HFSS Microstrip Tutorial

Kisa Avrutina

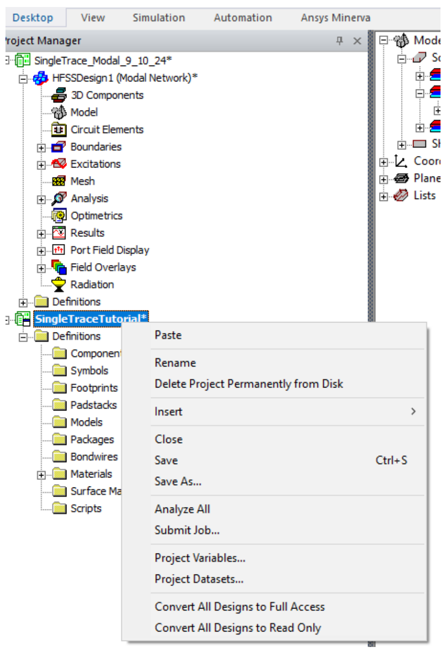
October 2024

Contents

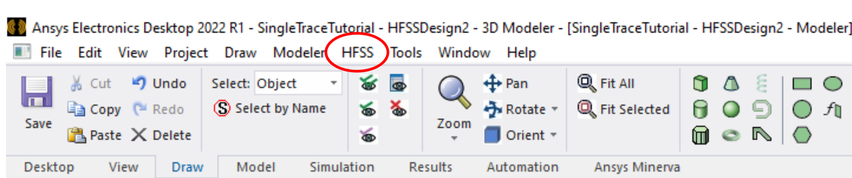
Setting Things Up

1. Open Ansys Electronics Display

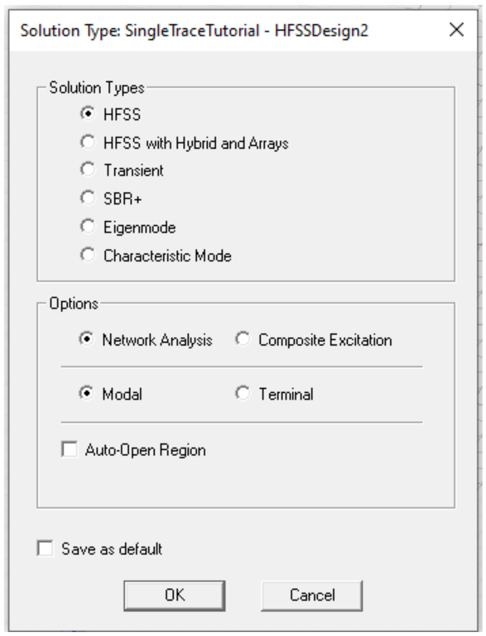
2. Create a new model file
    a. Hover over file in the top banner, and select New
    b. Right click on file name in Project Manager, select Insert, select Insert HFSS design



3. Check solution type
    a. On top banner, click on HFSS



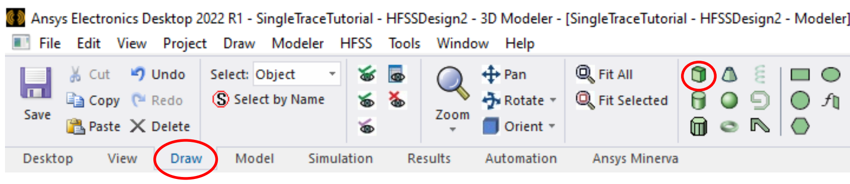
b. Select Solution Type
c. For wave port method, select Modal
d. Check “Save as default” to ensure new models will be of the modal solution type
e. Click Ok



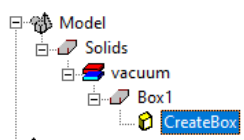
Building the Model

Ground plane:

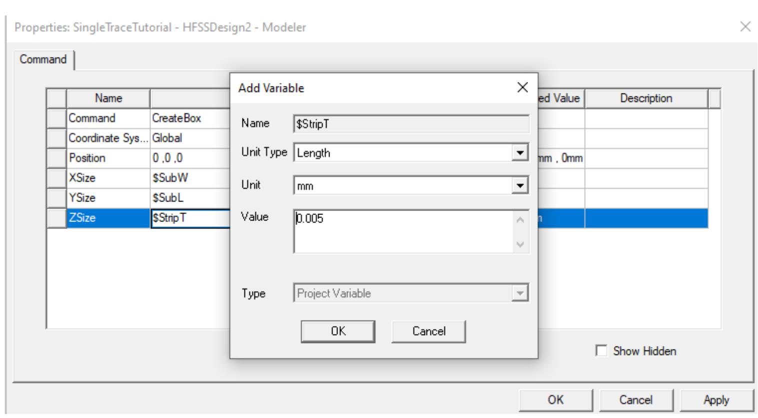
1. Select the Draw tab on the top banner
2. Select the rectangular prism



