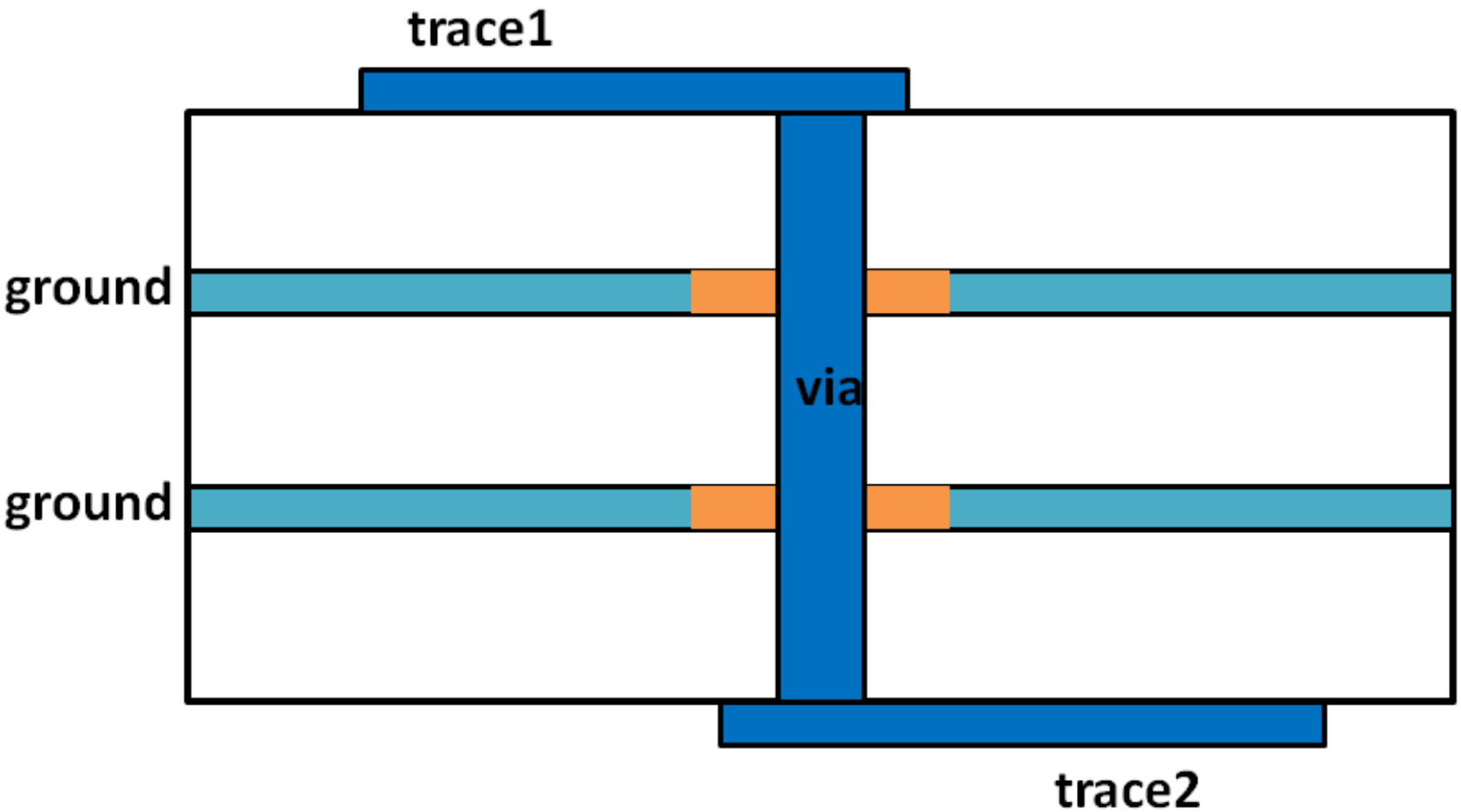
ECE 546: ANSYS HFSS Tutorial

Simulate and Analyze an Example of Trace and Via

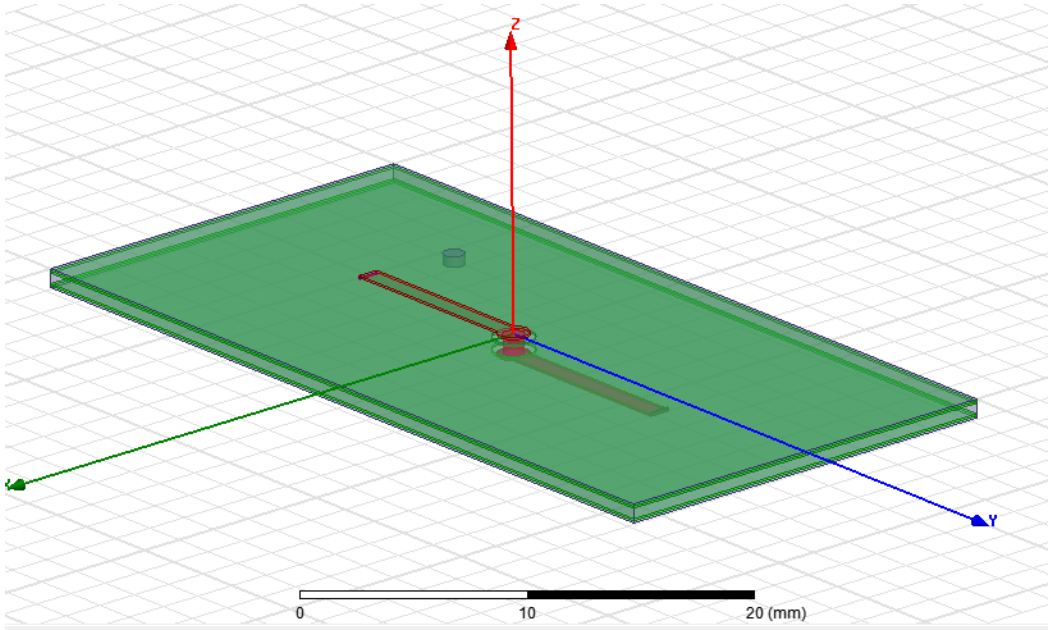
Tianjian Lu

01/30/2014

Trace and Via



Nominal Design



- Trace: length=10mm, width=1mm, thickness=0.1mm
- Via: radius=0.5mm, height=0.9mm
- Ground: thickness=0.1mm
- Board: thickness=0.9mm, relative permittivity ϵ_r=1

Getting Started

1. **Launch HFSS**:
   Start button ⇒ All Programs ⇒ ANSYS Electromagnetics ⇒ HFSS 15.0 ⇒ Windows 64-bit ⇒ HFSS 15.0
2. **Set HFSS options**:
   Tools ⇒ Options ⇒ HFSS Options ⇒ General Tab
   - check ✓Use Wizards for data input when creating new boundaries
   - check ✓Duplicate boundaries/mesh operations with geometry
   - click OK

Getting Started

3. **Set Modeler options**:
   Tools $\Rightarrow$ Options $\Rightarrow$ Modeler Options $\Rightarrow$
   Operation Tab
   - check ✓Automatically cover closed polylines
   - check ✓Select last command on object select

   Drawing Tab
   - check ✓Edit properties of new primitives

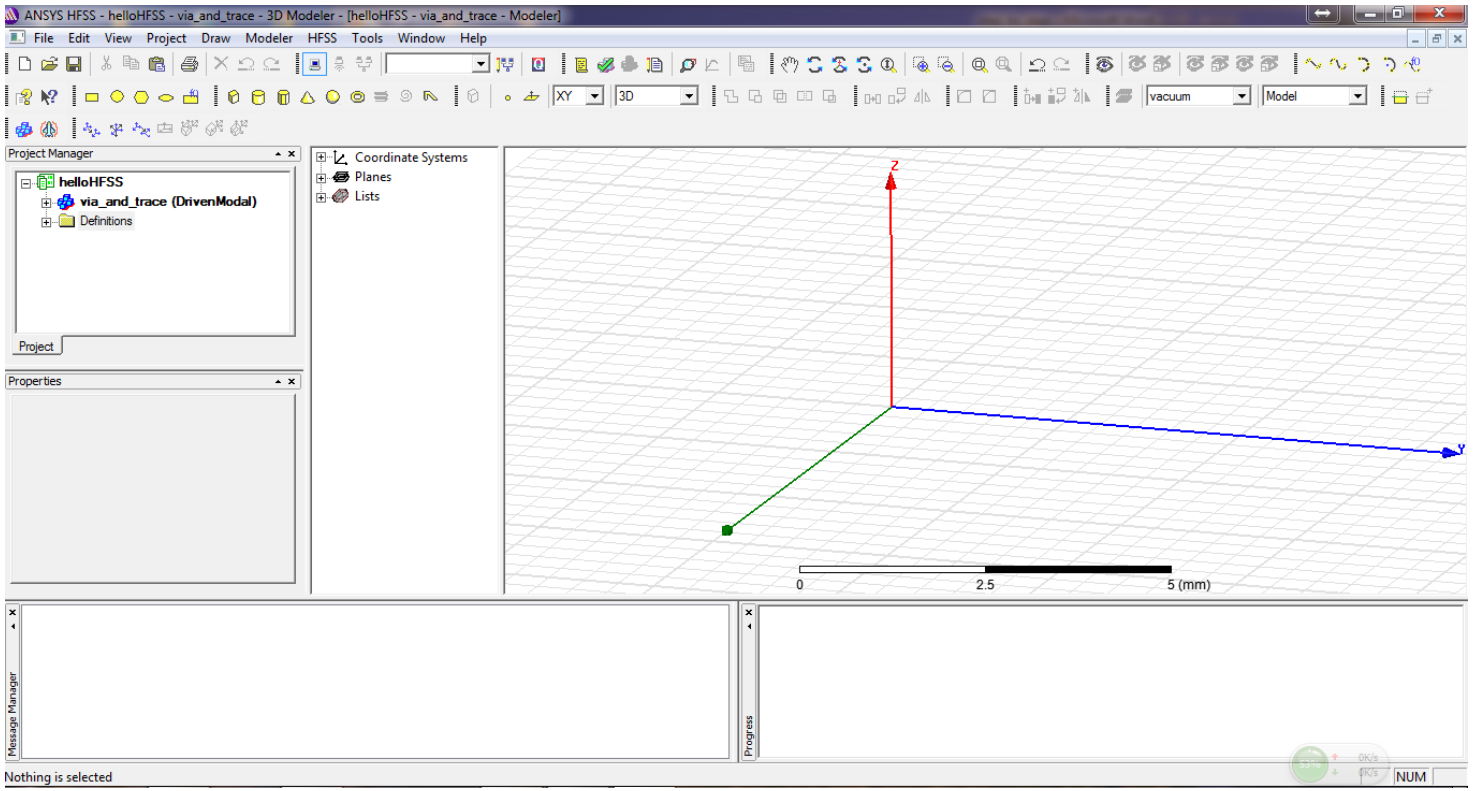
   Click OK

Getting Started

4. **New Project**
   - A new project called Project 1 is created by default, or you can create a new project from File $\Rightarrow$ New
   - Right click on Project 1 and select Rename (F2), you can rename the project as helloHFSS
5. **New HFSS Design**
   - Right click on helloHFSS $\Rightarrow$ Insert $\Rightarrow$ Insert HFSS Design
   - Rename the HFSS Design as *via_and_trace*

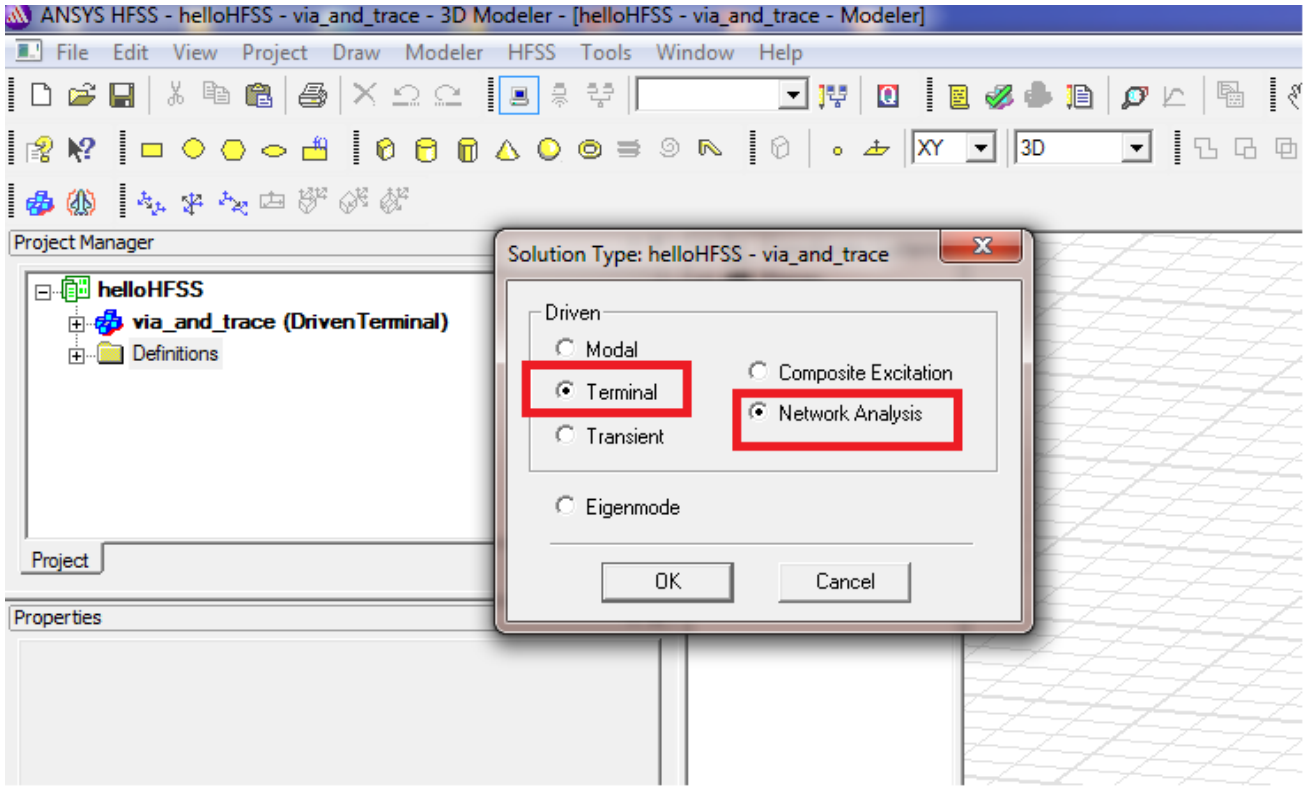
Hello HFSS!

