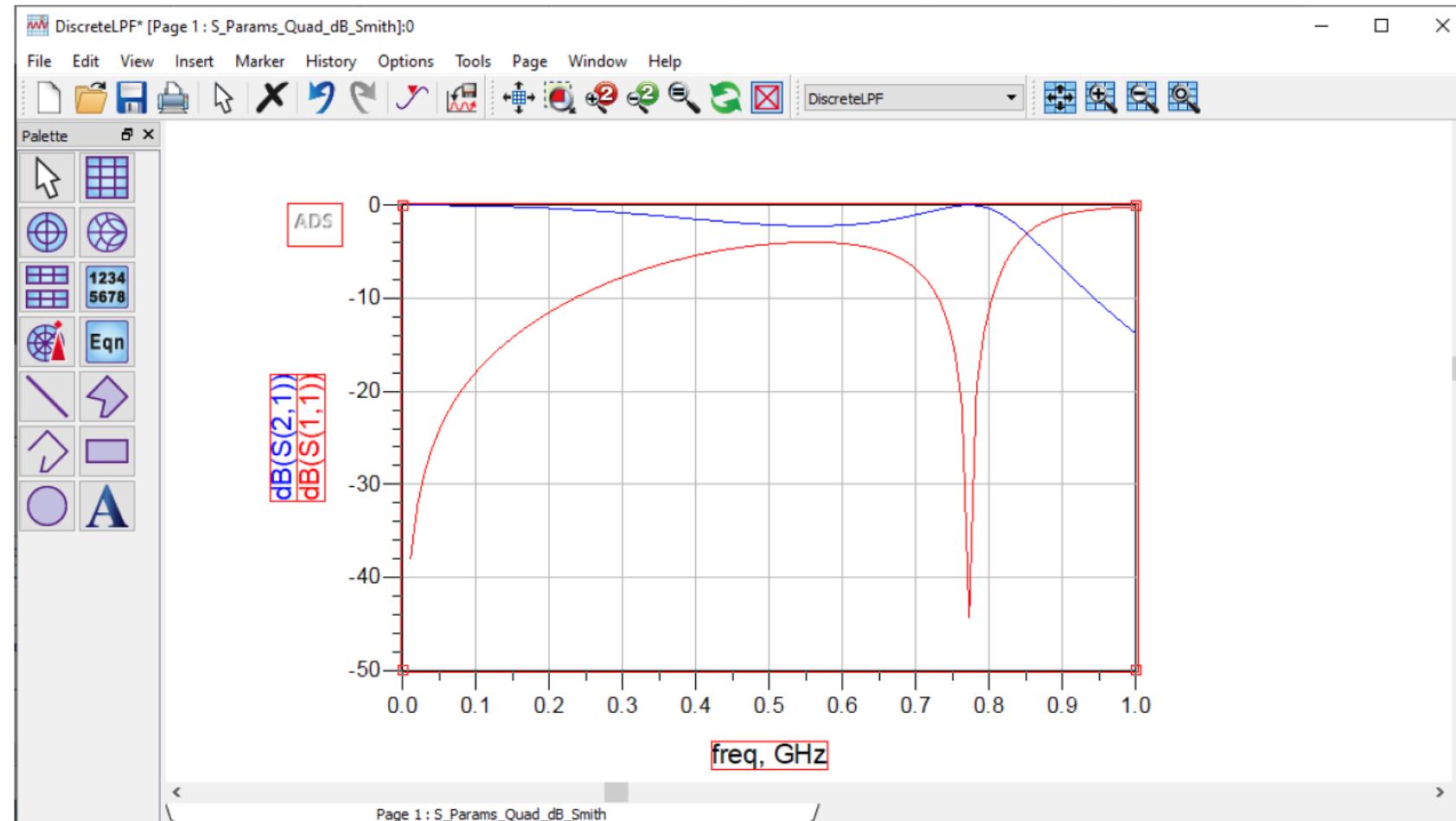
Data Display Palette



Adding S(1,1) and S(2,1)

# Tuning in PathWave ADS

4. Click OK. Figure shows the resulting data display.

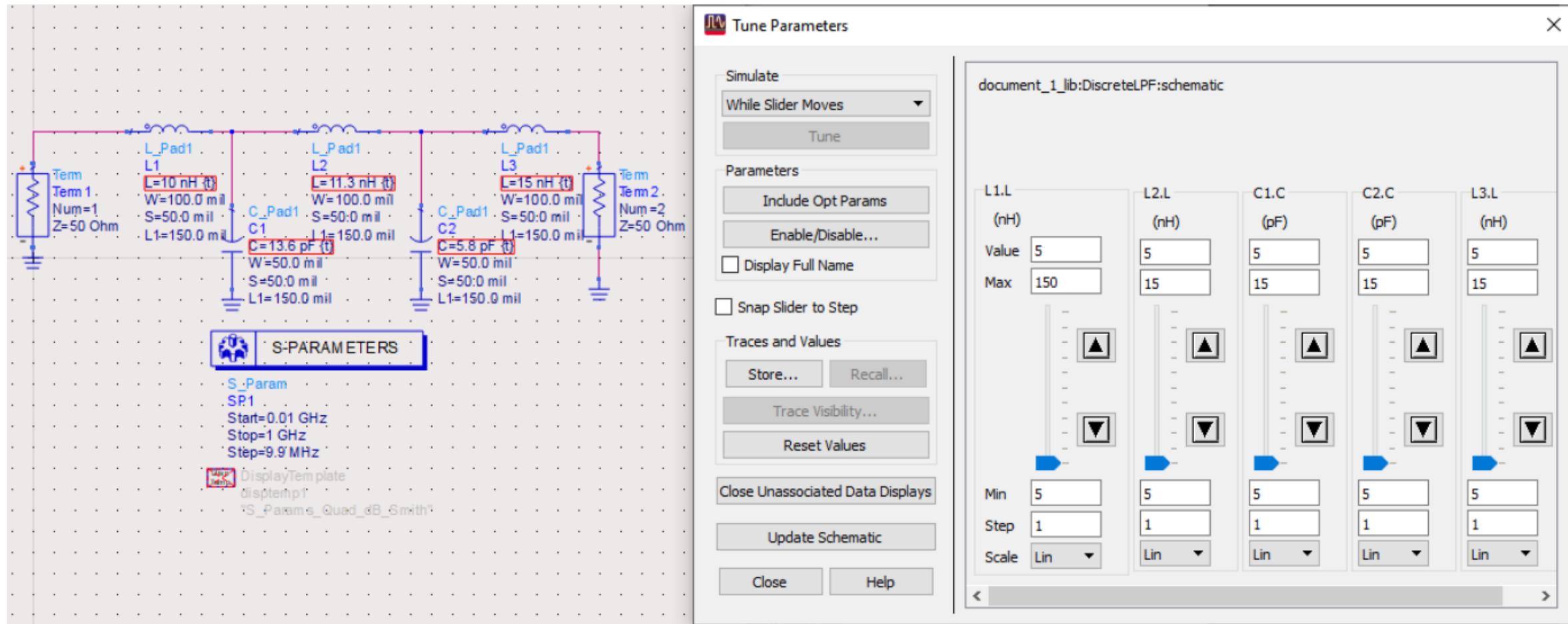


Plot of S(1,1) and S(2,1)

# Tuning in PathWave ADS

5. Return to the schematic window. Start tuning by pressing the **Tuning** icon  . Position the Tune Parameters and the Schematic windows side by side.
6. Once the Tune Parameters window is open, select the capacitors and inductors from the schematic window one-by-one. After each selection, a small window will open. This window will allow you to select the Tunable Parameters for the component. In this window, click the box to select the appropriate values (i.e. select C for the capacitors and L for the inductors) and press OK. Change the max values for all components to be 150.

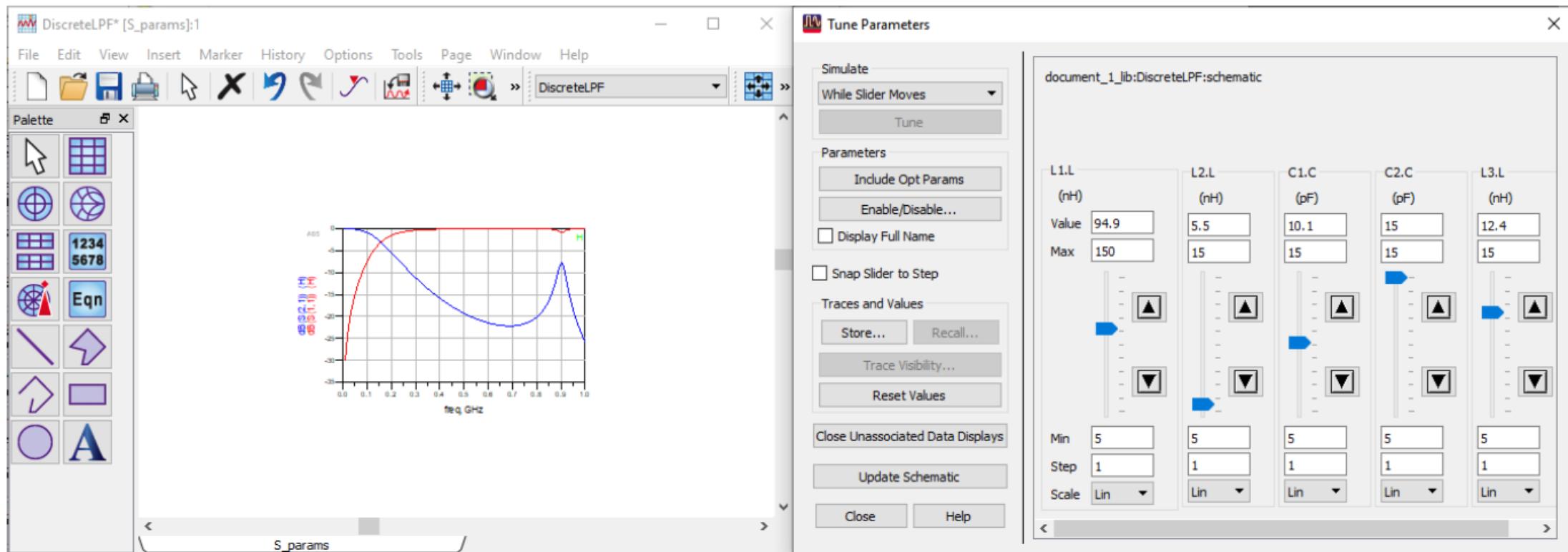
# Tuning in PathWave ADS



Tuning Window

# Tuning in PathWave ADS

7. Open the data display window. In the Tune Parameters window, move the component value slider. The S(1,1) and S(2,1) values will update as you tune the components. This layout is shown in Figure.



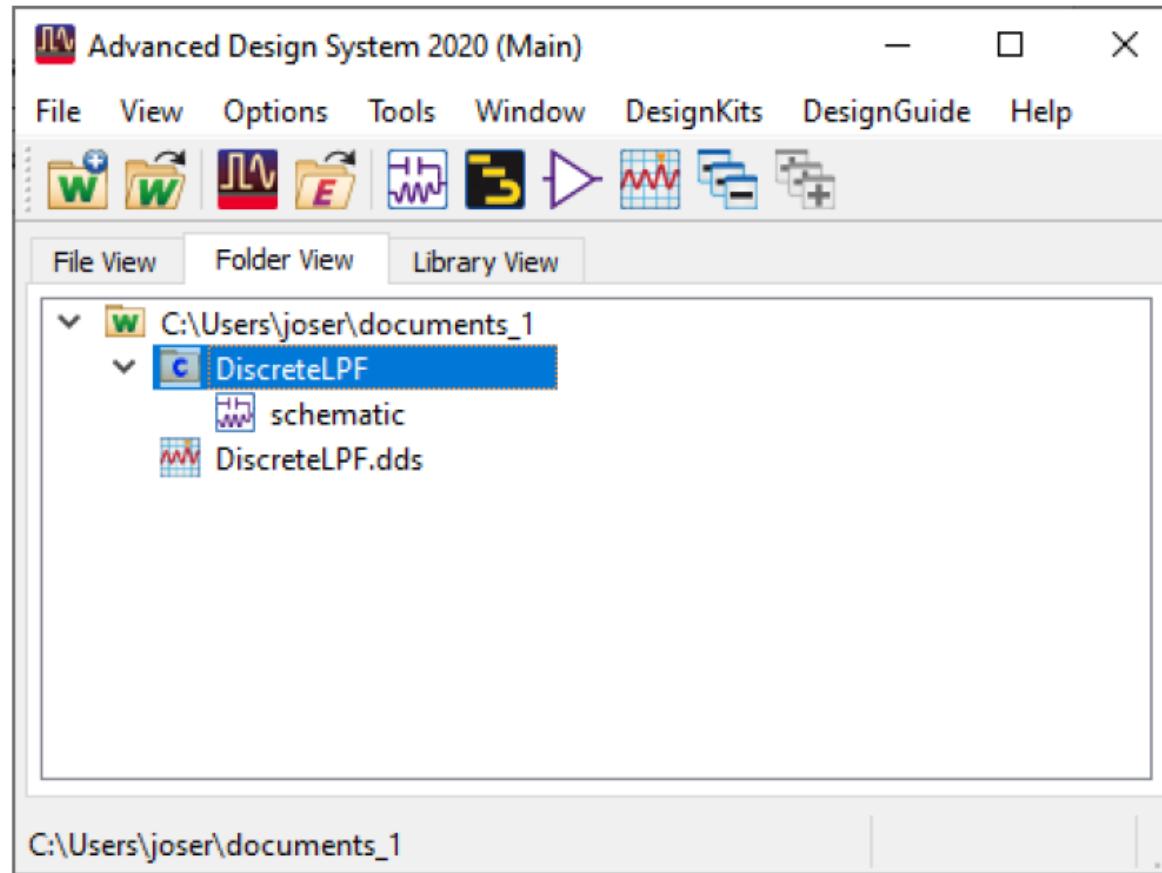
Plot different tuning values

# Tuning in PathWave ADS

8. Once the desired results have been achieved, click **Update Schematic** to update the values on the schematic. If you close the Tune Parameters window without pressing **Update Schematic**, the values will return to their original values.
9. Click **Close**. Note that the component values in the schematic have been updated to the tuned values.

# Optimisation in PathWave ADS

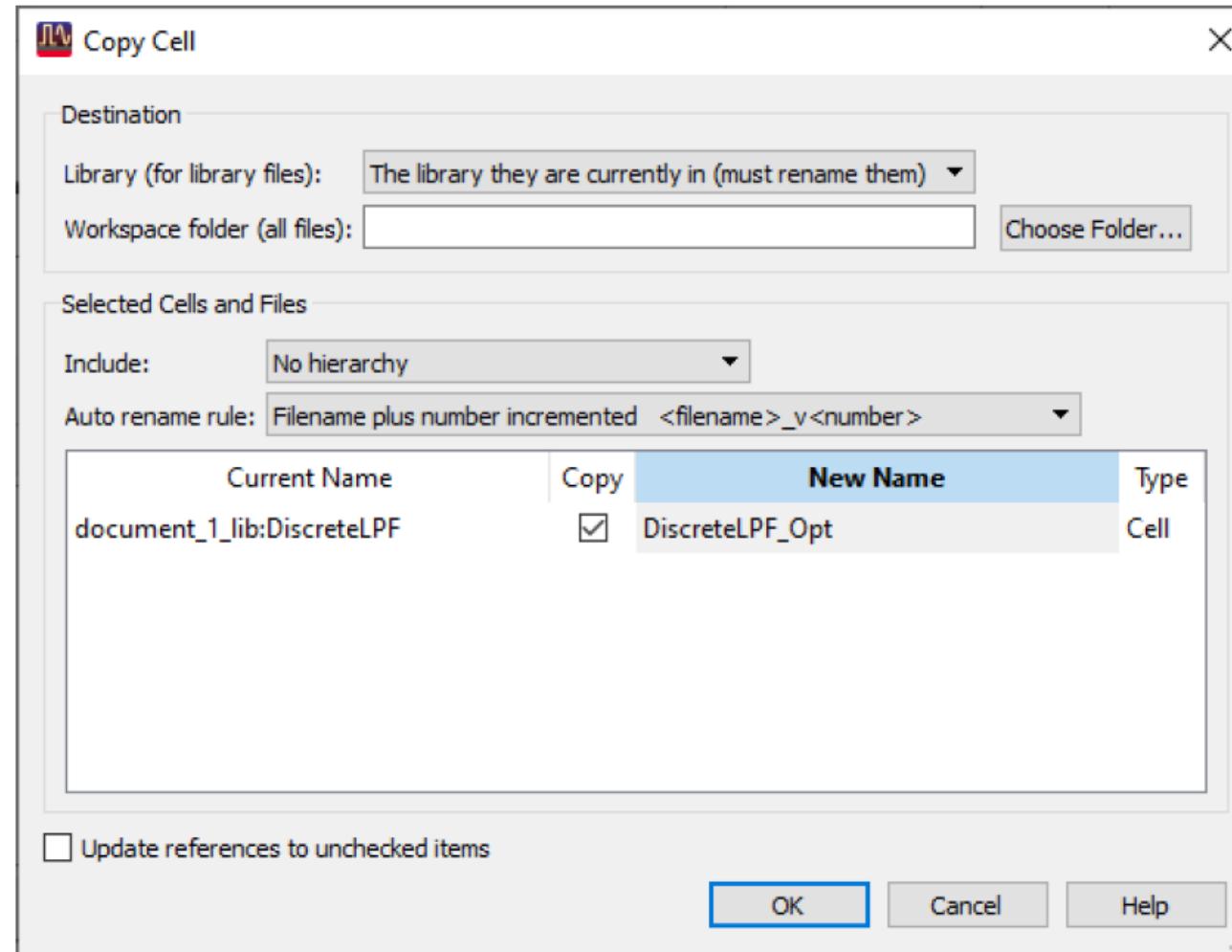
1. Go to the PathWave ADS Main Window. Right-click the LPF cell (in this case, **DiscreteLPF**) and select **Copy Cell**.



Main Window showing cell to be copied

# Optimisation in PathWave ADS

2. The Copy Cell window will open. Name the copied cell **DiscreteLPF\_Opt**. Note that if the cell hierachal, then we should use the option **Include Hierarchy....** If the workspace has folders, then you can copy this into the specific folder by clicking on **Choose Folder**. Click **OK**.



# Optimisation in PathWave ADS

3. Return to the Main Window. The cell has been copied and appears in the list with the new name. Open the schematic. In the parts selector (on the left), select Optim/Stat/DOE. This window is shown. Click on Goal. Click to place the Goal block in the schematic. Use the same process to add a second Goal block to the schematic.



# Optimisation in PathWave ADS

## Step 1 - Setting Goals

1. We are going to set two goals. Open the first goal block. Set the following parameters for the goal. When finished, press **OK**.

Expression:  $\text{dB}(\text{S}(1,1))$

Analysis: SP1

Limit 1: < -20

2. Open the second goal block. Set the following parameters for the goal. When finished, press **OK**.

Expression:  $\text{dB}(\text{S}(2,1))$

Analysis: SP1

Limit 1: > -1 (from 0.01 GHz to 0.2 GHz)

Limit 2: < -30 (from 0.4 GHz to 1.0 GHz)

3. From the Parts palette, insert an **Optim** block. This is needed to start the optimization.

# Optimisation in PathWave ADS

Optim Goal Input:2

ads\_simulation:Goal Instance Name  
OptimGoal1

Goal Information   Display

Expression: dB(S(1,1))   Help on Expressions

Analysis: SP1

Weight: 1.0

Sweep variables: freq    freq    time   Edit...

**Limit lines**

Name	Type	Min	Max	Weight	freq min	freq max
1	limit1	<	-20	1.0		

Add Limit   Delete Limit   Move Up   Move Down

OK   Apply   Cancel   Help

Goal 1

Optim Goal Input:2

ads\_simulation:Goal Instance Name  
OptimGoal2

Goal Information   Display

Expression: dB(S(2,1))   Help on Expressions

Analysis: SP1

Weight: 1.0

Sweep variables: freq    freq    time   Edit...

**Limit lines**

Name	Type	Min	Max	Weight	freq min	freq max
1	limit1	>	-1	1.0	0.01G	0.2G
2	limit2	<	-30	1.0	0.4G	1G

Add Limit   Delete Limit   Move Up   Move Down

OK   Apply   Cancel   Help

Goal 2

# Optimisation in PathWave ADS

## Step 2 – Setting Controller

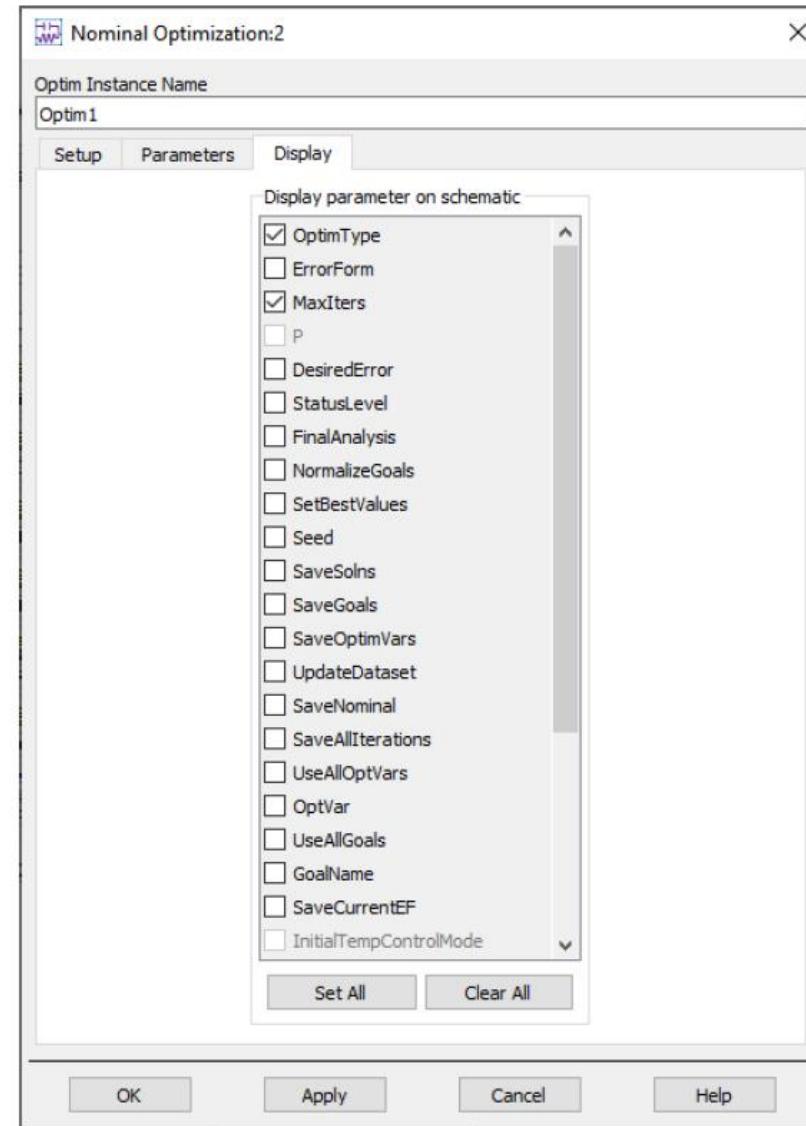
1. Double-click the Optimization controller and set parameters as below:

Optimization Type = Gradient

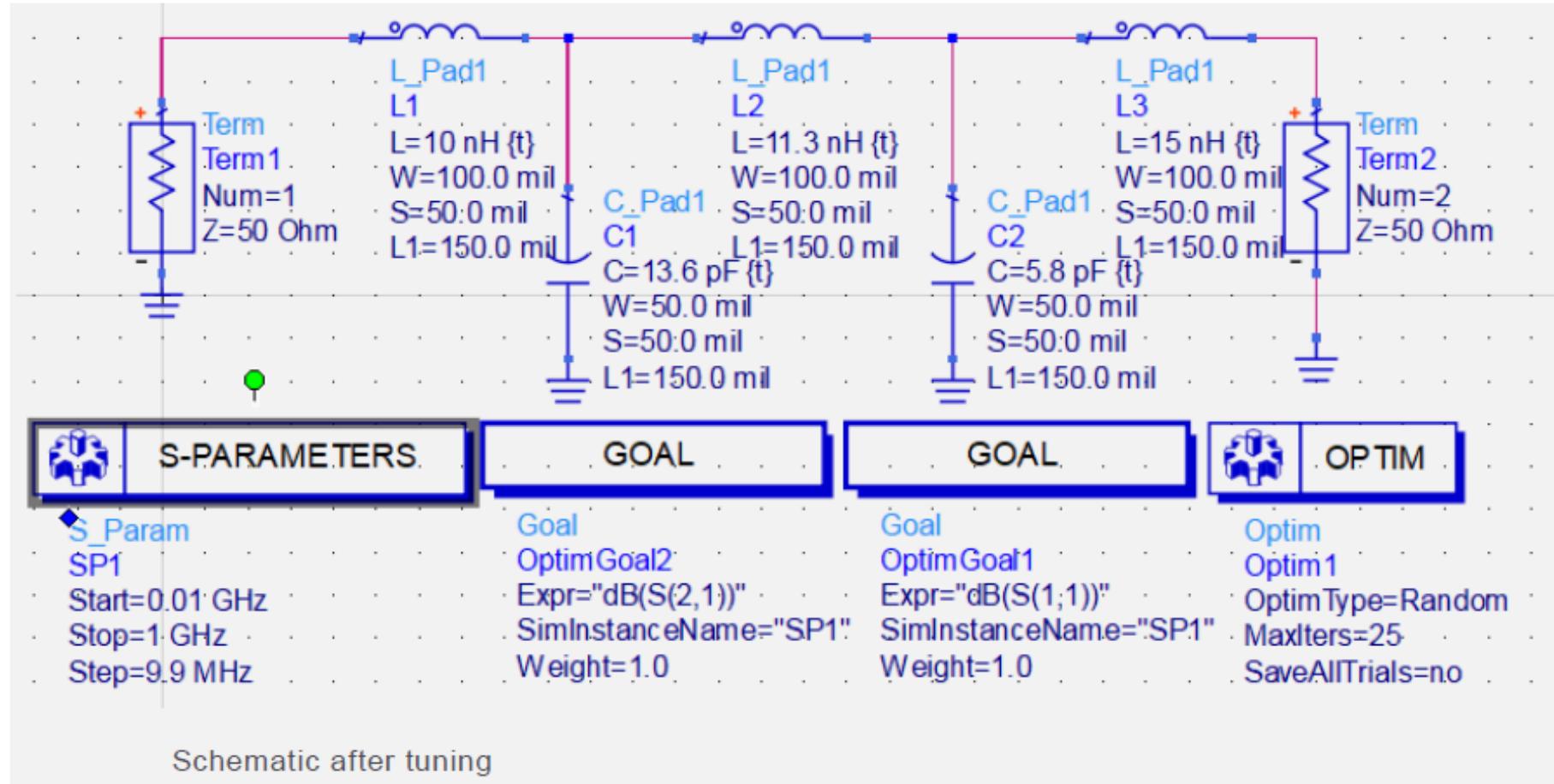
Number of Iterations = 2000

2. Go to the **Display** tab and select **Clear All**, which will uncheck all options.
3. Select **OptimType** and **MaxIter**. This will clean up the schematic, so that we only see the important information. Click **OK**. The updated schematic is shown.