3. Construct the shape anywhere on the work plane
4. The properties window should pop up. Adjust Position, XSize, YSize, and ZSize.
   a. **To access the properties window, double click "CreateShape"** (where Shape is refers to whatever shape your object is—such as Box, Cylinder, etc). **This can be found by expanding the object's tree.**

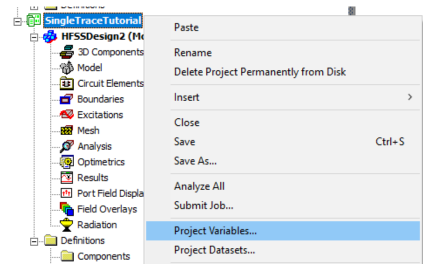


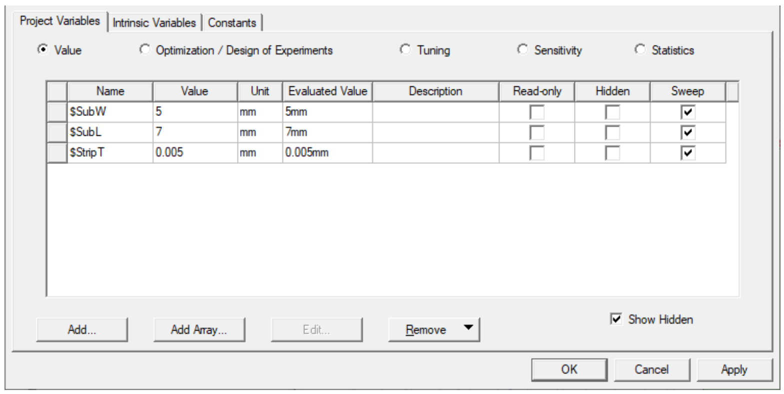
5. Create variables for all important properties
   a. Type "$" followed by your variable name into any of the size or position boxes. Click the enter key. Ansys will recognize any non-numerical value as a variable, and prompt you to define it.
   b. In the Add Variable window, set the variable's value. Make sure to set the units appropriately.
      i. $ ensures the variable is saved as a "Project Variable", making it easy to access later.



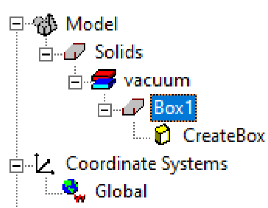
ii. **To access variables later:**

- Right click on the file name in Project Manager, and select Project Variables.
- The Project Variables window allows you to view and edit all project variables.