Now you should have a workspace as below



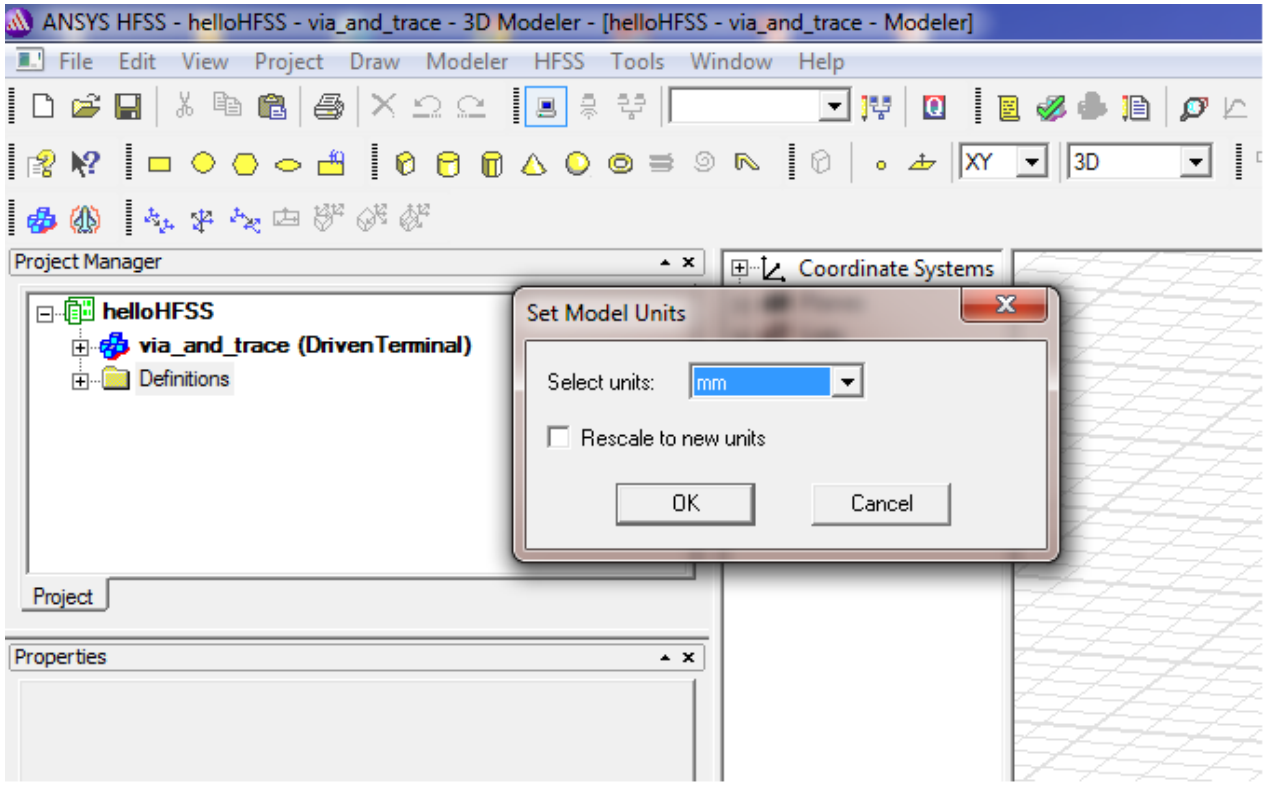
Creating the 3D model

1. **Set solution type**:
   HFSS ⇒ Solution Type ⇒ Driven Terminal ⇒ OK



Creating the 3D model

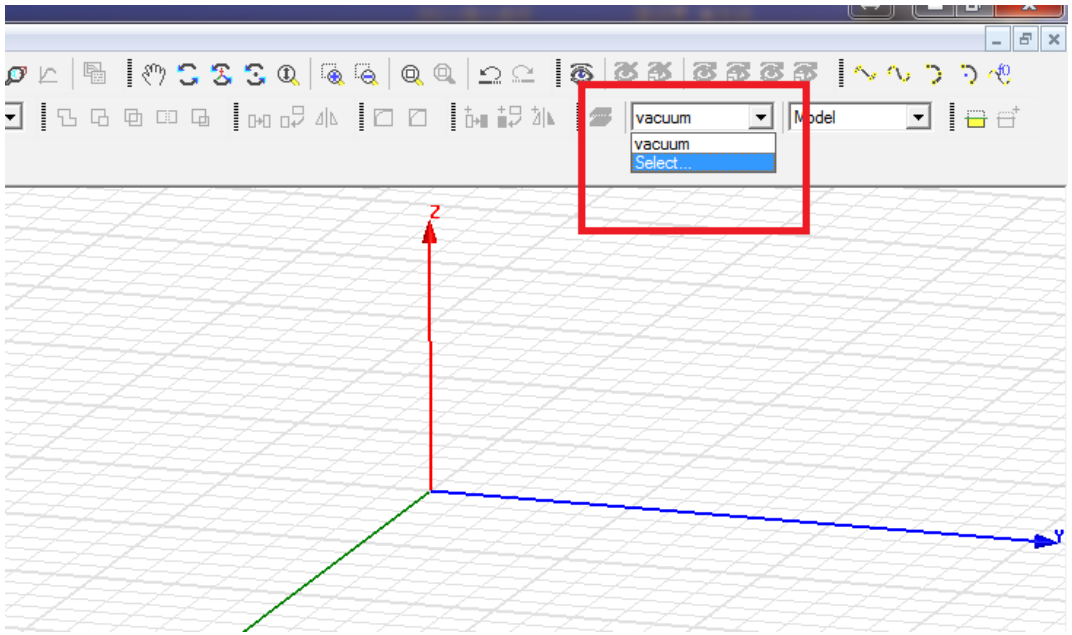
2. **Set model unit**:
   Modeler ⇒ Units ⇒ Select mm ⇒ OK



Creating the 3D model

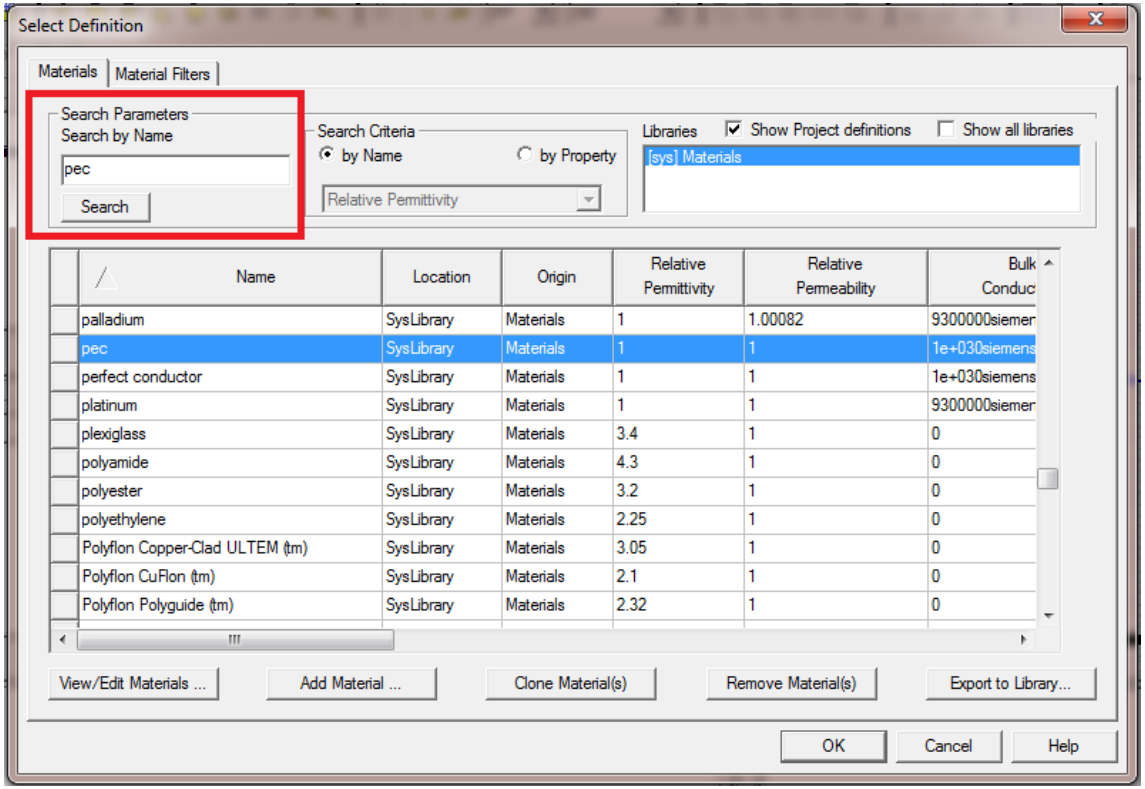
3. **Set default material**:
   Open material tool bar ⇒ type **pec** in the **Search by Name** field ⇒ Click OK

Open material tool bar



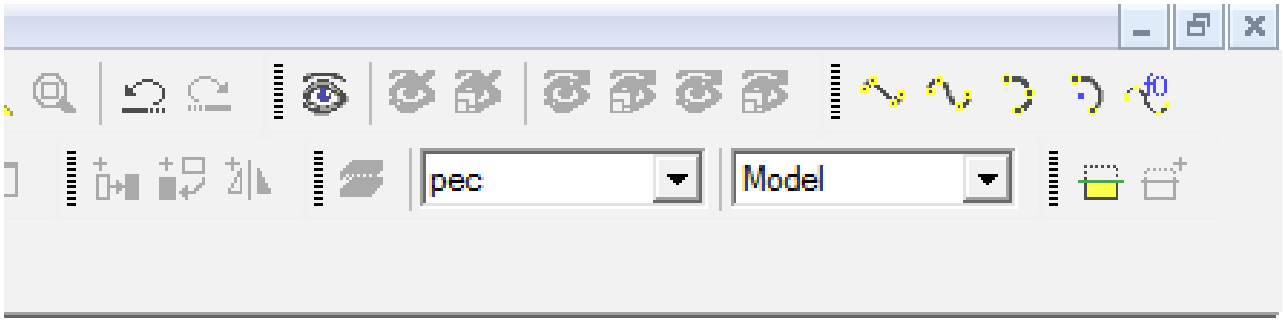
Creating the 3D model

type **pec** in the **Search by Name** field



Creating the 3D model

Click Ok and the default material has been changed to **pec**



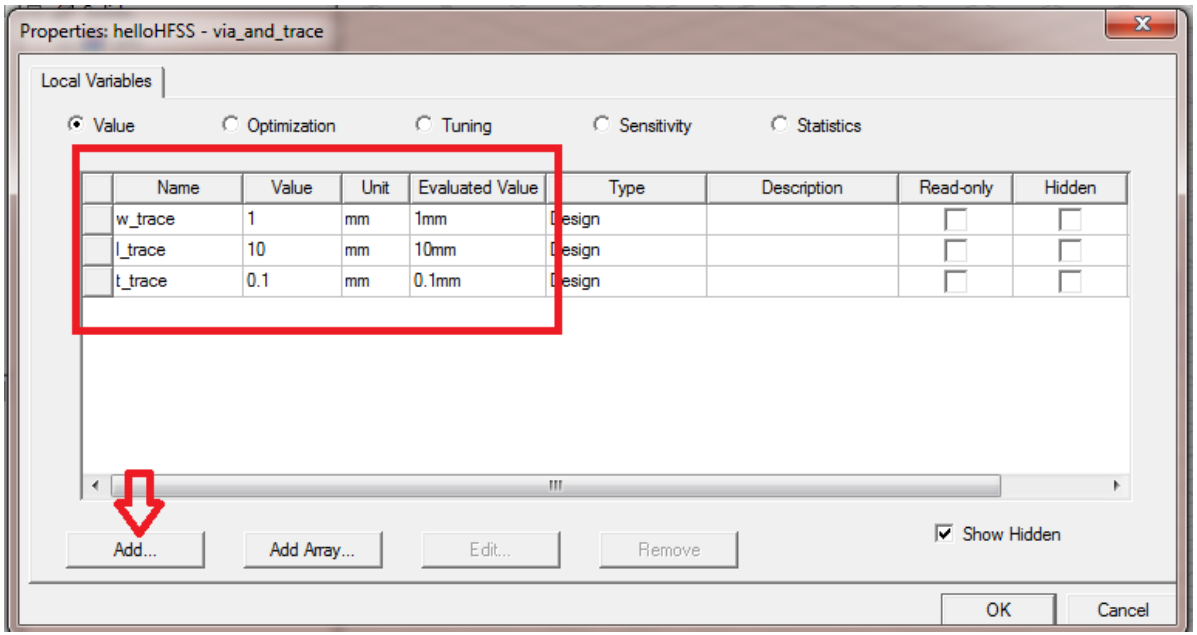
Create Trace

There are multiple ways to specify the position and dimensions of a trace, or any shape in general

- One way is to enter the information in the **coordinate entry fields**. Variables are not allowed in the coordinate entry fields.
- Another way is to draw an arbitrary box first and modify its position and dimensions with **variables**.
- Using variables is encouraged as it is easier for changes to be made on the designs.
- Here, we first create a box in the 3D modeler window and then redefine its position and dimensions with variables.

