Workshop 6.1: Microstrip Bend Lumped Ports and Simulation

HFSS Getting Started

Release 2020 R2

Outline - Microstrip Bend Lumped Ports and Geometry Construction

This HFSS workshop starts with a microstrip transmission line with a right-angle bend built in the geometry construction workshop.

The steps covered in this workshop:

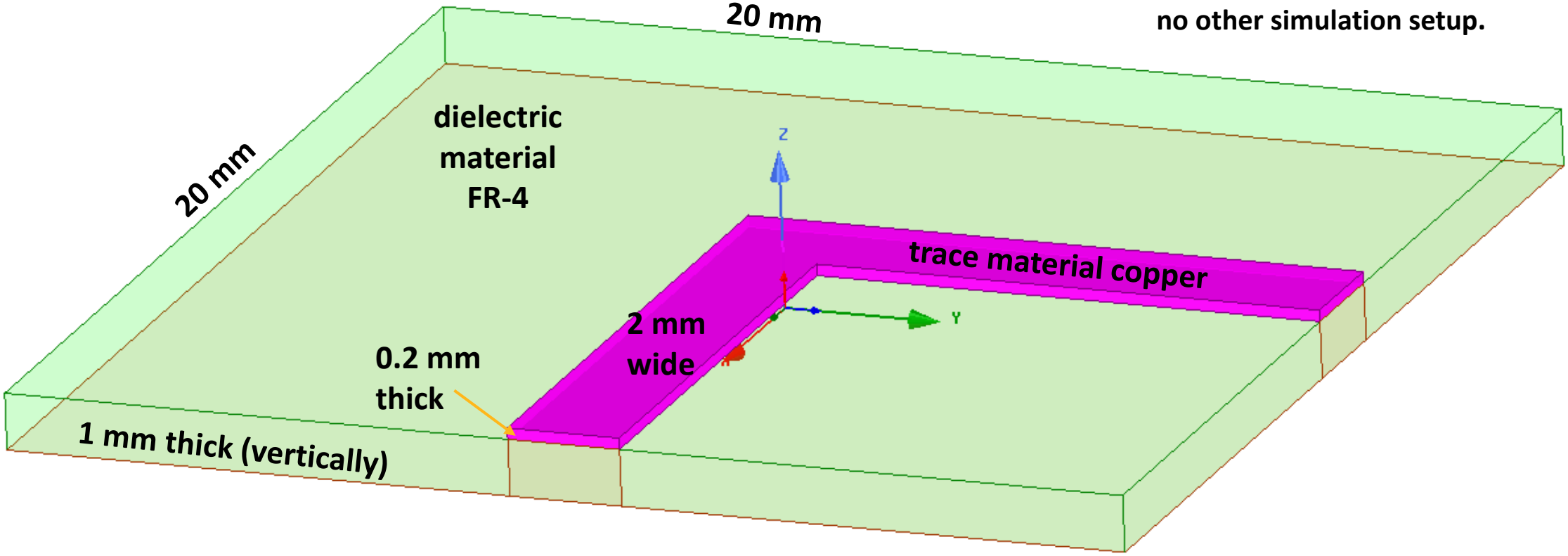
1.	Add Lumped Ports and an Open Region	**MicrostripBend6.aedt**
2.	Add a Solution Setup and Frequency Sweep	**MicrostripBend7.aedt**
3.	Simulate (Analyze)	**MicrostripBend7.aedt**
4.	View S-Parameters	**MicrostripBend7.aedt**
5.	View Field Plot	**MicrostripBend7.aedt**
6.	Animate Field	**MicrostripBend8.aedt**

This workshop starts with **MicrostripBend5.aedt**. The ending HFSS project file name will be: **MicrostripBend8.aedt.**

Additional detailed information on geometry construction in the HFSS modeler is available in the Help document ***HFSS.pdf*** which can be found in the HFSS online Help and in the installation directories in the ***Help/HFSS*** directory. RTMT - Read the manual too!

Microstrip Bend Model Geometry Overview

MicrostripBend5.aedt has the complete geometry and no other simulation setup.



The magenta (pink) color indicates that the object is selected.

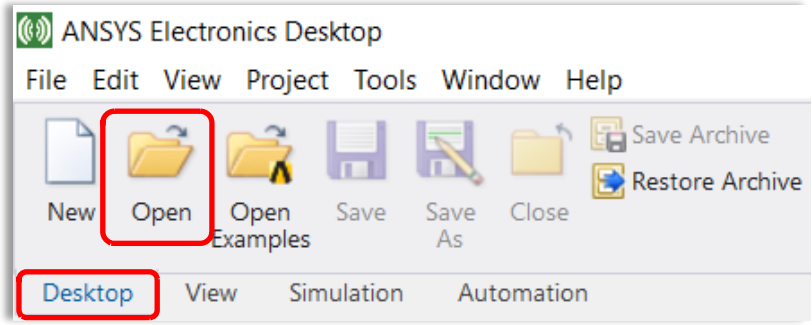
perfect electrical conductor boundary condition ground plane along the bottom of the substrate

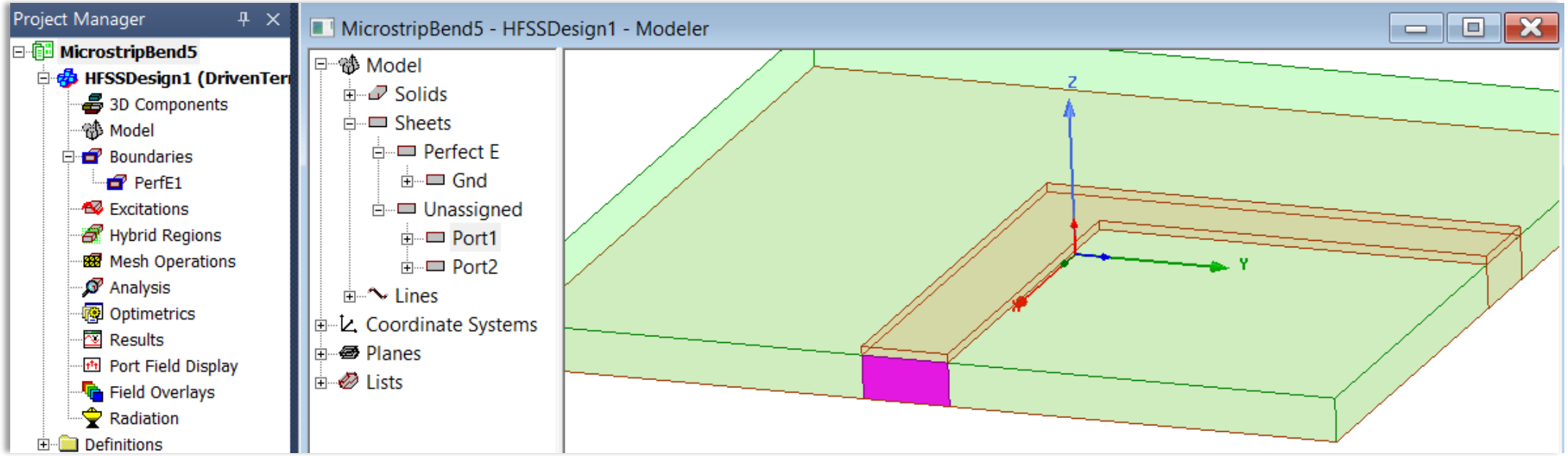
Ansys

Open AEDT Project MicrostripBend5.aedt

– In the ***Desktop*** tab of the Ribbon, click on the ***Open*** button and select ***MicrostripBend5.aedt*** from WS5.1.

This can also be accessed from the pull-down menus by selecting ***File > Open****.*