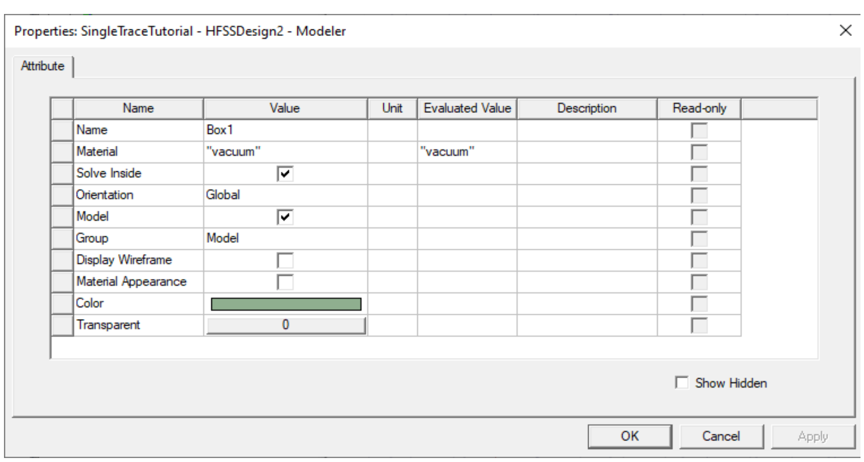




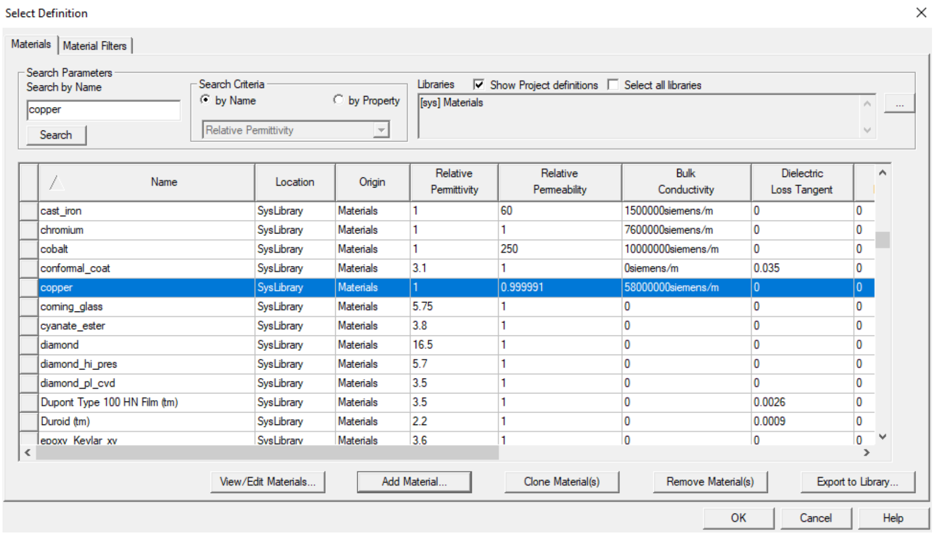
6. Assign materials
   a. Double click on the object's name in the model tree.



   b. The properties window will pop up. You can assign a name, appearance, and material. For ease of visibility later on, you may want to raise the transparency.



c. Expand the material drop-down. The material should be set to “vacuum” by default.
d. Select “Edit…”
e. You can select a material from the library, or add your own material and manually assign values to material properties. The ground plane’s material is typically copper.



f. Click Ok

Substrate:

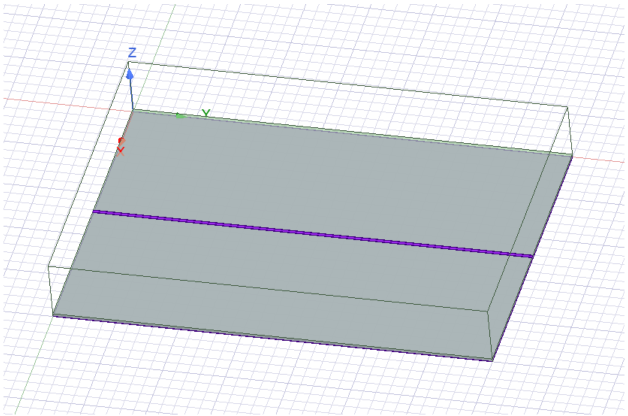
1. Construct the shape the same way you did for the ground plane. Use variables in the properties window to assign size and position.
2. Assign a material. This is often aluminum nitride, called “Al_N” in Ansys.

Microstrip:

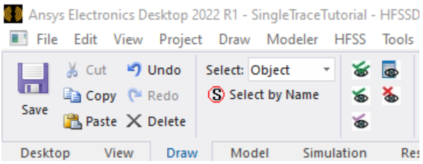
1. Construct the shape. Use variables to assign size and position. The trace’s thickness is typically the same as the ground plane.
2. Assign a material. This is typically copper.

Create an air boundary:

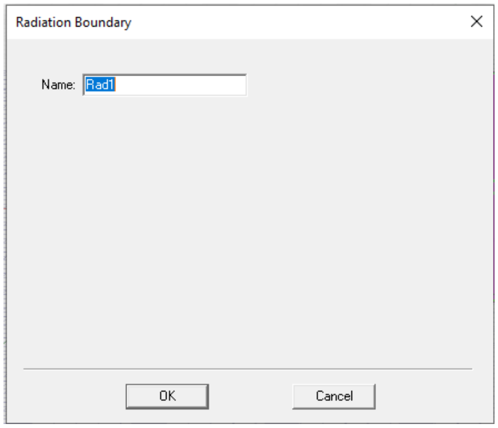
1. Create a box of the same x and y-dimensions of the ground-plane, but significantly bigger in the z-dimension (about 1-1.5 mm is usually good). This box should be placed on the same work plane as the ground plane, and overlap the ground plane and substrate exactly.



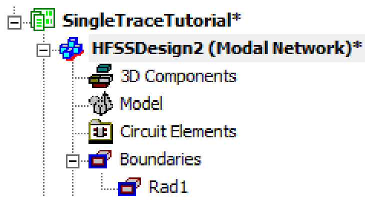
2. Assign the material of the box to be air.
3. Make sure Object is chosen in the Select menu found under the Draw tab on the top banner. You may want to change this later to select specific faces or edges.