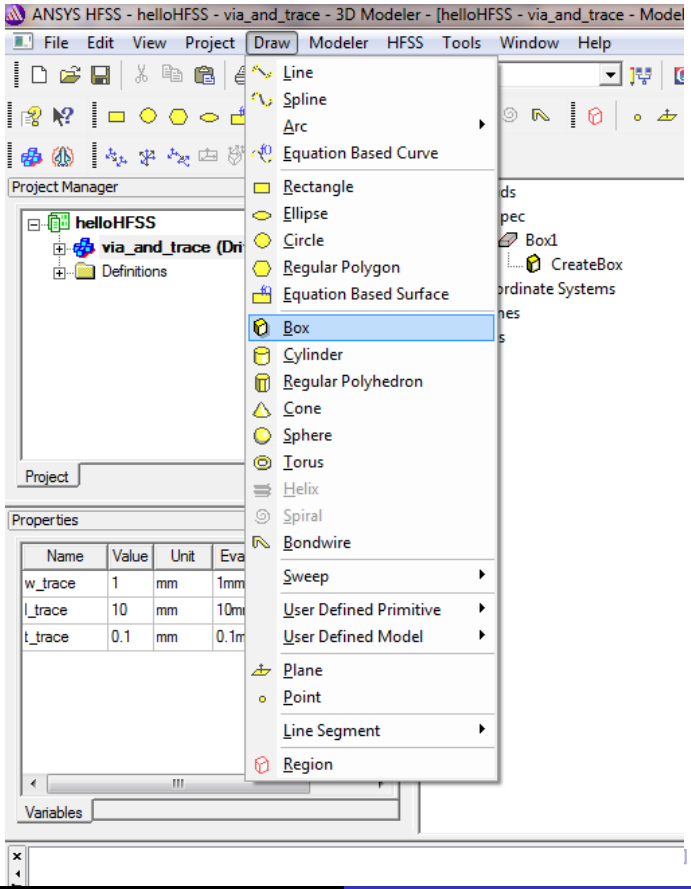
Create Trace

1. **Define variables**
   HFSS $\Rightarrow$ Design Properties $\Rightarrow$ add $w\_trace = 1$ mm, $l\_trace = 10$ mm, and $t\_trace = 0.1$ mm as local variables; w_trace, l_trace, and t_trace represent the width, length, and thickness of the trace, respectively



Create Trace

2. **Draw a box**
   Draw ⇒ Box

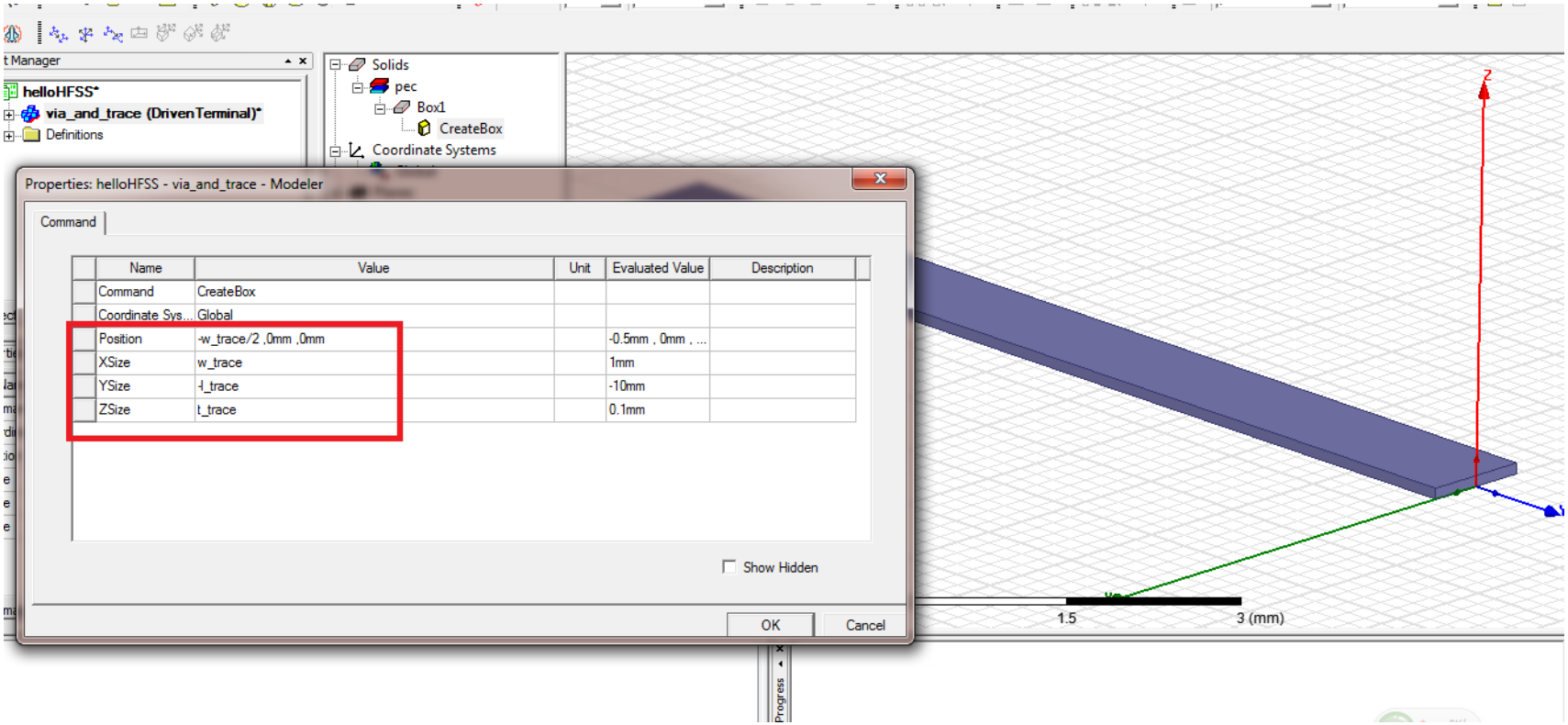


Create Trace

3. **Click three times in the 3D modeler window to create an arbitrary box**
    - first click to define the position of the box
    - second click to specify the lateral dimensions
    - third click to create the height of the box

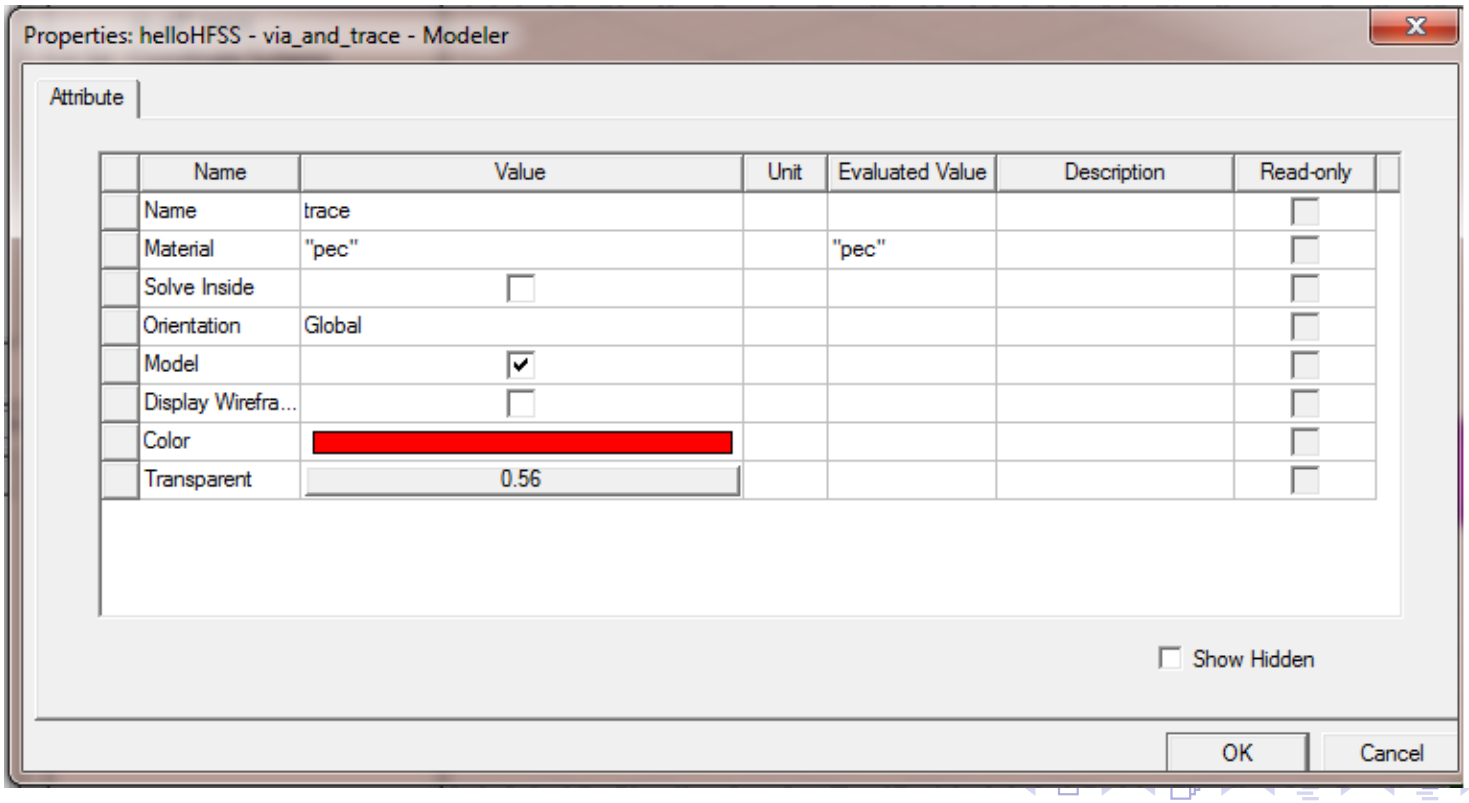
Create Trace

4. **Specify the dimensions and position with variables**



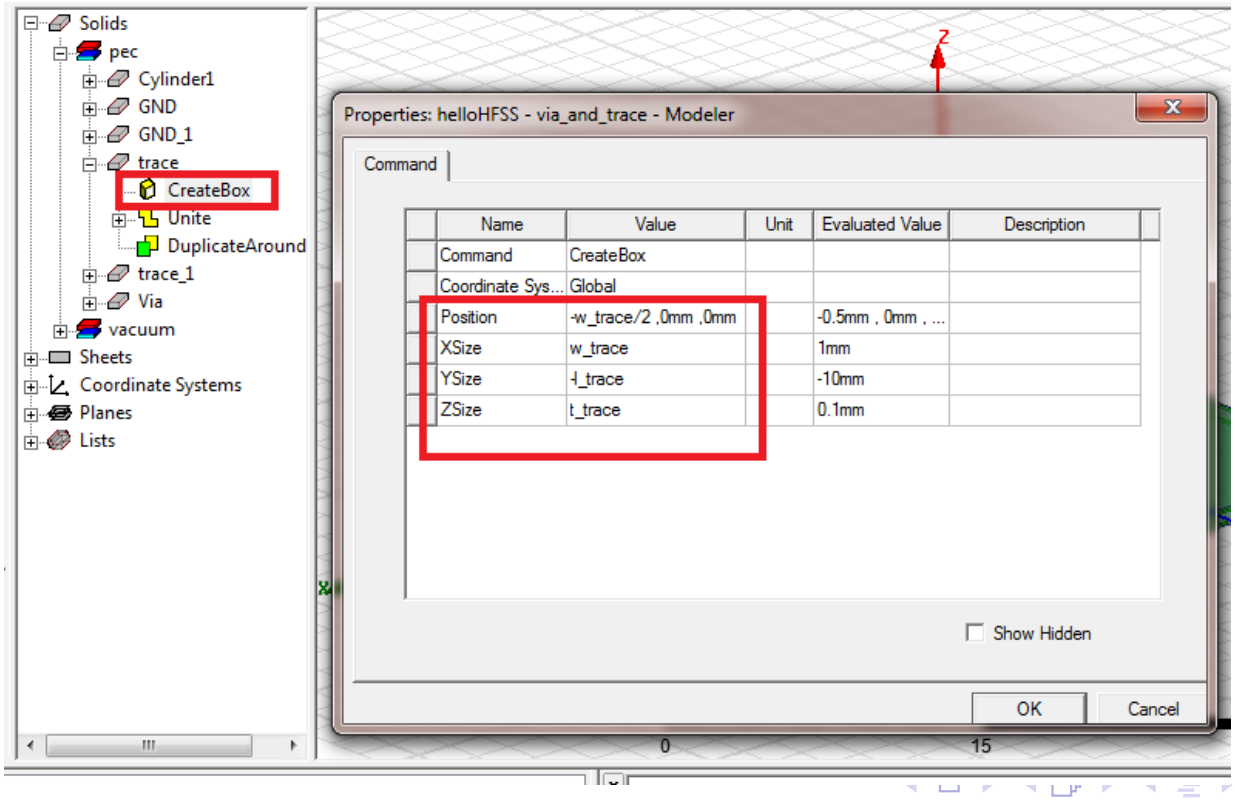
Create Trace

5. **Change the attributes**
   Double click on **Box1** for the attribute menu, change the name to **trace**, color to **red**, and set the transparency to 0.5.