# Optimisation in PathWave ADS



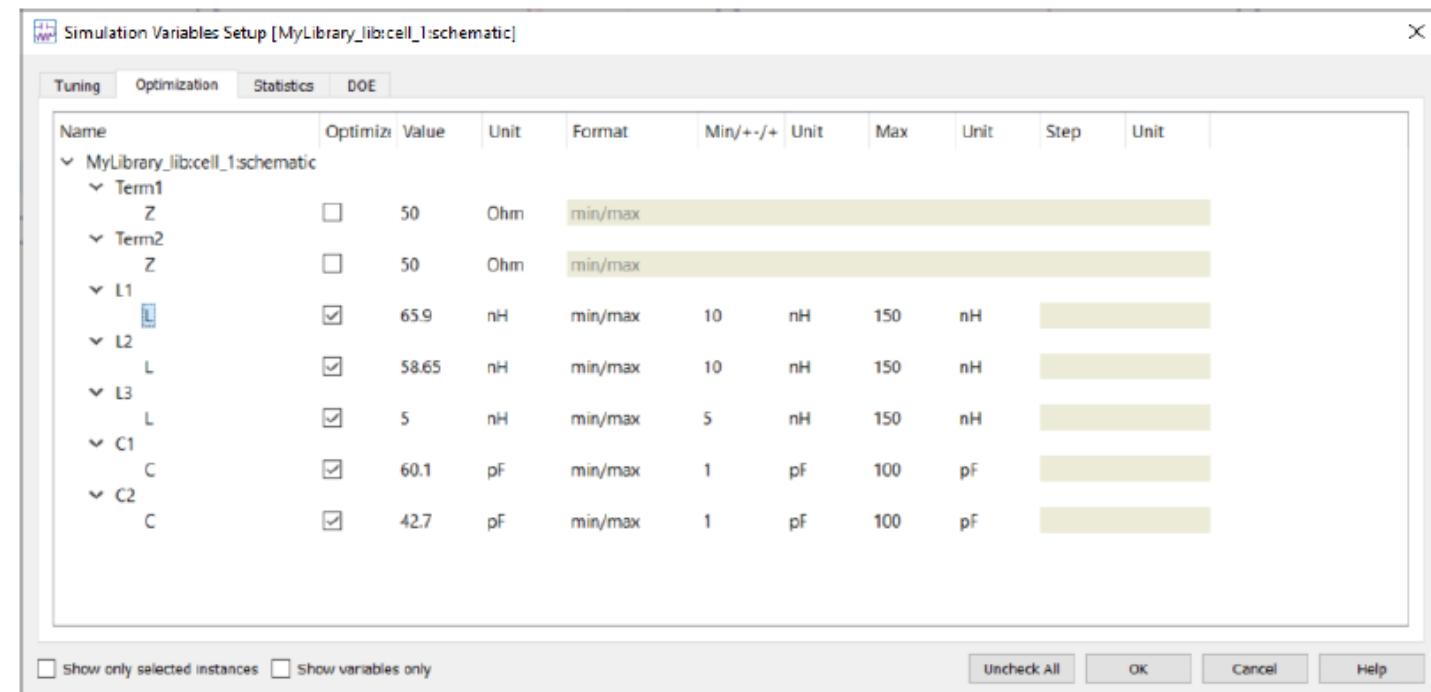
# Optimisation in PathWave ADS



# Optimisation in PathWave ADS

## Step 3 – Define Component Values

1. Go to Simulate > Simulation Variables Setup.
2. Instead of setting the optimization for each component individually, this window allows you to set several tuning or optimization variables at once. Click the Optimization tab. Check the Optimize box for the inductors and capacitors. Set the optimization values to match Figure. Press OK.

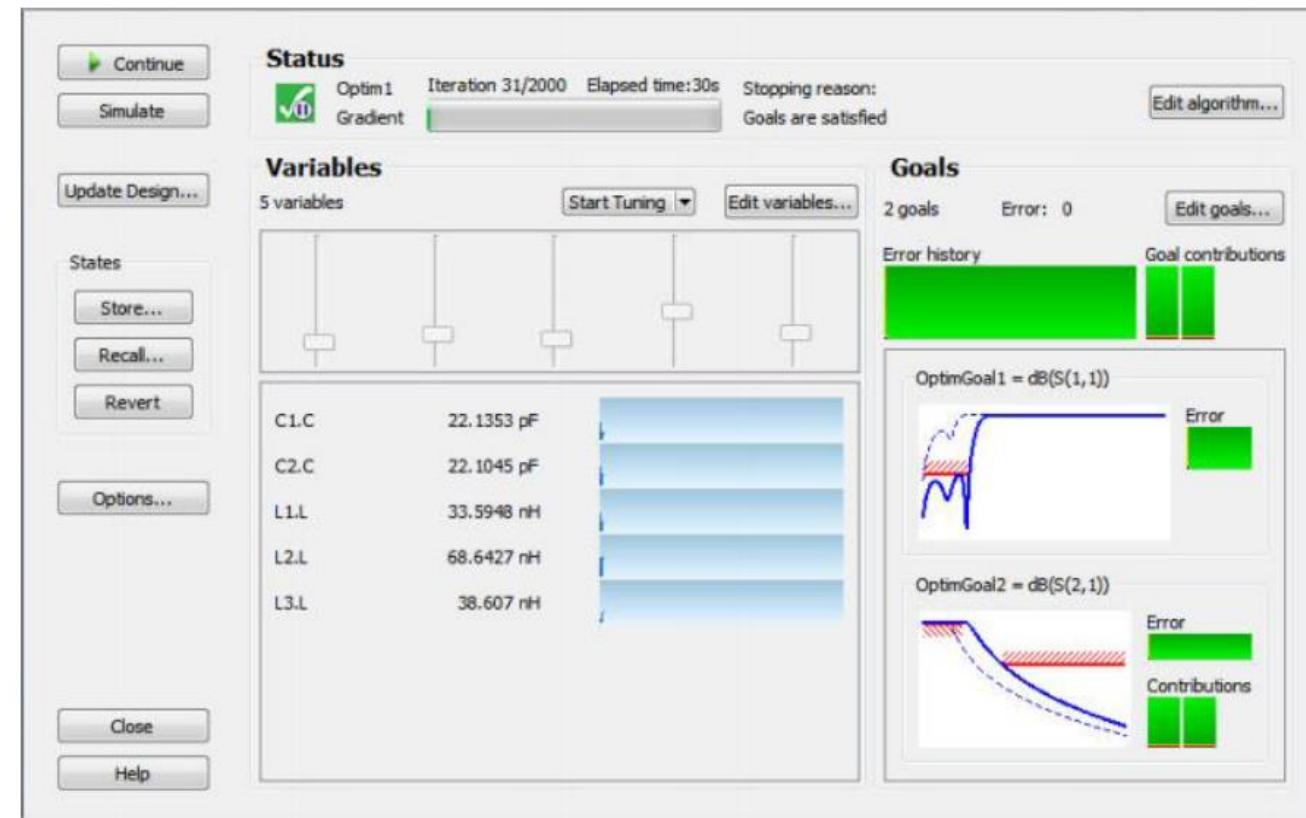


Variables Schematic

# Optimisation in PathWave ADS

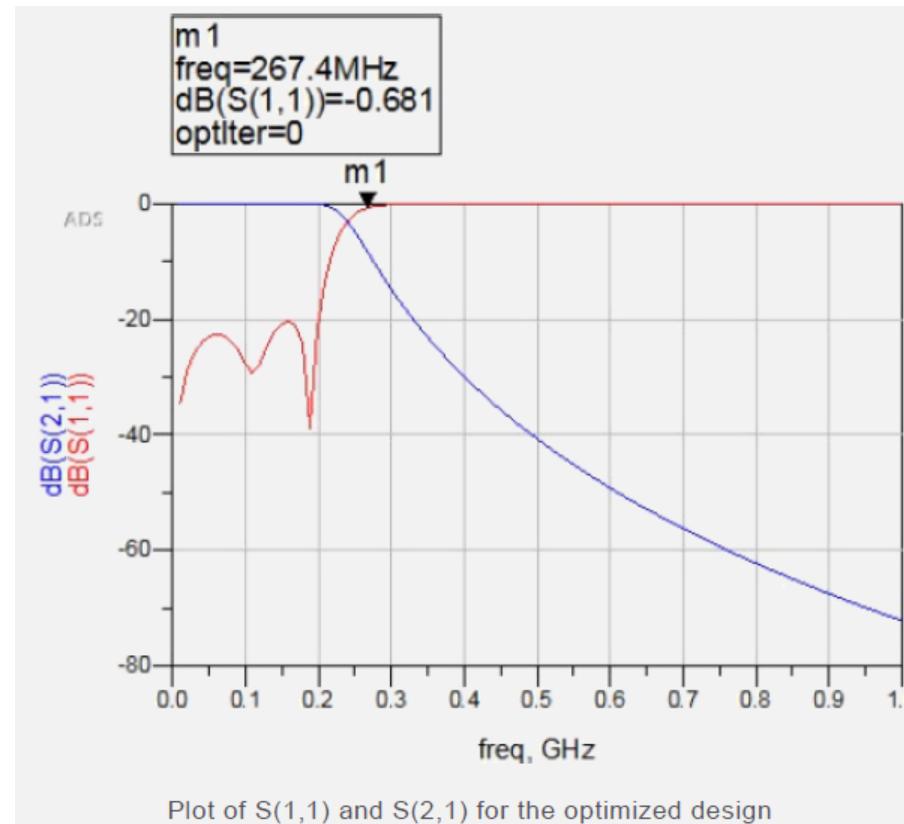
## Step 4 – Optimize

1. Click the Optimize button  in the schematic toolbar.
2. The Optimization Cockpit window will open. This window shows the process of the optimization, as well as the current component values as the optimization progresses. The optimization will continue until the goals are met or the maximum number of iterations is reached.



# Optimisation in PathWave ADS

3. Once the optimization stops, click **Close**. Select **Update the Design** when prompted.
4. In the data display, add rectangular plots and show  $S(1,1)$  and  $S(2,1)$ . Check the circuit performance against the optimization goals. Add markers from the Marker toolbar  to analyze the graphs. The final plot is shown.
5. Save all your work by going to **File > Save All** from the PathWave ADS Main Window.



# Optimisation in PathWave ADS

## Notes

You can simplify the Optimization Goals setup, involving Optimization Goals and Controller, by saving this as a template. To do so, place the components on a new blank schematic and go to File > Save Design as Template.

This template can be inserted into any workspace by going to Insert > Template.

# SIGNAL INTEGRITY VALIDATION IN MODELLING AND SIMULATION

Module 2: ANSYS HFSS Basics

# Introduction to HFSS

# Introduction to ANSYS, ANSYS Electronic Desktop and HFSS

## • ANSYS Introduction

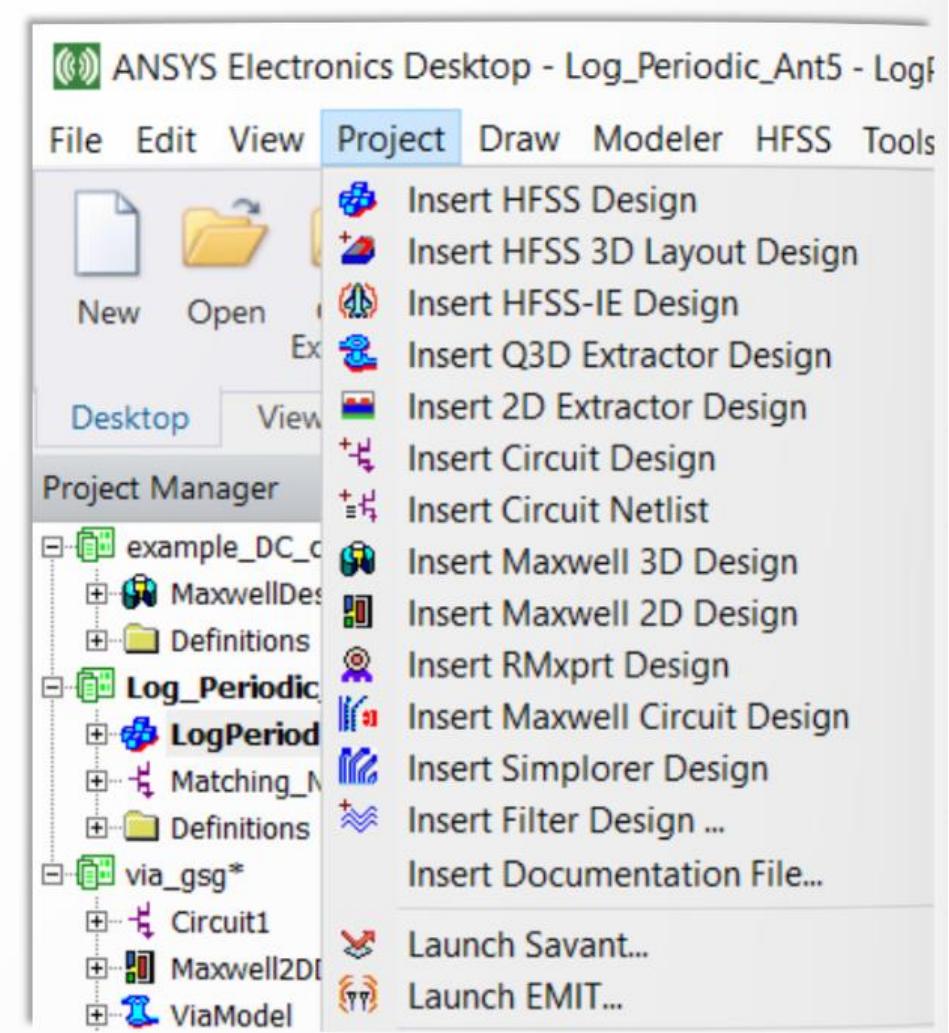
- Multiple Disciplines and Multiphysics Simulation Software
- Publicly Traded: ANSS
- Worldwide Headquarters in Canonsburg, PA USA
- [www.ansys.com](http://www.ansys.com)

## • AEDT Introduction - ANSYS Electronic Desktop

- Common Graphical User Interface for Multiple Products
- Common File Extension \*.aedt and \*.aedtz for Zip Archive
- Multiple Projects and Different Simulators Can Be Open

## • HFSS Finite Element Method (FEM)

- HFSS Includes Several Different Electromagnetic Simulation Solvers.
- HFSS Finite Element Method (FEM) is the Subject of this Course
- Two Different Approaches and GUI Feature Sets:
  - HFSS MCAD - Fully Arbitrary 3D - ***This course***
  - HFSS 3D Layout - Layered Structures



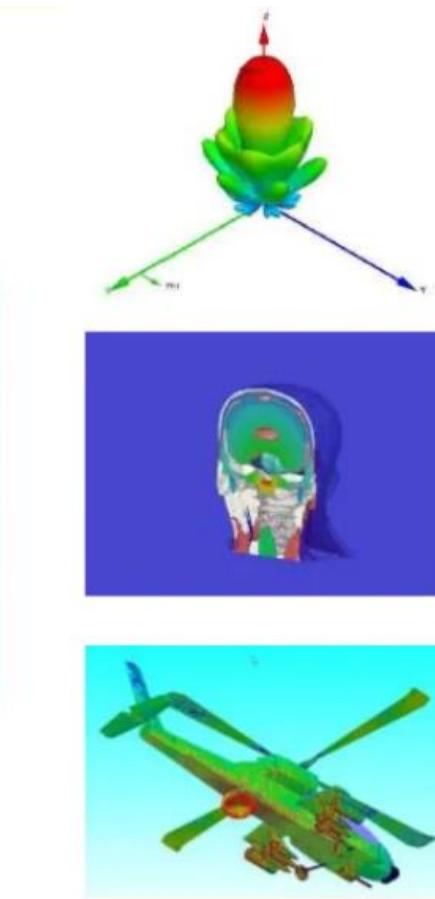
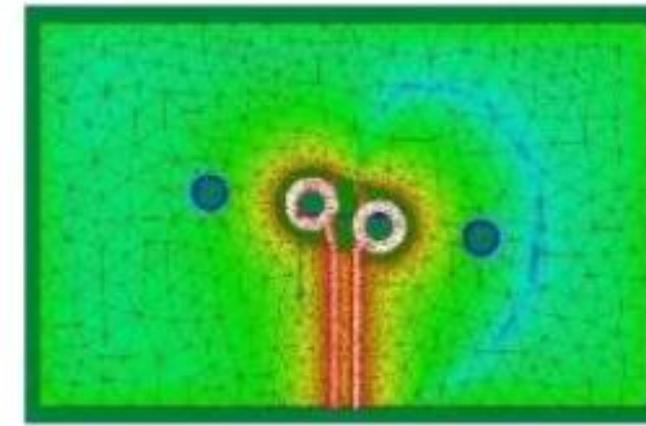
## Background

- Introduced from 90s
- Simulation tool for complex 3D geometries
- Using Finite Element Method
- Adaptive Mesh generation & refinement
- 2 Main Vendors—Agilent & Ansoft
- Merger from May 1
- Transfer to Ansoft HFSS after Nov. 1, 2001

# Introduction to High Frequency Structure Simulator HFSS

## Applications

- Antennas
- Microwave transitions
- Waveguide components
- RF filters
- Three-dimensional discontinuities
- Passive circuit elements



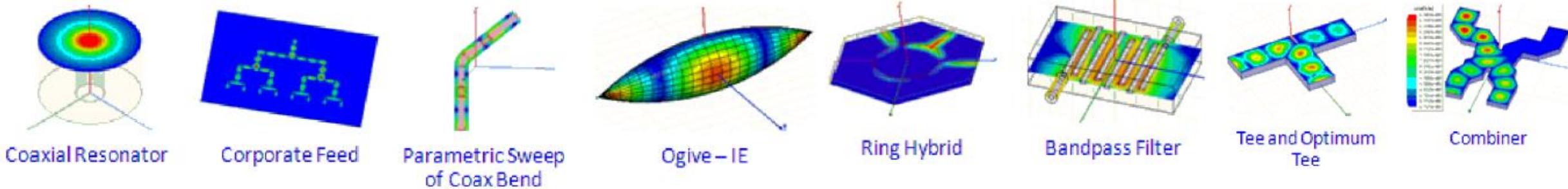
# Introduction to High Frequency Structure Simulator HFSS

Full-wave frequency-domain 3-D field solver based upon finite element method

- Industry-standard accuracy
- Adaptive meshing of arbitrary geometry
- Fully parametric modeling
- Optimization and HPC
- Multi-physics via Ansys Workbench

Widely used for RF/microwave design

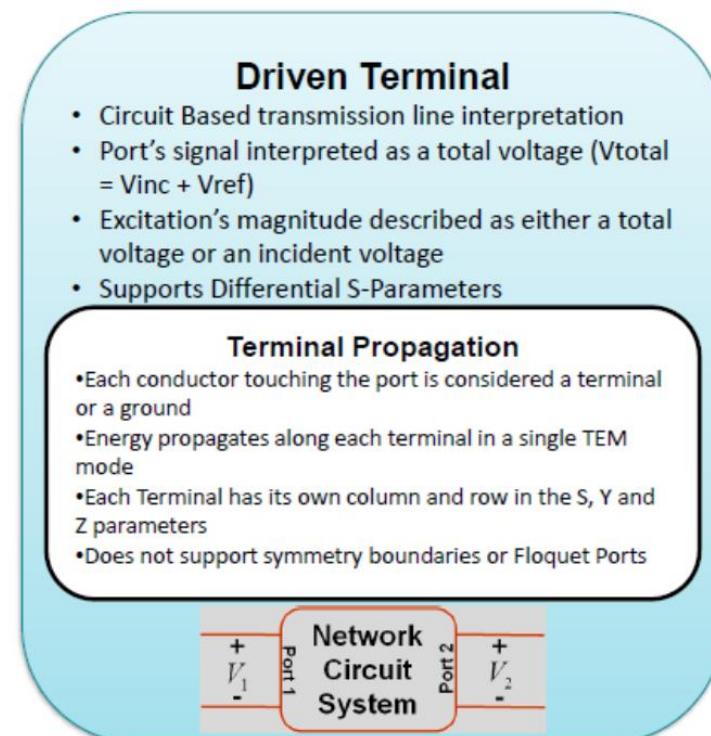
- Antenna design and platform integration
- Filters and waveguide structures
- Electronic packages and PCBs
- Connectors and transitions
- EMC/EMI
- Radar cross-section



# Introduction to High Frequency Structure Simulator HFSS

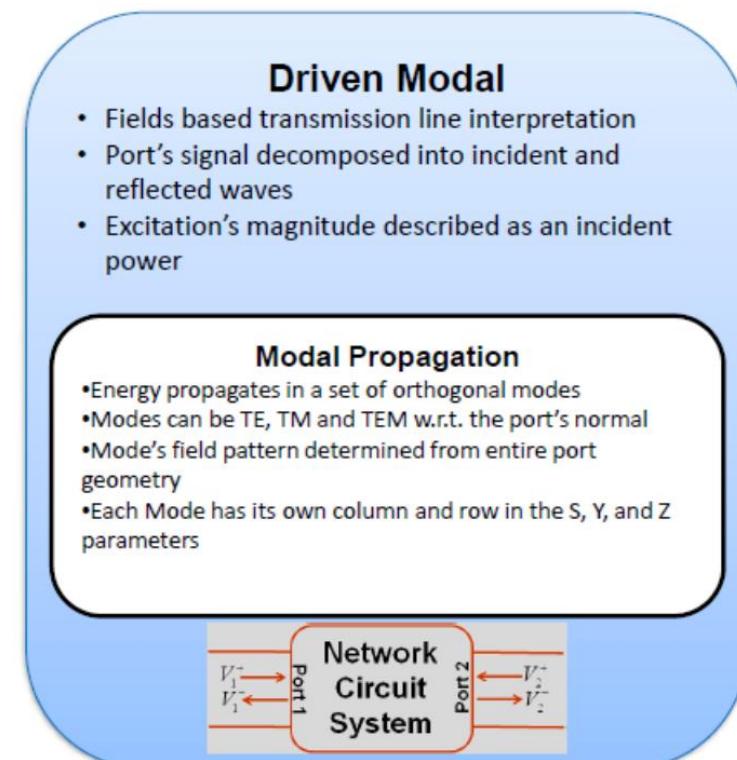
## Eigenmode solution

- Solves for natural resonances of structure based on geometry, materials, and boundaries
- Provides modal frequencies, unloaded Qfactors, and fields



## Driven solution

- Port or incident field used to excite the structure
- *Driven modal* method commonly used for RF/microwave designs
- *Driven terminal* method commonly used for multi-conductor transmission lines
- Provides S-parameters and fields



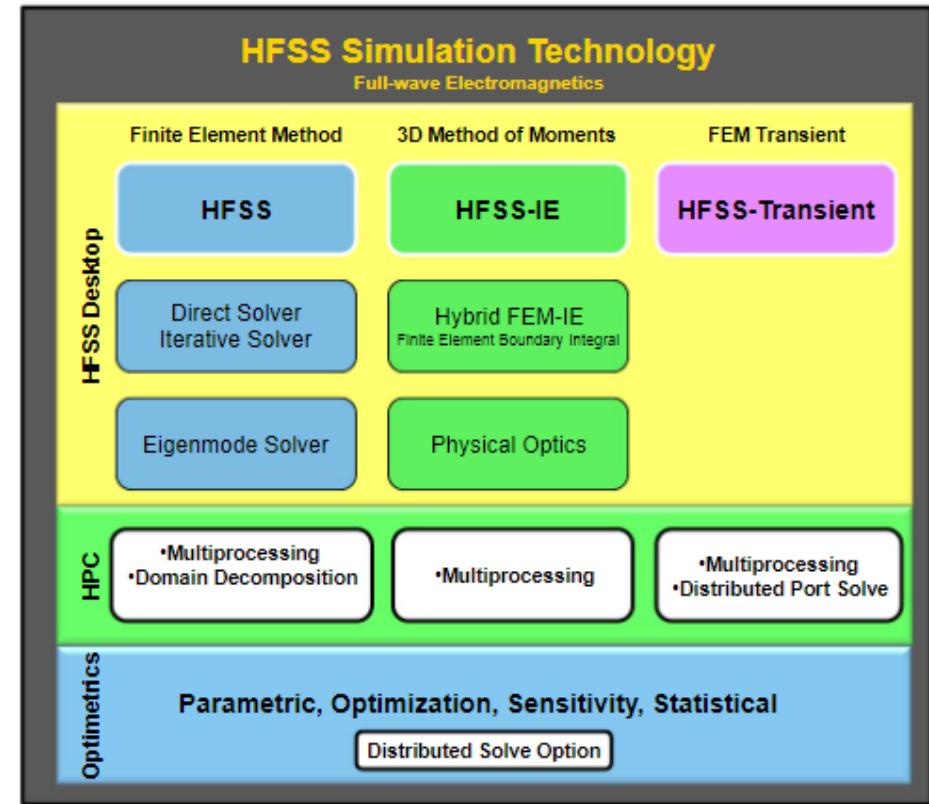
# Introduction to High Frequency Structure Simulator HFSS

Ansoft HFSS can be used to calculate parameters such as S-Parameters, Resonant Frequency, and Fields. Typical uses include:

- Antennas/Mobile Communications – Patches, Dipoles, Horns, Conformal Cell Phone Antennas, Quadrafilar Helix, Specific Absorption Rate (SAR), Infinite Arrays, Radar Cross Section (RCS), Frequency Selective Surfaces (FSS)
- Waveguide – Filters, Resonators, Transitions, Couplers
- Filters – Cavity Filters, Microstrip, Dielectric Package Modeling – BGA, QFP, Flip-Chip
- EMC/EMI – Shield Enclosures, Coupling, Near- or Far-Field Radiation
- PCB Modeling – Power/Ground planes, Mesh Grid Grounds, Backplanes
- Silicon/GaAs - Spiral Inductors, Transformers
- Connectors – Coax, SFP/XFP, Backplane, Transitions