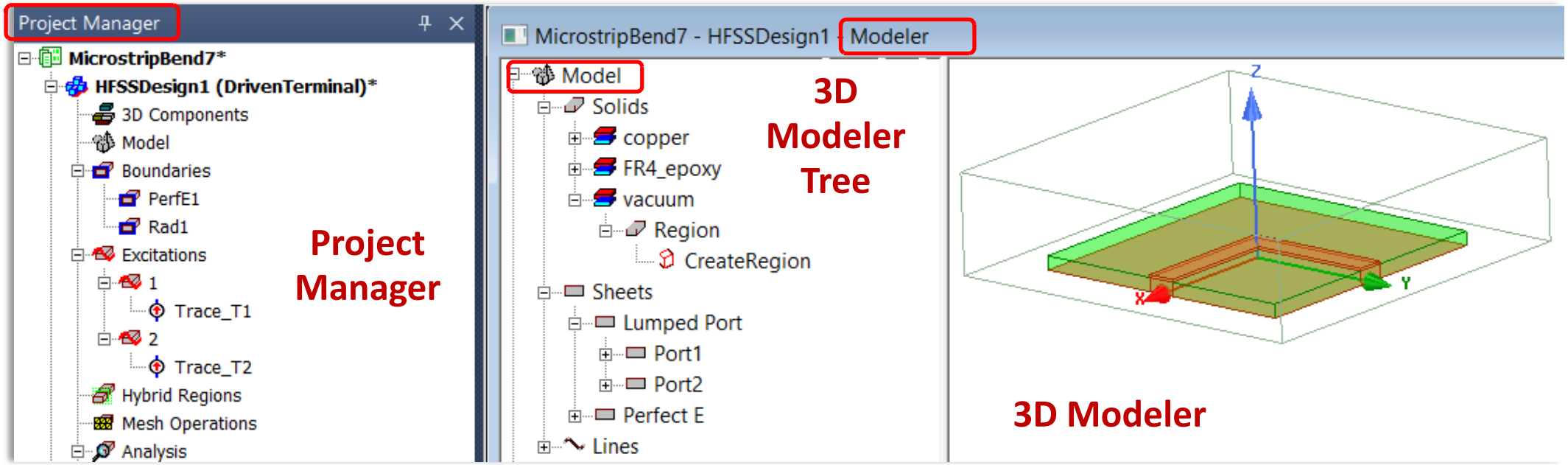




Ansys

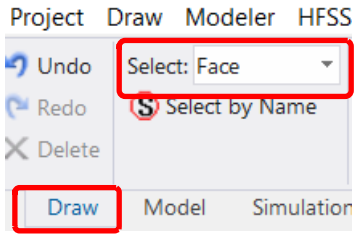
HFSS *Project Manager, 3D Modeler Tree*, and *3D Modeler*

Terminology clarification…

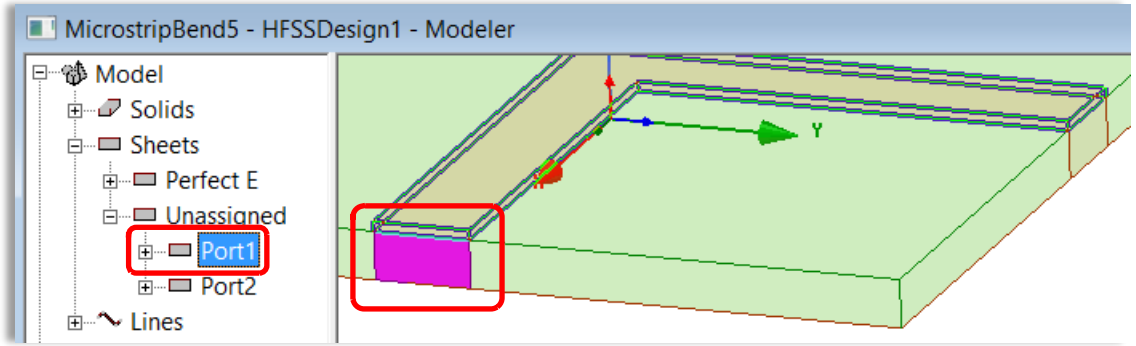


Ansys

Select Port Rectangle and Add Two Lumped Ports



Make sure the ***Select*** mode is set to ***Face.***

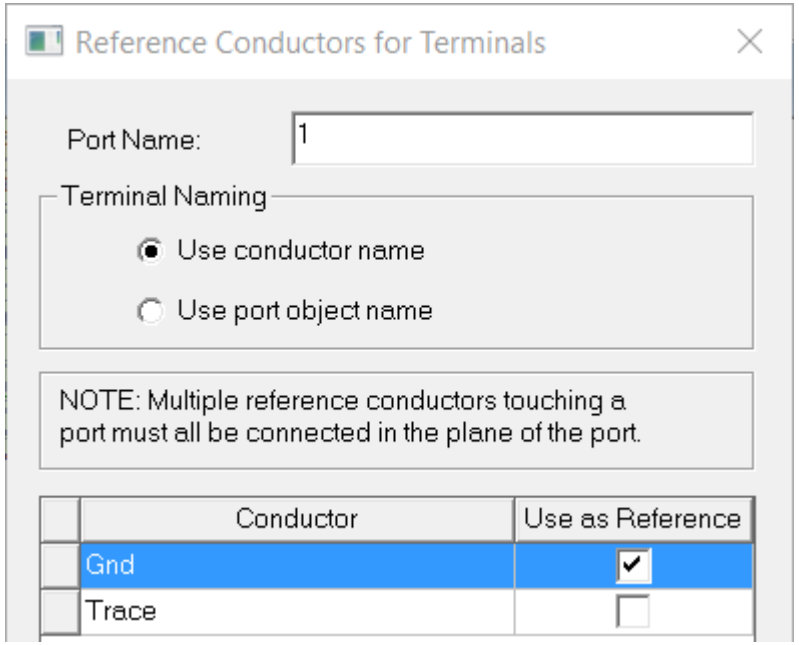


- In the ***3D Modeler,*** select the ***Port1*** rectangle, either from the ***3D Modeler Tree*** or by clicking on the ***Face*** on the geometry.
- Right-click on selected ***Port1*** and choose ***Assign Excitation > Lumped Port…***

*This can also be accessed from the pull-down menus by selecting **HFSS > Excitations > Assign > Lumped Port…***

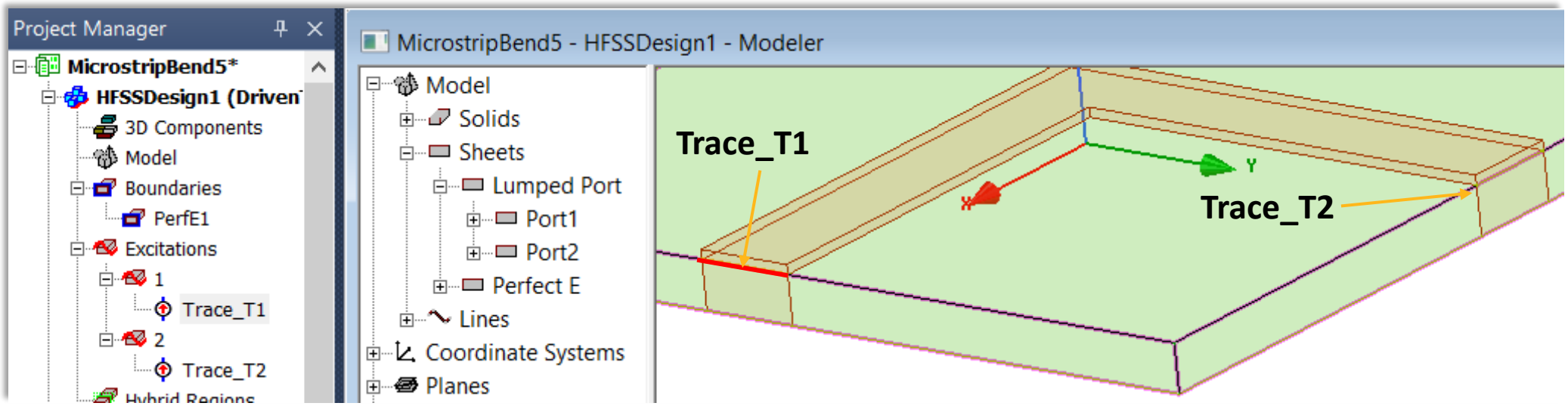
In the ***Reference Conductors for Terminals*** dialog box that appears…..

- Click ***Gnd*** for ***Use as Reference***
- Click ***OK***
- Repeat for second port.



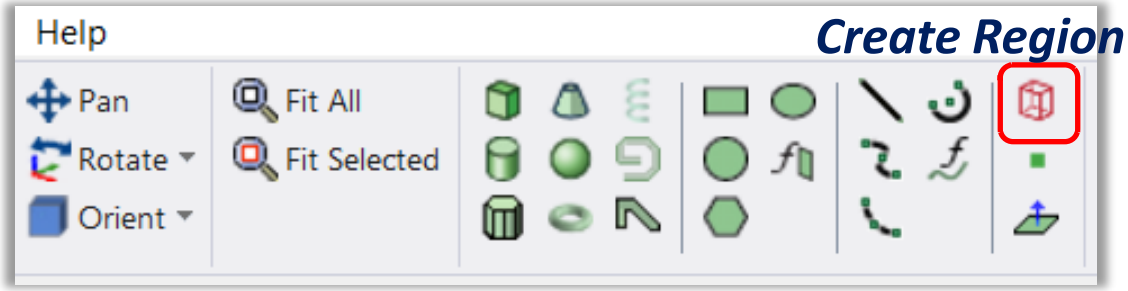
Ansys

Access Open Region Dialog Box from Ribbon Draw Tab



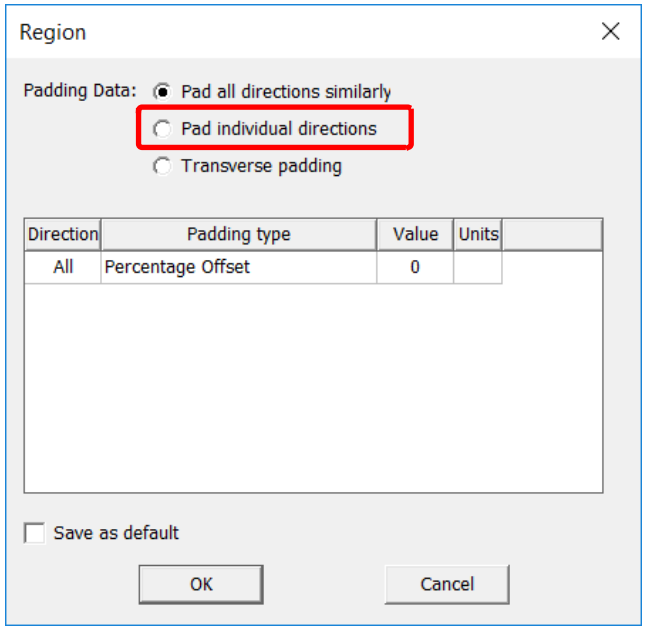
With two lumped ports in place, the next step is to create at open region.