Create Trace

6. **View the information of the created trace**
   Double click on **Create Box** under **trace**, one is able to see the Position, XSize, YSize, and ZSize of the created trace in the figure below



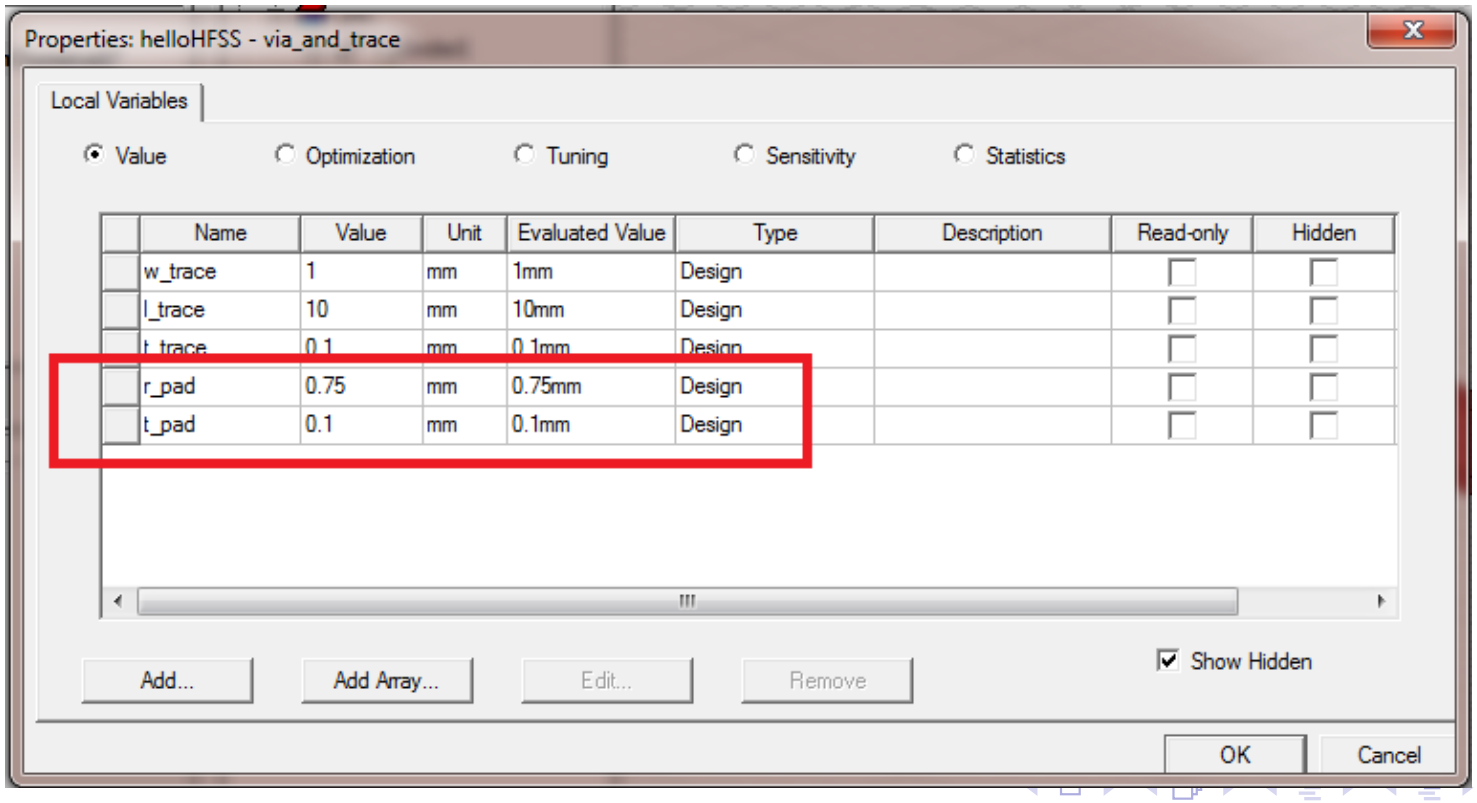
Create Trace

7. **Fit the view in the 3D modeler window**
   CTRL + D
8. **Done with the trace!**
   Try the two useful tricks in the 3D modeler window
   - Shift + Mouse left
   - Alt + Mouse left

   What did you find?

Create Via Pad

1. **Define variables**
   HFSS $\Rightarrow$ Design Properties $\Rightarrow$ add $r\_pad = 0.75$ mm, $t\_pad = 0.1$ mm, as the radius and the thickness of the via pad, respectively.

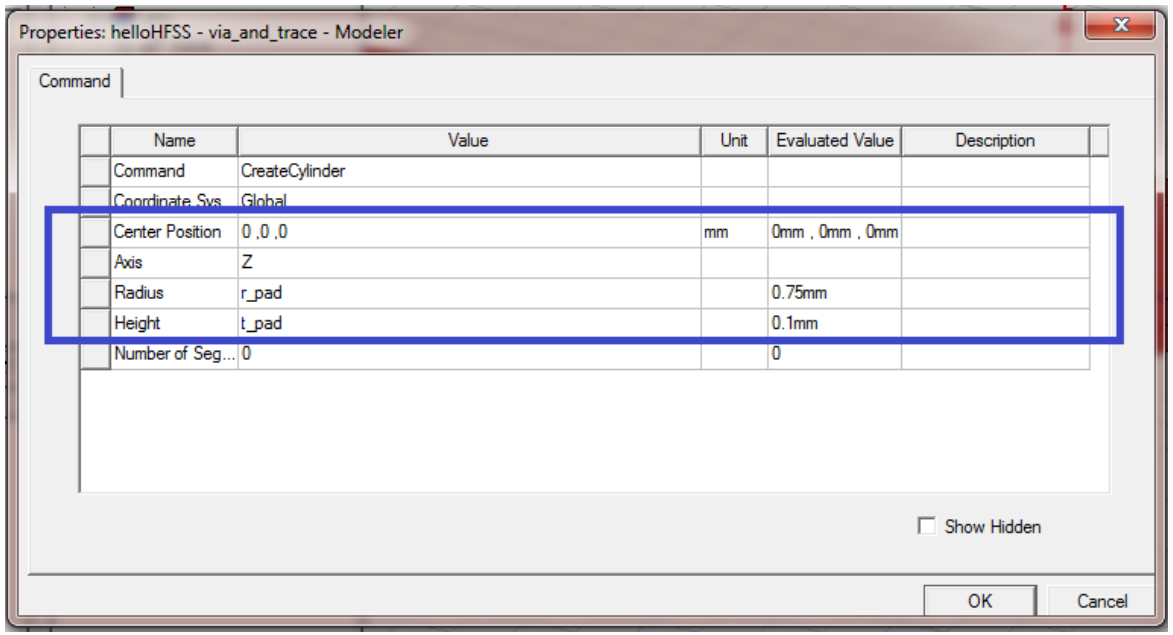


Create Via Pad

2. **Draw a cylinder**
   - Draw $\Rightarrow$ Cylinder
   - Again, three clicks in the 3D modeler window: the first click to specify the center of the cylinder, the second click to define the radius, and the third click for the height

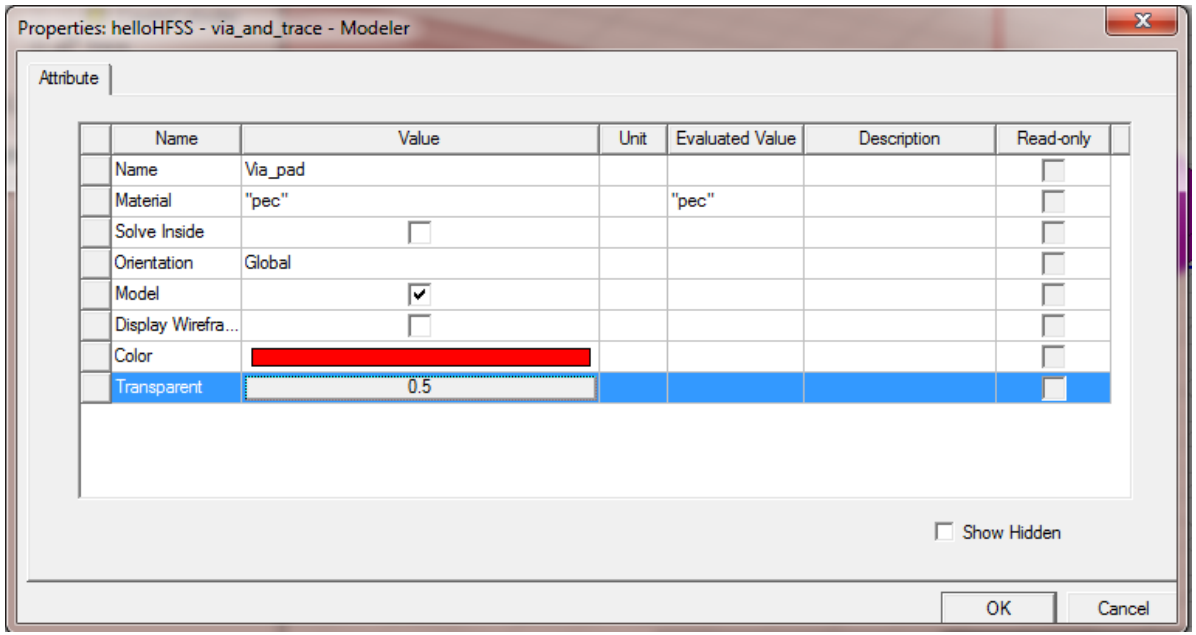
Create Via Pad

3. **Position and Dimensions**
   Enter the position and the dimensions of the created cylinder as shown in the figure below.



Create Via Pad

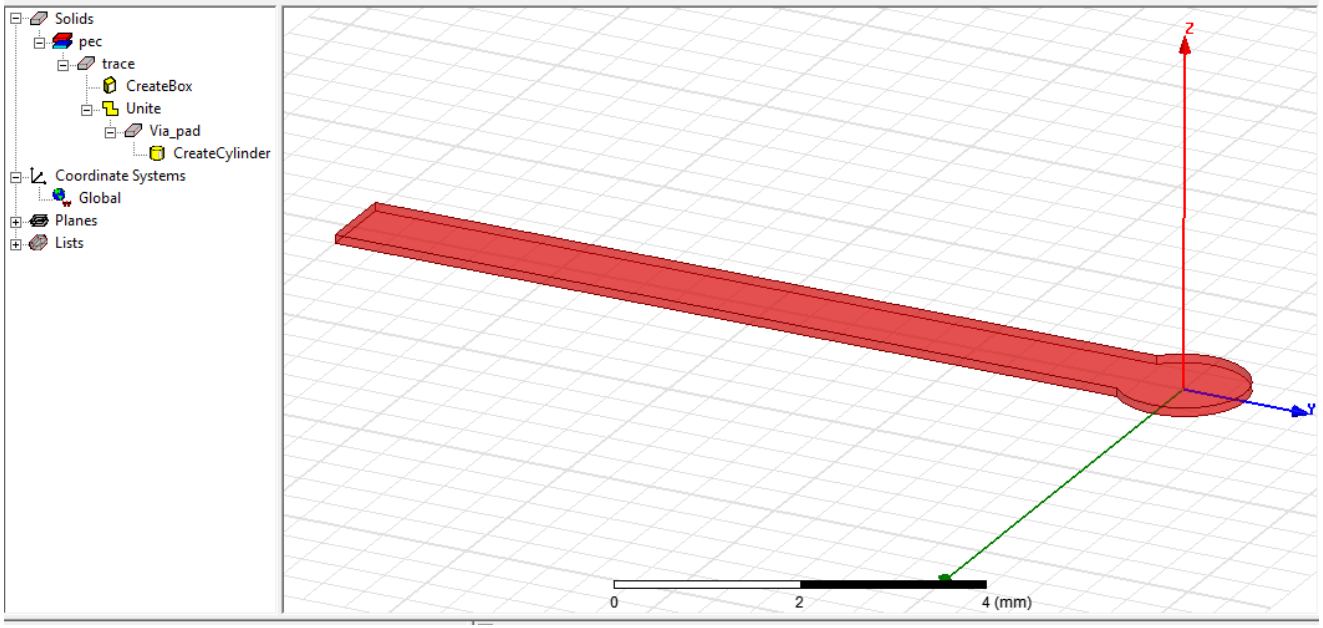
4. **Change the attributes**
   Double click on **Cylinder 1** for its attribute menu, change the name to *Via_pad* and color to **red**, and set the transparency to 0.5.