4. Select your air box by clicking on it in the work plane.
5. Right click on the box, select Assign Boundary, and select "Radiation…"
6. The Radiation Boundary pop-up will prompt you to name your boundary. Choose an appropriate name and click OK.

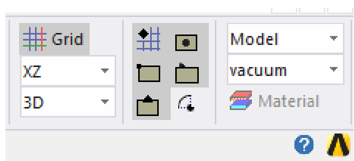


7. Check that your boundary appears under Boundaries in the design tree in Project Manager.

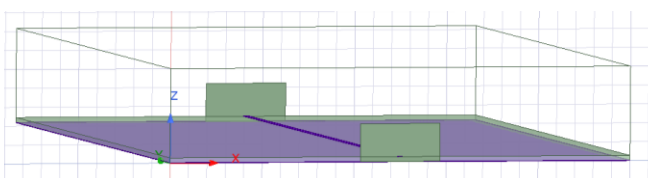


Create ports:

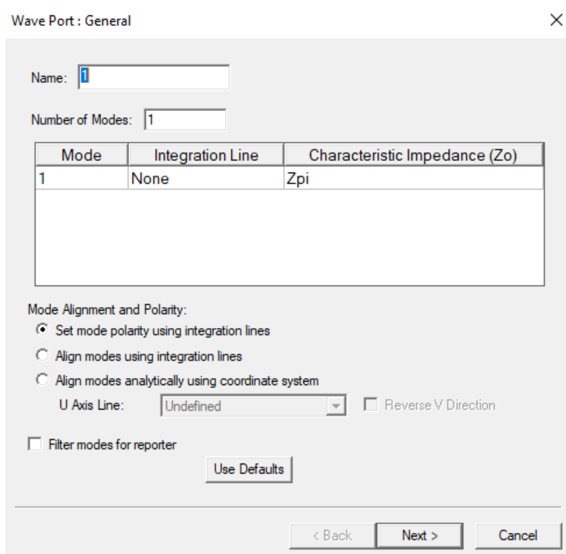
1. Creating the wave ports involves drawing rectangles at either end of the microstrip. 2D shapes are drawn along the work plane grid, which is oriented on the XY-plane by default. To draw a shape along a different plane, you will need to change the grid. This can be done under the Draw tab.
    a. You will most likely want the XZ grid.



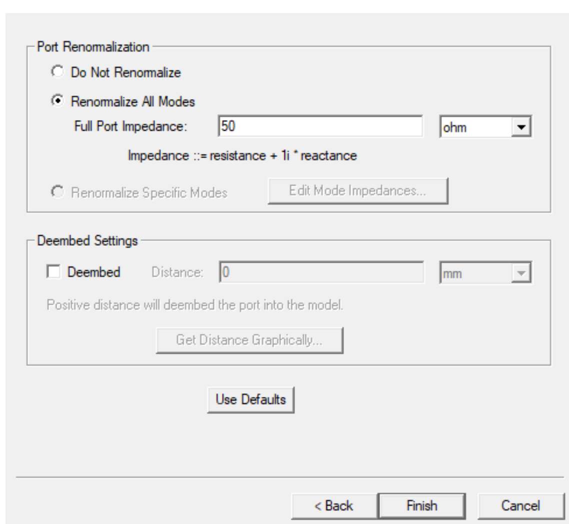
2. Draw a rectangle at one end of your microstrip, then assign appropriate position and dimensions.
    a. The wave port should be centered at the center of your microstrip. Its bottom end should be aligned with the top edge of the ground plane.
    b. <u>Ideal Dimensions:</u>
        i. The height should be >4h, where h is the thickness of your substrate.
        ii. The width should be ~5w, where w is the width of your microstrip.
        iii. You can go bigger than this! If your simulation results look strange, it may be worth increasing the size of your wave port.



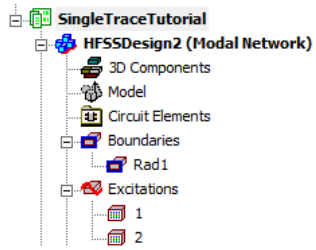
3. Assign excitations
    a. Select the created rectangle. Right click on the selection, click Assign Excitation, then Port, then Wave Port.
    b. Ansys allows you to set a number of modes and set an integration line for each mode. The default setting are fine for many basic applications.
        i. You can also change the characteristic impedance. The options are Zpi, Zpv, Zvi, and Zwave. These will generally give different results. Zpv and Zvi will only appear if an integration line is drawn such that a unique voltage can be computed for that mode. Zwave is for homogenous waveguides.
    c. For this model, I used 1 mode with Zpi characteristic impedance.



    d. Click Next. Renormalize all modes to port impedance of 50 Ohm. Click Finish.



e. You will be able to find the created rectangle under Sheets in the model tree. You will find the wave ports under Excitations in the design tree in Project Manager.