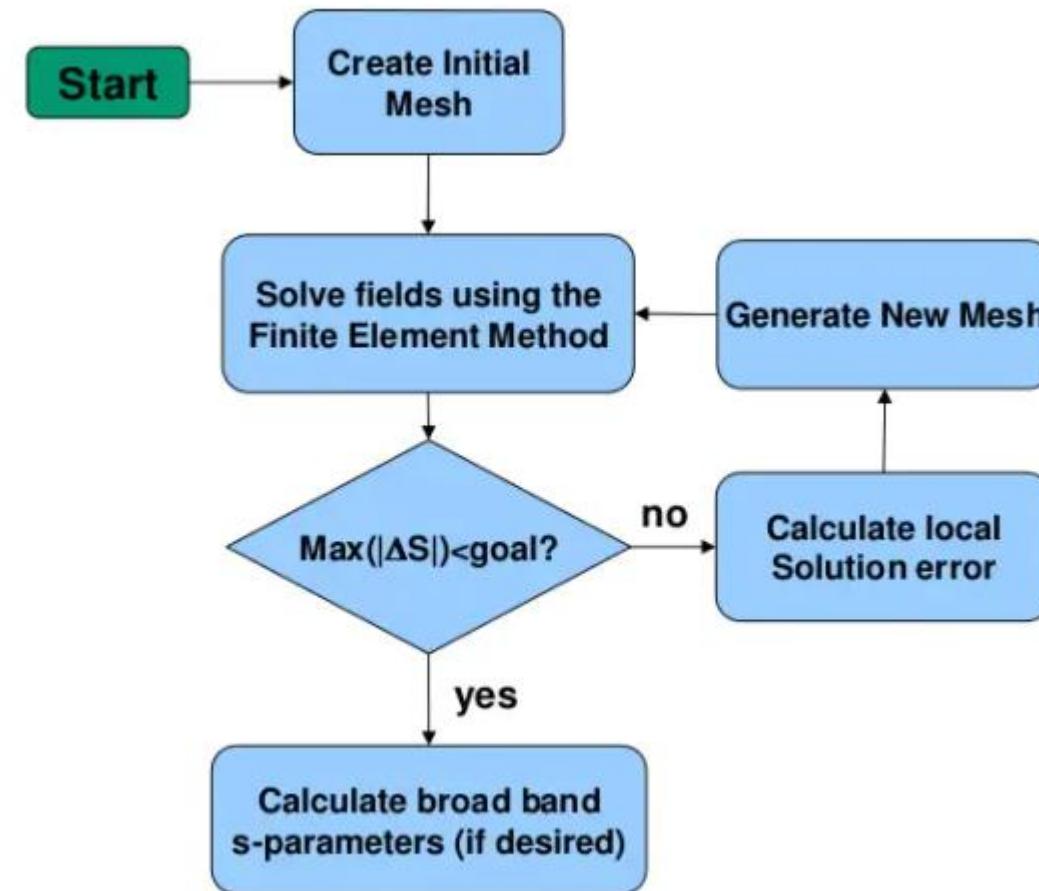
# Introduction to High Frequency Structure Simulator HFSS

- HFSS is an interactive simulation system whose FEM solution uses the tetrahedron mesh element. This allows you to solve any arbitrary 3D geometry, especially those with complex curves and shapes, in a fraction of the time it would take using other techniques.
- Ansoft pioneered the use of the Finite Element Method (FEM) for EM simulation by developing/implementing technologies such as tangential vector finite elements, adaptive meshing, and Adaptive Lanczos-Pade Sweep (ALPS). Today, HFSS continues to lead the industry with innovations such hybrid FEM-IE solution, FEM based transient solution and domain decomposition methods



# Introduction to High Frequency Structure Simulator HFSS

## Automated Solution Process



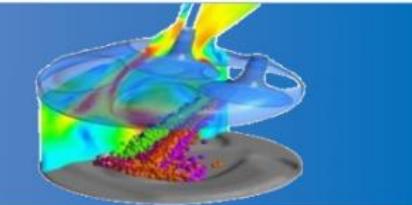
# Breadth of Technologies



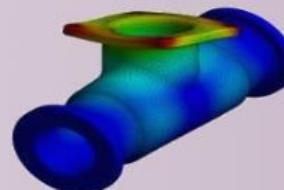
**Fluid Mechanics:**  
From Single-Phase Flows



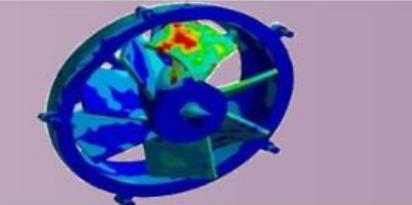
To Multiphase  
Combustion



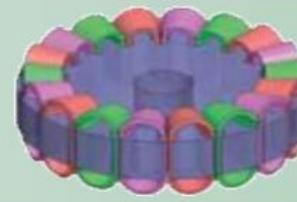
**Structural Mechanics:**  
From Linear Statics



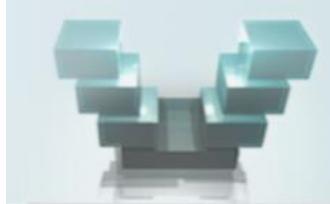
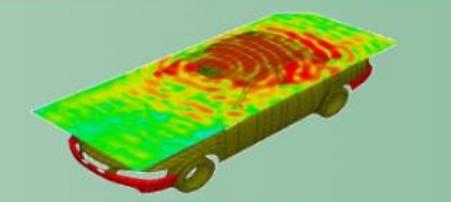
To High-Speed Impact



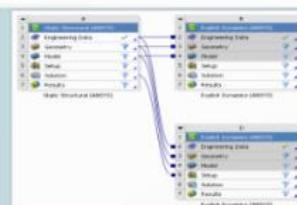
**Electromagnetics:** From  
Low-Frequency Windings



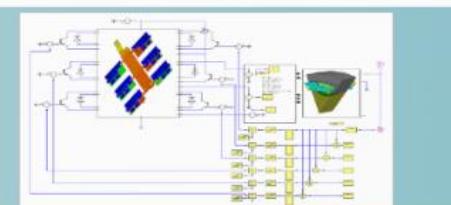
To High-Frequency  
Field Analysis



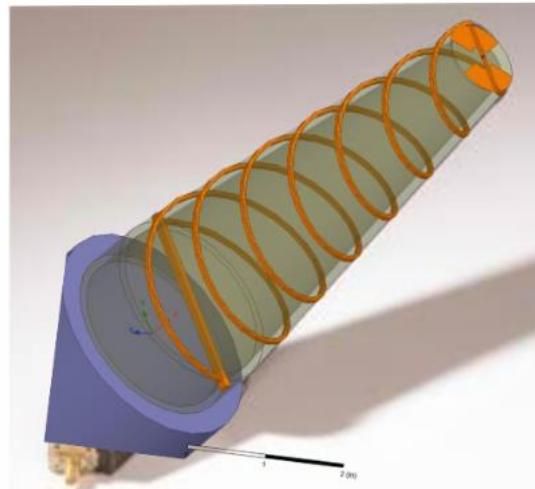
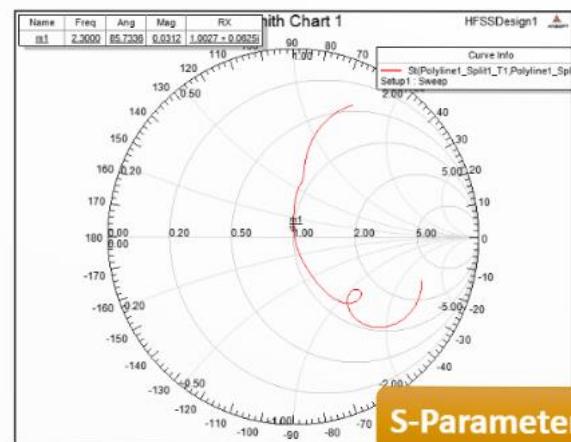
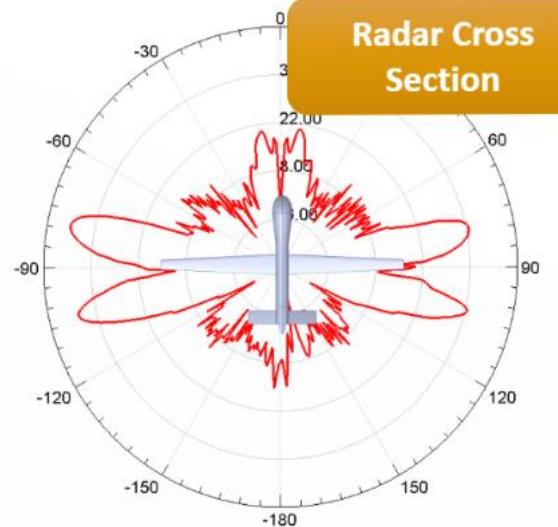
**Systems:**  
From Data Sharing



To Multi-Domain  
System Analysis

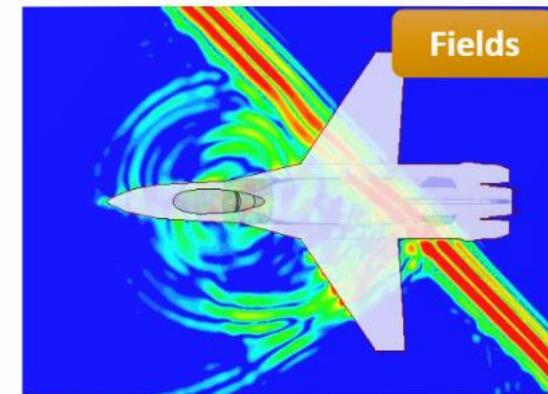
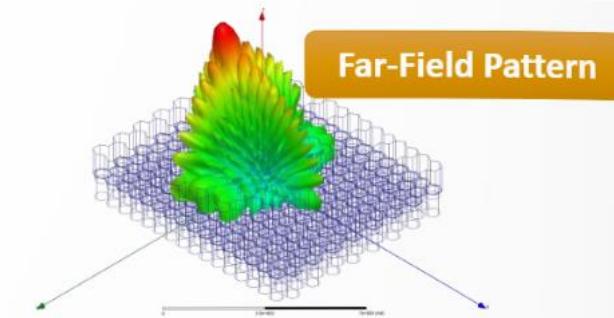


# Virtual Prototypes



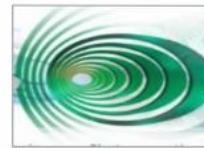
HFSS Virtual Prototype

Virtual Compliance



# Multi-Physics for Electronics

Electromagnetics  
(HFSS, SIwave, Savant,  
Q3D Extractor, Maxwell,  
Designer, EMIT, Simplorer)



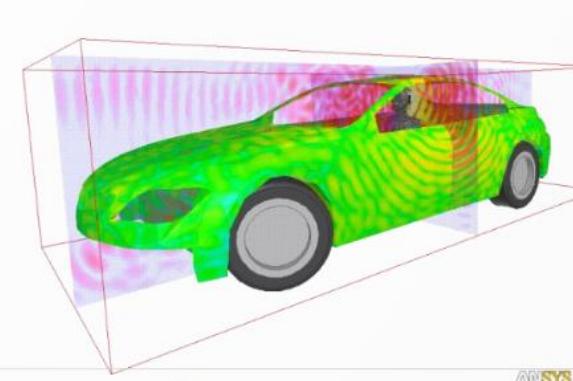
Thermal  
(ANSYS Mechanical, Icepak)



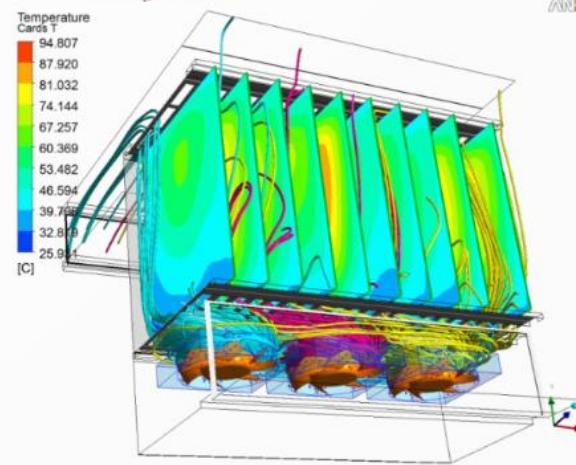
Structural  
(ANSYS Mechanical)



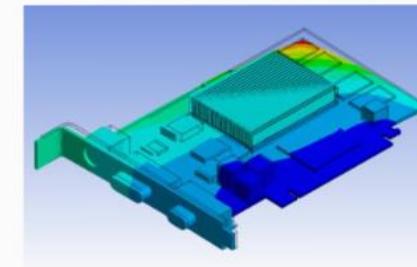
EM Fields  
(HFSS)



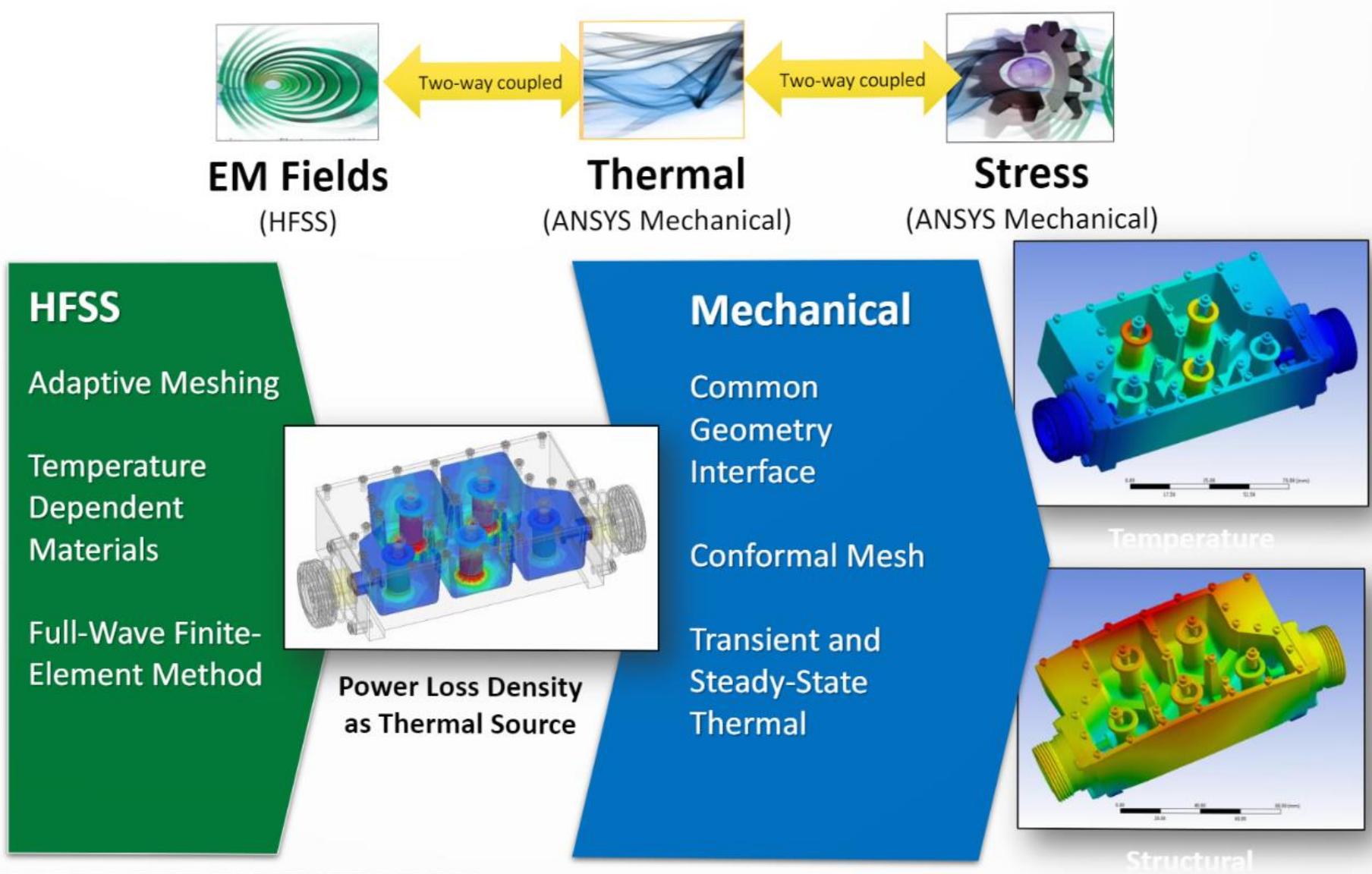
Air Flow  
(Icepak)



Stress  
(Mechanical)

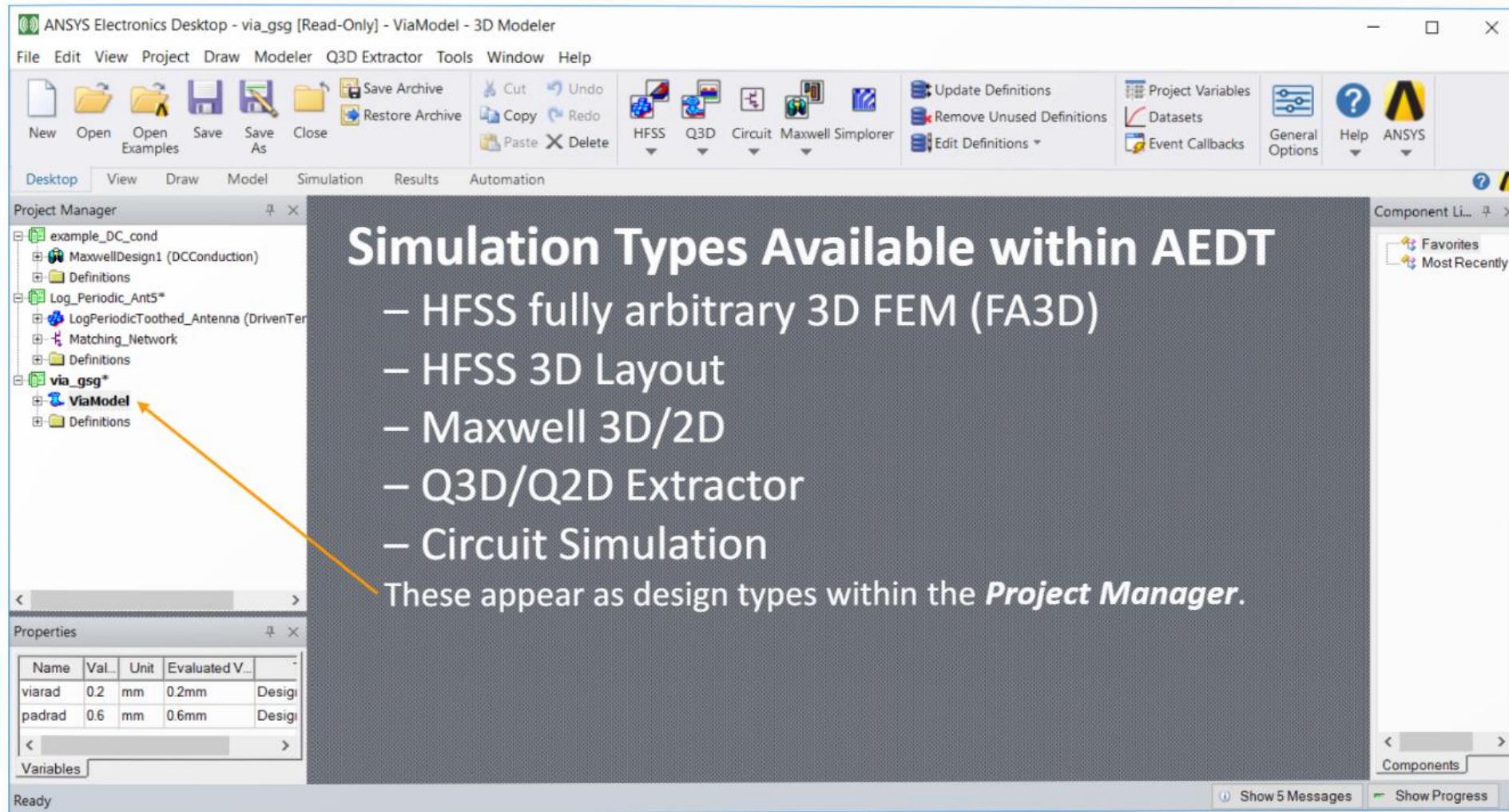


# Multi-Domain: Multiple Physics – Icepak for Thermal with HFSS



# HFSS in ANSYS Electronics Desktop (AEDT)

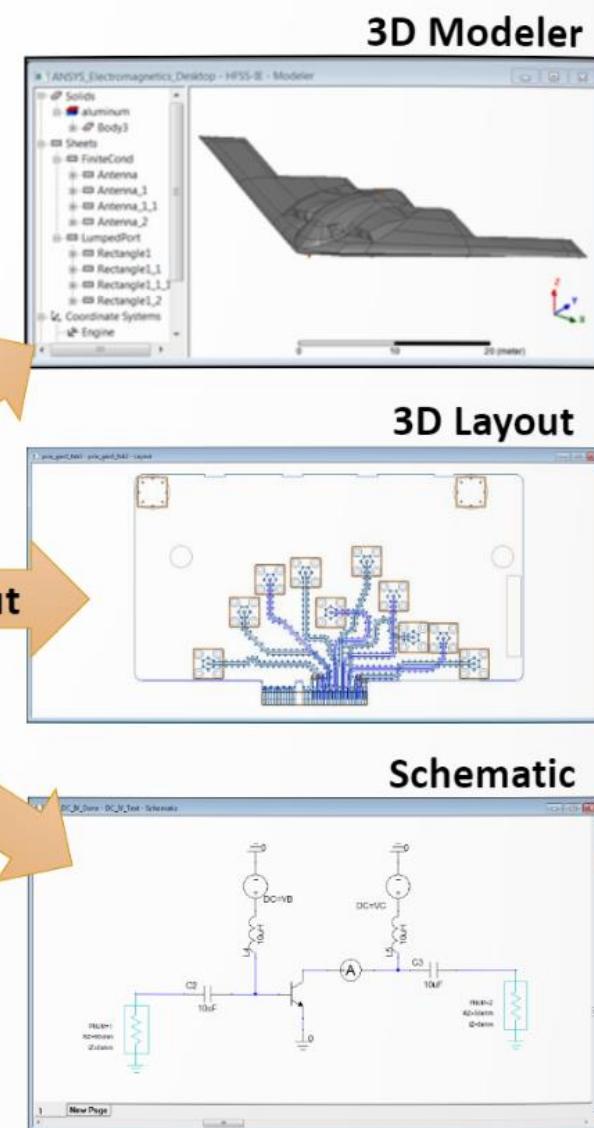
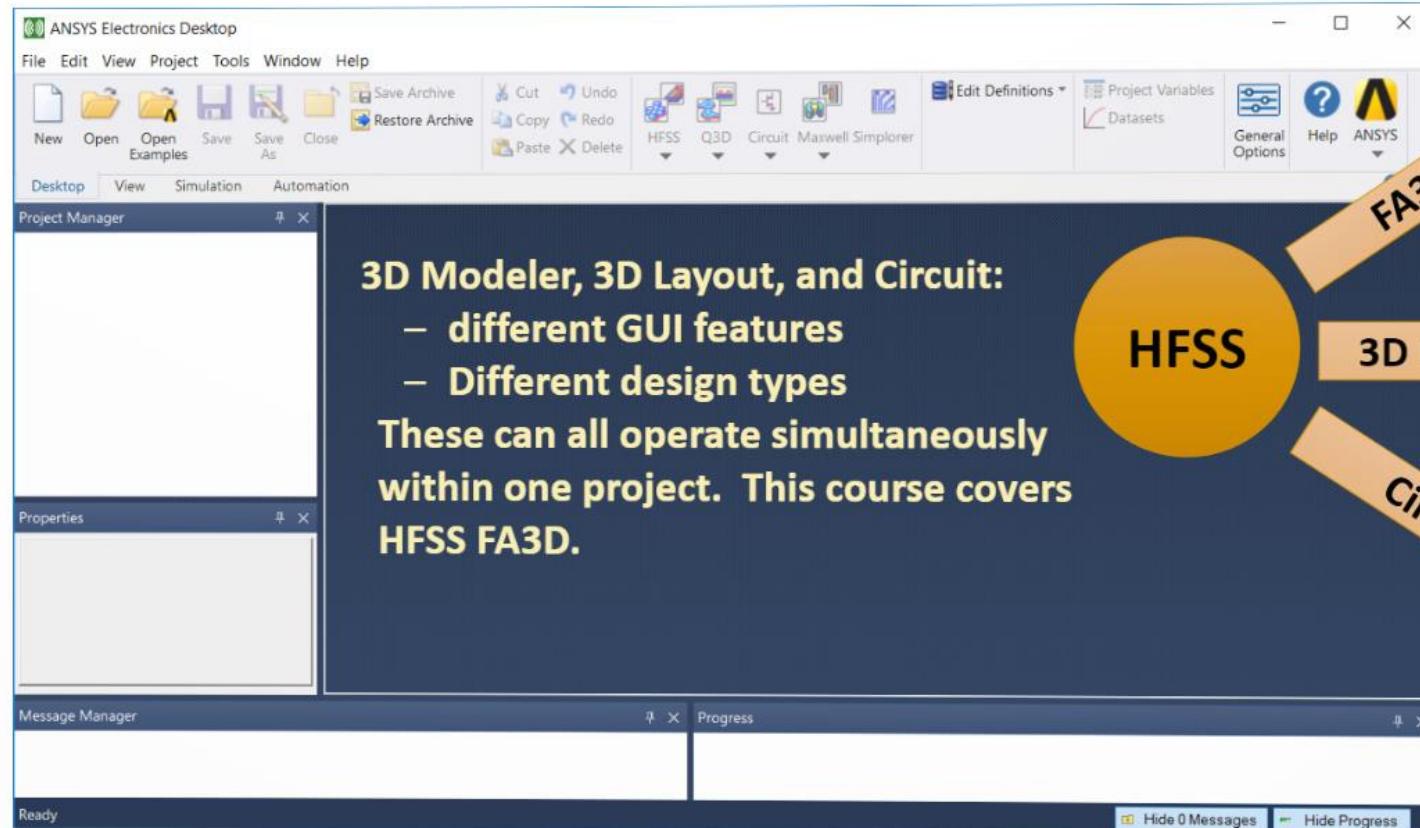
- The ANSYS Electronic Desktop is a graphical user interface (GUI) common to many electronic simulation tools.