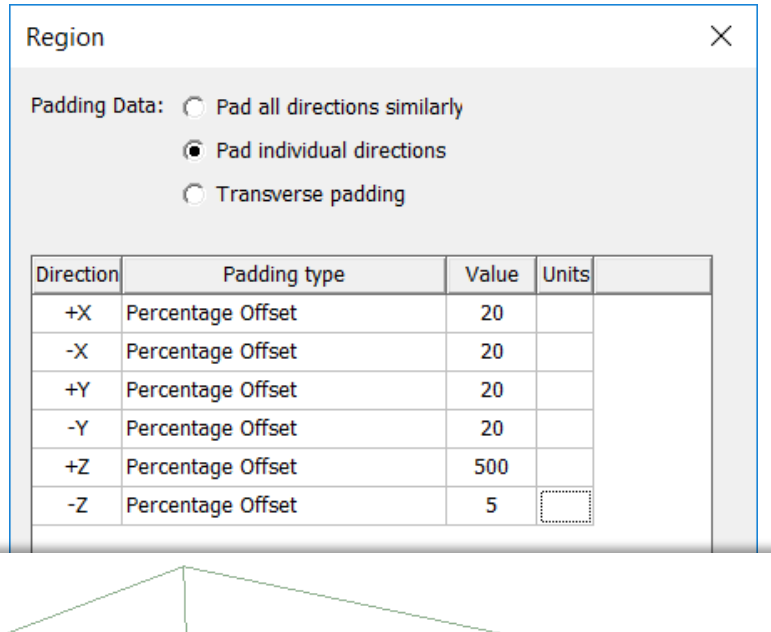
– From the Draw tab in the Ribbon, click on ***Open Region*** to bring up the ***Region*** dialog box.



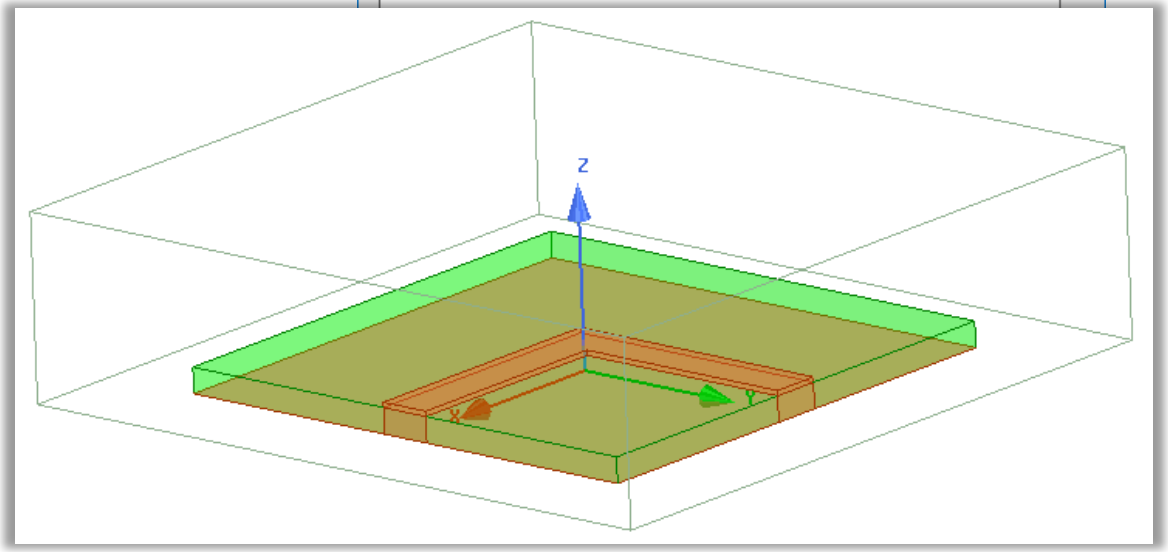
Ansys

Specify Region Dialog to *Percentage Offset*





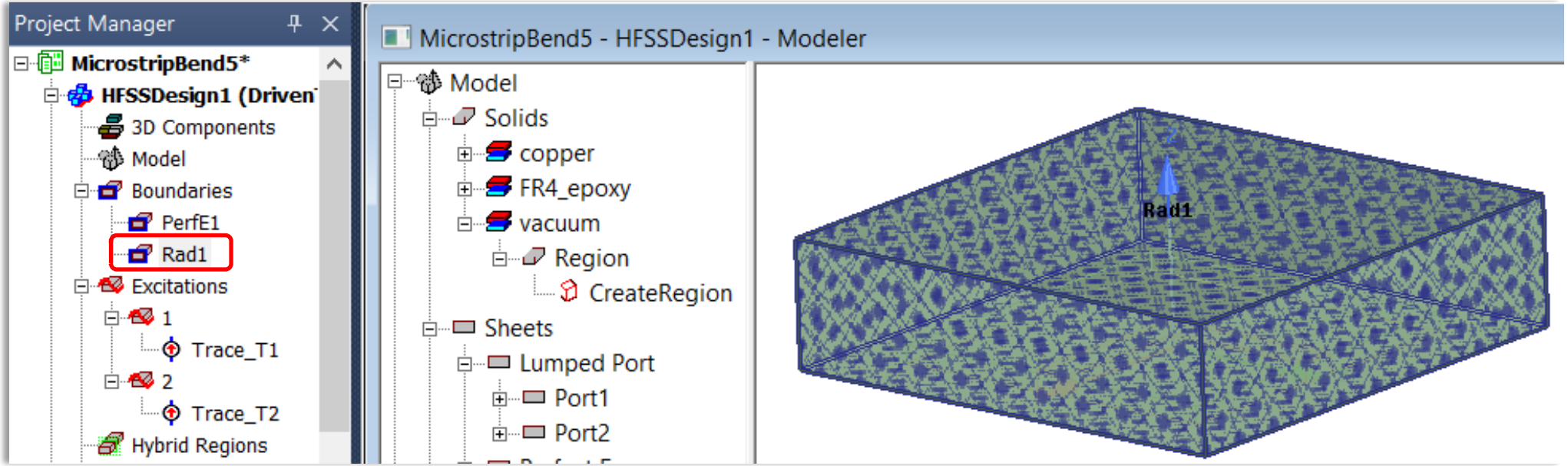
- When the Region dialog box appears, select ***Pad Individual Directions***
- Fill in the values: ***20%*** for ***+X, -X, +Y, -Y***
- For ***-Z*** (bottom) ***5%***
- For ***+Z: 500 %***
- Click ***OK***



Ansys

Assign *Radiation* Boundary to *Region* - MicrostripBend6.aedt

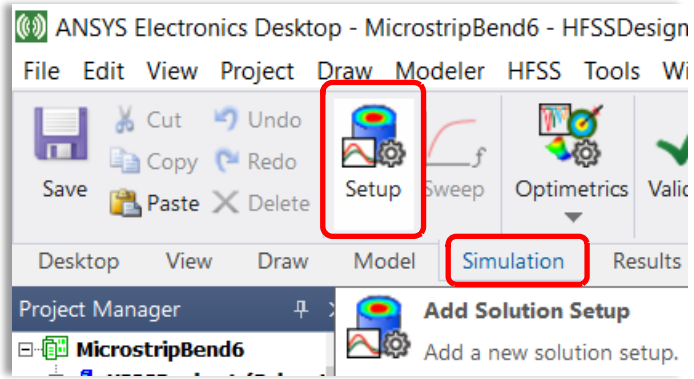
- Select the ***Region***, either in the ***3D Modeler Tree*** (under **vacuum**) or in the ***3D Modeler***.
- Right click and choose ***Assign Boundary > Radiation.***
- When the ***Radiation Boundary*** dialog box appears, the default name **Rad1** can be used.
- Click ***OK*** to close the ***Radiation Boundary*** dialog box.
- Select ***File > Save As*** **Mictrostripbend6.aedt**