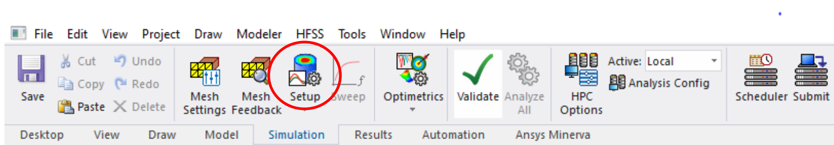


f. Repeat this process for the second port.

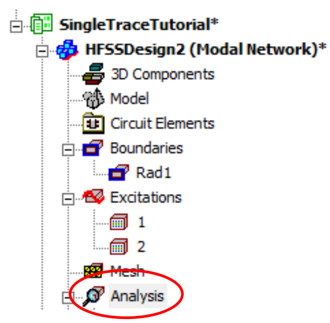
Running Simulations

Simulation Setup:

1. Go to the Simulation tab on the top banner. Click Setup and select Advanced.

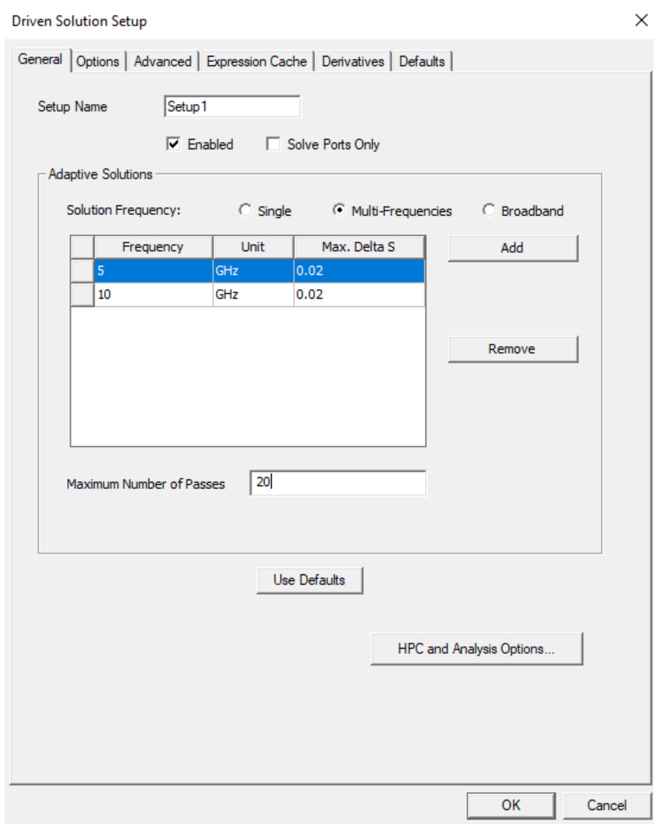


2. Alternatively, right click on Analysis under the design tree in Project Manager. Hover over Add Solution Setup and select Advanced.



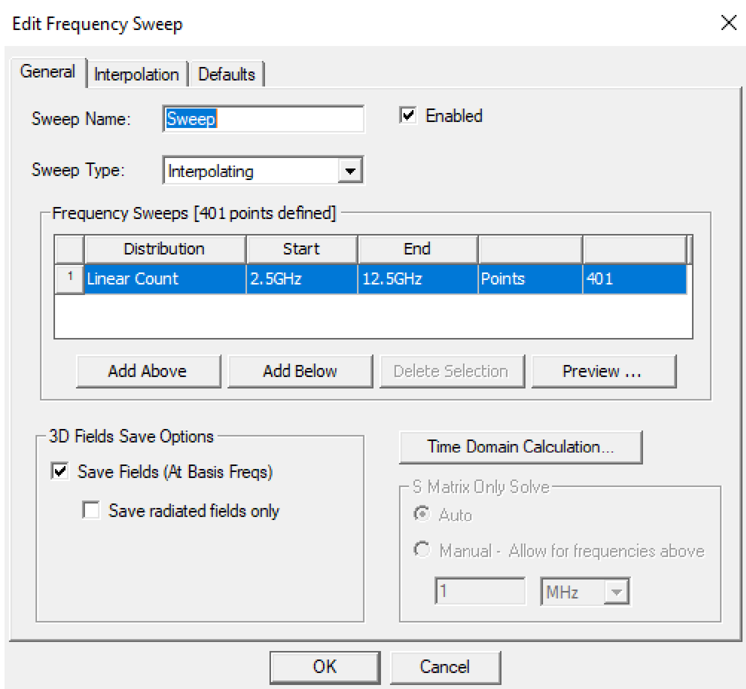
3. You can solve for either a single frequency or multiple frequencies. Ansys will sweep through a range of frequencies either way, and use this to generate data for S-parameter plots. At this stage, you are selecting frequencies at which you will be able to plot the H-field on the model itself.
   a. Multiple frequencies can be useful when trying to diagnose problems in the model.
4. In the Driven Solution Setup window, you can adjust the frequencies you solve for, maximum delta S, and maximum number of passes.