Create Via Pad

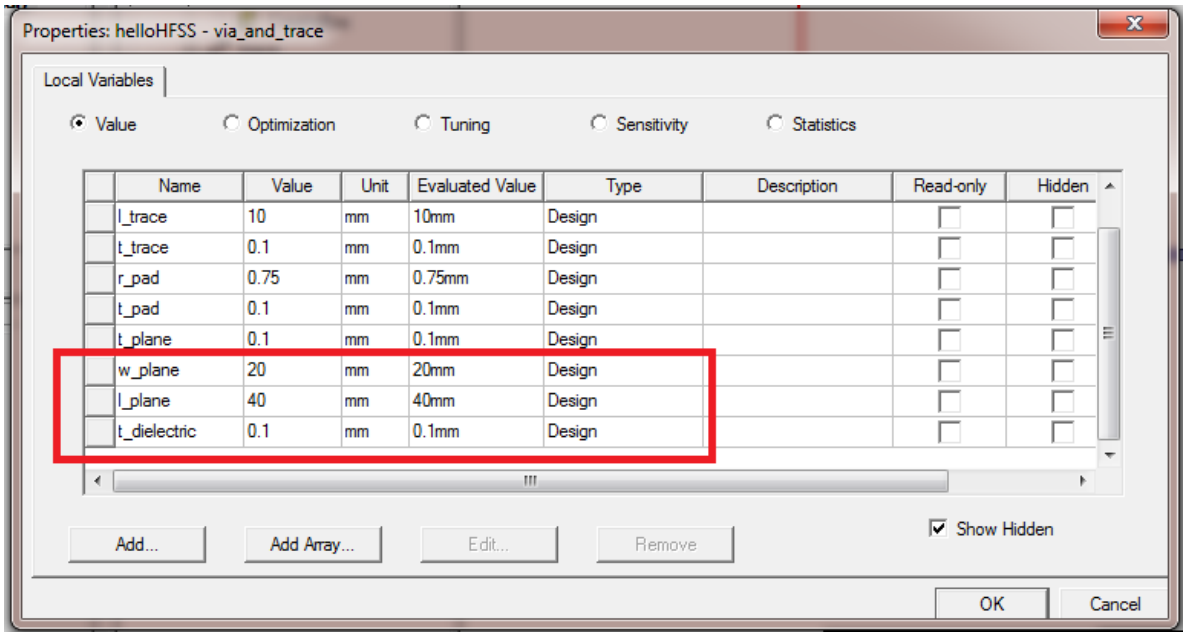
5. **Unite trace and pad**

   - Edit ⇒ Select All Visible or CTRL+A
   - Modeler ⇒ Boolean ⇒ Unite



Create Ground Plane

1. **Define variables**
   HFSS $\Rightarrow$ Design Properties $\Rightarrow$ add *t_plane* $= 0.1$ mm, *w_plane* $= 20$ mm, *l_plane* $= 40$ mm, and *t_dielectric* $= 0.1$ mm

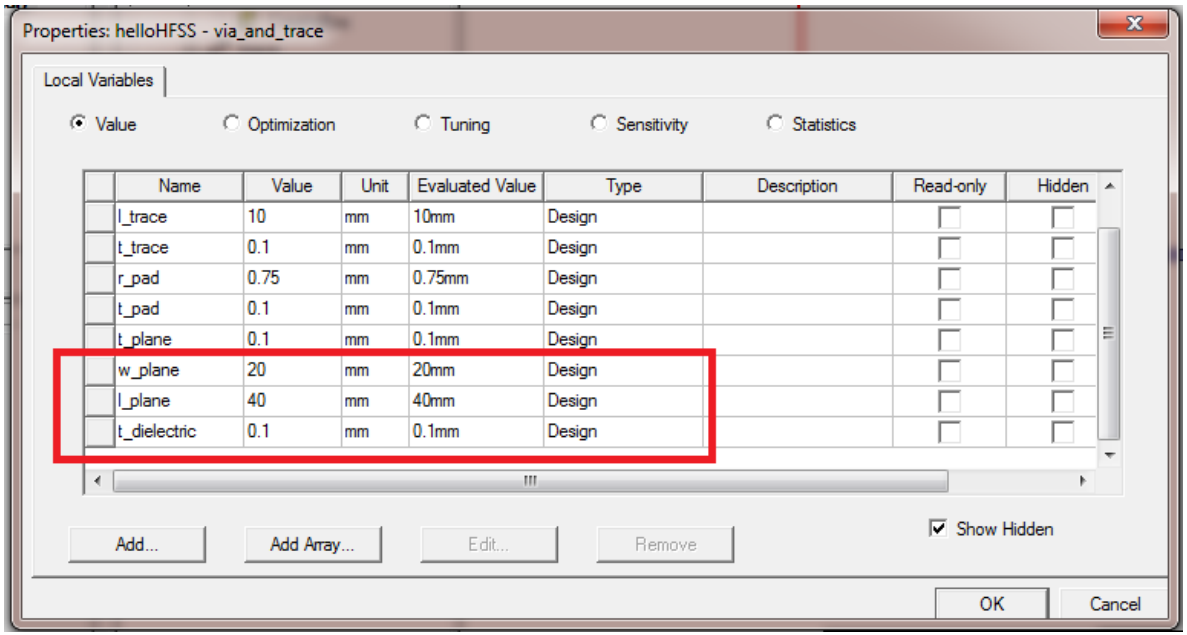


Create Ground Plane

- *t_plane*, *w_plane*, and *l_plane* denote the thickness, the width, and the length of the plane, respectively;
- *t_dielectric* denotes the thickness of the dielectric layer between the trace and its neighboring ground;
- the lateral dimensional of the dielectric layer is chosen as the same as that of the ground plane.

Create Ground Plane

2. **Draw a box**
   Draw ⇒ Box

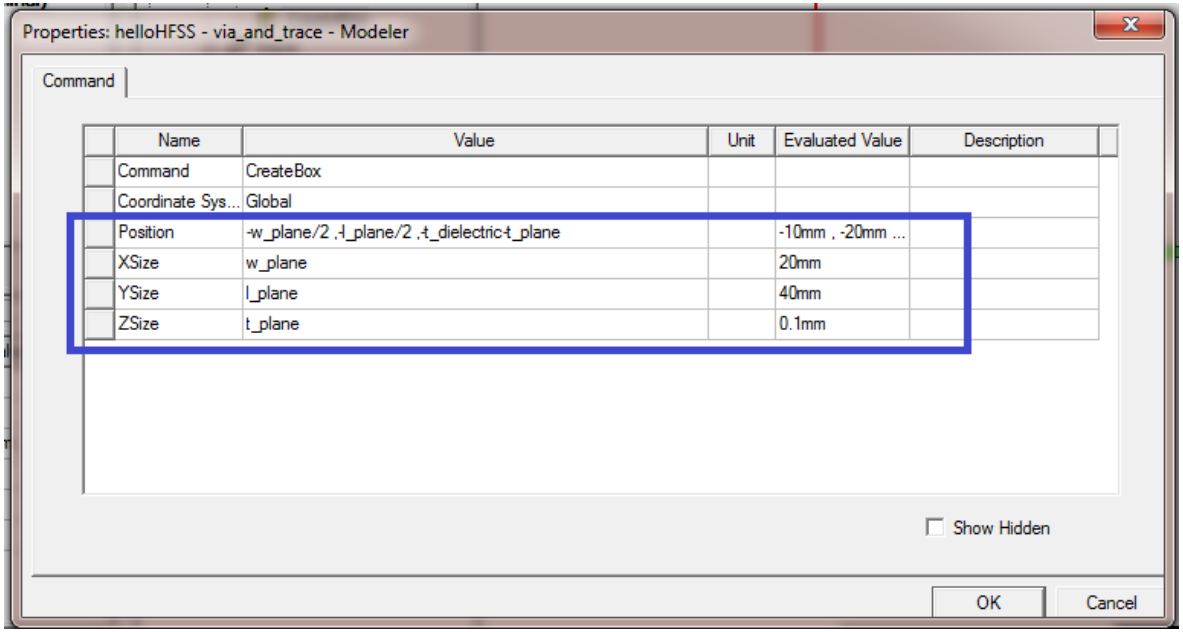


Create Ground Plane

3. **Similar to creating the trace, click three times in the 3D modeler window for an arbitrary box**
   - first click to define the position of the box
   - second click to specify the lateral dimensions
   - third click to create the height of the box

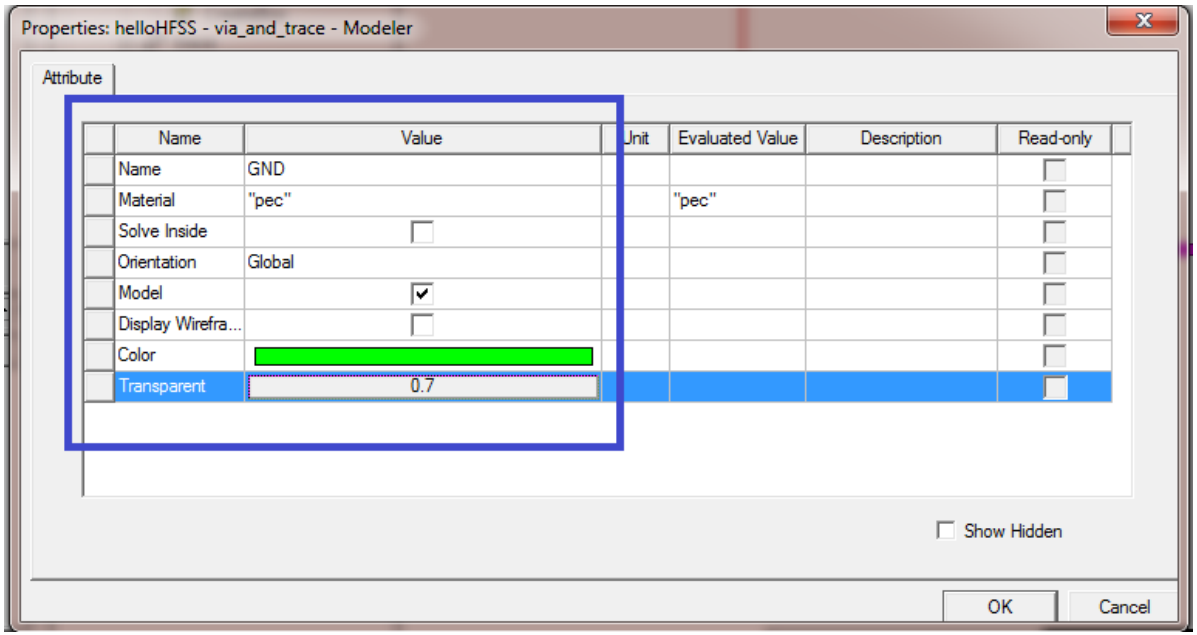
Create Ground Plane

4. **Specify the dimensions and position with variables**



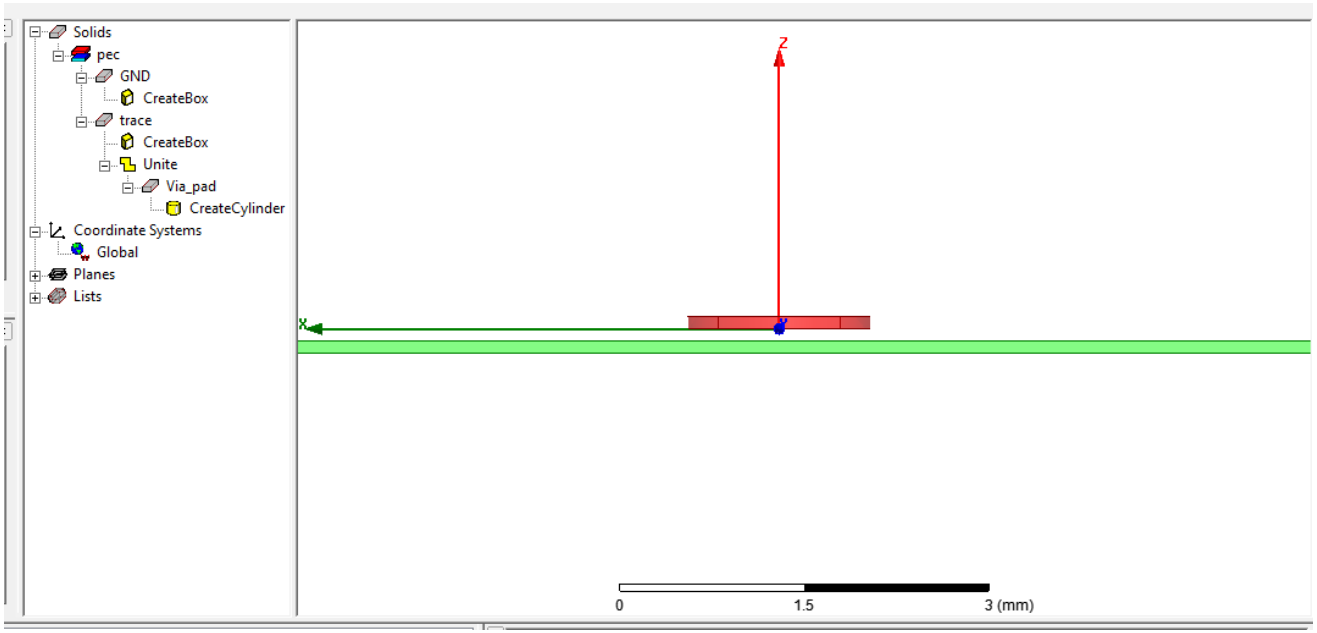
Create Ground Plane

5. **Change the attributes**
   Double click on **Box 1**, rename it as **GND**, set the color to **green**, and modify the transparency to 0.7.