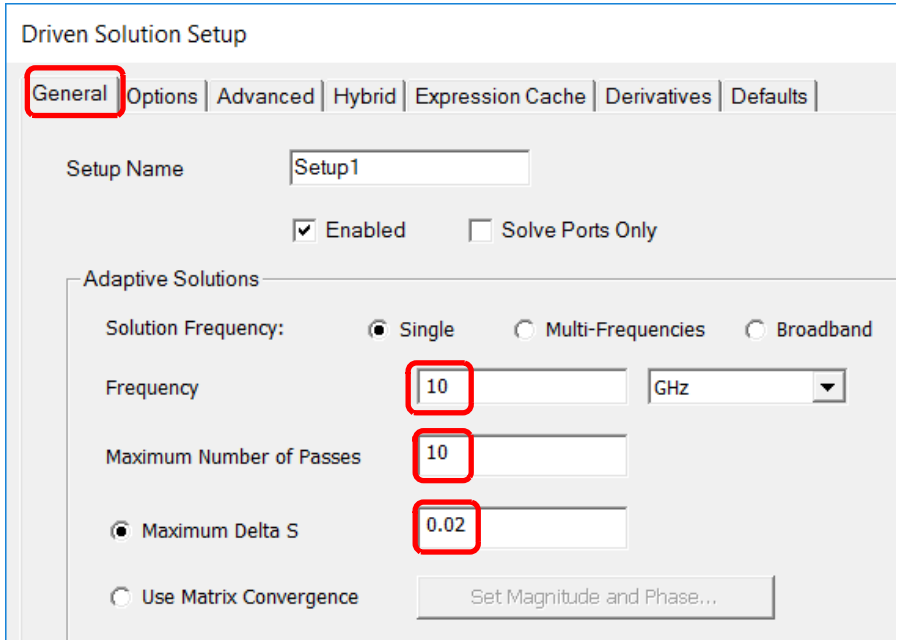


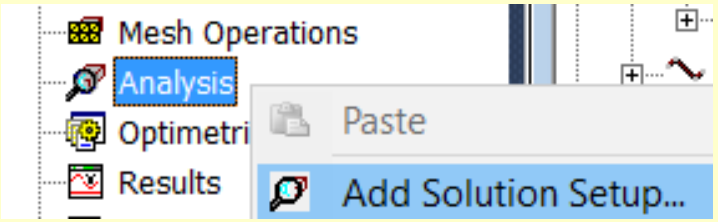
Ansys

Create a *Solution Setup* - 10 GHz - *Delta S* 0.02

- In the ***Ribbon***, under the Simulation tab, select ***Setup > Advanced.***
- When the ***Driven Solution Setup*** dialog box appears, in the ***General*** tab, specify
  - ***Frequency*** **10 GHz**
  - ***Maximum Number of Passes:*** **10**
  - ***Maximum Delta S:*** **0.02**
- Click ***OK***
  … which brings up the ***Edit Frequency Sweep*** dialog box.





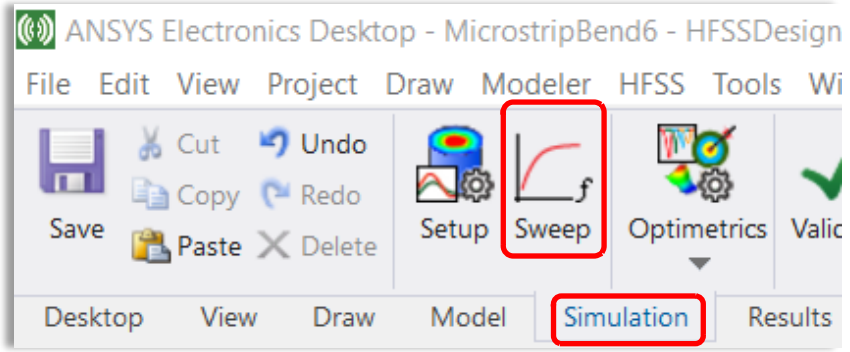


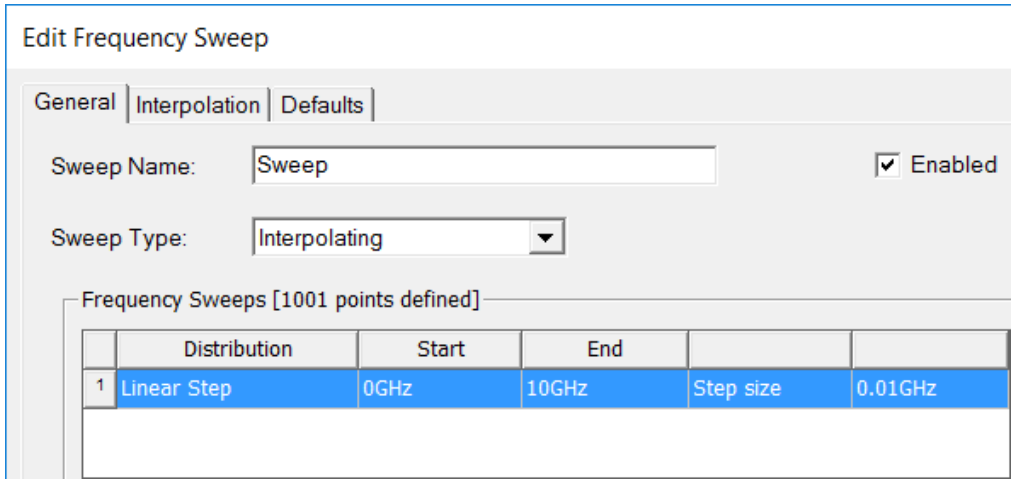
Solution Setup is also accessible from the ***Project Manager*** by right-clicking on ***Analysis*** and choosing ***Add Solution Setup.***

Ansys

Add a *Frequency Sweep* from 0 to 10 GHz in 0.1 GHz *Steps*

- If the ***Edit Frequency Sweep*** window doesn't appear …
- In the ***Project Manager***, under ***Analysis***, select the new **Setup1**.
- In the ***Ribbon***, under the Simulation tab, select ***Sweep***
- When the ***Edit Frequency Sweep*** dialog box appears, in the ***General*** tab, specify:
  - ***Sweep Type:*** **Interpolating**
  - ***Frequency Setup Type:*** **Distribution: Linear Step**
  - ***Start*** **0.0 GHz**
  - ***End:*** **10 GHz**
  - ***Step:*** **0.1 GHz**
- Click ***OK***
  If a message appears about using causal materials, click ***OK***.
- Click ***OK***
  to close the ***Edit Frequency Sweep*** dialog box.





Edit Frequency Sweep is also accessible from the ***Project Manager,*** under ***Analysis*** by right-clicking on **Setup1** and choosing ***Add Frequency Sweep.***

Set Causal Materials in *HFSS > Design Settings*

To address this recommendation, we'll set HFSS to ***Automatically use causal materials***.

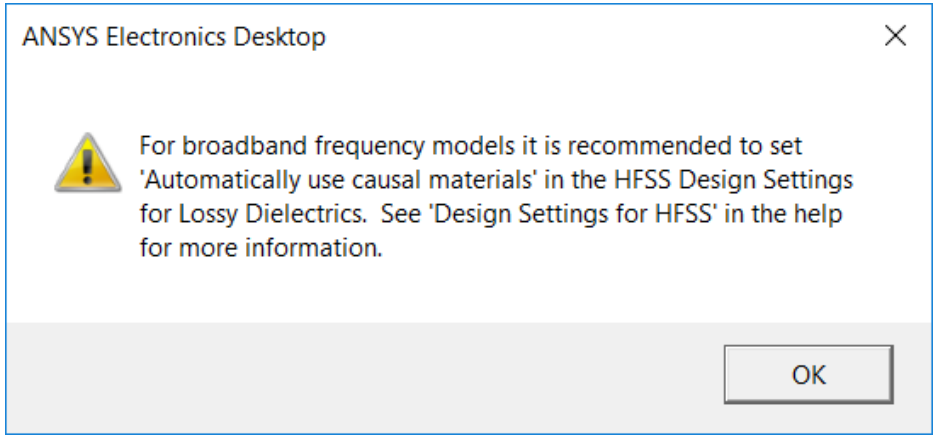
– Select ***HFSS > Design Settings*** to bring up the ***HFSS Design Settings*** tab dialog box.