a. Delta S characterizes the mesh sensitivity for the S-matrix, which controls the convergence of the adaptive solution. A smaller delta S results in a more accurate solution.
b. For most cases, a delta S of 0.02 is best.
c. The maximum number of passes is the maximum amount of times HFSS will refine the mesh as it attempts to converge. If the allowed amount of adaptive passes is low, the solution will not fully converge, and HFSS will return a warning.
d. 20 passes is generally sufficient. You may go lower if the solution is taking especially long. If your solution does not fully converge, increase the number of allowed passes and re-run the simulation.

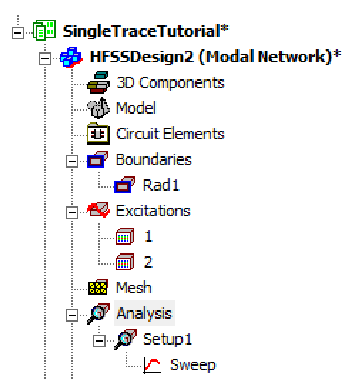


5. The Edit Frequency Sweep window should appear once you confirm your solution setup.
   a. If this window does not pop up on its own, right-click on your solution setup (which should have appeared under Analysis) and select Add Frequency Sweep.
   b. Set the sweep type to "Interpolating".

c. Set your sweep start and end to reasonable values. Make sure the frequencies you chose in your solution setup fall within this range.
d. ~400 points for 10 GHz typically works well for smooth plotting.
e. **Check the box for Save Fields (At Basis Freqs)**
    i. This is what allows you to plot the magnetic field onto your model! This is very useful for assessing whether the simulation has run correctly.

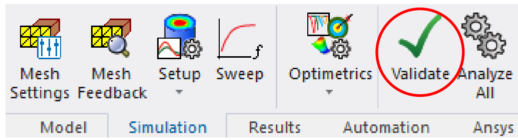


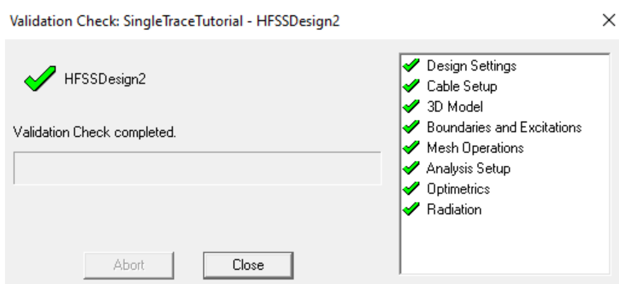
f. You will be able to find your Setup and Sweep under Analysis under the design tree in Project Manager. Double click on either of them to edit.



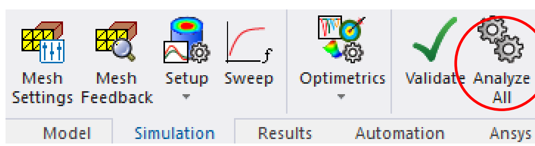
Running the Simulation:

1. Under the Simulation tab in the top banner, click Validate. This runs a validation check. Make sure there are no errors or warnings.