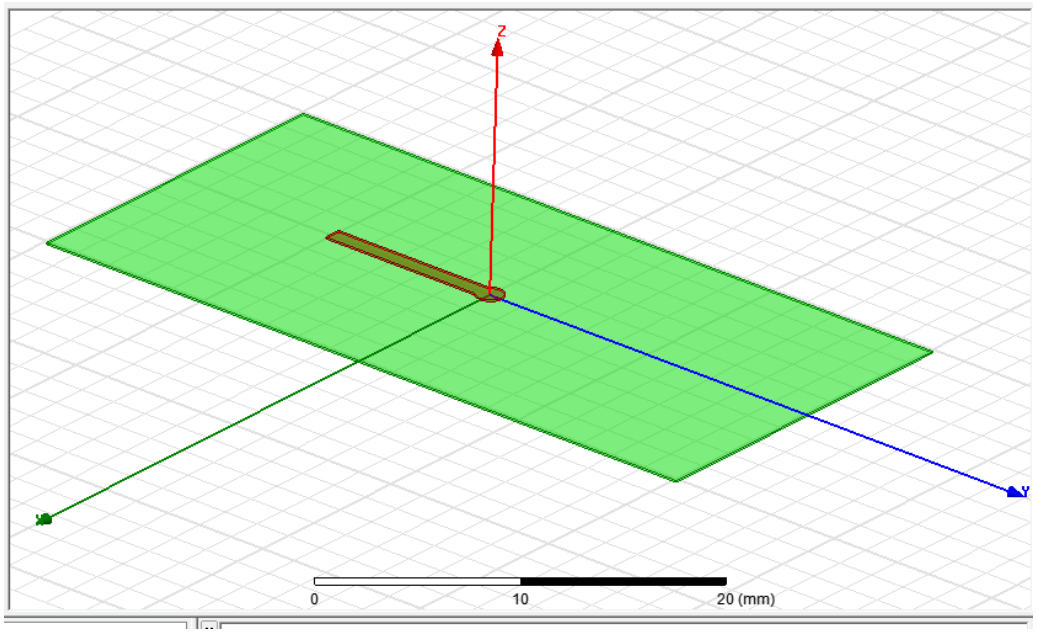
Create Ground Plane

Side-view of the ground plane and the trace(pad)



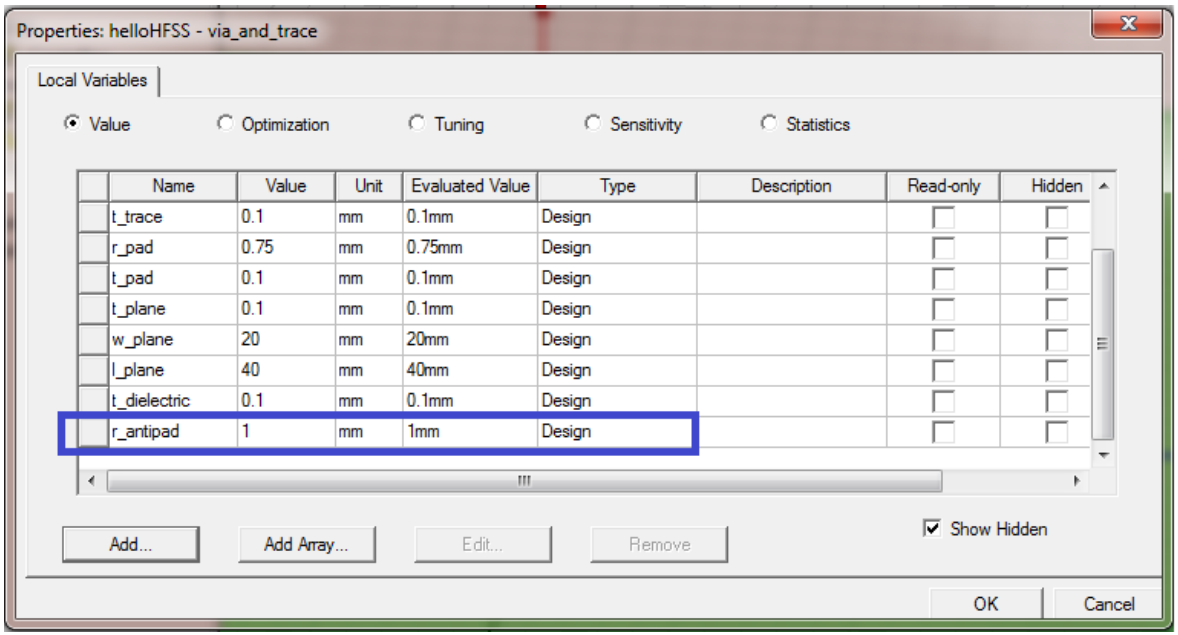
Create Ground Plane

3D-view of the ground plane and the trace(pad)



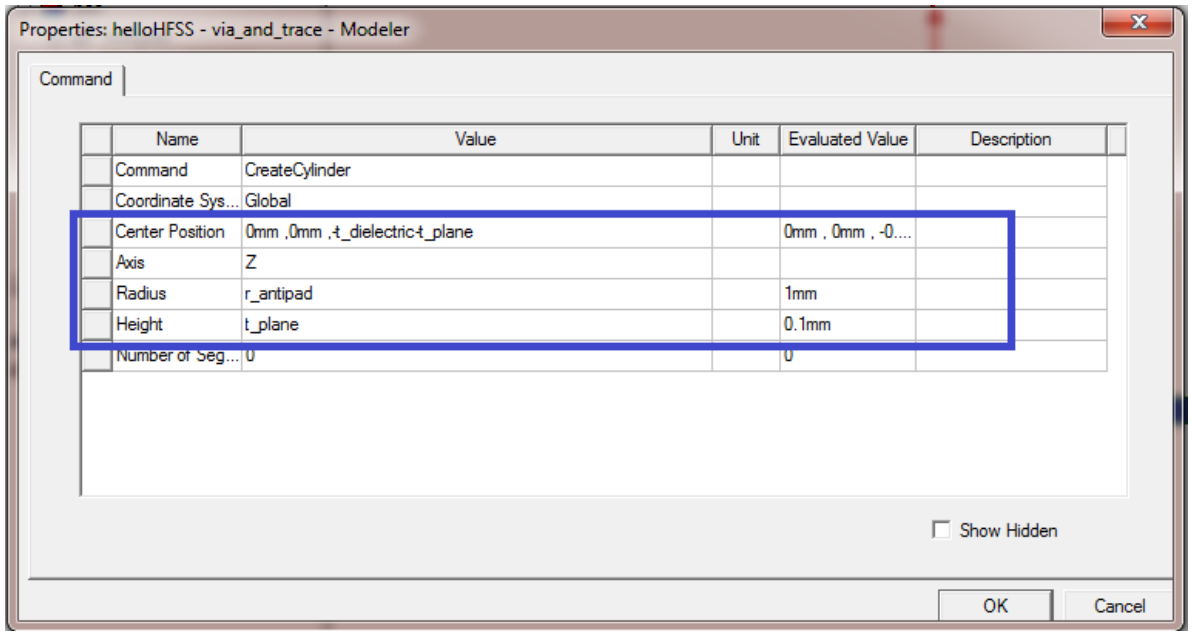
Create Anti-Pad

1. **Define variables**
   HFSS $\Rightarrow$ Design Properties $\Rightarrow$ add *r_antipad* $= 1$ mm, the height of the anti-pad should be identical to the thickness of the ground plane



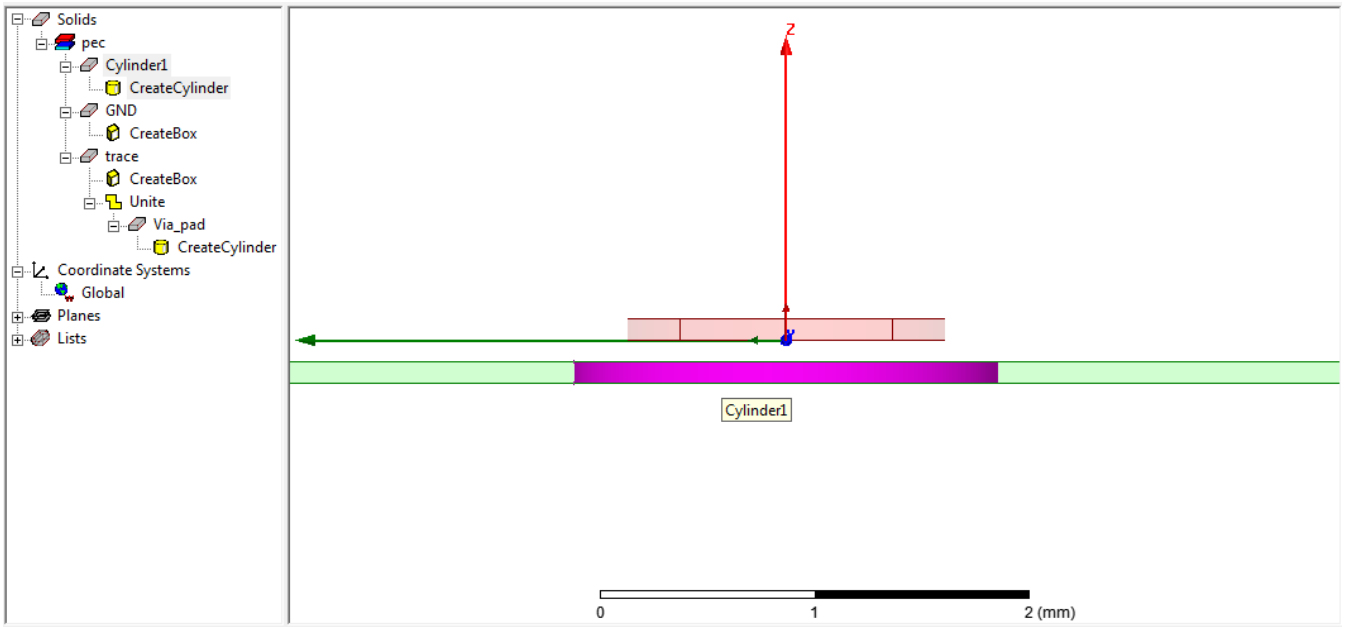
Create Anti-Pad

2. **Draw a cylinder**
   Draw ⇒ Cylinder and specify the dimensions with variables.



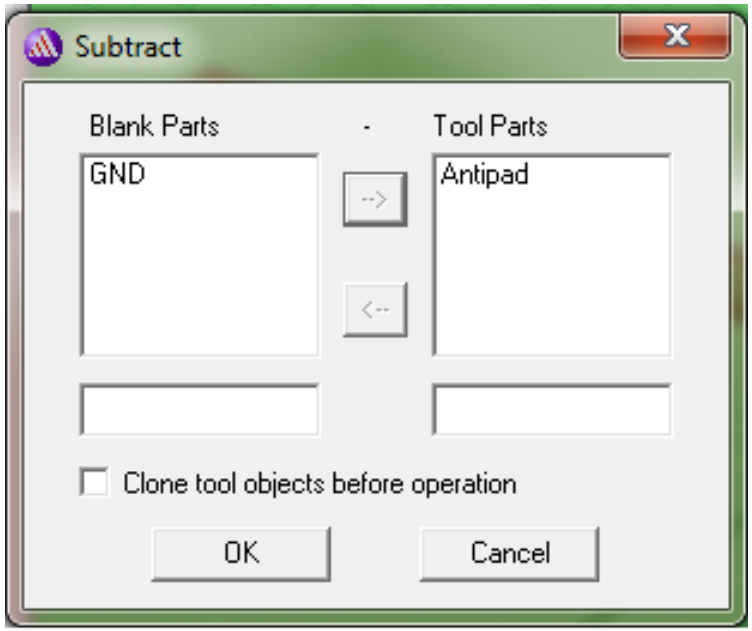
Create Anti-Pad

3. **Change the attributes**
   Double click on **Cylinder 1** for its attribute and rename it as **Antipad**

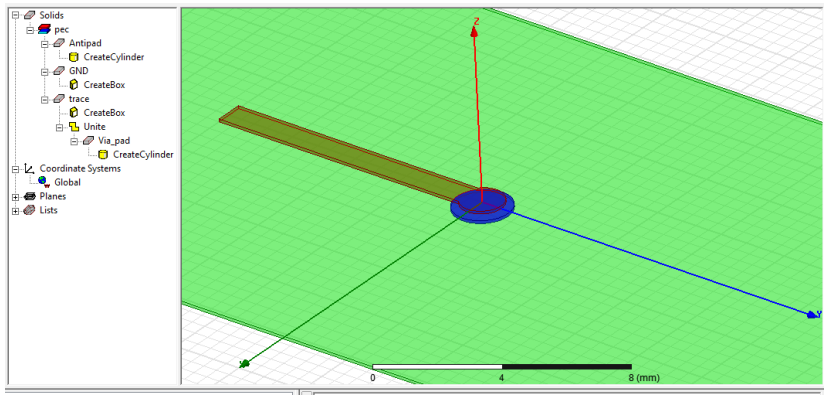


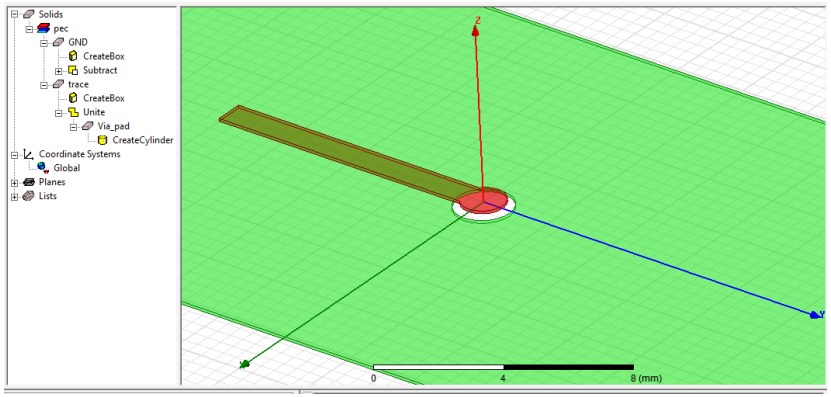
Create Anti-Pad

4. **Boolean Operation**
   - Edit ⇒ Select ⇒ By Name ⇒ select GND and Antipad
   - Modeler ⇒ Boolean ⇒ Subtract ⇒ Bland parts: GND; Tool parts: Antipad ⇒ Click OK