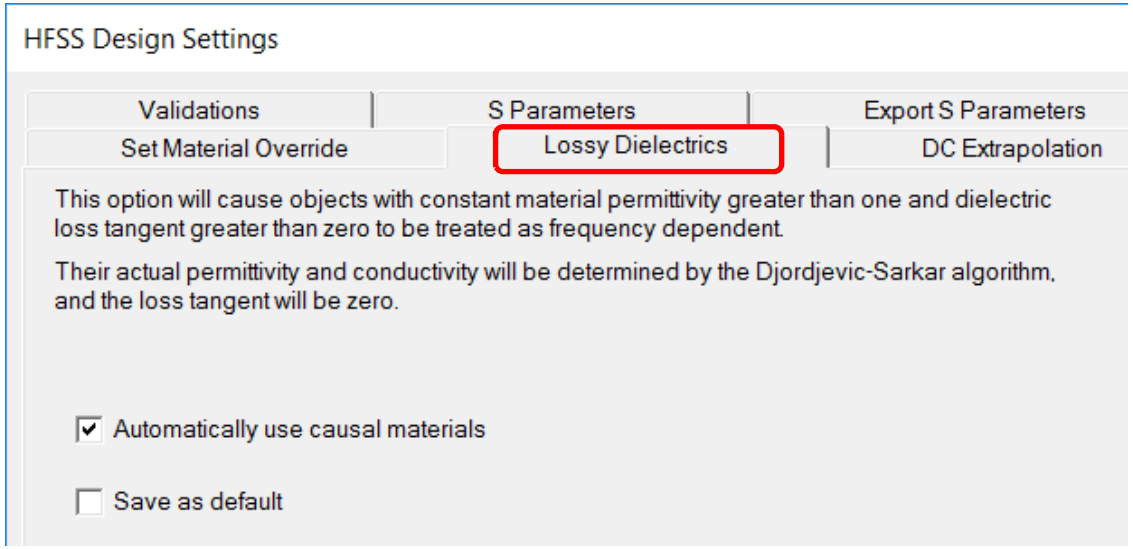
– Select ***Lossy Dielectrics*** tab.

– Check the box for ***Automatically use causal materials.***

– Click ***OK*** to close the ***HFSS Design Settings*** dialog box.

For more information see ***HFSS.pdf***, ***HFSS Technical Notes > Introduction to Causality Issues for Simulations***.



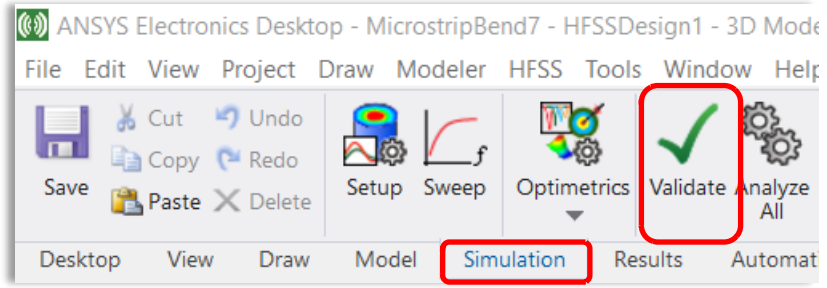


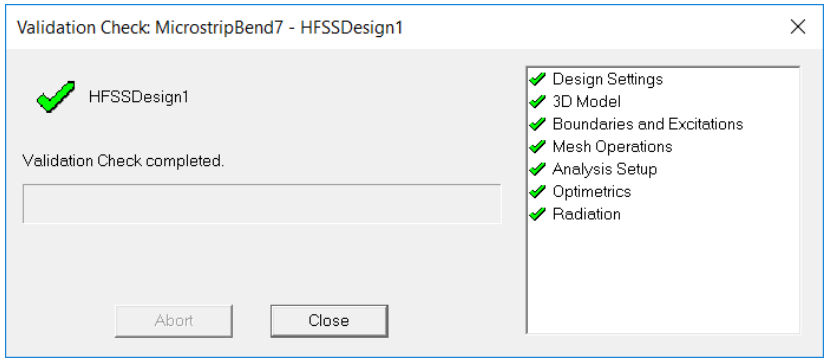
Save As MicrostripBend7.aedt, *Validate*, and *Simulate*

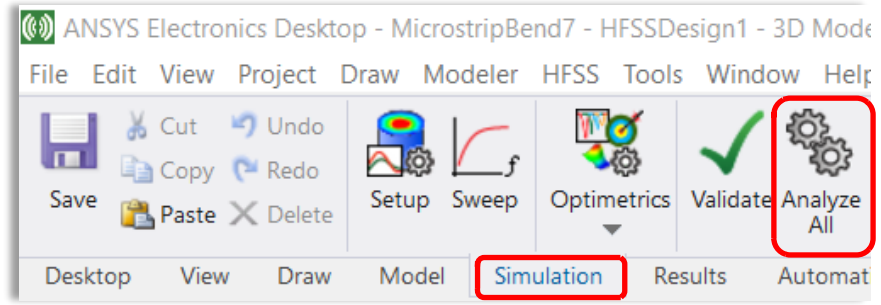
- Click the ***Validate*** green check mark in the ribbon (with Simulation chosen) to validate the project.
- Select ***File > Save As*** and save project renaming file as: **MicrostripBend7.aedt**

  Keep all HFSS workshop simulation files; future workshops continue with these files.
- 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).
- Save **MicrostripBend7.aedt** when the simulation finishes.







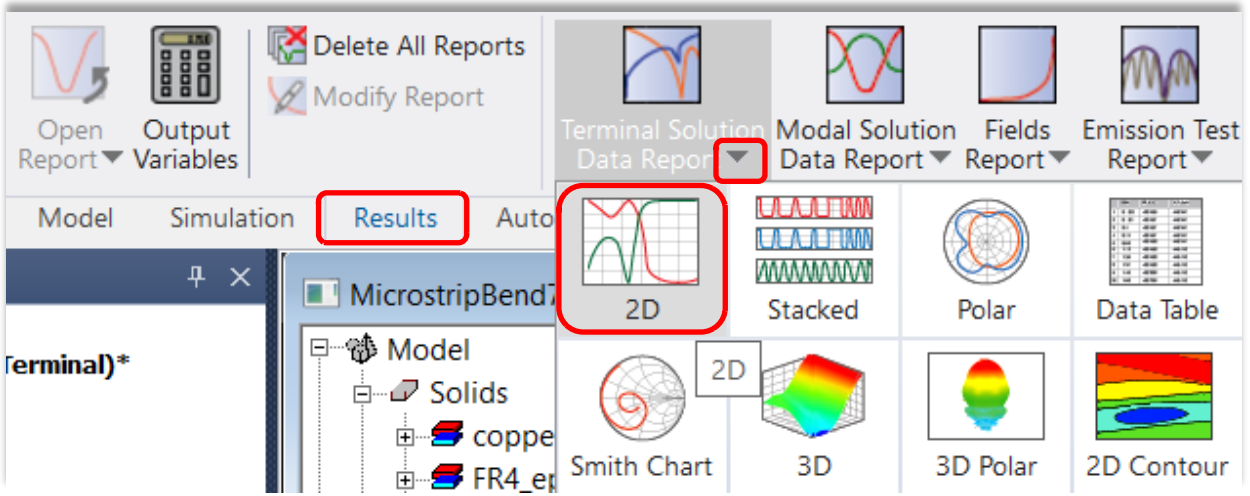
Click on ***Show Messages*** and in the bottom right of the GUI in order to see the simulation progress.

Ansys

Bring Up *New Report* Dialog Box for Plotting S-Parameters

- In the ***Ribbon***, in the Results tab, click on ***Terminal Solution Data Report*** and select ***2D.***
- When the ***Report*** dialog box appears, select the reflection and transmission S-parameters for port1/T1.

...continued on next page....



The ***New Report - New Traces*** dialog box is also accessible from the ***Project Manager.*** Right-click on ***Results*** and select under ***A Create Terminal Solution Data Report > Rectangular Plot.***