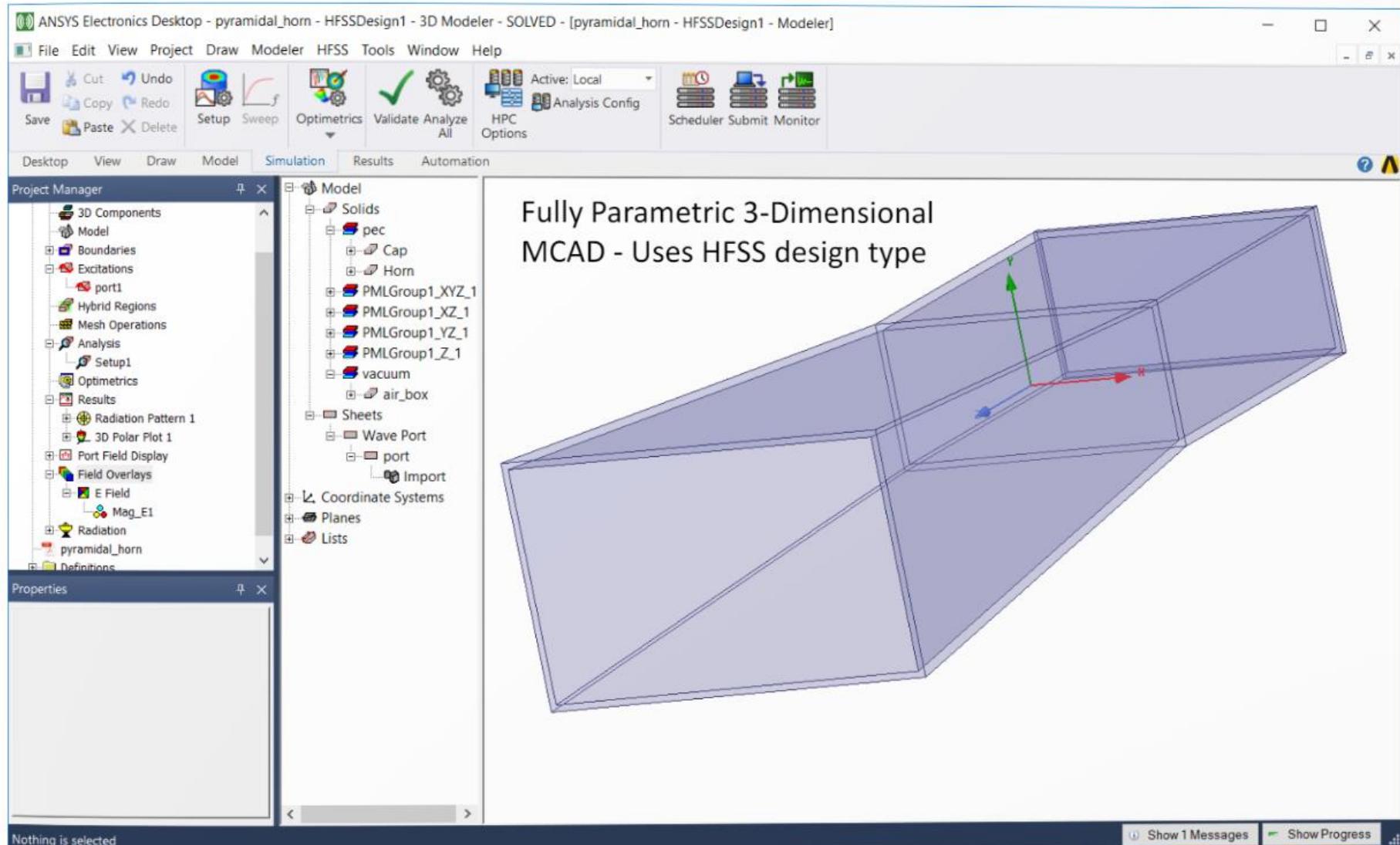
HFSS runs  
in AEDT.  
An HFSS  
project file  
extension  
is \*.aedt

# HFSS in ANSYS Electronics Desktop (AEDT)

3 Basic Interfaces - 1 Desktop



# HFSS 3D Renderer - Mechanical CAD (MCAD)



HFSS 3D MCAD is the most general...and the subject of this course.

For additional background on the basic operations in the ANSYS electronic desktop, AEDT, including file operations, there are a number of resources that come with HFSS.

In the HFSS install directories, such as

`AnsysEM19.X\Win64\Help\HFSS\GSG`

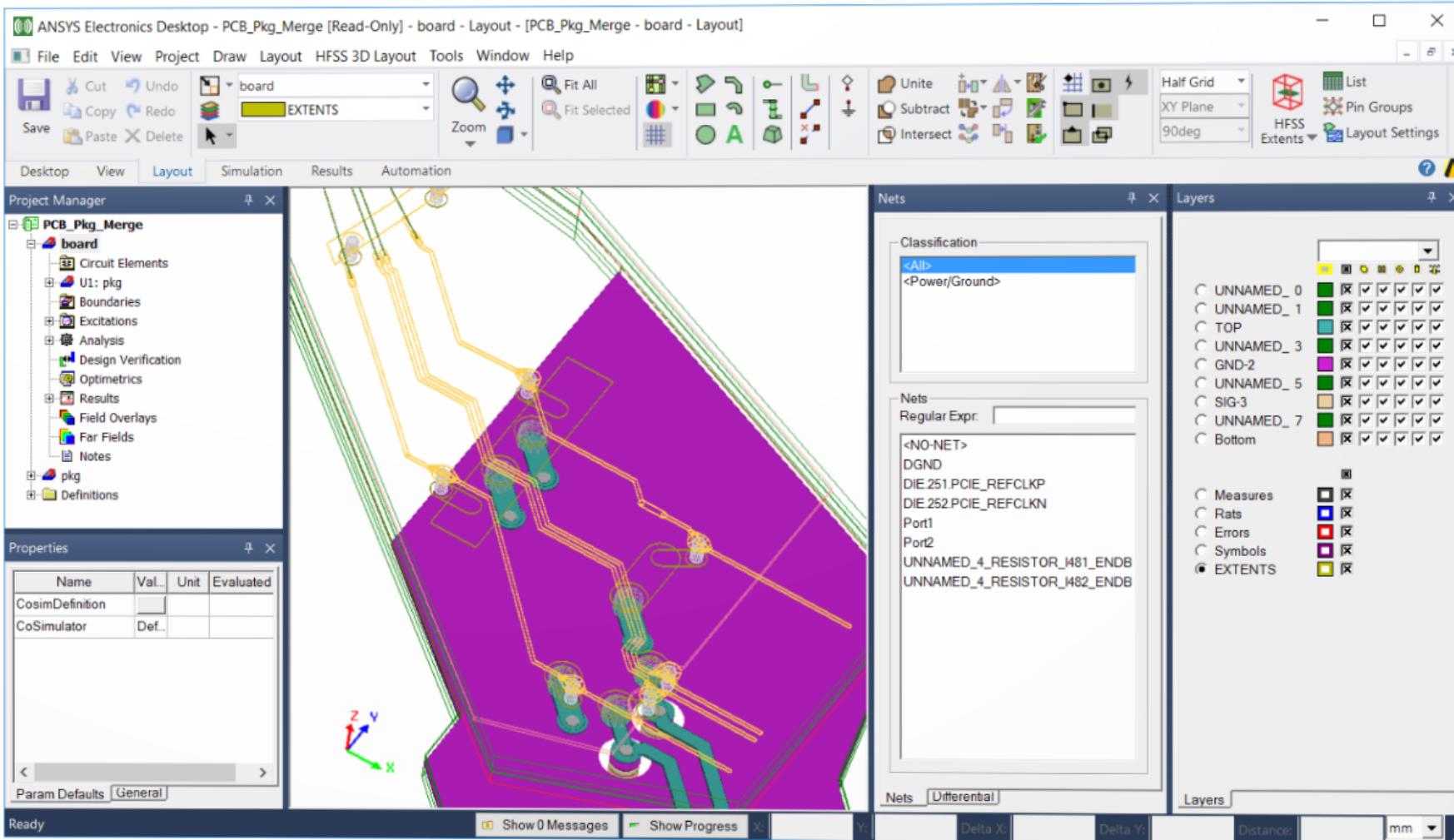
there is an HFSS help document "HFSS.pdf" which includes sections:

***2 - Working with ANSYS Electronics Desktop Projects***

including opening, closing, and saving project

# HFSS 3D Layout Editor

## 3-Dimensional Electrical CAD (ECAD) - Fully Parametric

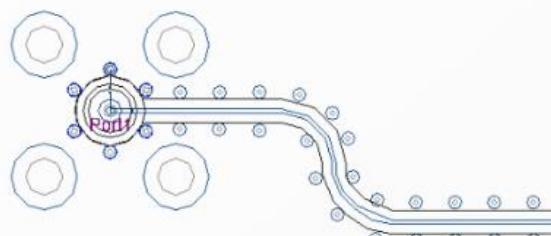
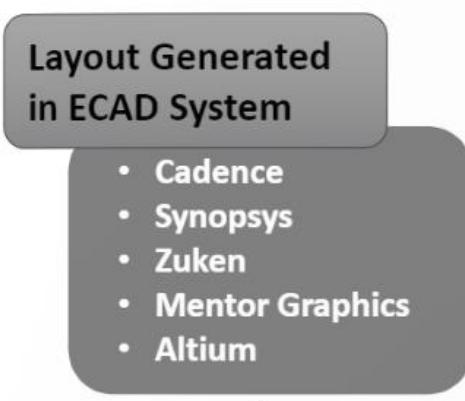


There is an HFSS 3D Layout design type. The **Layers** and **Nets** functionality is available in HFSS 3D Layout.

# HFSS 3D Layout Editor

## HFSS 3D Layout Integration (ANSYS Designer)

- Native Layout Editor for 3D HFSS simulations
  - Cadence, Mentor, Zuken, Altium, DXF, GDSII

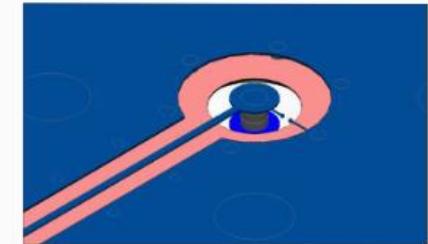


Native Layout Editor

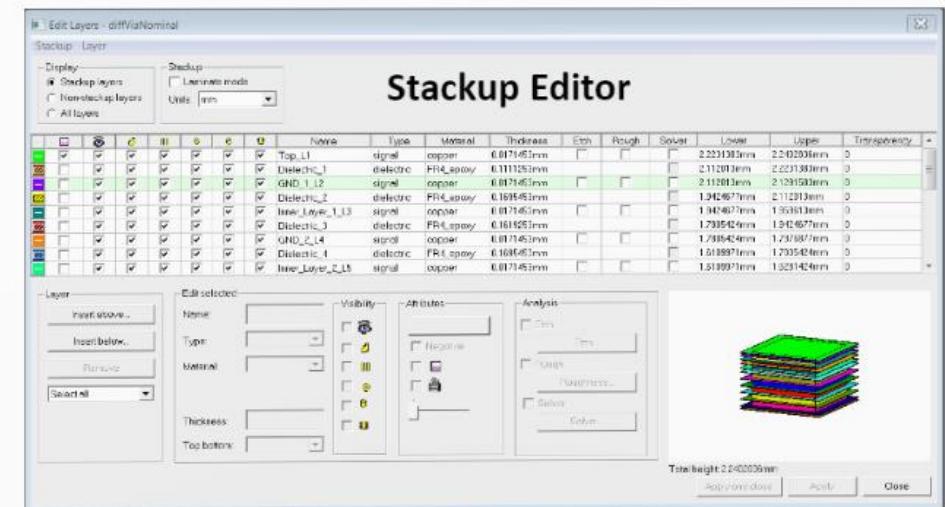


3rd Party Layout Translation

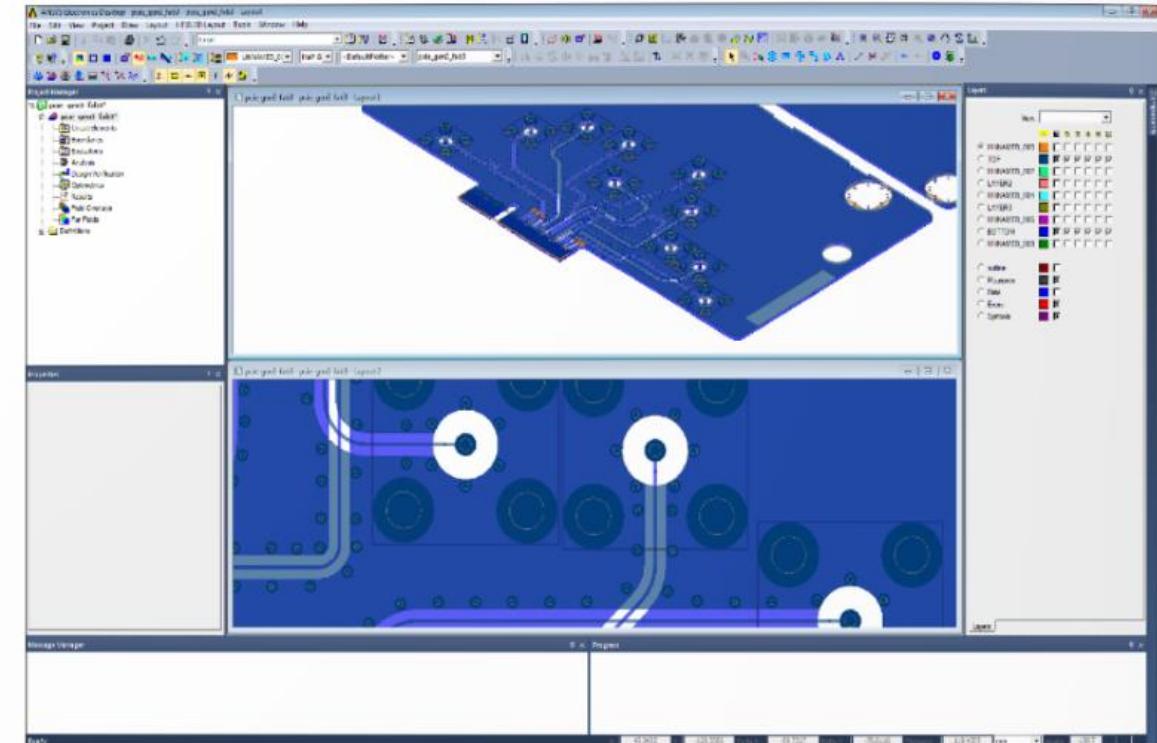
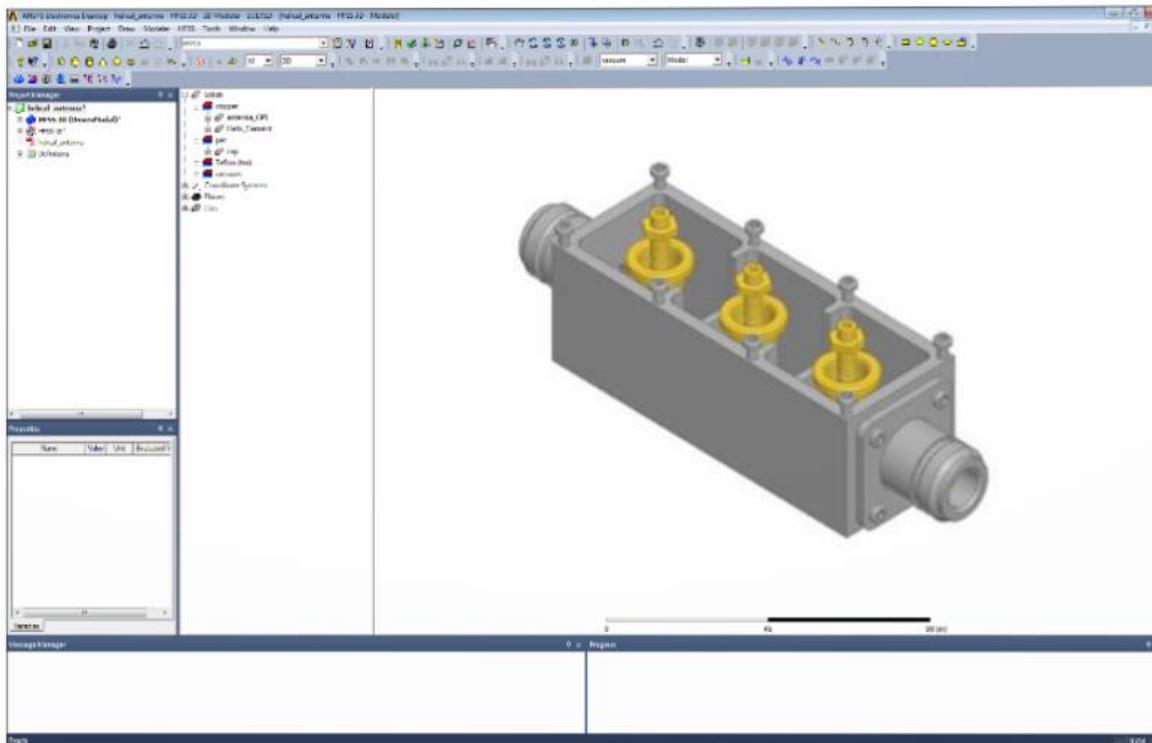
Alinks for EDA



HFSS Circuit Port



# Different Interfaces – Same HFSS FEM Solver

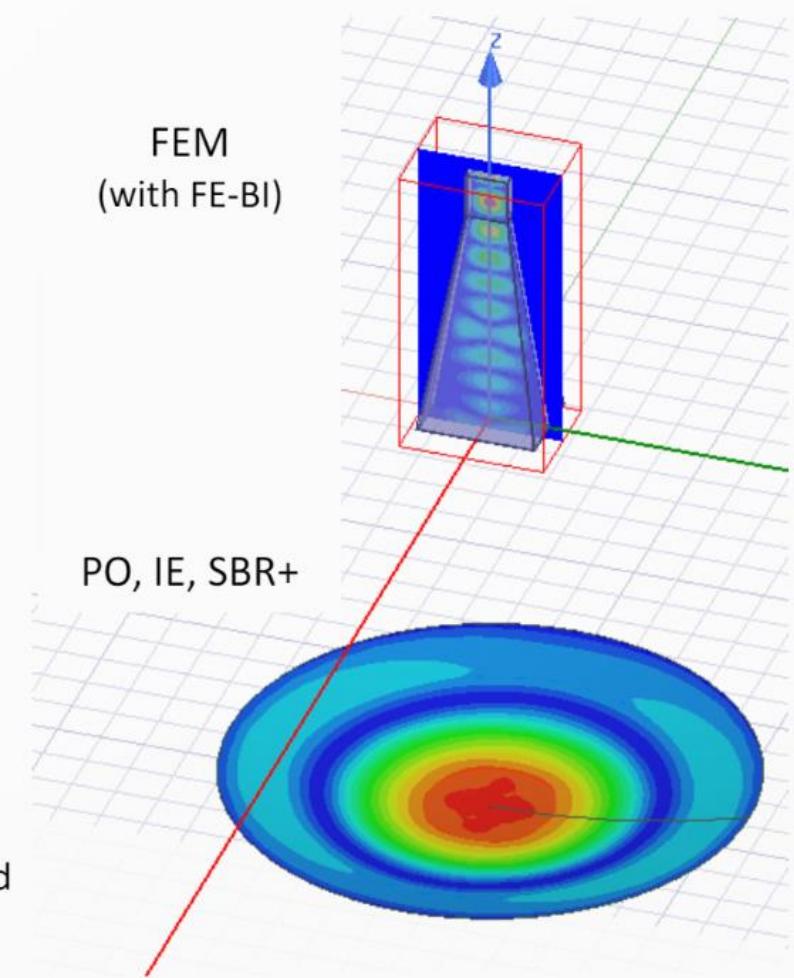


Regardless of which User Interface is used, engineers have access to:

- full parametric modeling to aid in design space exploration
- the same Finite Element Method (HFSS-FEM) field solver
- HFSS's Automatic Adaptive Meshing Process for unparalleled accuracy

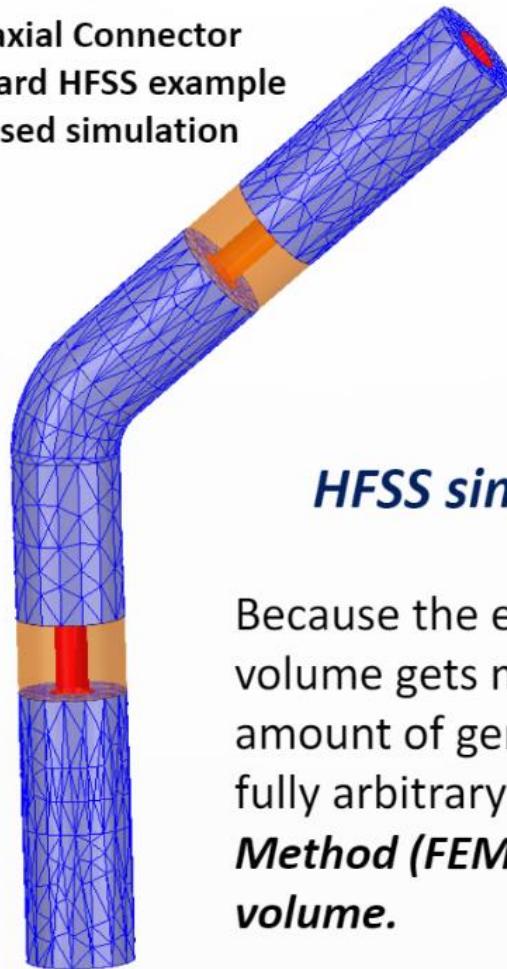
# HFSS Includes Multiple EM Solvers

- **HFSS FEM (Finite Element Method)**
  - Fully arbitrary 3D - the whole simulation space gets meshed
  - Used for microwave, antenna, and PCB signal integrity applications
  - HFSS is also a “design type” within the HFSS product.
- **HFSS IE (Integral Equation) Solver**
  - 3D surface meshing – but only meshes surfaces
  - Commonly used for antenna applications
  - Available within the HFSS design type
- **HFSS PO (Physical Optics) and SBR+ (Shooting Bouncing Ray) Solvers**
  - Approaches wave propagation in terms of rays
  - Commonly used for antenna applications
  - Available within the HFSS design type
- **HFSS Transient Solver**
  - Time domain formulation that can employ pulsed excitations
  - Commonly used for applications such as EMI (electromagnetic interference)
- **HFSS Eigenmode Solver**
  - Used to obtain fields in cavities and periodic structures along with the associated dispersion curves
  - No excitation needed - not a driven solution



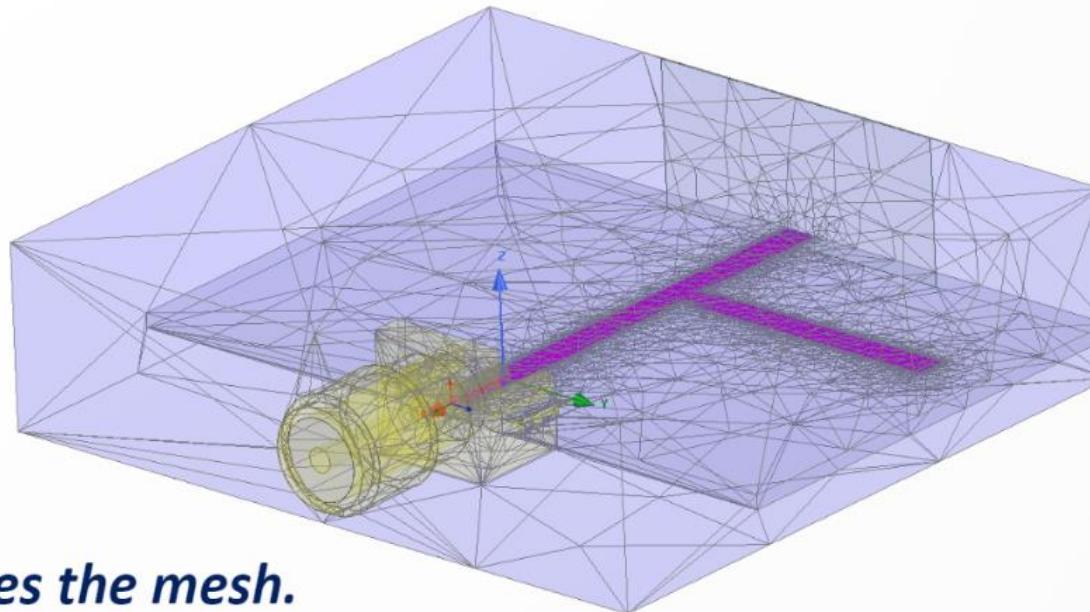
# HFSS Fully Arbitrary 3D FEM Meshes the Entire Simulation Space

Coaxial Connector  
Standard HFSS example  
Closed simulation



*HFSS simulates the mesh.*

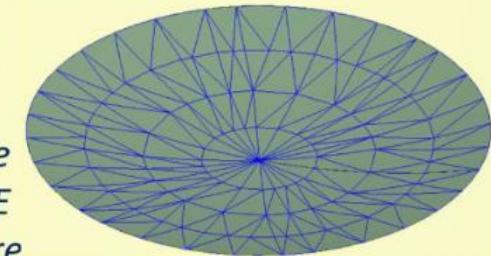
Because the entire computational volume gets meshed, the greatest amount of generality is available – fully arbitrary 3D. **Finite Element Method (FEM) solves for fields in a volume.**



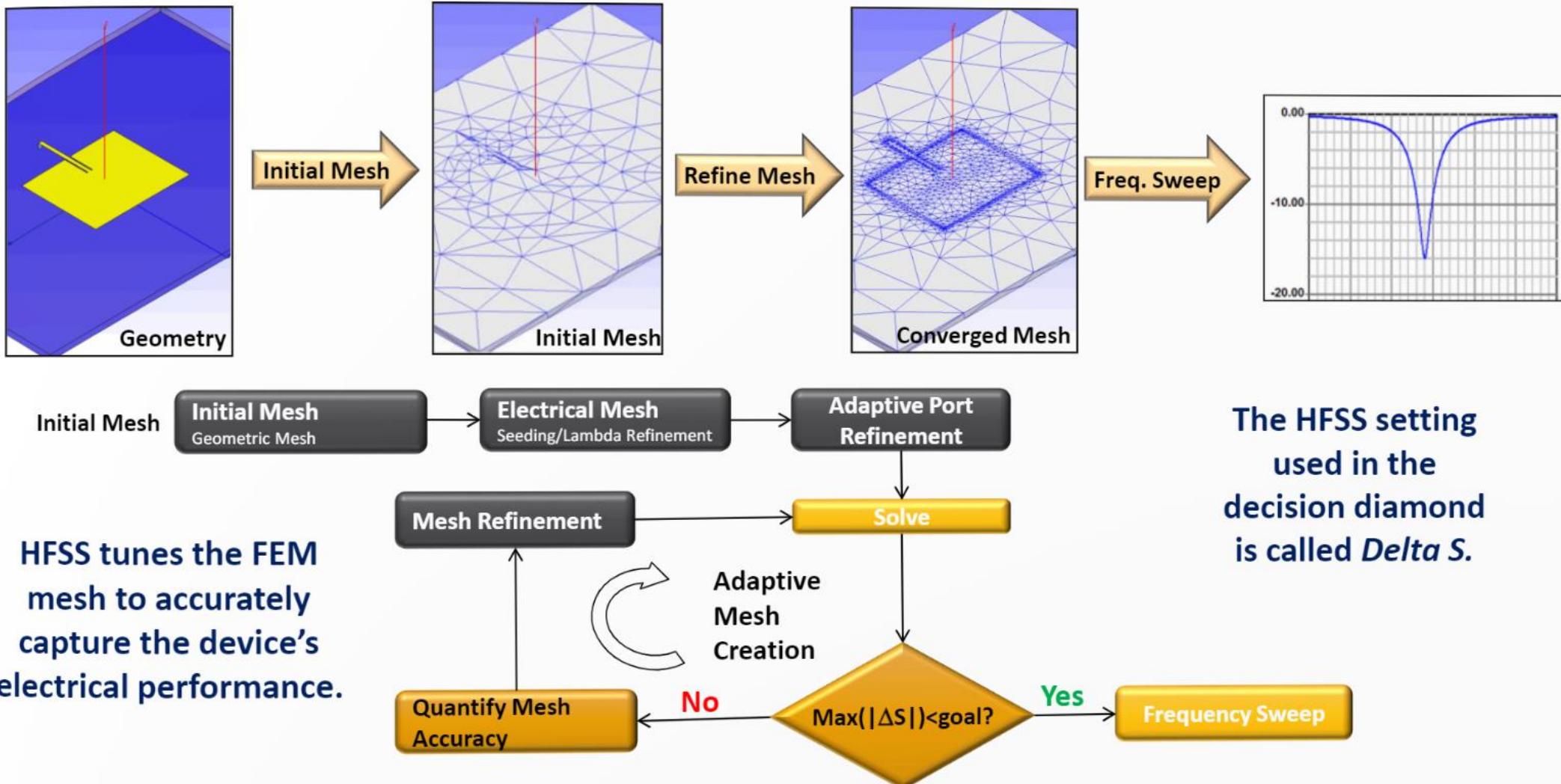
SMA coaxial to  
microstrip  
transition  
Open simulation