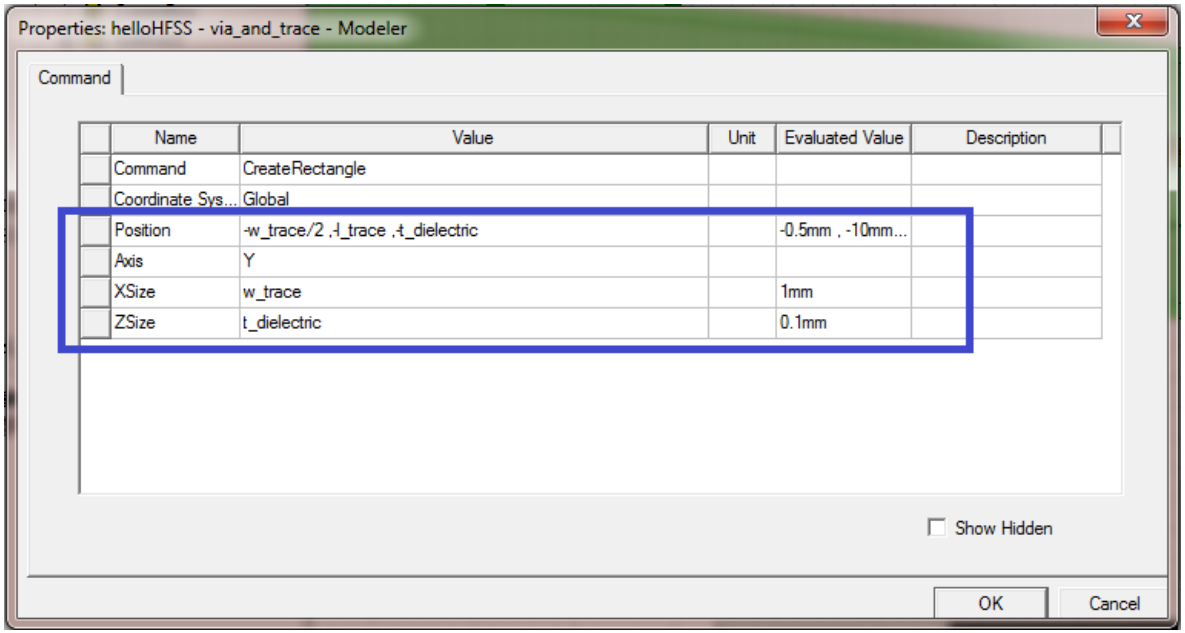
Create Anti-Pad





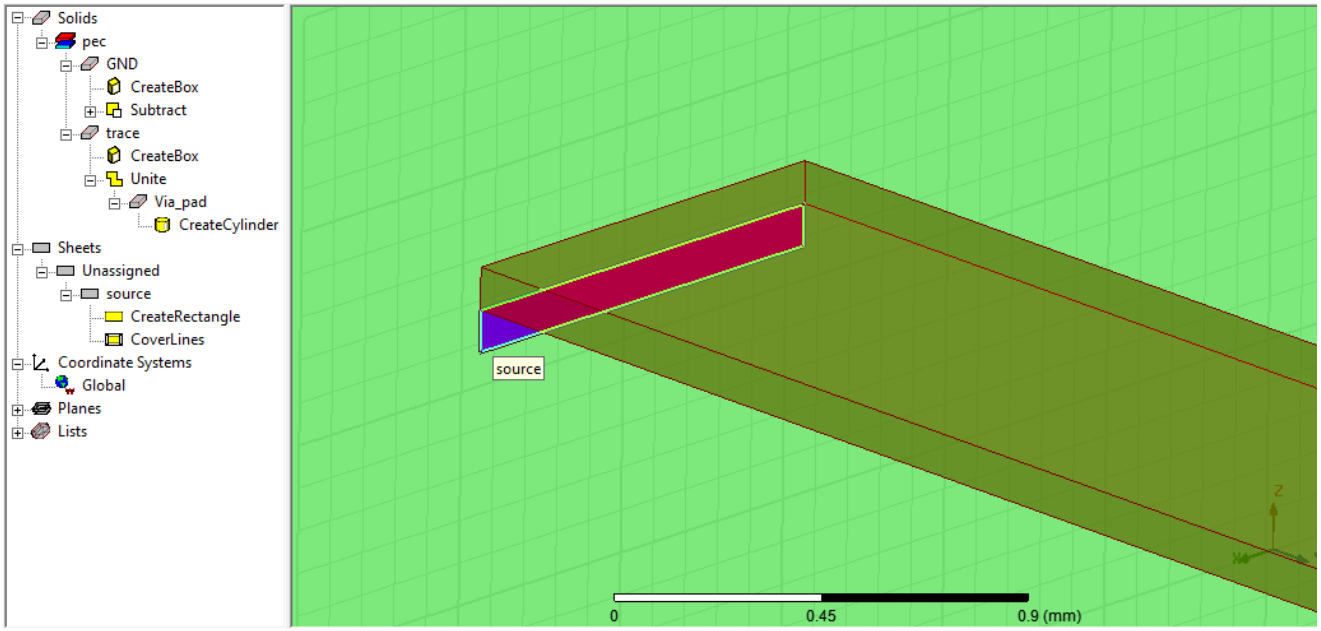
Assign Excitation

1. **Draw Rectangle**
   - Modeler ⇒ Grid Plane ⇒ XZ
   - Draw ⇒ Rectangle
   - Specify the dimensions as the figure below



Assign Excitation

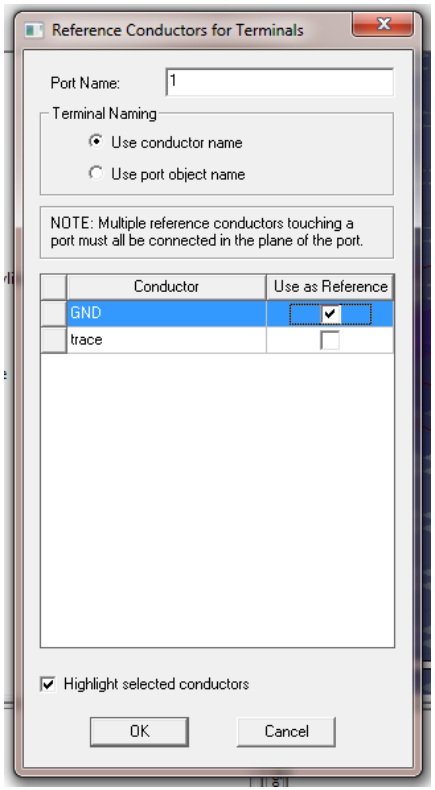
2. **Rename as source**
   Double click on **Rectangle 1** and rename it as **source**



Assign Excitation

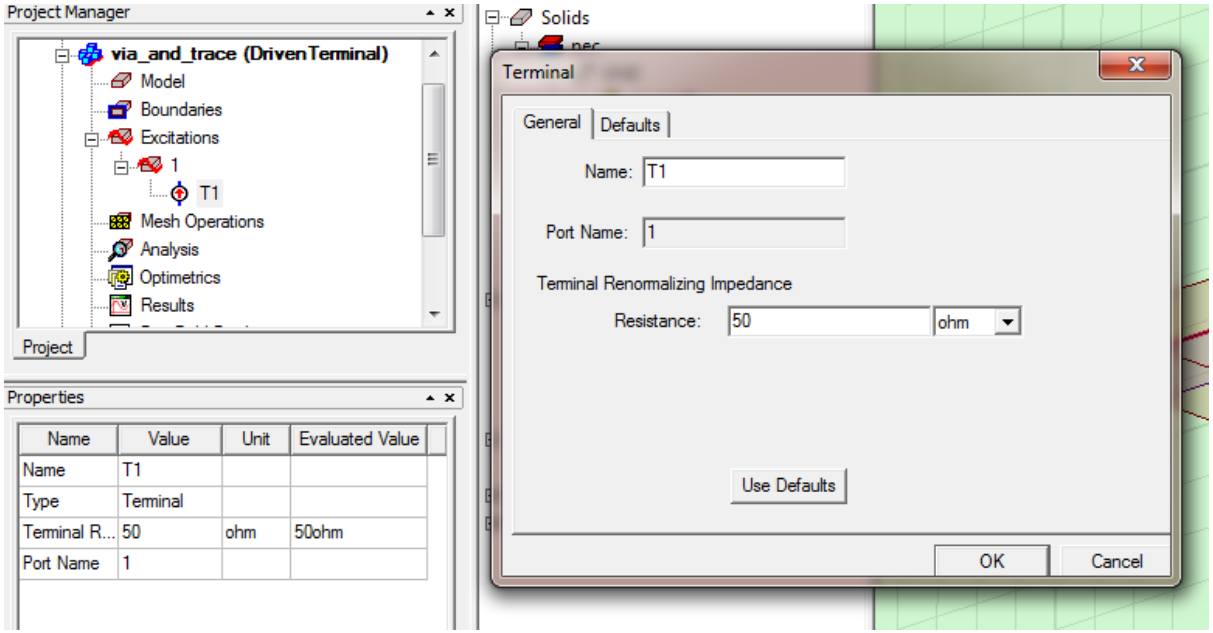
3. **Assign lumped port**
   - Edit ⇒ Select ⇒ By Name ⇒ select **source** ⇒ HFSS ⇒ Excitations ⇒ Assign ⇒ Lumped port
   - Set the lumped port as shown in the picture:
     Port name to 1;
     select conductor **GND** and check ✓**Use as Reference**;
     check ✓**Highlight selected conductors** ⇒ Click OK

Assign Excitation



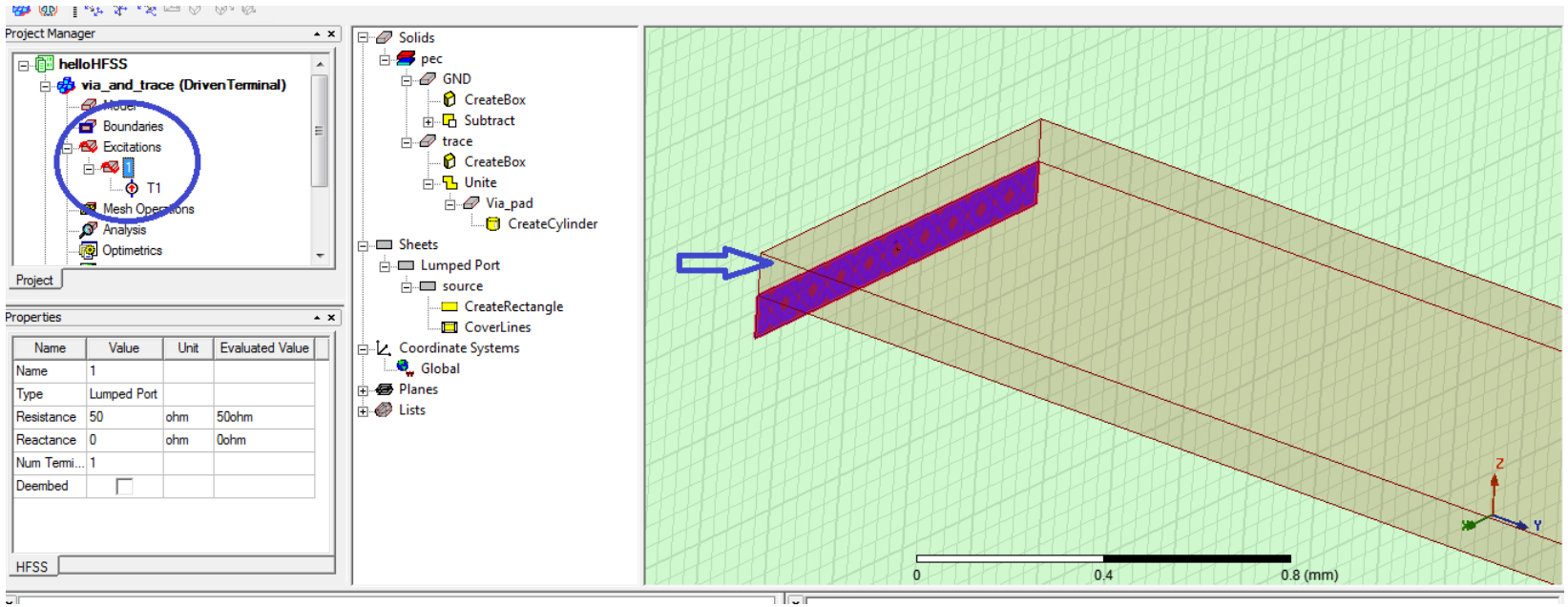
Assign Excitation

4. **Renormalizing impedance**
   - In the Project Manager, expand Excitations and rename *trace_T1* as **T1**
   - The renormalizing impedance of the lumped port is by default 50 Ohms



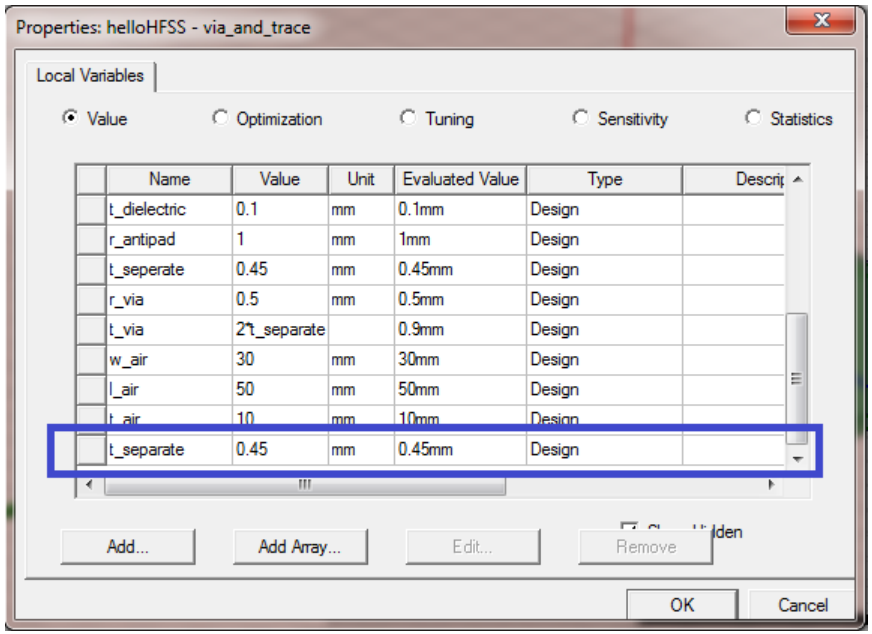
Assign Excitation

5. **Highlight the lumped port**
   Again under Excitations in the Project Manager, if you click on 1, the lumped port is highlighted.