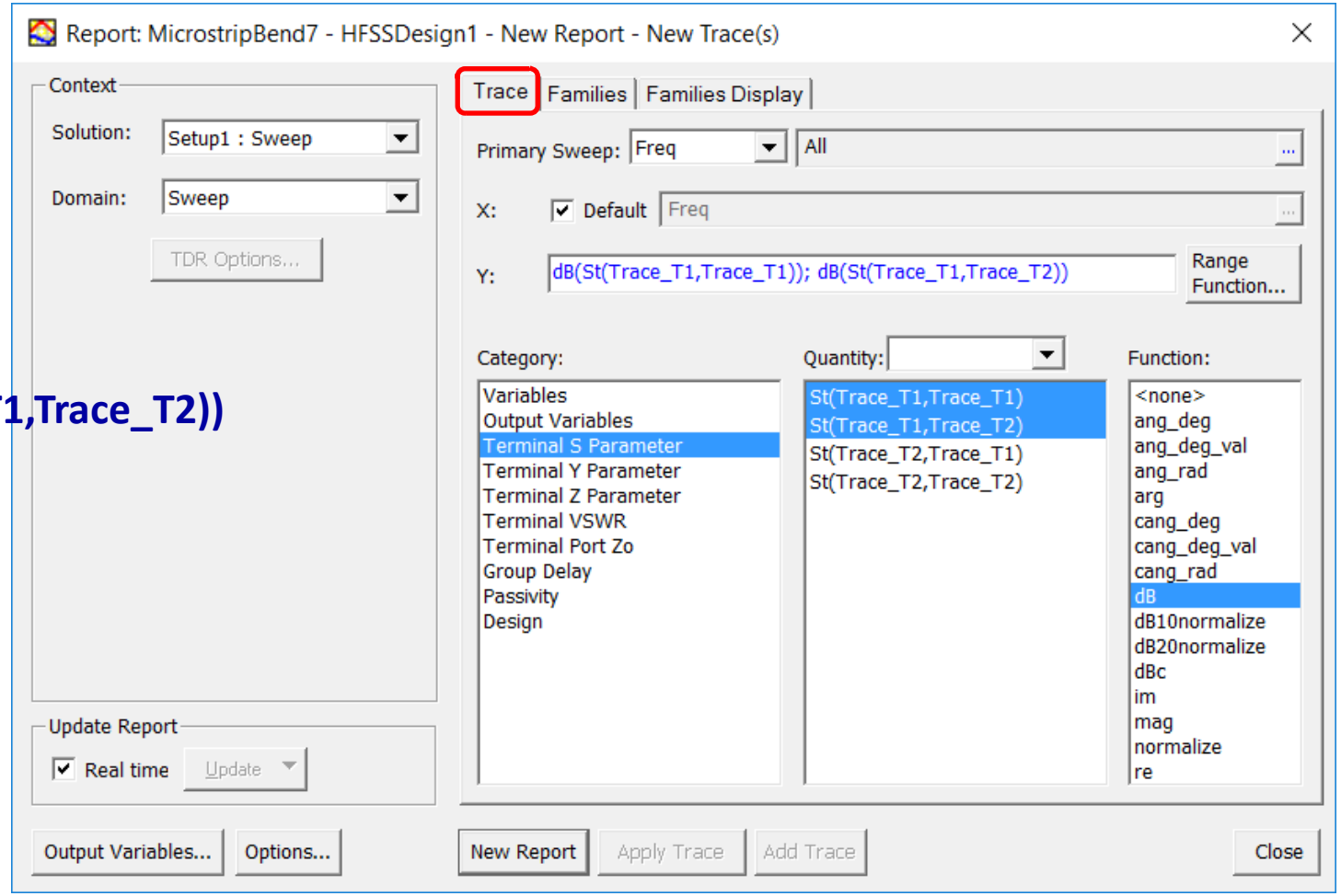
OR

From the top pull downs, select ***HFSS > Results > Create Terminal Solution Data Report > Rectangular Plot.***

Plot MicrostripBend7 S-Parameter Simulation *Results*

In the ***New Report*** dialog box, ***Trace*** tab set:

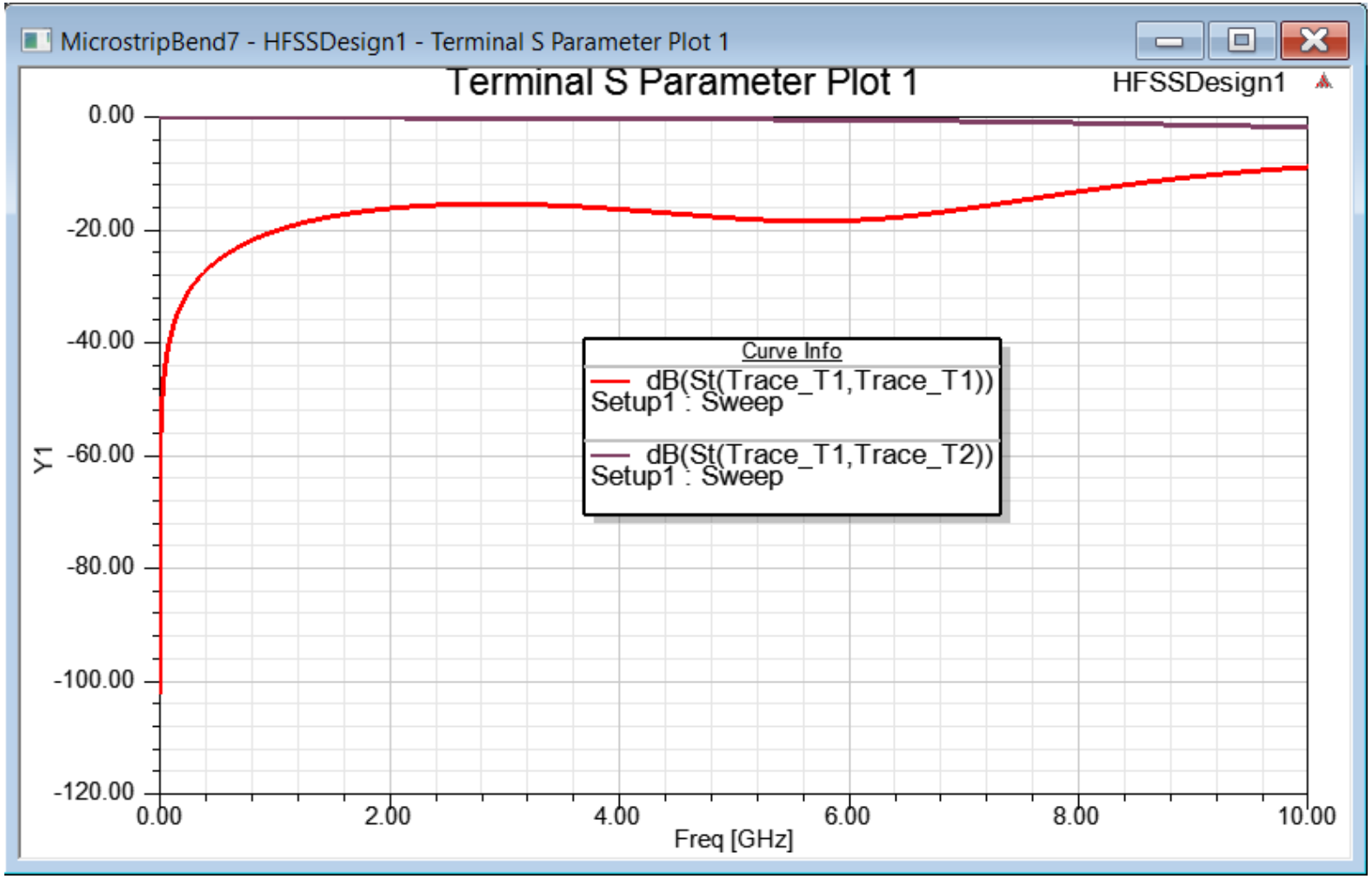
- ***Solution***: **Setup1: Sweep**
- ***Domain***: ***Sweep***
- ***Primary Sweep: Freq***
- ***Category***: **Terminal S-Parameter**
- ***Quantity***:
  dB(St(Trace_T1,Trace_T1)); dB(St(Trace_T1,Trace_T2))
  Use ***Ctrl-D*** to multiple select traces.
- ***Function***: ***dB***
- Click on ***New Report***.
- Click on ***Close***.



View MicrostripBend7 S-Parameter Simulation Results

As we expect, the transmission S-parameter magnitude dB(St(Trace_T1,Trace_T2)) (like an S_{12}) is near 100% and decreases as frequency goes up.

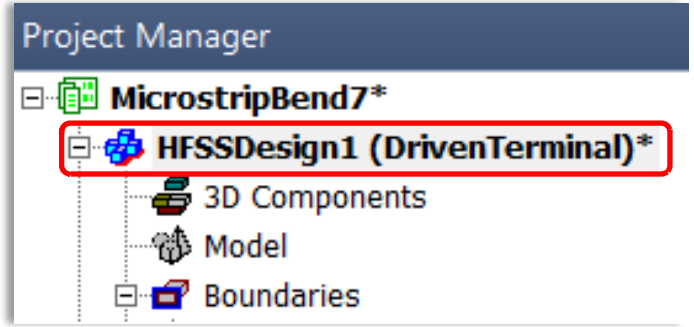
The reflection S-parameter dB(St(Trace_T1,Trace_T1)) (like an S_{11}) is lower and increases as frequency goes up.