*What else is there...besides fully arbitrary 3D?*

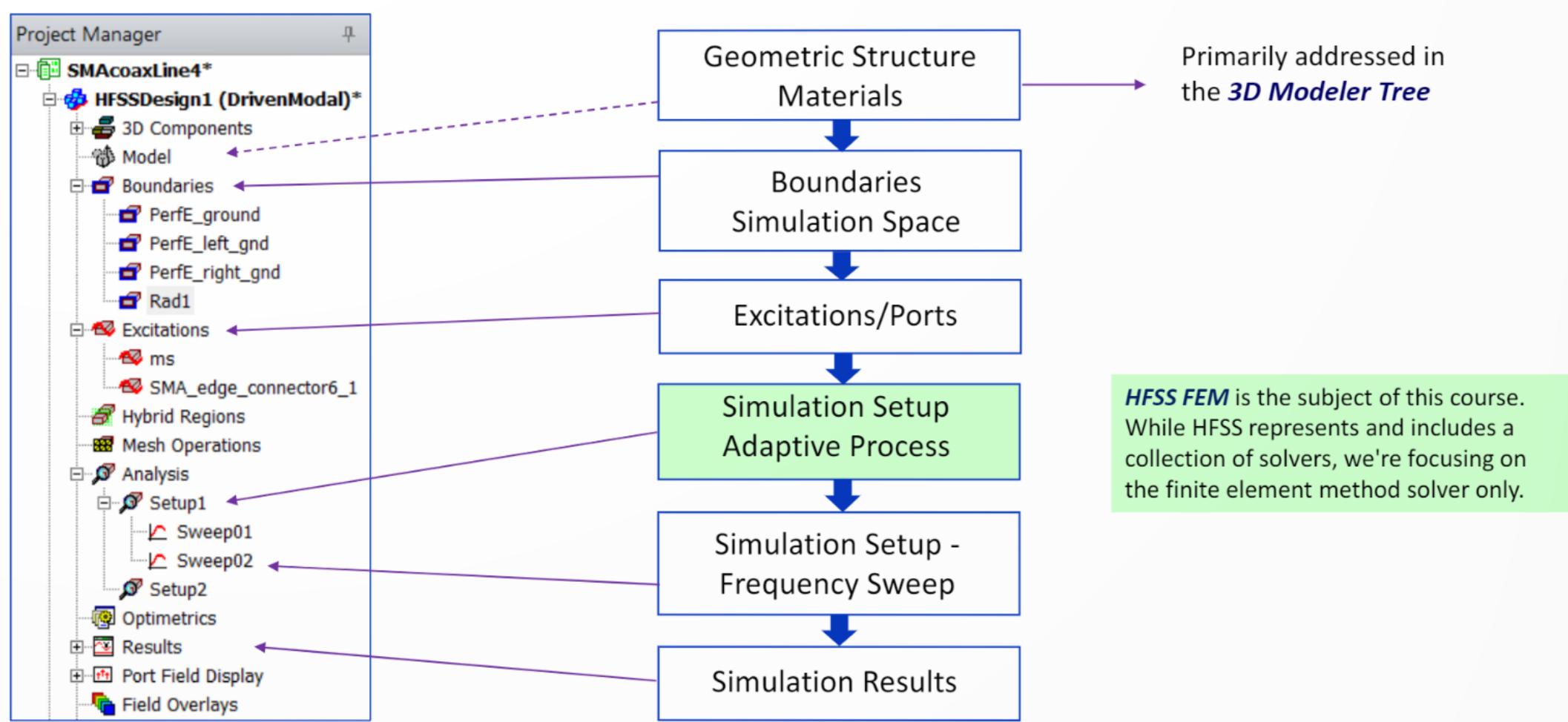
For comparison, the HFSS IE solver meshes surfaces and simulates fields at a distance away from that structure. IE reflector antenna shown here.



# HFSS FEM Automated Solution Adaptive Meshing Process



# The HFSS Project manager Reflects EM Simulation Workflow

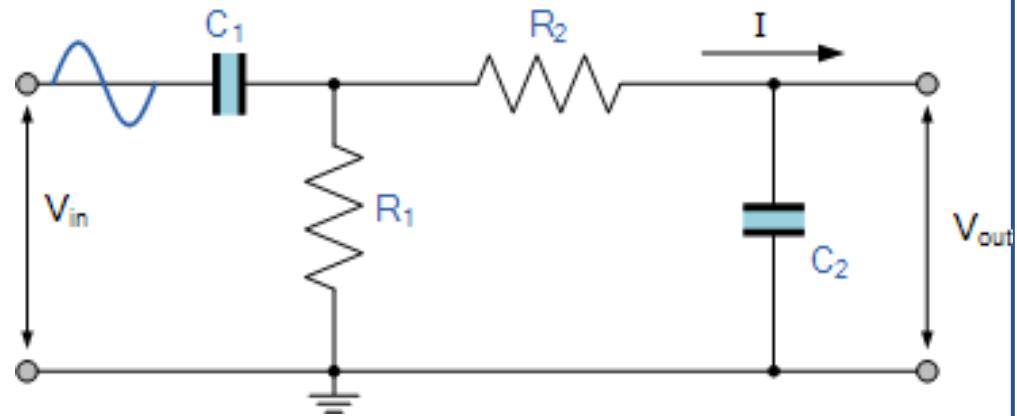


# Band Pass Filter Simulation

# Band Pass Filter Simulation

## Passive Bandpass Filter

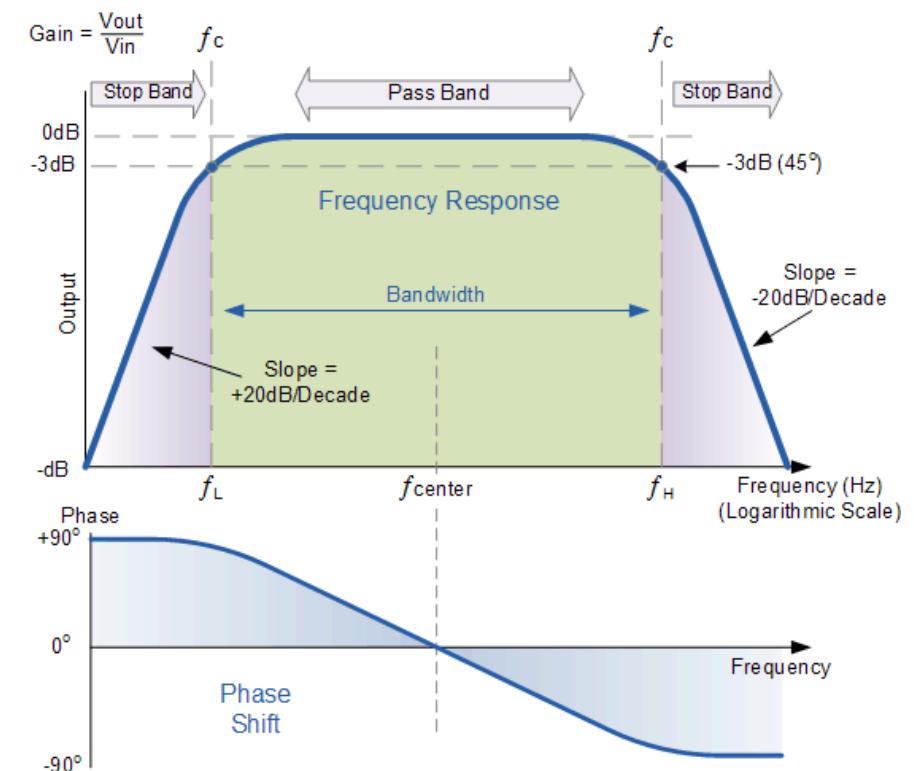
- Band Pass Filters can be used to isolate or filter out certain frequencies that lie within a particular band or range of frequencies. The cut-off frequency or  $f_c$  point in a simple RC passive filter can be accurately controlled using just a single resistor in series with a non-polarized capacitor, and depending upon which way around they are connected, we have seen that either a Low Pass or a High Pass filter is obtained.
- One simple use for these types of passive filters is in audio amplifier applications or circuits such as in loudspeaker crossover filters or pre-amplifier tone controls.



# Band Pass Filter Simulation

## Frequency Response of a 2nd Order Band Pass Filter

- The Bode Plot or frequency response curve above shows the characteristics of the band pass filter. Here the signal is attenuated at low frequencies with the output increasing at a slope of +20dB/Decade (6dB/Octave) until the frequency reaches the “lower cut-off” point  $f_L$ . At this frequency the output voltage is again  $1/\sqrt{2} = 70.7\%$  of the input signal value or -3dB ( $20 \cdot \log(V_{OUT}/V_{IN})$ ) of the input.
- The output continues at maximum gain until it reaches the “upper cut-off” point  $f_H$  where the output decreases at a rate of -20dB/Decade (6dB/Octave) attenuating any high frequency signals. The point of maximum output gain is generally the geometric mean of the two -3dB value between the lower and upper cut-off points and is called the “Centre Frequency” or “Resonant Peak” value  $f_r$ . This geometric mean value is calculated as being  $f_r^2 = f(\text{UPPER}) \times f(\text{LOWER})$ .



# Band Pass Filter Simulation

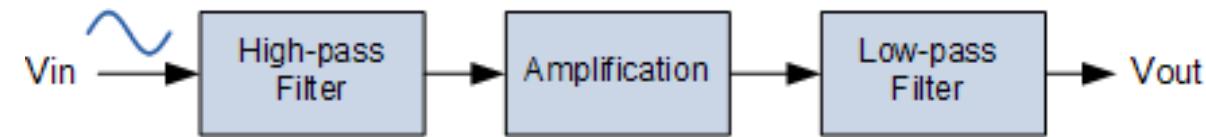
- A band pass filter is regarded as a second-order (two-pole) type filter because it has “two” reactive components within its circuit structure, then the phase angle will be twice that of the previously seen first-order filters, ie, 180o. The phase angle of the output signal LEADS that of the input by +90o up to the centre or resonant frequency,  $f_r$  point were it becomes “zero” degrees (0o) or “in-phase” and then changes to LAG the input by -90o as the output frequency increases.
- The upper and lower cut-off frequency points for a band pass filter can be found using the same formula as that for both the low and high pass filters, For example

$$f_c = \frac{1}{2\pi RC} \text{ Hz}$$

# Band Pass Filter Simulation

## Active Bandpass Filter

- For a low pass filter this pass band starts from 0Hz or DC and continues up to the specified cut-off frequency point at -3dB down from the maximum pass band gain. Equally, for a high pass filter the pass band starts from this -3dB cut-off frequency and continues up to infinity or the maximum open loop gain for an active filter.
- However, the Active Band Pass Filter is slightly different in that it is a frequency selective filter circuit used in electronic systems to separate a signal at one particular frequency, or a range of signals that lie within a certain “band” of frequencies from signals at all other frequencies. This band or range of frequencies is set between two cut-off or corner frequency points labelled the “lower frequency” (  $f_L$  ) and the “higher frequency” (  $f_H$  ) while attenuating any signals outside of these two points.



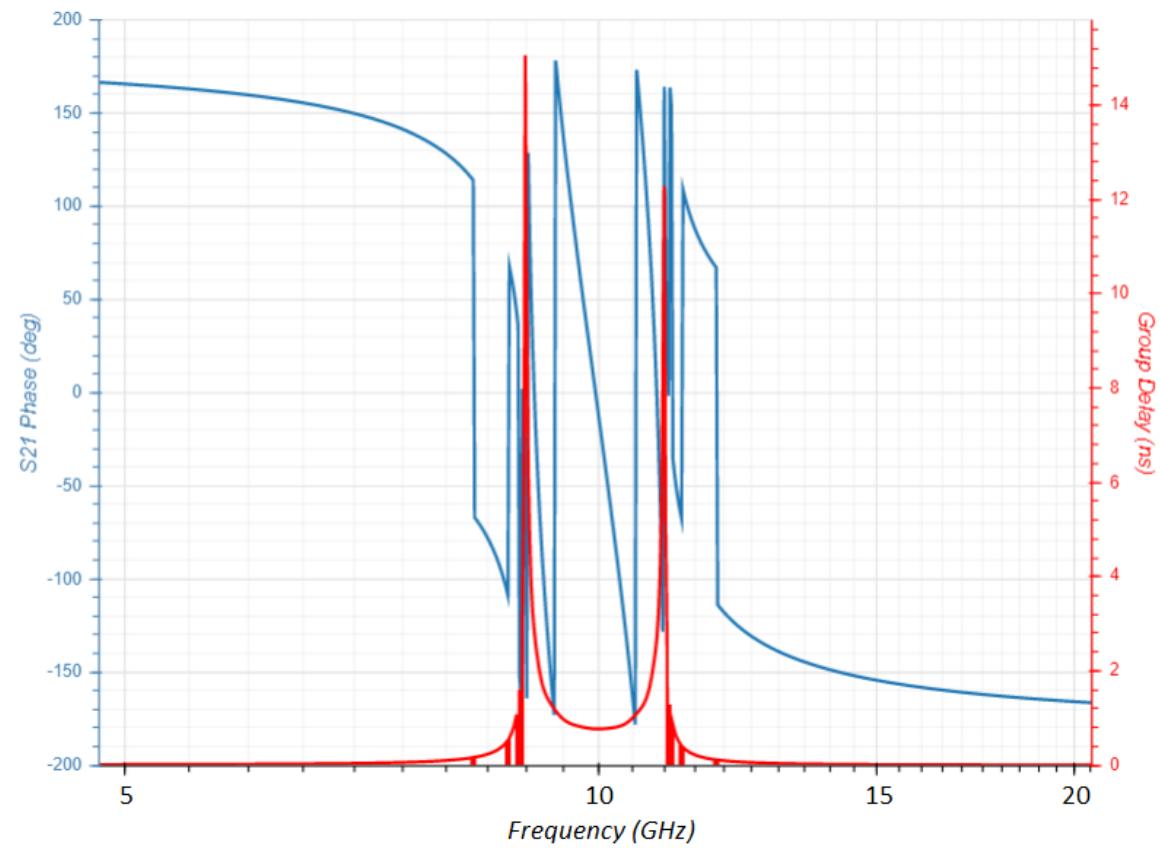
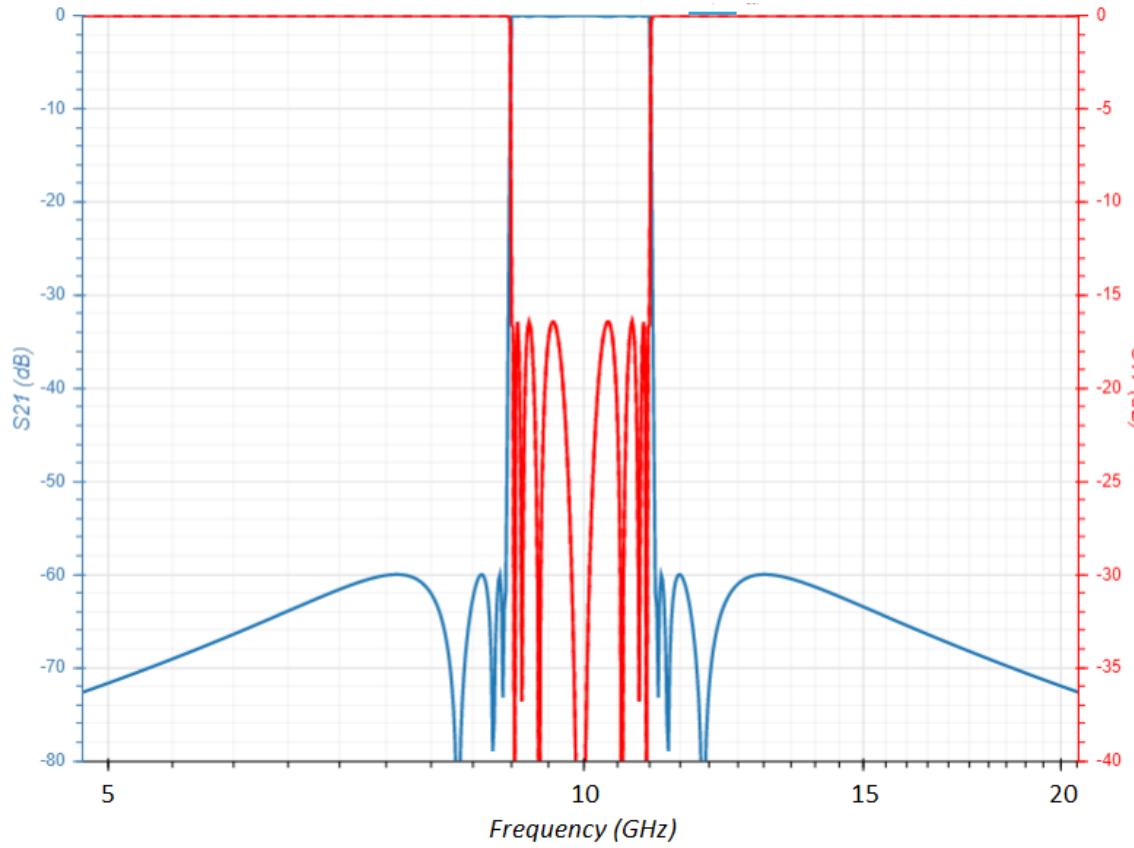
# Band Pass Filter Simulation

## ***Passive RF Bandpass Filter Simulation***

- With RF/wireless devices, the goal is to make the return loss seen by the propagating signal as low as possible, meaning that the scattering parameter S11 should be as negative as possible when measured in dB. Ideally, an RF bandpass filter simulation will help you design the filter so that the return loss spectrum nicely overlaps the carrier signal and the modulation signals (in frequency modulation). Similarly, you want to bring insertion loss (S21) as close to 0 dB as possible.
- With antennas, you want to pass as much of the carrier signal to the antenna as possible, bringing the voltage standing wave ratio (VSWR) as close to 1 as possible. Commercially available antennas that include filtration/impedance matching circuits typically quote VSWR values between 1 and 2 for these components.
- Just like any filter, signals that fall within the pass band and are output from an RF bandpass filter will acquire a phase shift. Filters also produce a group delay, but this is generally ignored when working with a purely sinusoidal signal of constant amplitude. With multiplexed amplitude-modulated signals (for example, quadrature amplitude modulation), the group delay becomes very important as this represents the delay in the time-dependent amplitude of specific AC frequencies.

# Band Pass Filter Simulation

## Passive RF Bandpass Filter Simulation



# Band Pass Filter Simulation

## Determining Losses, Group Delay, and Phase Delay

With any filter, the response can be determined using frequency sweeps at particular sinusoidal frequencies. Once you have a measurement of the output voltage and the input voltage at each frequency, you can determine the phase difference between the two signals. You can use the ratio of these voltages to determine insertion loss and return loss. If you are working with an antenna, you'll also want to calculate the VSWR from the return loss:

### Insertion Loss

$$IL = 20 \log \left( \left| \frac{V_{out}}{V_{in}} \right| \right) \text{ in dB}$$

### Return Loss

$$RL = -20 \log \left( \left| \frac{V_{ref}}{V_{in}} \right| \right) \text{ in dB}$$

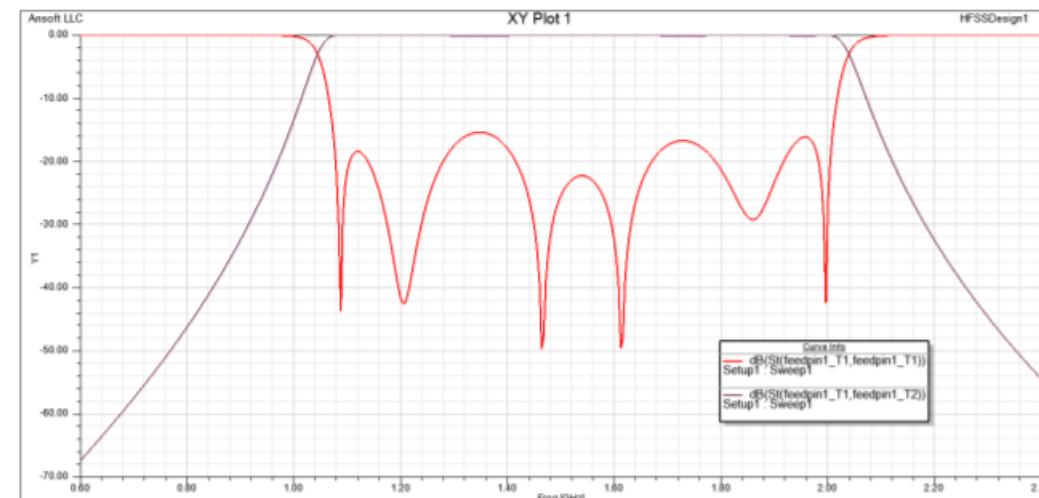
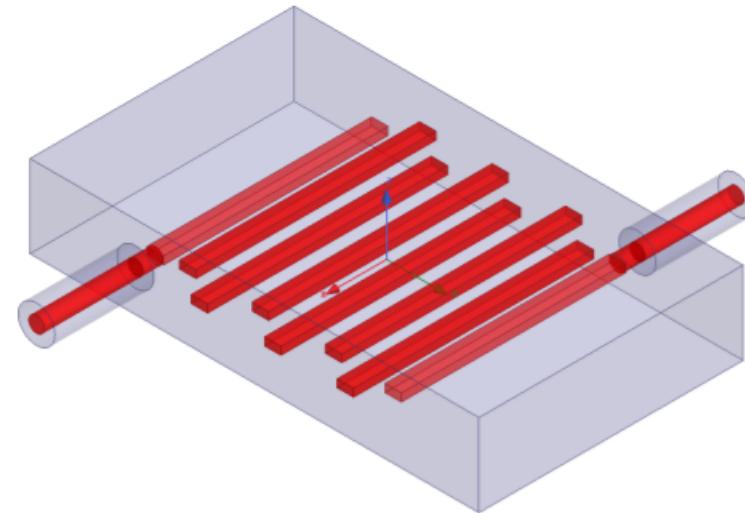
### VSWR

$$VSWR = \frac{1 + \left| \frac{V_{ref}}{V_{in}} \right|}{1 - \left| \frac{V_{ref}}{V_{in}} \right|}$$

# Outline - Band Pass Filter HFSS FEM Simulation Workshop

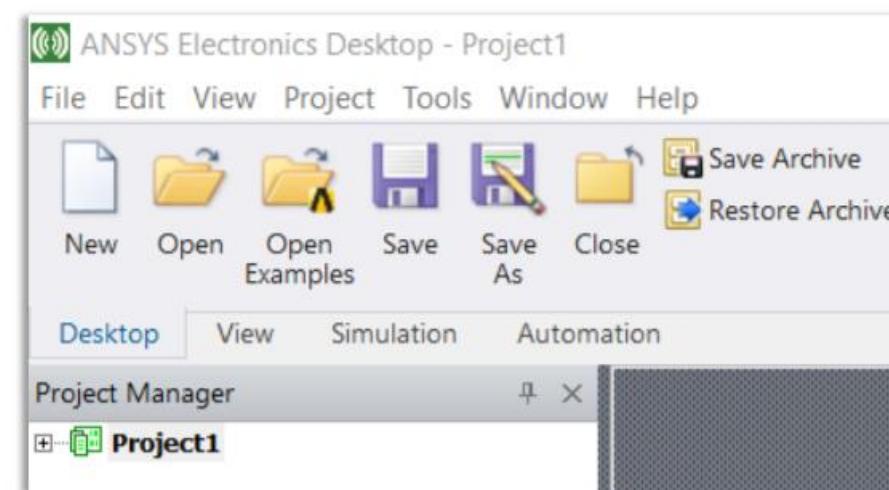
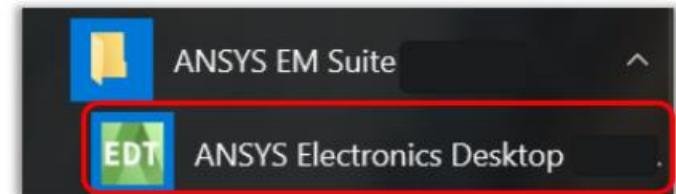
Here are the steps that this workshop will do:

- Invoke HFSS and open the installed Example ***bp\_filter.aedt***
- Save filter project to working directory as ***bp\_filter1.aedt***
- Expand **Project Manager** if not already, including **Design**
- Click on **Show Messages** and **Show Progress**. (Or pull bar up).
- Click **Simulation** tab in the ribbon
- Click on the **Validate** check mark and then **Analyze All**
- Expand **Messages** to see simulation completion.
- Expand **Results** and double click on **XY Plot 1**
- Optional: Make S-parameter traces thicker
- Optional: Change title of plot



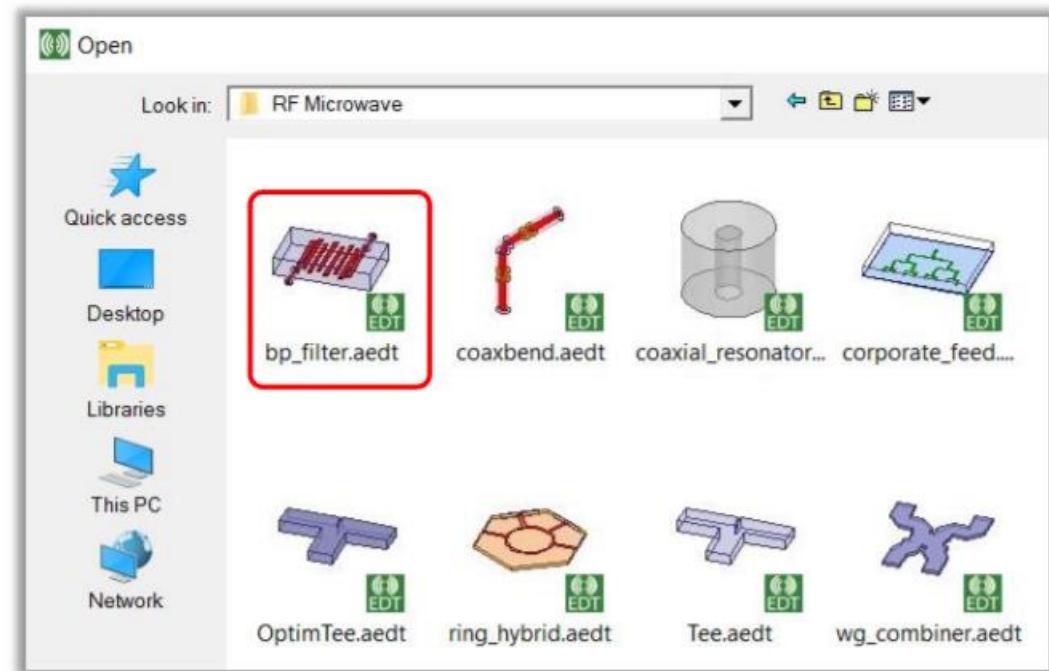
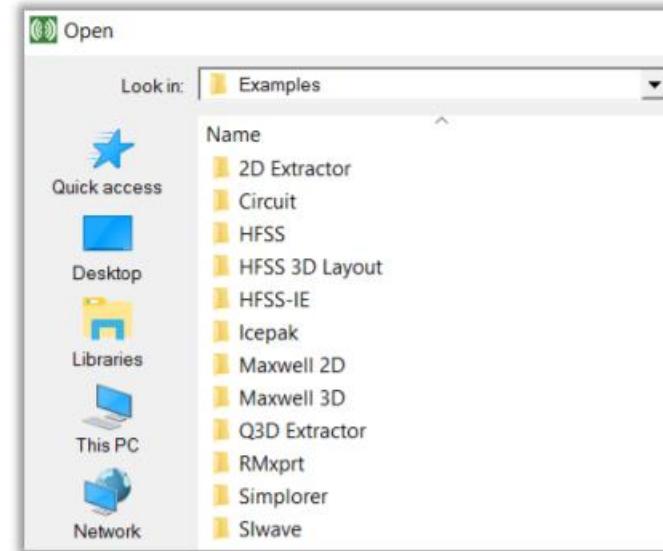
# Launch AEDT – Blank Project

- To access HFSS click the Microsoft Start button, select  
**Programs > ANSYS EM Suite 20XXRY > ANSYS Electronics Desktop**  
A new **Project1** appears under the **Project Manager**.