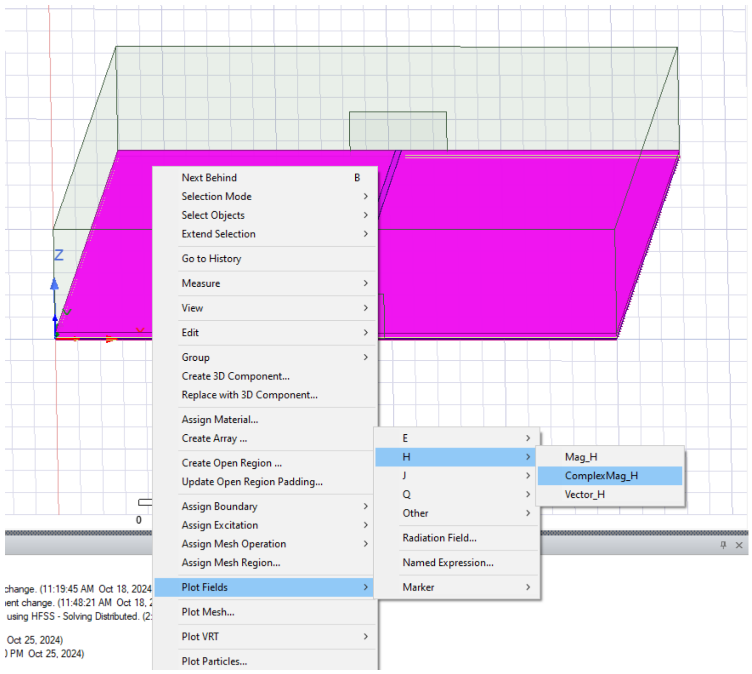
2. Under the Simulation tab, click Analyze All.



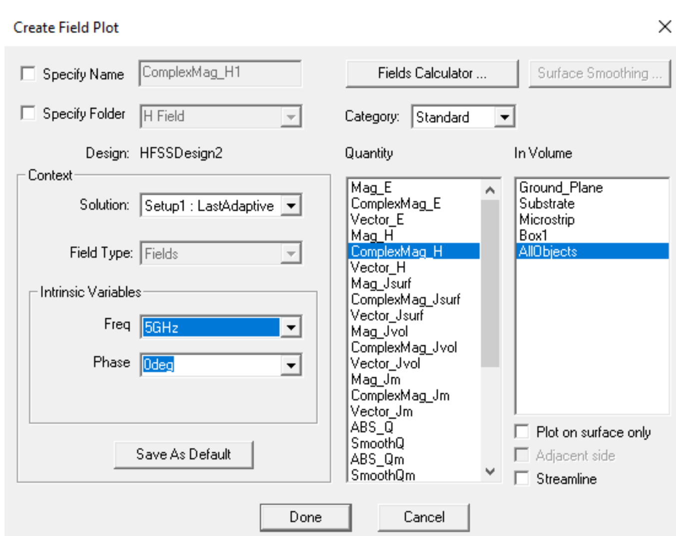
3. Wait for the simulation to finish running!

Viewing Solutions:

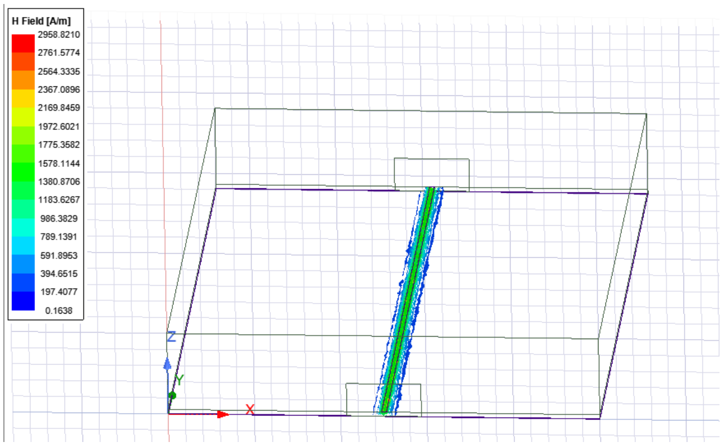
1. At this point, it is best to make sure all model objects are transparent.
2. **Plotting magnetic field**
   a. Select any geometry on your model. Right click, hover over Plot Fields, hover over H, and select ComplexMag_H.



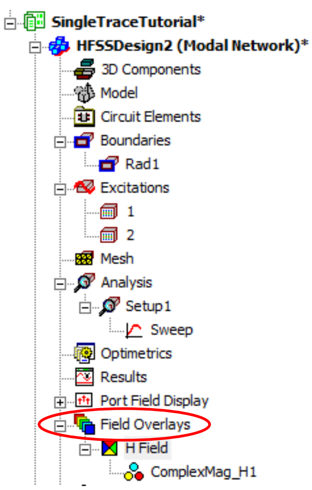
   b. The Create Field Plot window will pop up. You can plot the magnetic field at any of your basis frequencies and choose which model object to plot on.



c. You should be able to view the magnetic field overlay on your model.