Ansys

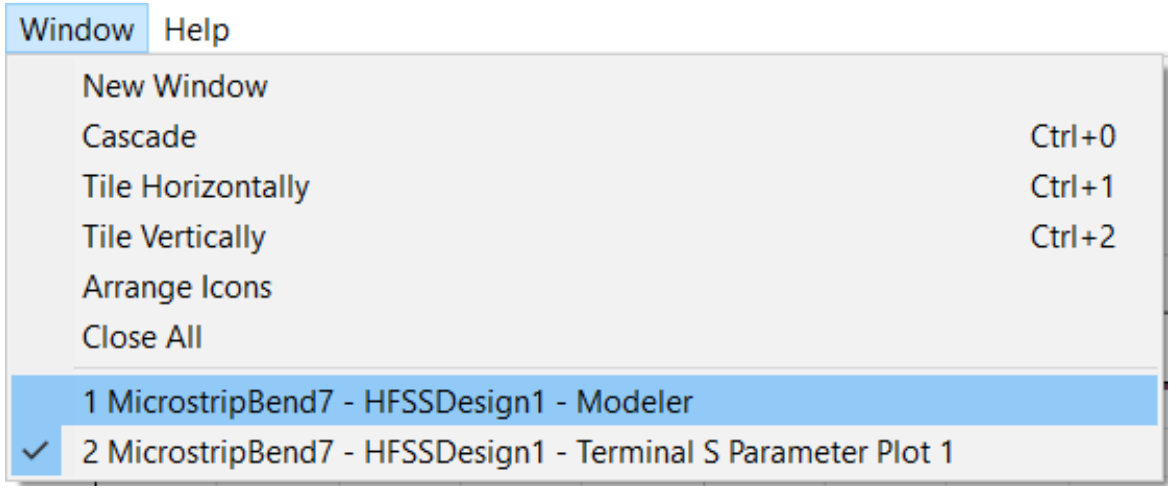
Return to the Modeler View

- From our view of the S-parameters, we want to return to the ***Modeler***. This can be done by either:
- Double-clicking on the design name ***HFSSDesign1 (DrivenTerminal)*** in the ***Project Manager***:

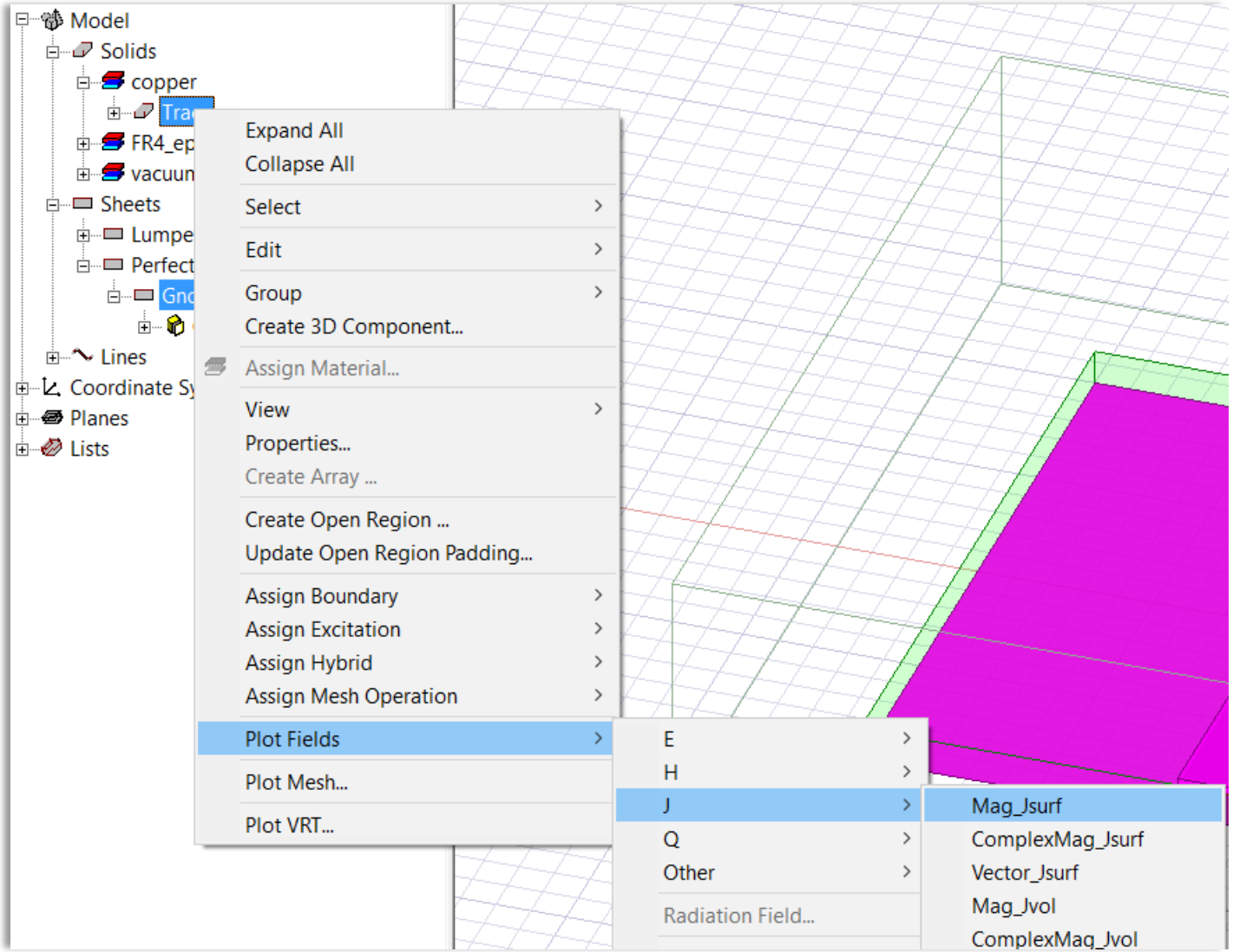


OR

- Selecting ***Window > MicrostripBend 7 - HFSSDesign1 - Modeler***



Bring Up the Create Field Plot Dialog Box



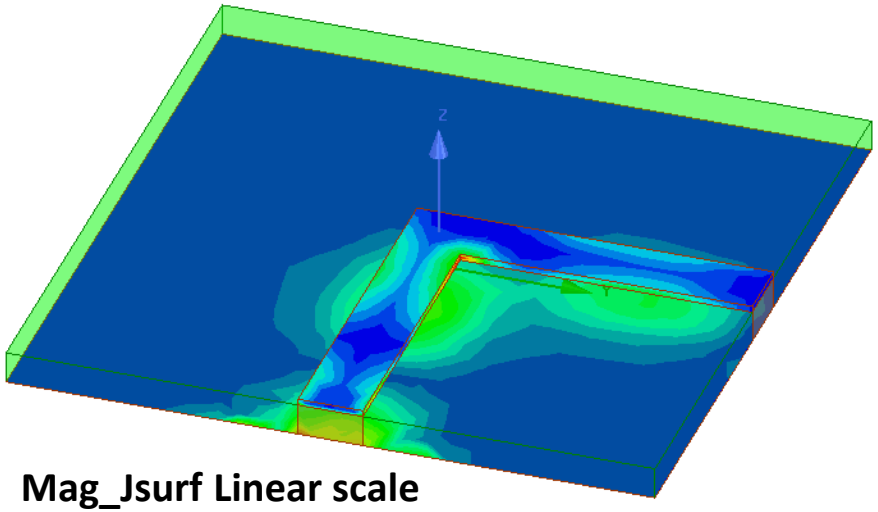
- Select both the ***Trace*** and the ***Gnd*** in the ***Model tree***. Use ***Ctrl-D*** for multiple select.
- Right click on either of them (or in the ***Project Manager*** on ***Field Overlays***)
- Select ***Plot Fields > J > MagJsurf*** to bring up the ***Create Field Plot*** dialog box.