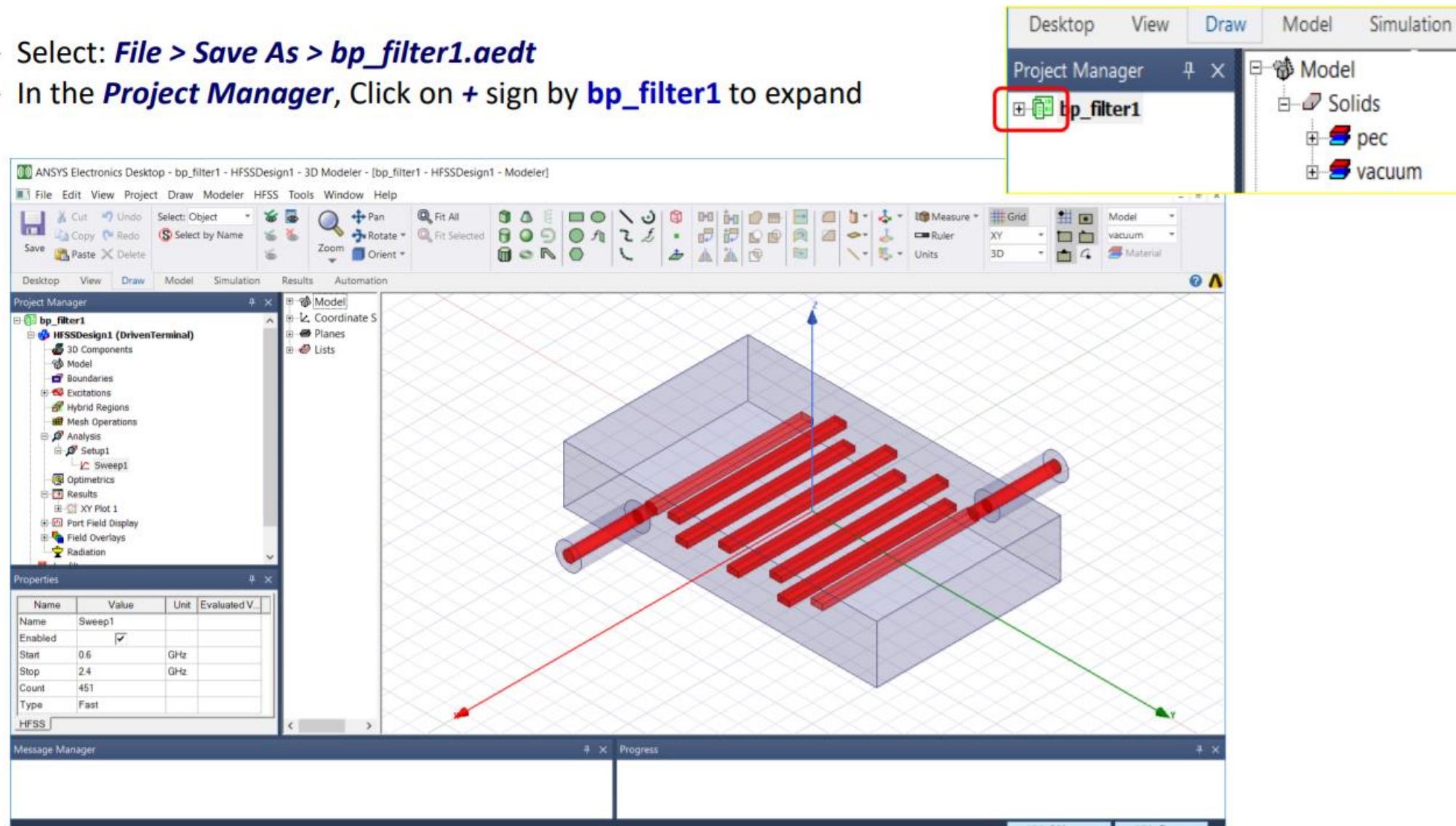
# Open Installed HFSS Band Pass Filter Example

- Select: **File > Open Examples > HFSS > RF Microwave**
- Click on **bp\_filter.aedt**
- Select **Open**



# Save As bp\_filter1.aedt to a Working Directory

- Select: **File > Save As > bp\_filter1.aedt**
- In the **Project Manager**, Click on + sign by **bp\_filter1** to expand



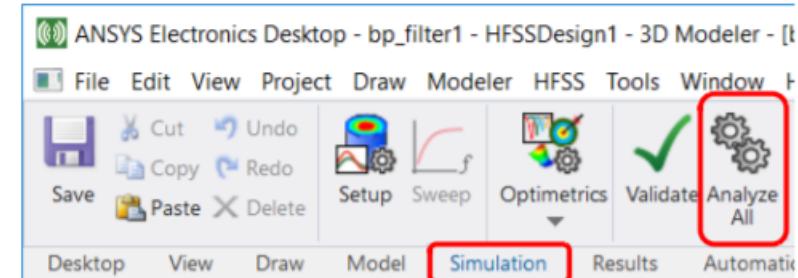
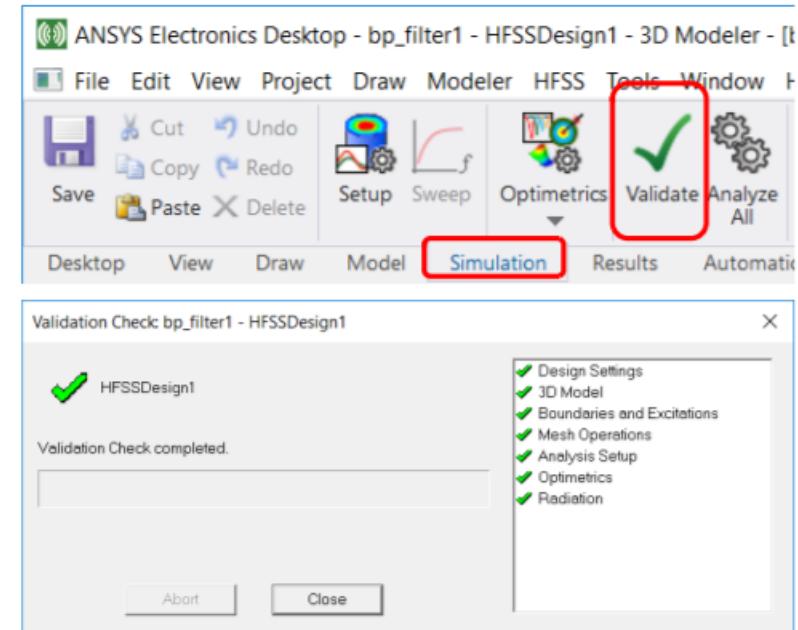
# Validate and Analyse HFSS Project bp\_filter1.aedt

- In the **Ribbon**, with the **Simulation** tab selected, click the **Validate** green check mark to validate the project.
- Click on **Analyze All** in the **Ribbon** to start the HFSS simulation.

The **Validation Check** and **Analyze All** operations are also available from the **HFSS** pull-down at the top of the graphical user interface (GUI).

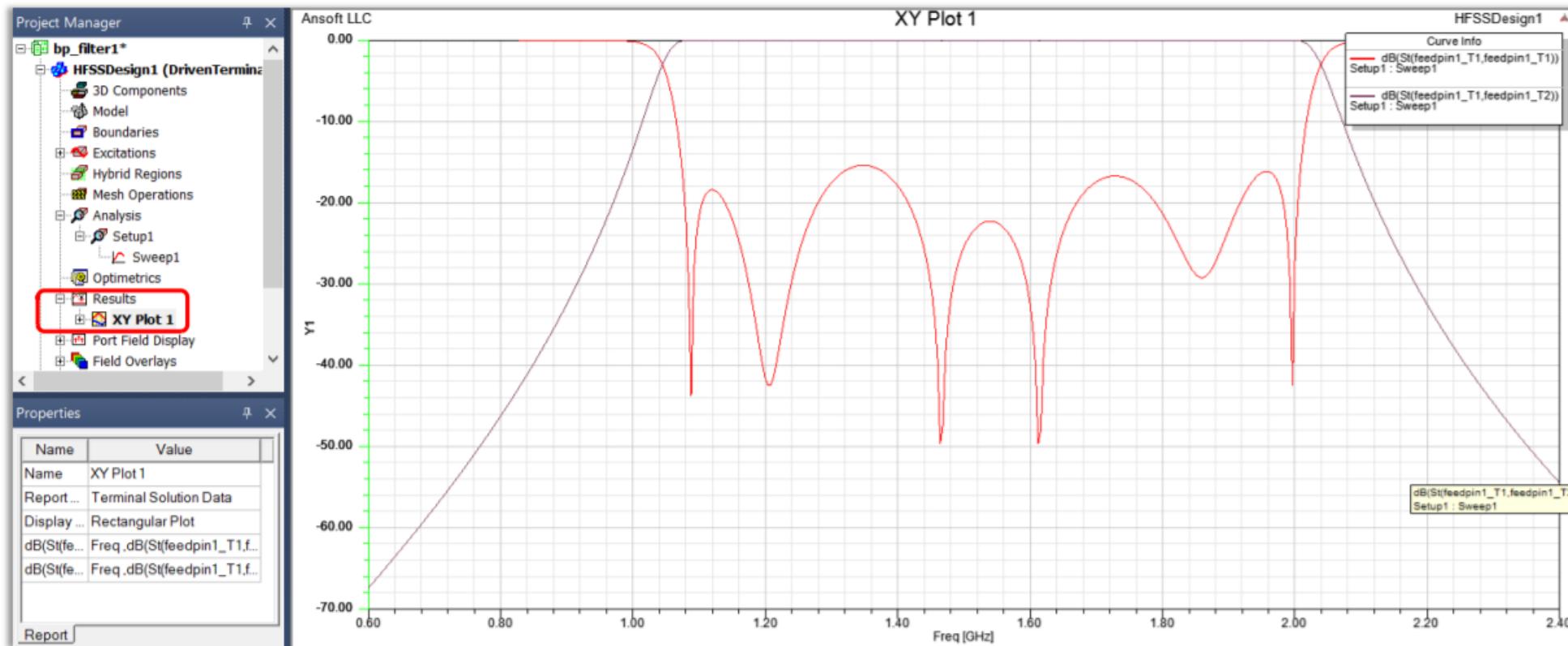
- Watch the **Progress Window** and the **Message Manager** to see when the simulation finishes.

*Keep all HFSS workshop simulation files; future workshops continue with these files.*



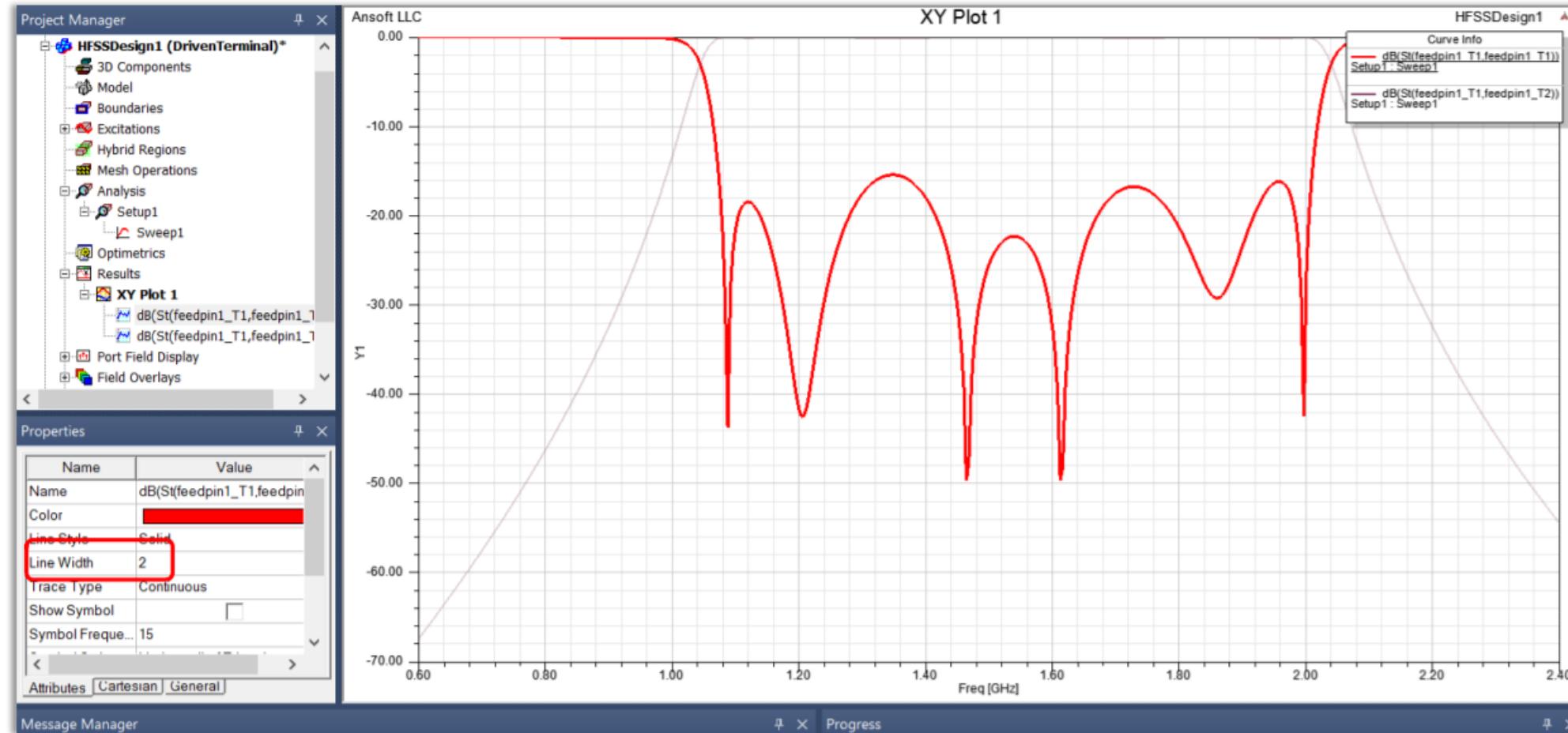
# View S-Parameter Results

- In the **Project Manager**, under **Results** (expand by clicking if necessary), double click on XY Plot 1 to make the S-parameter plot (Report) appear.
- Save **bp\_filter1.aedt**. *Keep all HFSS workshop simulation files; future workshops continue with these files.*



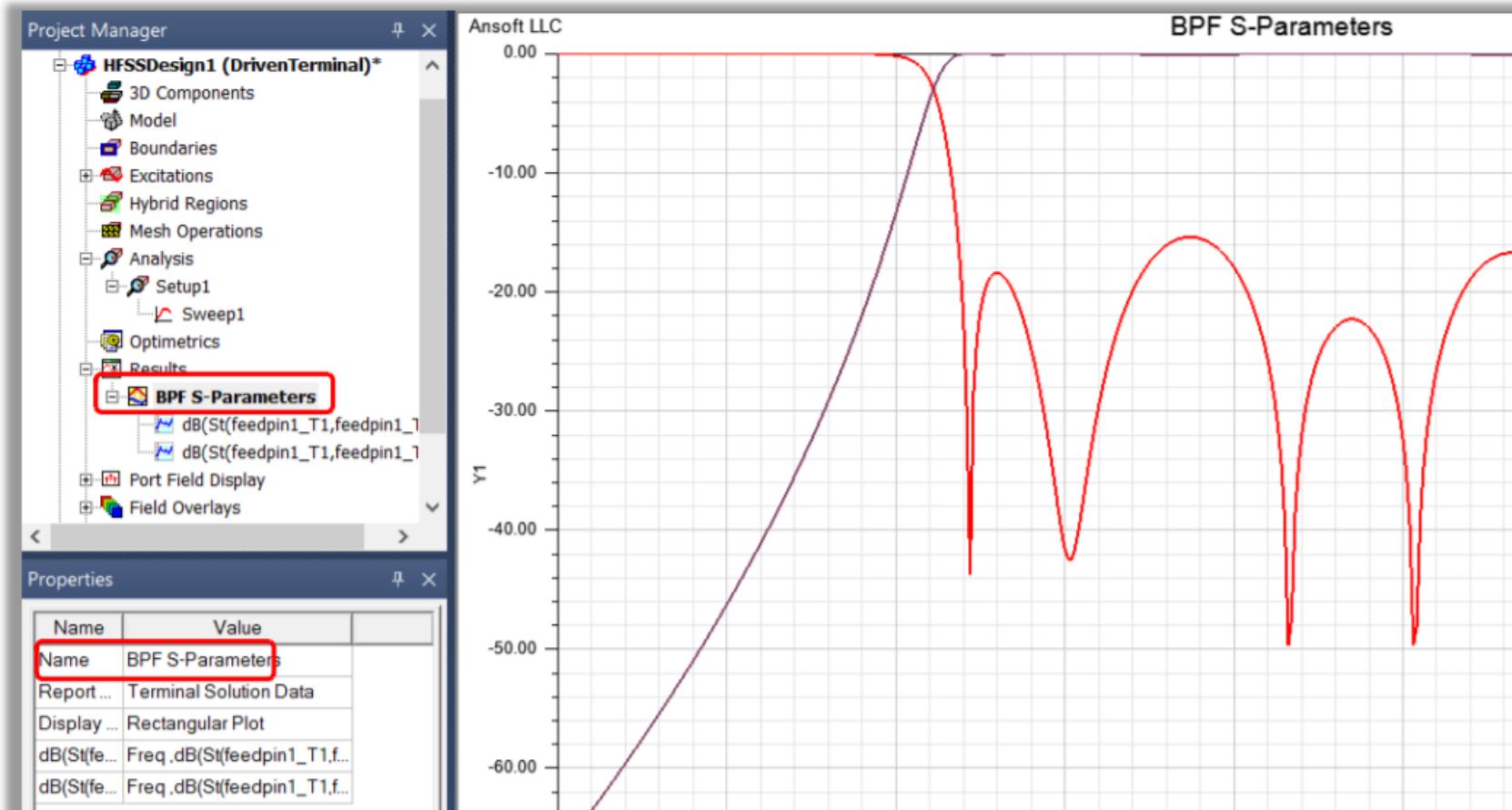
# Optional: Make S-Parameter Traces in Report Thicker

- In the **Report** (Plot), click either on the signal trace itself, or on the signal in the **Curve Info** legend. This makes the trace properties show in the lower left Properties window.
- Change the **Line Width** to a higher number, like from **1** to **2**. The higher the number, the thicker the trace.



# Optional: Change the Name of the Report (Plot)

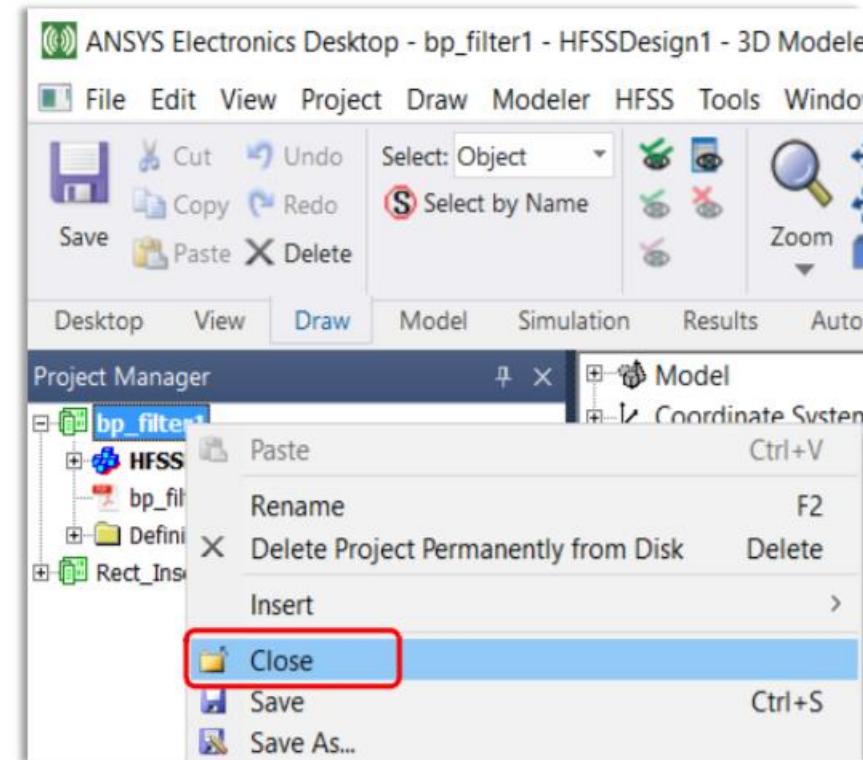
- In the **Project Manager**, under **Results** (expand by clicking if necessary), right click on **XY Plot 1** and select **Rename**, and change the name to **BPF S-Parameters**.
- or select **XY Plot 1** in the Project Manager and change the name in the Properties window below.



# Close Project without Exiting HFSS

- An HFSS project can be closed without closing the whole HFSS AEDT (ANSYS Electronic Desktop).
- In the **Project Manager**, select (left-click) the **bp\_filter1** project.
- Right-click on **bp\_filter1** and choose **Close**  
... OR Select **File > Close**

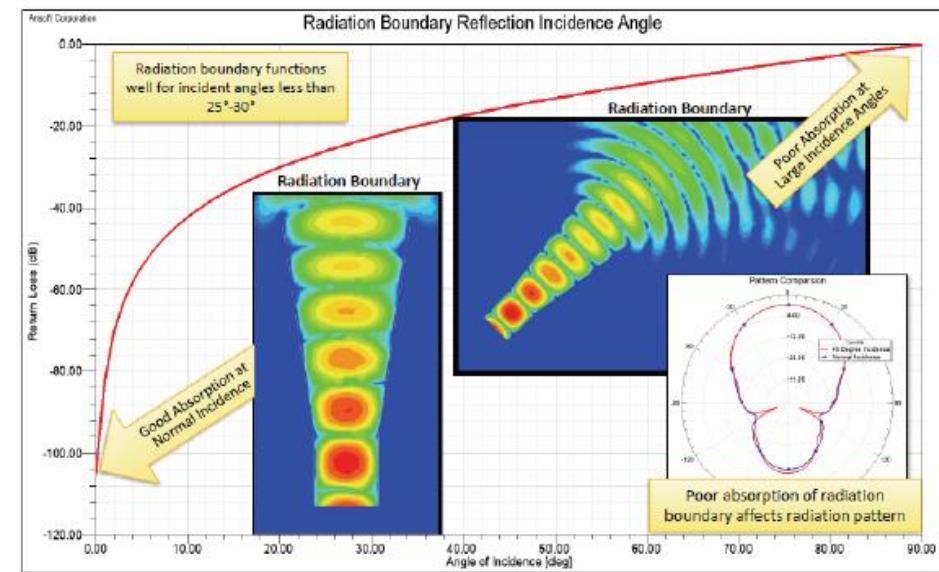
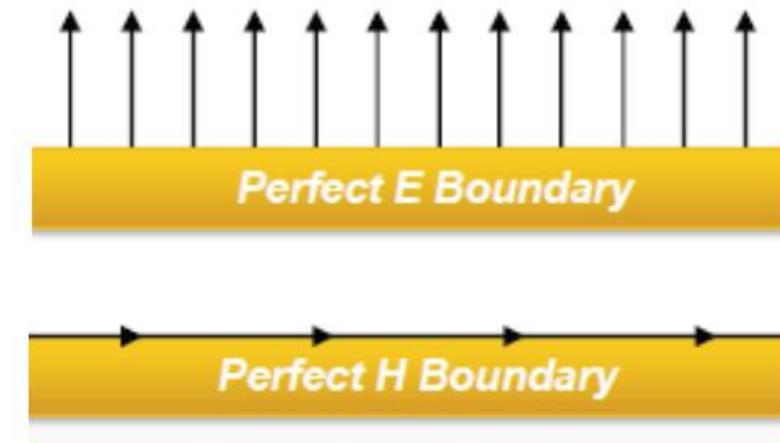
HFSS can have multiple projects open at once. The project needs to be selected in order to close it.