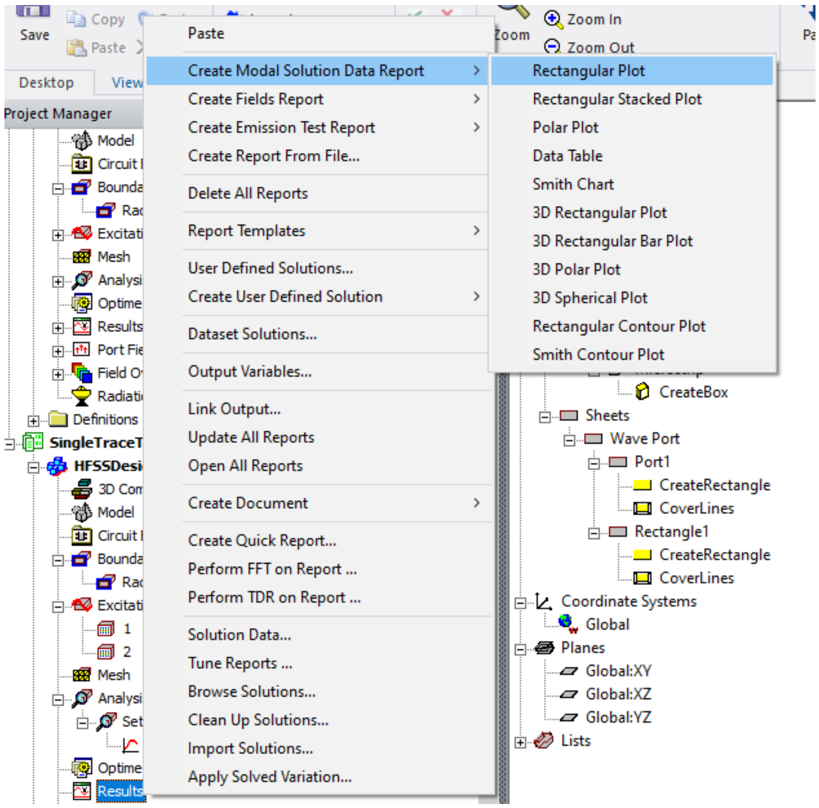


d. You will find plotted fields under Field Overlays under the design tree in project manager. Right click on a plot to modify or delete it.

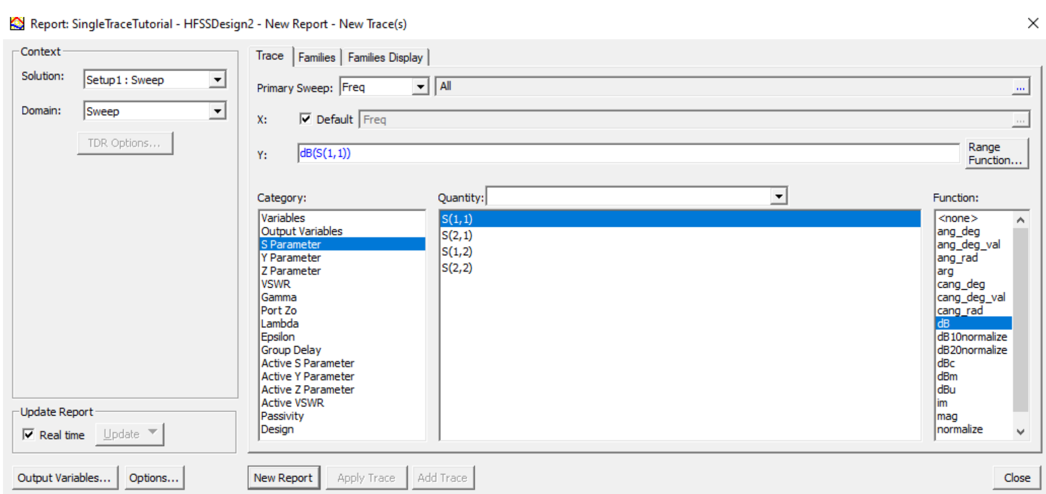


3. **Plotting S-parameters.**

   a. Right click on Results under the design tree in project manager. Hover over Create Modal Solution Data report, and select Rectangular Plot.



   b. You will be prompted to choose a variable and units. Choose these appropriately and click New Report.



   c. The appropriate plot will be generated.

Miscellaneous

1. To delete an object: Right click on the object name, hover over Edit, then select Delete.

2. Create global variables for all important properties!
    a. Type "$" followed by your variable name into any size or position box in the properties window and click the enter key. Ansys will recognize any non-numerical value as a variable, and prompt you to define it.
        i. $ ensure the variable is saved as a "Project Variable" and can be accessed from Project Manager (right click on the file name, and click Select Project Variables).

3. To easily orient to a specific view of your model:
    a. Right click on your model, hover over View, and select Apply Orientation.