Ansys

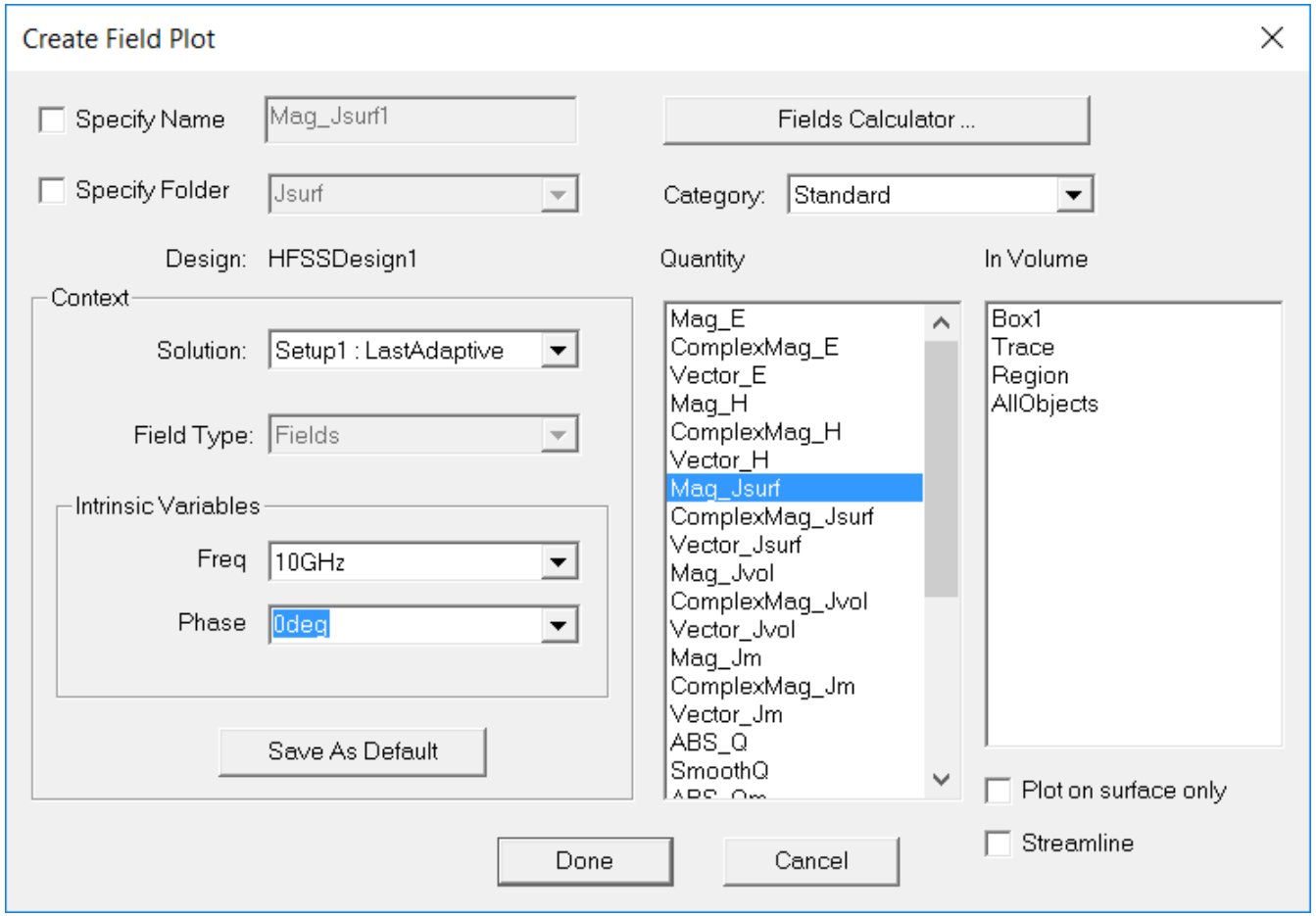
Access *Create Field Plot* Dialog Box

In the ***Create Field Plot*** dialog box, verify or select:

- ***Solution:* Setup1: LastAdaptive**
- ***Freq* 10GHz**
- ***Quantity Mag_Jsurf***
- Click ***Done***.



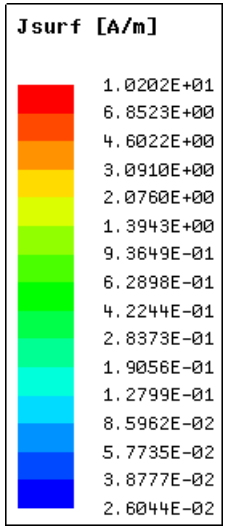
Mag_Jsurf Linear scale



Ansys

In *Attributes Scale* Tab Set *Scale* to *Log*

– In the ***3D Modeler***, double-click on the color scale to bring up the ***Jsurf*** tab dialog box.

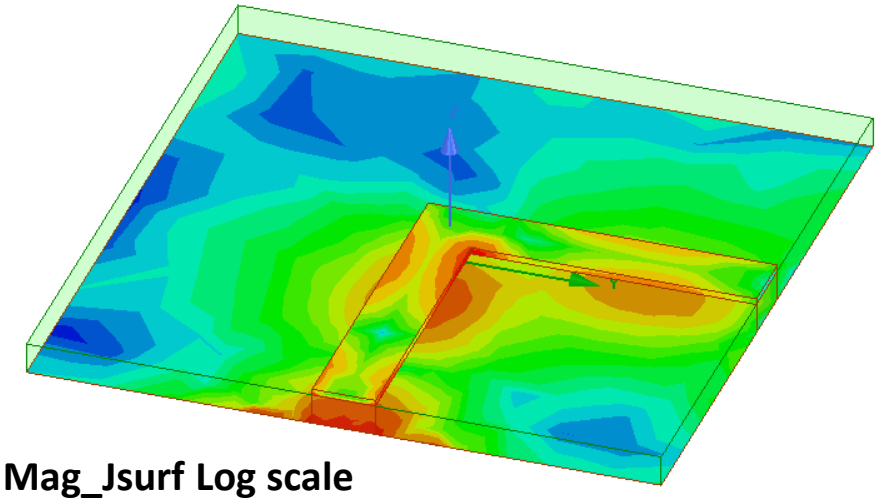


In ***Jsurf*** tab dialog box:

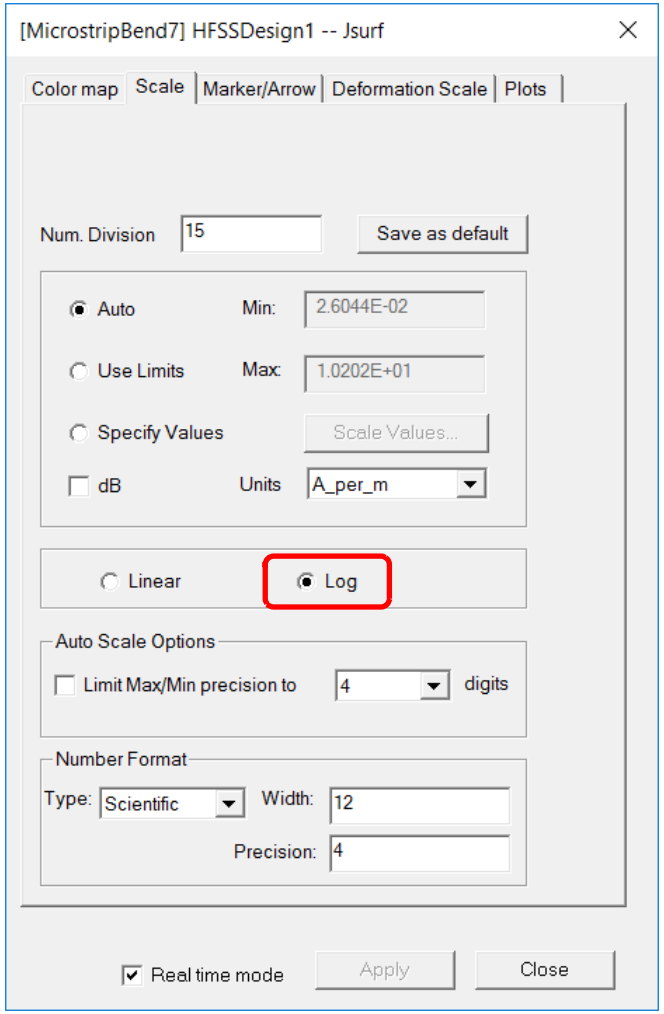
– Select the ***Scale*** tab.

– Near the middle, click on the ***Log*** radio button (instead of ***Linear***).

– Click ***Close.***



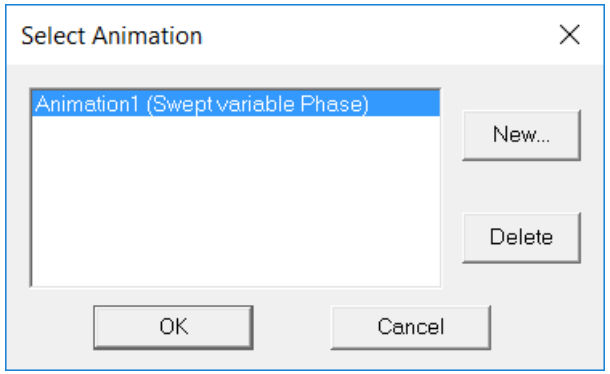
Mag_Jsurf Log scale

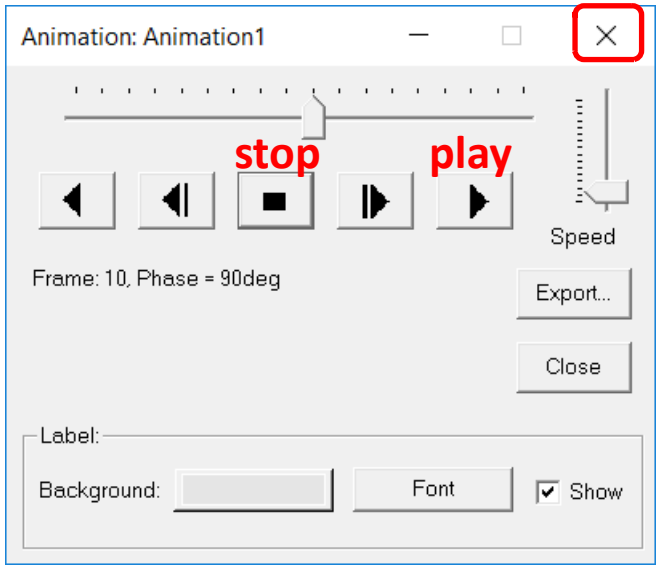


Ansys

Animate Fields and *Save As* MicrostripBend8.aedt

- In the ***Project Manager***, right-click on ***Field Overlays*** and select ***Animate***.
- In the ***Select Animation*** menu, click ***OK***.
- The ***Animation*** player comes up and the field animation starts playing.
- When finished, click the square stop button and the ***X*** to close the ***Animation*** player.
- ***Save As*** **MicrostripBend8.aedt** when the simulation finishes.





End of Presentation