# Boundaries and Simulation Space

# Boundaries and Simulation Space

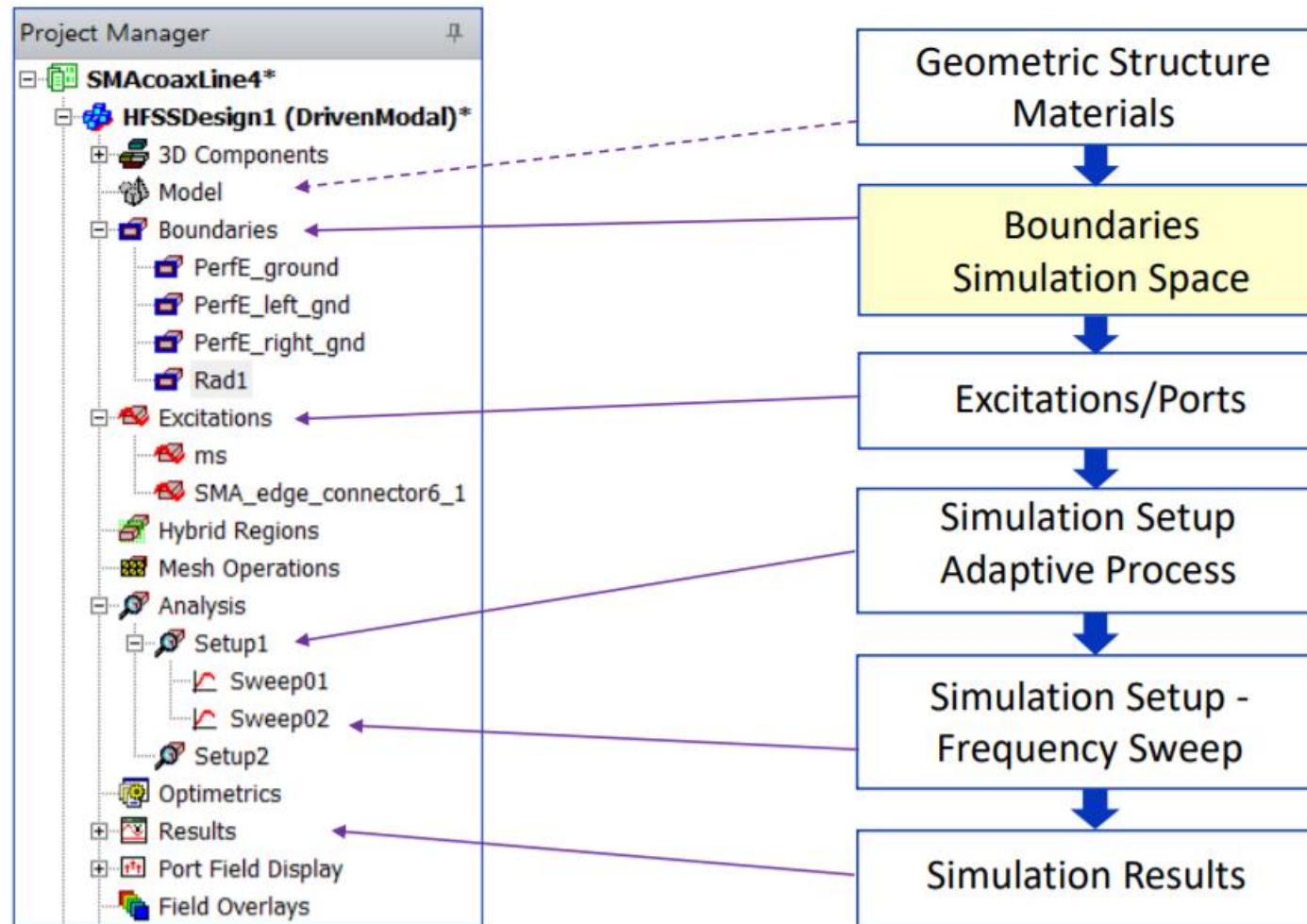
- Used to simplify geometry or make meshing more efficient
- The purpose of using boundary conditions in HFSS is to define the behaviour of the electromagnetic field on the object interfaces and at the edges of a problem region. Defining boundary conditions reduces the electromagnetic or geometric complexity of the model.
- A closed model represents a structure or a solution volume where no energy escapes except through an applied port.



# Boundaries and Simulation Space

- Boundaries Define HFSS Finite Element Method (FEM) Computational Volume
  - Boundaries define the simulation space (also called computational volume or solution space).
  - PEC boundaries can serve as the outer surface of the geometry of a closed model.
  - Air around a structure can be meshed in the computational volume (simulation space).
  - **Radiation** boundaries allow energy out of the solution space.
- Creating Boundaries and Simulation Space in HFSS FEM
  - **Create Open Region** makes a geometric region and assigns a boundary condition.
  - **Draw > Region** creates a **Region** in the model with padding.
  - **PML** (Perfectly Matched Layer) has a setup wizard.
  - Assigning boundary conditions to a radiating surface bounding box
- Types of Boundaries in HFSS
  - Boundary types: PEC, finite conductivity, **Radiation**, **PML**, and **FE-BI** are some
  - PEC - perfect electrical conductor
- Boundaries versus Materials
  - PEC boundaries can be used in place of ideal conductor material on a 3D solid
  - 2D sheets with boundary assignments can be inside of an HFSS simulation space

# Boundaries and Simulation Space

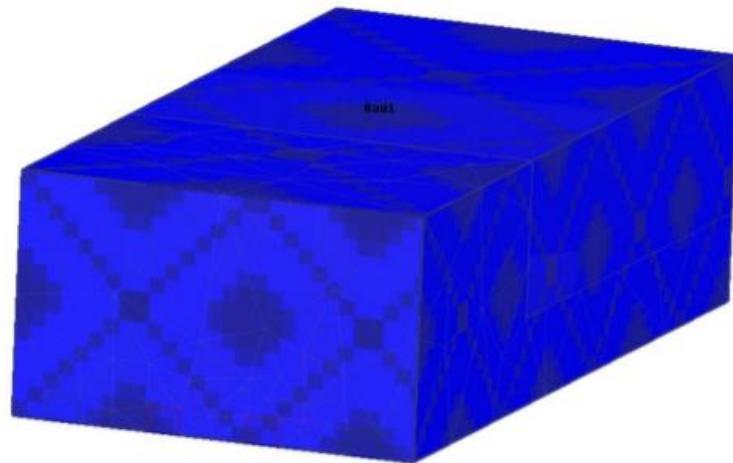
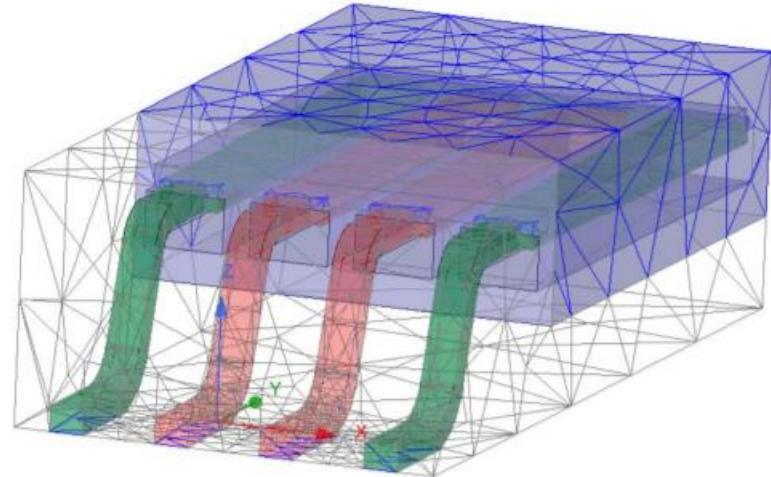


This presentation focuses on **Boundaries** in HFSS.

The HFSS Boundaries define what gets meshed and included in the FEM simulation.

Boundaries relate to the HFSS model geometry and structure. Sometimes boundaries extend beyond the geometry and sometimes boundaries are defined on the outside of the model geometry.

# Boundaries and Simulation Space



- HFSS finite element method (FEM) first discretizes a simulation problem, creating a mesh.
  - The mesh captures the electromagnetic character of the structure.
  - The mesh is what gets simulated.
  - A meshed connector is shown to the left.
- 
- The mesh gets surrounded and enclosed by a boundary.
  - The boundary defines the extents and boundary condition of the meshed simulation space.
  - Boundaries and their properties provide the boundary conditions to the mathematics of the finite element method simulation.
  - Boundaries and their properties are part of the accurate modeling of a given problem to be simulated.

# The Purpose of Boundaries in HFSS

The purpose of using boundary conditions in HFSS is to define the behavior of the electromagnetic field on the object interfaces and at the edges of a problem region. Defining boundary conditions reduces the electromagnetic or geometric complexity of the model.

A closed model represents a structure or a solution volume where no energy escapes except through an applied port. For an Eigenmode simulation, this closed model represents a cavity resonator and for a driven modal or terminal solution, the model can be a waveguide or some other fully enclosed structure.

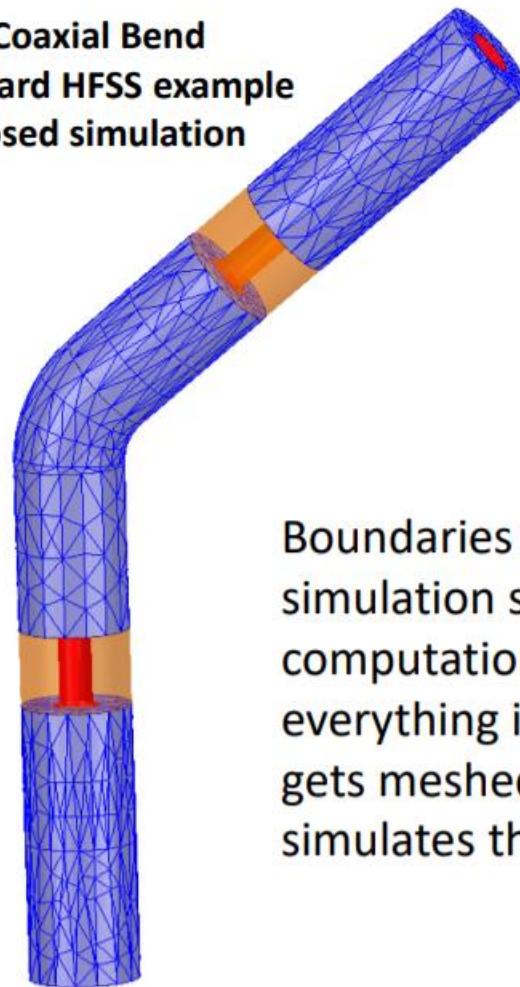
An electromagnetically open model allows energy to emanate or radiate away. Common examples include an antenna, a printed circuit board, or any structure that is not enclosed within a closed cavity.

By default HFSS treats any given model as closed since all outer surfaces of the solution space are covered with a perfect electric conducting boundary. In order to create an open model, you must specify a boundary on the outer surfaces that overwrites the default perfect electric conducting boundary.

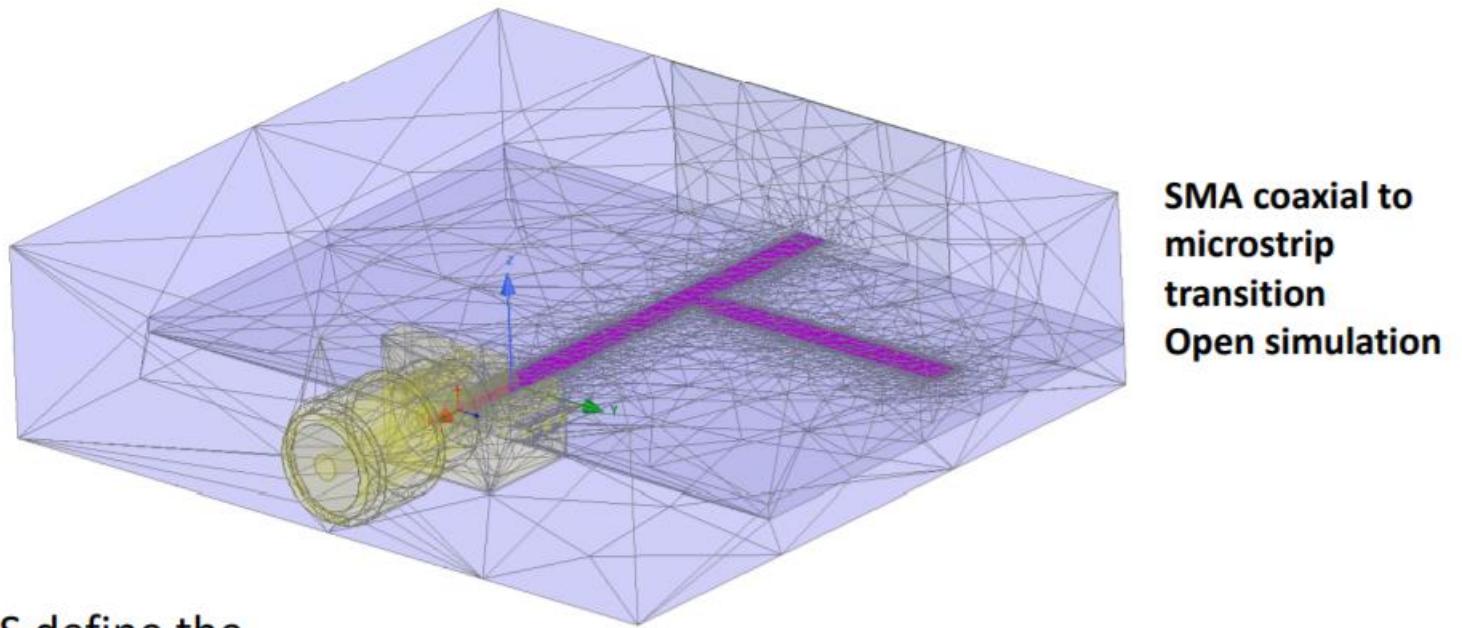
Boundary conditions are assigned on 2D sheet objects and surfaces of 3D objects.

# HFSS FEM Boundaries Define the Simulation Space

Coaxial Bend  
Standard HFSS example  
Closed simulation



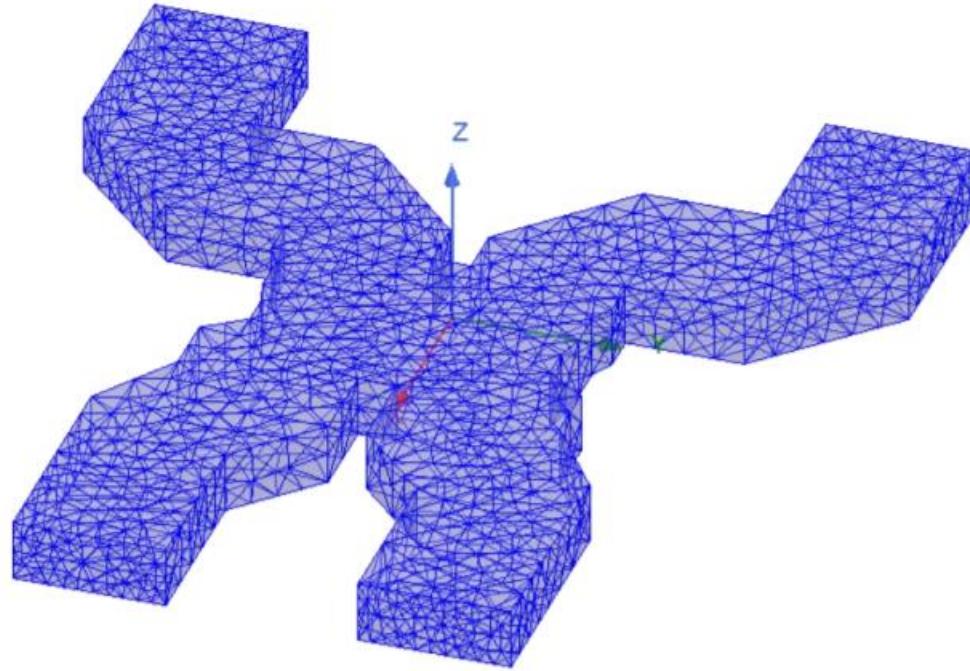
Boundaries in HFSS define the simulation space, also called the computational volume, because everything inside the boundaries gets meshed and simulated. HFSS simulates the mesh.



The meshed region may be larger than the physical boundaries of the structure – or not. The coaxial connector boundary *is* the outer conductor. By contrast, the microstrip can have a boundary above and below the microstrip structure. Air above and below this microstrip gets meshed and simulated.

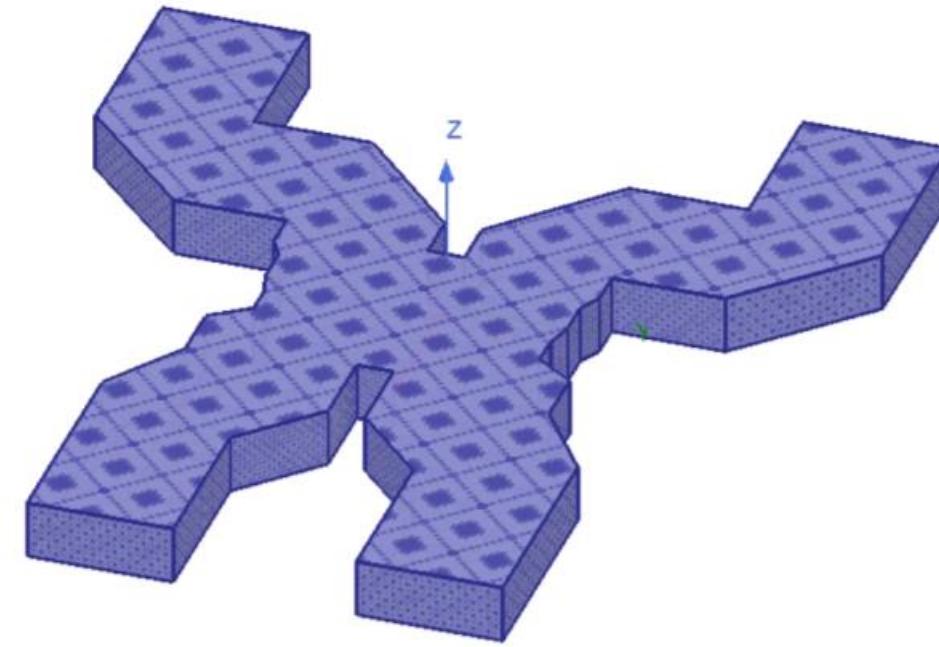
# HFSS FEM Boundary Around the Simulation Space – Closed Model

The computational volume gets meshed.



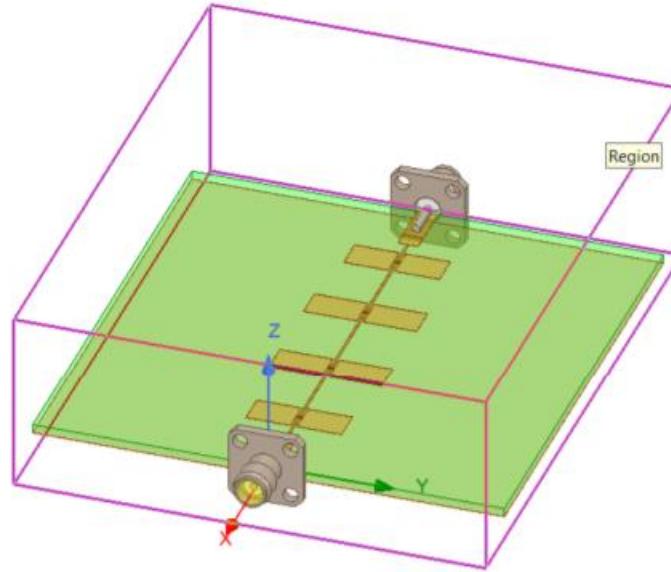
The entire volume, inside of this four-port waveguide combiner, gets meshed and simulated with finite element method (FEM).

A boundary surrounds the computational volume.

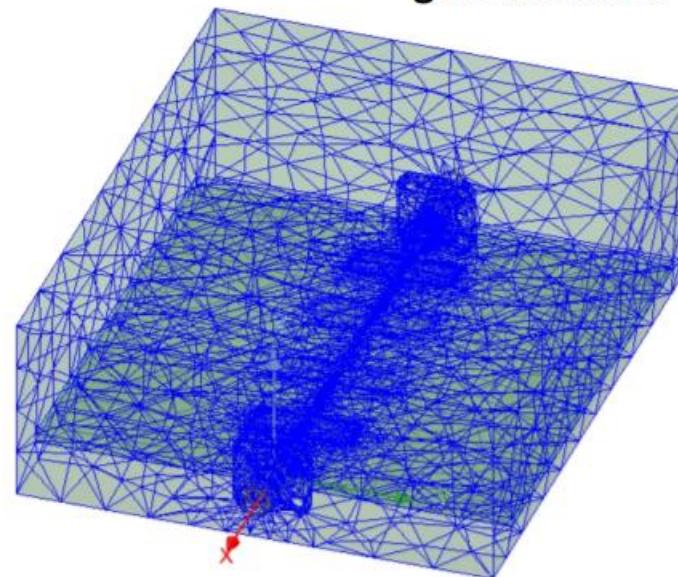


The outside faces of this waveguide combiner have a finite conductivity boundary that appears in the **Project Manager** under **Boundaries**.

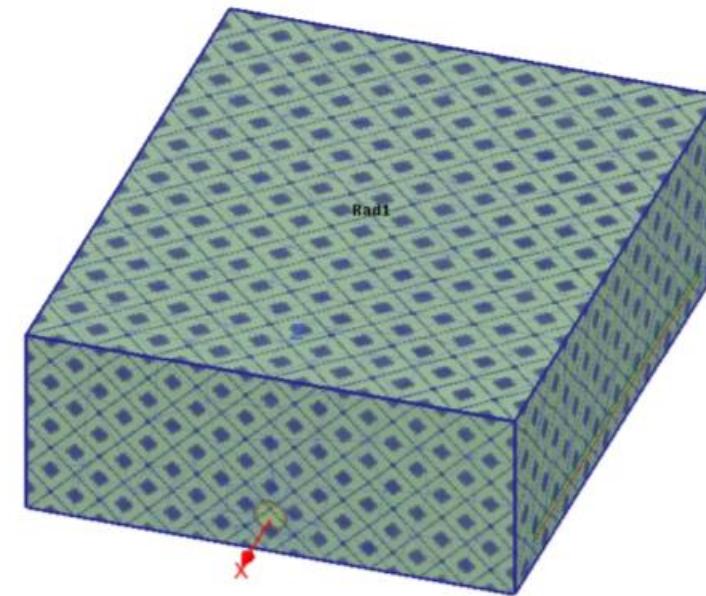
# HFSS FEM Boundaries Around the Simulation Space – Open Model



The structure is a 1.25 GHz low pass filter with SMA connectors. The **magenta purple** box is a **Region** created in the simulation specifically to define the boundary around the simulation space.



The computational volume gets meshed.



A boundary surrounds the computational volume.

# Boundaries and Simulation Space

## Anisotropic Impedance

Anisotropic impedance boundary condition represents a sheet with different impedance values from different tangential directions taking the polarization of the incident field into account.

## Finite Conductivity

Finite Conductivity boundary condition allows creation of single layer conductors. Finite conductivity boundaries represent imperfect conductors and is valid only if the conductor being modeled is a good conductor.

## Impedance

An impedance boundary condition represents a resistive surface; it allows creation of ohm per square material layers.

## Layered Impedance

Layered impedance boundary is used to model multi-layered conductors and thin dielectrics.

## Lumped RLC

Lumped RLC allows creation of ideal circuit components. It represents any combination of lumped resistor, inductor, and/or capacitor in parallel on a surface.

## Master

Master represents a surface on which the E-field at each point is matched to another surface (the slave boundary) within a phase difference. It's used with slave boundary to model infinitely large repeating array structures.

# Boundaries and Simulation Space

## Perfect Electric Conductor (PEC)

PEC is the default HFSS boundary representing a perfectly conducting surface that fully encloses the solution space and creates a closed model.

## Perfect H

Perfect H represents a surface where the tangential component of H is zero.

## Radiation

A radiation boundary is used to create an open model. It represents an open boundary by way of an absorbing boundary condition ABC that absorbs outgoing waves.

## Slave

Slave represents a surface on which the E-field at each point has been forced to match the Efield of another surface (the master boundary) within a phase difference. It's used with master boundary to model large infinitely repeating array structures.

## Symmetry

Symmetry boundary represents a perfect E or perfect H plane of symmetry.

## Perfectly Matched layer (PML)

A PML is used to create an open boundary condition using several layers of specialized materials for absorbing outgoing waves. PML boundary conditions are preferred for antenna simulations.

# Mesh and Sweep

# Mesh & Sweep

## Mesh

Meshing is really hard because it's intrinsically tied to CAD, and CAD has limited tolerance. That, in turn, limits the design engineer's ability to mesh a large geometric scale. So, while working on system-level designs such as antenna-on-platform, IC packages or any other complex electromagnetic (EM) system, engineers have to deal with a mix of different types of CADs. And they have to be extra careful about how they prepare the CAD and how they align different parts.

Ansys, which has developed different meshing technologies over the years, has introduced a solution that drives rapid and fully-coupled simulation of complex EM systems. HFSS Mesh Fusion combines ICs, packaging, PCBs, connectors, and antennas in a single high frequency structure simulator (HFSS) analysis tool to predict any EM interactions.

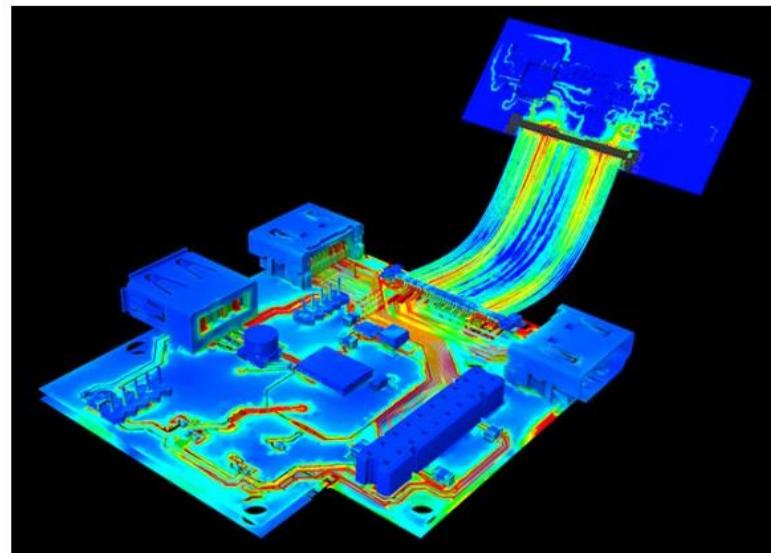


Figure 1 Mesh Fusion can perform signal integrity simulation of a multi-PCB system, including connectors and flex cable. Source: Ansys

# Mesh & Sweep

## Mesh

Tetrahedral mesh automatically generated and refined below userdefined electrical length

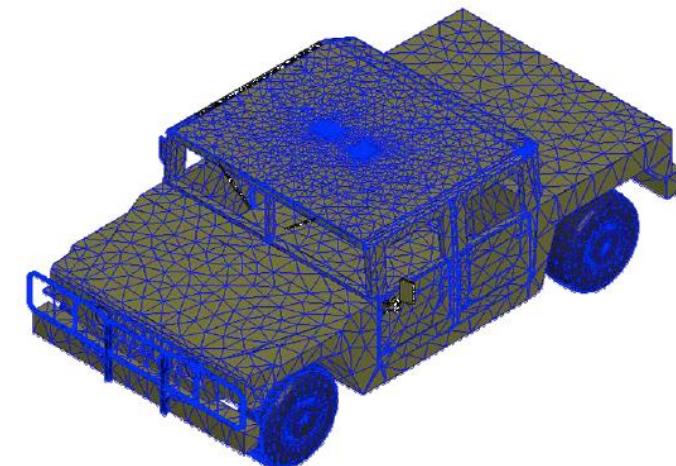
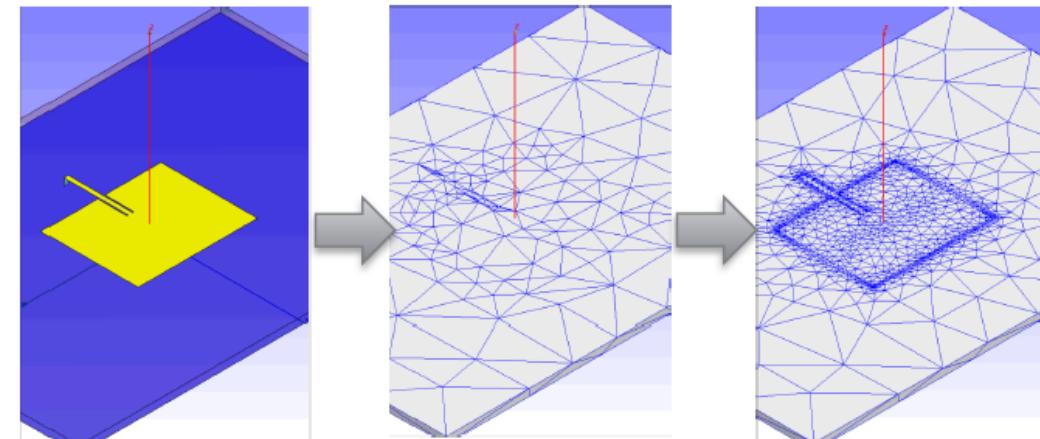
- Tetrahedral element shape conforms to arbitrary geometries

Iterative algorithm solves fields and refines mesh until user-defined convergence threshold value is reached

- Driven modal: S-parameter convergence
- Eigenmode: Frequency convergence

Produces graded mesh with fine discretization only where needed to accurately represent field behavior

- Efficient use of computational resources
- Tunes mesh to capture EM performance



# Mesh & Sweep

## Mesh Control

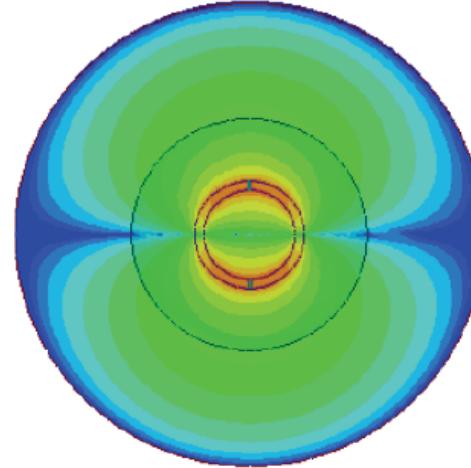
Mesh seeding allows user to directly influence initial mesh

- Reduce number of adaptive passes
- Focus mesh elements in critical areas
- Not required for accurate results
- Can create better-looking field plots
- Seeding radiation boundary can
- improve far-field data

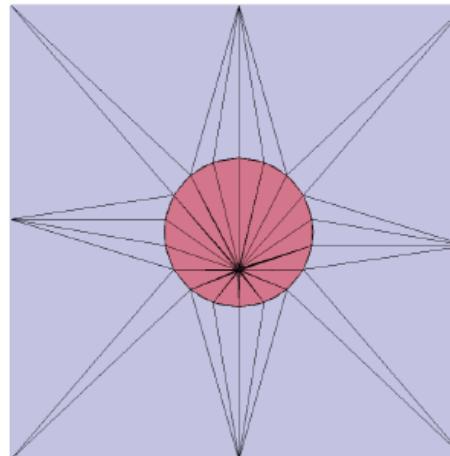
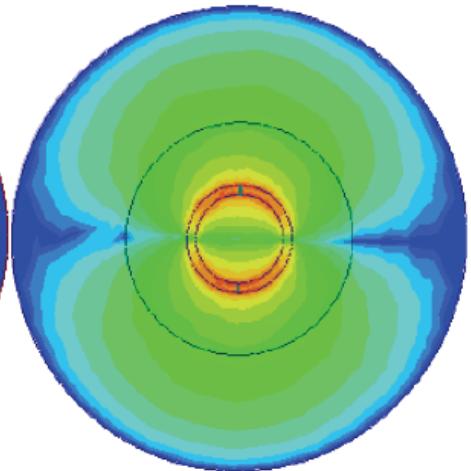
## Lambda refinement

- Ensures that initial mesh is refined to fraction of electrical wavelength
- Electrical size depends on solver basis order

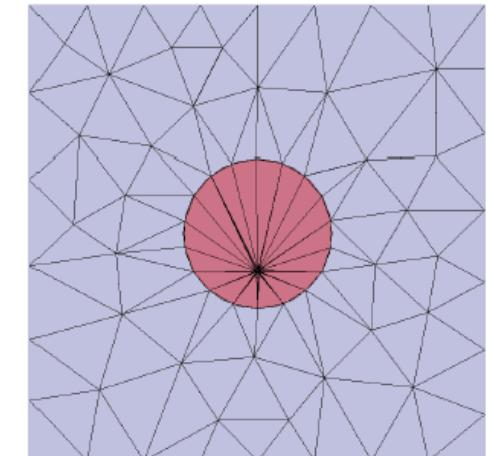
Field plot with seeding



Field plot without seeding



Initial geometric mesh

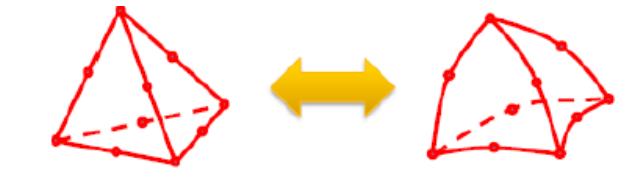


Electrical mesh after  
lambda refinement

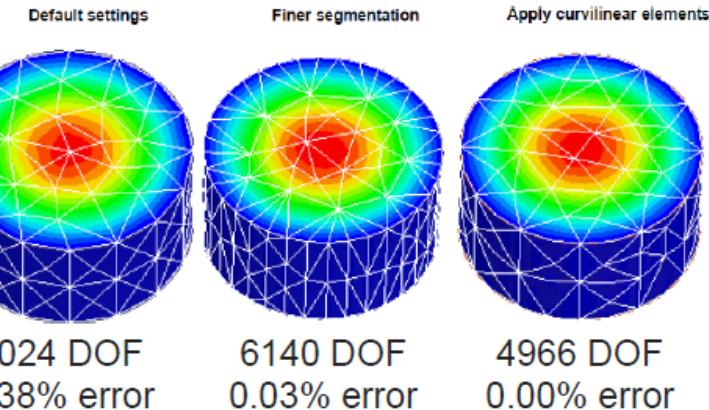
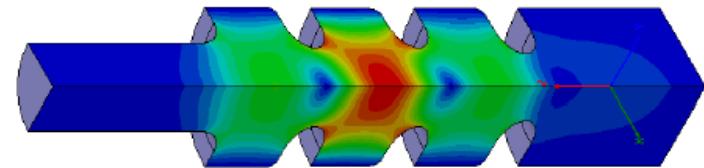
# Mesh & Sweep

## ***Curvilinear Mesh Elements***

- Global mesh approximation setting for all true surfaces in model
- Higher order (curvilinear) elements used to represent the geometry
  - Pulls midpoints of tetrahedra surfaces to true surface
- Pillbox resonator with analytical  $f_R = 22.950$  GHz for TM<sub>010</sub> mode
  - Default setting: 23.269 GHz
  - Finer segmentation: 23.012 GHz
  - Curvilinear elements: 22.950 GHz



Rectilinear mesh element      Curvilinear mesh element

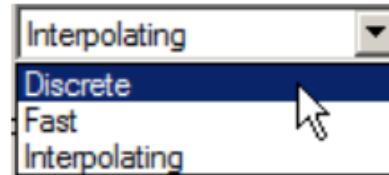


# Mesh & Sweep

## Frequency Sweeps

### Discrete sweep

- Solves adapted mesh at every frequency
- Matrix data and fields at every frequency



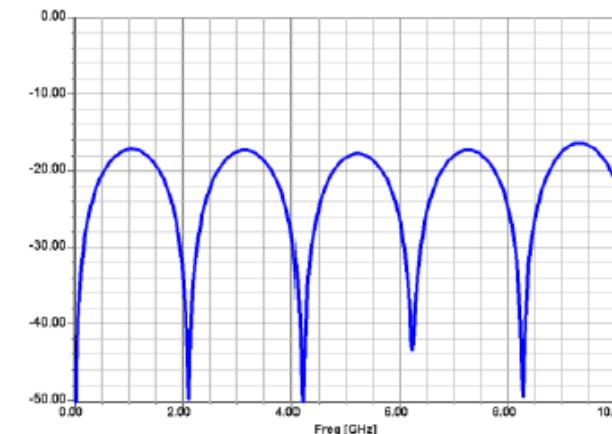
### Fast sweep

- Extrapolates rational polynomial function for electric field over specified range
- Usually valid over less than 10:1 BW
- Matrix data and fields at every frequency

$$S = \frac{\beta_q(s - z_q)(s - z_{q-1})...(s - z_1)}{\alpha_q(s - p_q)(s - p_{q-1})...(s - p_1)}$$

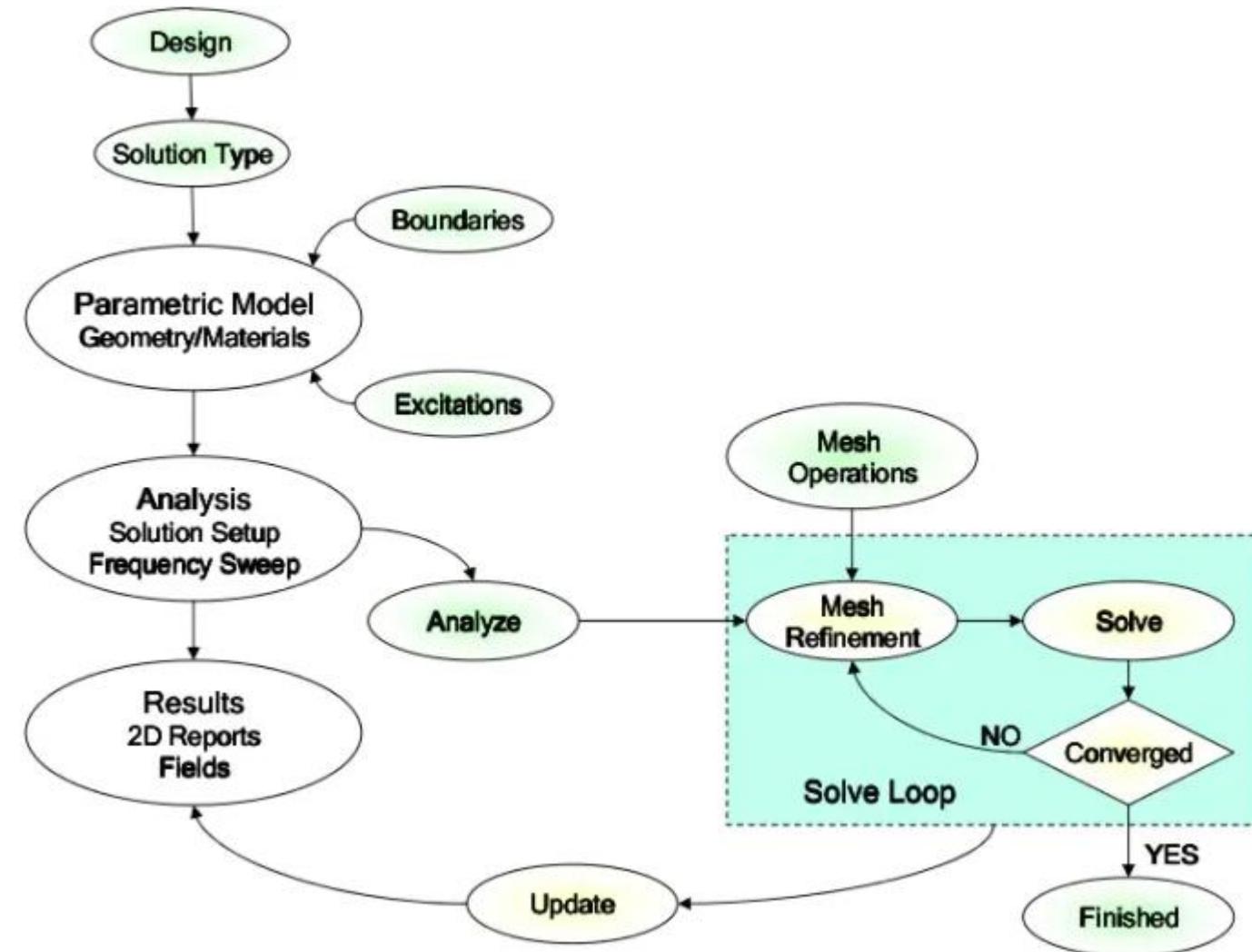
### Interpolating sweep

- Solves minimum number of frequencies to create polynomial fit for S-parameters
- Useful for very broadband S-parameters
- Matrix data at every frequency



# Mesh & Sweep

## HFSS Start to Finish



# Post-Processing, S-Parameters and Fields

# Post-Processing, S-Parameters & Fields

## ***Plotting S-parameter results***

- One of the most important outputs from HFSS is the S-parameter. Once a simulation has finished. S-parameters can be plotted at a single frequency or over a frequency sweep. HFSS generates S-parameters with matched loads. Matched load essentially means that the port is loaded by its characteristic port impedance. S-parameters of the matched loads can be renormalized by loading the ports with arbitrary impedance. When comparing HFSS wave port results to measured data, it is important to re-normalize the HFSS results to the loading impedances when the measurement was performed. If a given S-parameter is based on a lumped port, the S-parameters are normalized to the value of  $Z_0$  specified when the port was created.
- S-parameters are easily plotted using the HFSS results editor. However, depending on the port type, S-parameters are either generalized or normalized. The S-parameters for a wave port are, by default, generalized. Generalized S-parameters do not have a normalization constant but rather are normalized to the characteristic impedance of the corresponding wave port. As a result, when comparing HFSS wave port results to measured data, it is important to re-normalize the HFSS results to the normalization constant used when the measurement was performed. If a given S-parameter is based on a lumped port, the S-parameters are normalized to the value of  $Z_0$  specified when the port was created. When comparing measured and simulated data, it is again necessary to re-normalize the HFSS data if the user-specified  $Z_0$  is not equivalent to the normalization constant used in the measurement

# Post-Processing, S-Parameters & Fields

## ***Plotting Field Results***

HFSS can produce a plot of any standard electromagnetic quantity, such as the electric field, magnetic field, Poynting vector, or current density. Generally, fields are displayed on 2D objects, faces of 3D objects, or on coordinate system planes. Plots can be scalar quantity plots or vector quantity plots. Specific quantities based on mathematical operations on the basic field quantities can also be plotted by use of the fields calculator.

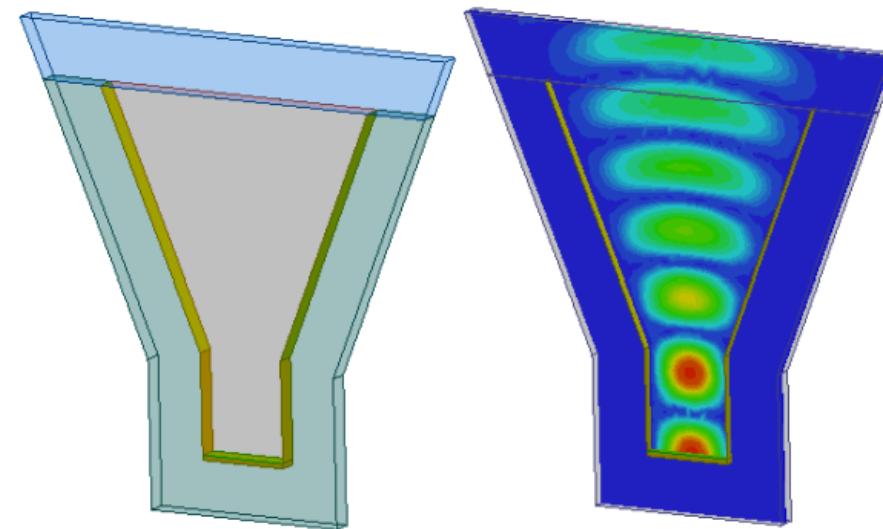


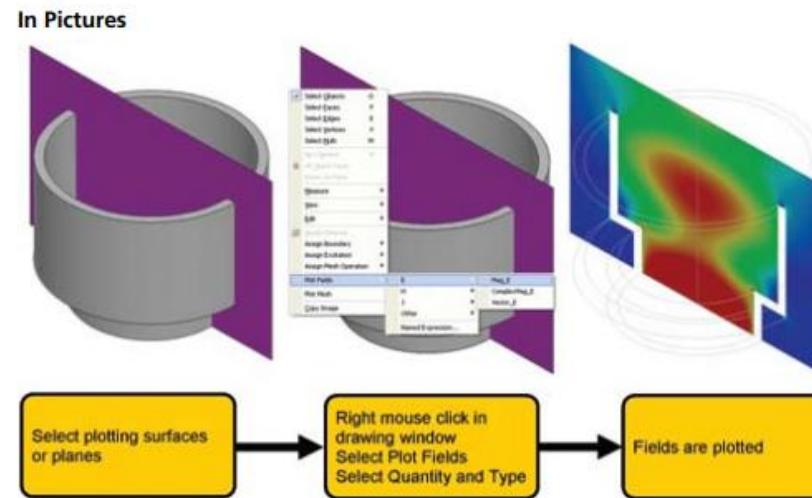
Figure 6-6 Example of an E-field plot on the XZ plane of a pillbox antenna

# Post-Processing, S-Parameters & Fields

## ***Plotting Field Results***

Field plots, or, more specifically, field overlays, are representations of the basic or derived field quantities on specific surfaces of objects or within an object for the current design variation.

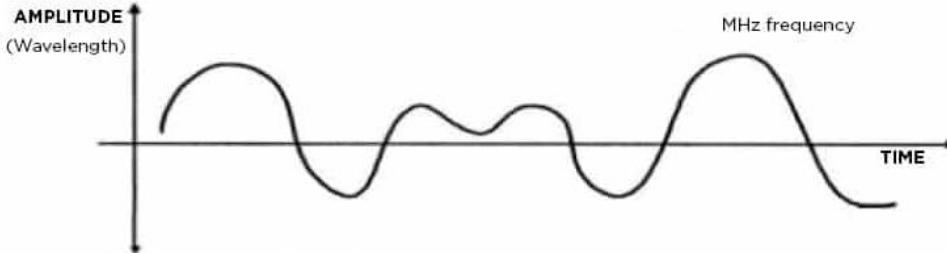
A field overlay's appearance can be changed by modifying the settings in the Plot attributes dialog. This dialog modifies a plot folder and all field overlays contained within that folder will use the same attributes. Field overlays can also be created by the use of the field calculator. The field calculator allows a user to create mathematical operations on the basic field quantities. These results can be plotted or exported depending on the needs of the user.



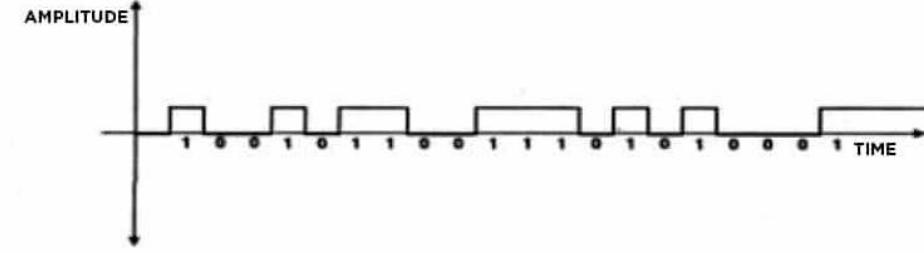
# Fundamentals of Signal Integrity

# Fundamentals of Signal Integrity

## ANALOG SIGNAL

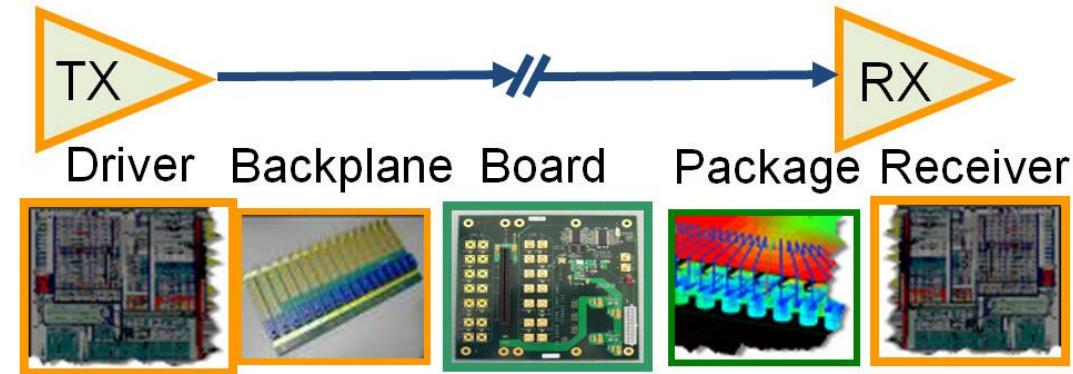


## DIGITAL SIGNAL



- In a system, signals travel through various kinds of interconnections (e.g., from chip to package, package to board trace, and trace to high-speed connectors).
- Signal integrity problems arise from the physical nature of interconnecting wires. Unlike a connection line drawn on a schematic, a real wire has resistance, capacitance to ground and other wires, and inductance. At higher frequencies, capacitance and inductance can cause the wire to act as a transmission line, and antenna effects can result in crosstalk and EMI.
- Signal integrity problems cause systems to fail or work only intermittently, producing "bad" data. As such, signal integrity issues are particularly important to find early in a design cycle because intermittent failures are very difficult to debug on prototypes.

# Fundamentals of Signal Integrity



***What could possibly go wrong?***

If care is not taken to ensure a high level of signal integrity when designing the PCB layout, then manufacturing problems can occur in that:

1. It will cause the design to work incorrectly in some cases, but not all cases.
2. The design might actually fail completely.
3. The design might operate slower than expected (and required).

# Fundamentals of Signal Integrity

Every Design Detail is Important. At clock frequencies in the hundreds of megahertz and above, every design detail is important to minimize signal integrity problems:

1. Clock distribution
2. Signal path design
3. Stubs
4. Noise margin
5. Impedances and loading
6. Transmission line effects
7. Signal path return currents
8. Termination
9. Decoupling
10. Power distribution

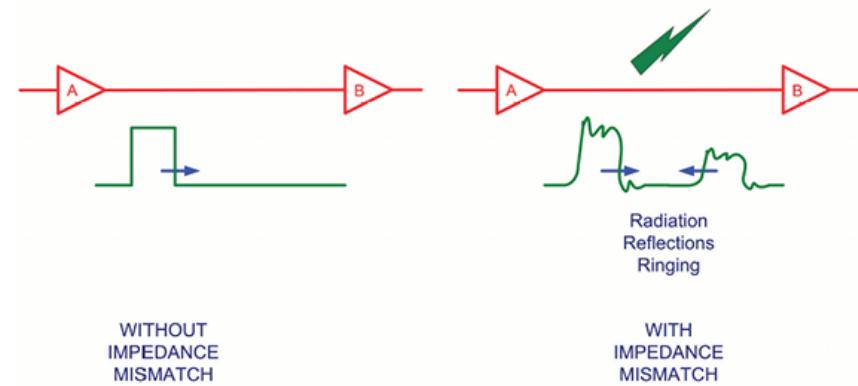
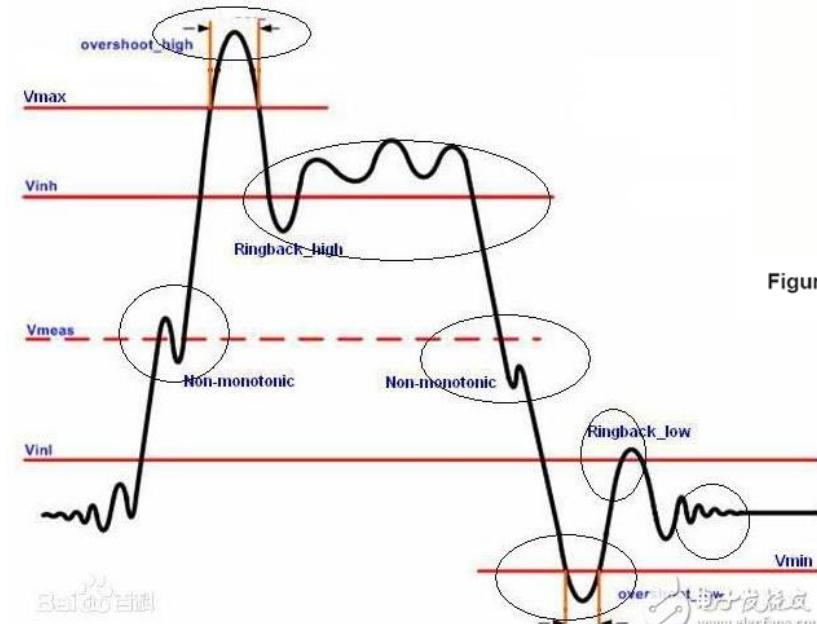


Figure 1: Transmission of a Digital Signal with and without Impedance Mismatch

# Fundamentals of Signal Integrity

There are six main areas of circuit design and layout that must be taken into consideration to ensure that the signal integrity of a board or circuit design are maintained:

1. Transmission line effects
2. Impedance matching
3. Simultaneous switching effects
4. Crosstalk
5. Electromagnetic Interference (EMI)
6. Ground Bounce

## Training Enquiry



[admin@psdc.org.my](mailto:admin@psdc.org.my)



[www.psdco.org.my](http://www.psdco.org.my)



04-643 7929

## Solution Enquiry

**ORIONPLEX**

**ORIONTRAIN**



[gary.lee@orionplex.com.my](mailto:gary.lee@orionplex.com.my)



[www.orionplex.com.my](http://www.orionplex.com.my)



019-4106712



Gary Lee Engineering Solutions Penang



espenang.6712

*Thank you*