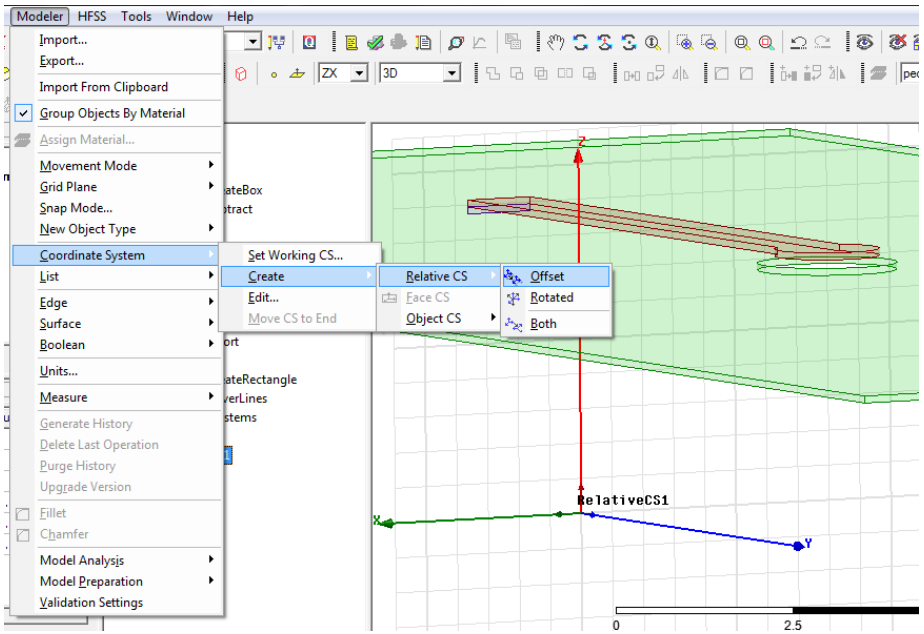
Create Multilayers by Duplication

1. **Define variables**
   HFSS $\Rightarrow$ Design Properties $\Rightarrow$ add $t\_separate = 0.45$ mm



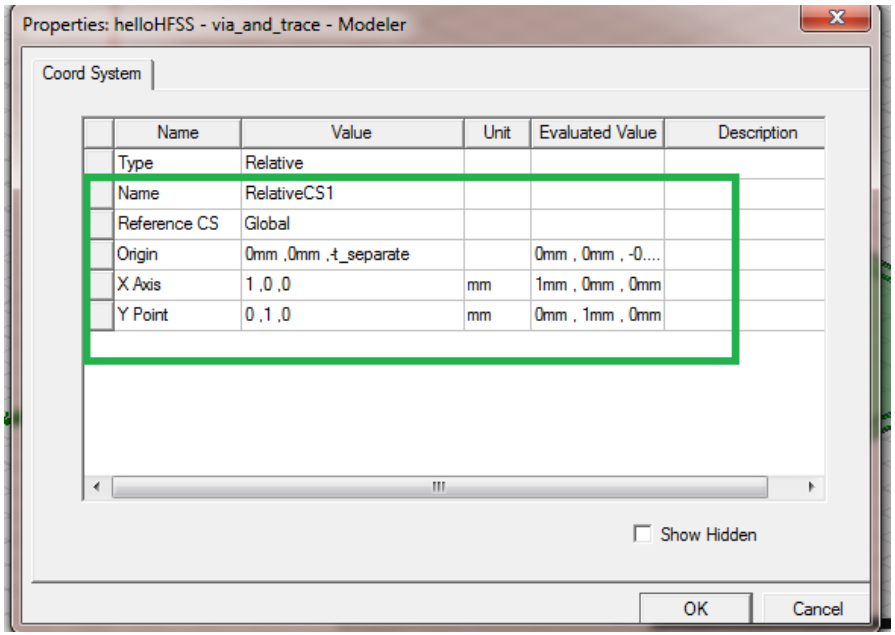
Create Multilayers by Duplication

2. **Create Offset Coordinate System**
   Modeler ⇒ Coordinate System ⇒ Create ⇒ Relative CS ⇒ Offset



Create Multilayers by Duplication

3. **Set the Offset Coordinate System**
   Click on the 3D modeler window and set the RelativeCS1 as shown in the figure

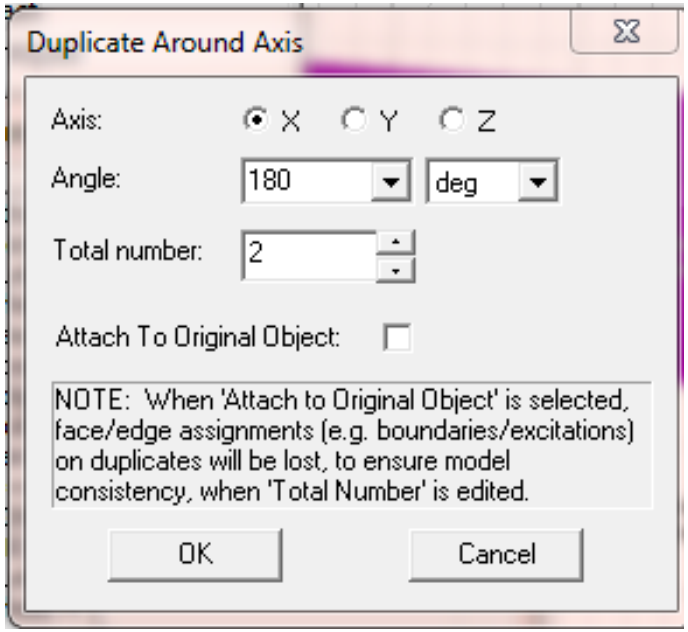


Create Multilayers by Duplication

4. **Perform the Duplication**
   - Edit $\Rightarrow$ Select All Visible or CTRL + A
   - Modeler $\Rightarrow$ Coordinate System $\Rightarrow$ Set Working CS $\Rightarrow$ RelativeCS1
   - Edit $\Rightarrow$ Duplicate $\Rightarrow$ Around Axis
   - Set the duplication parameters as shown in the picture in the next page

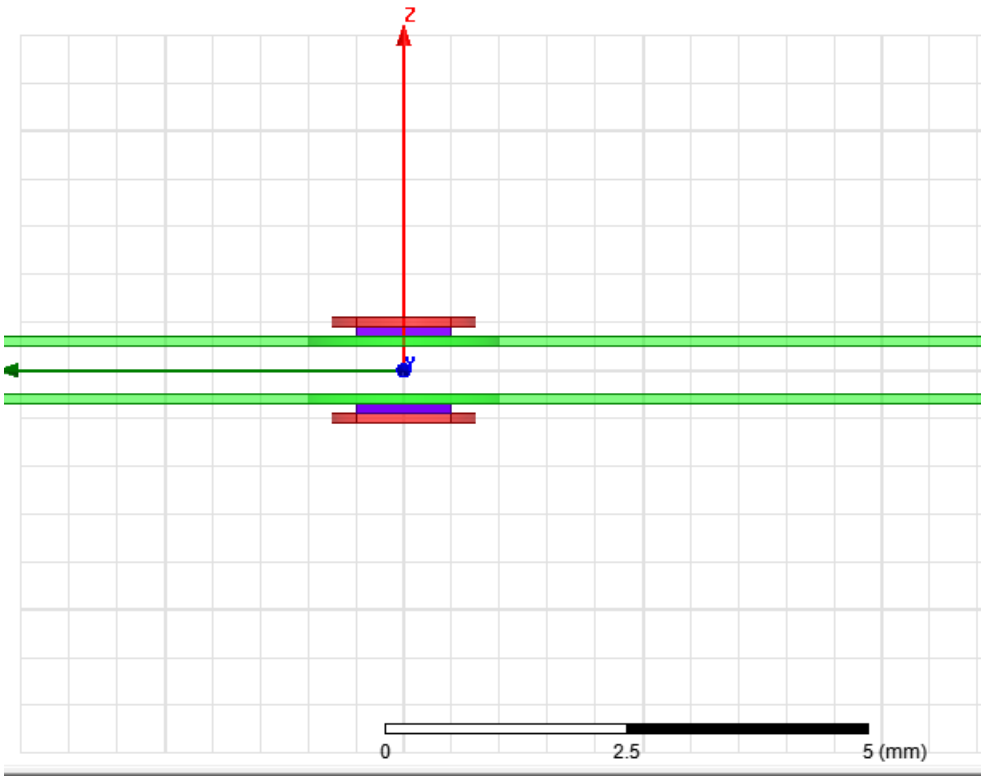
Create Multilayers by Duplication

4. **Perform the Duplication**



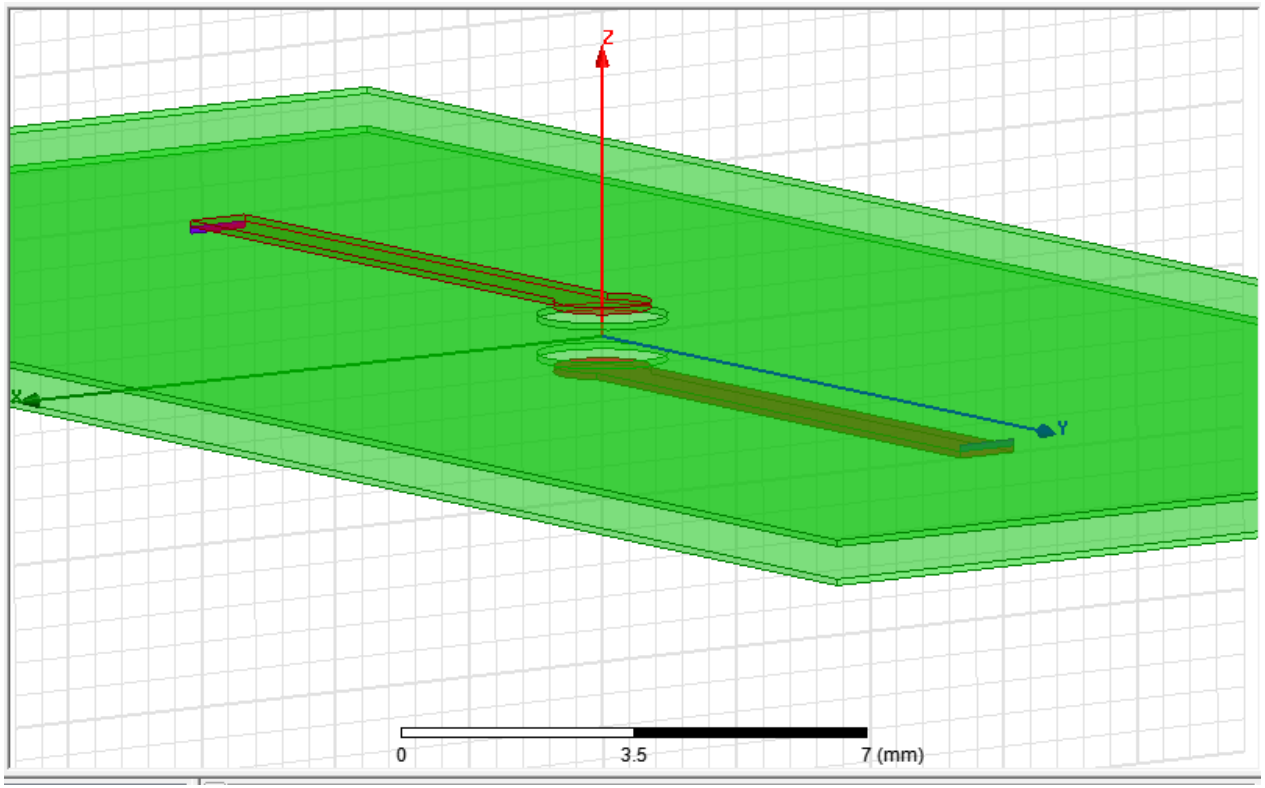
Create Multilayers by Duplication

Side-view of the multilayered structure



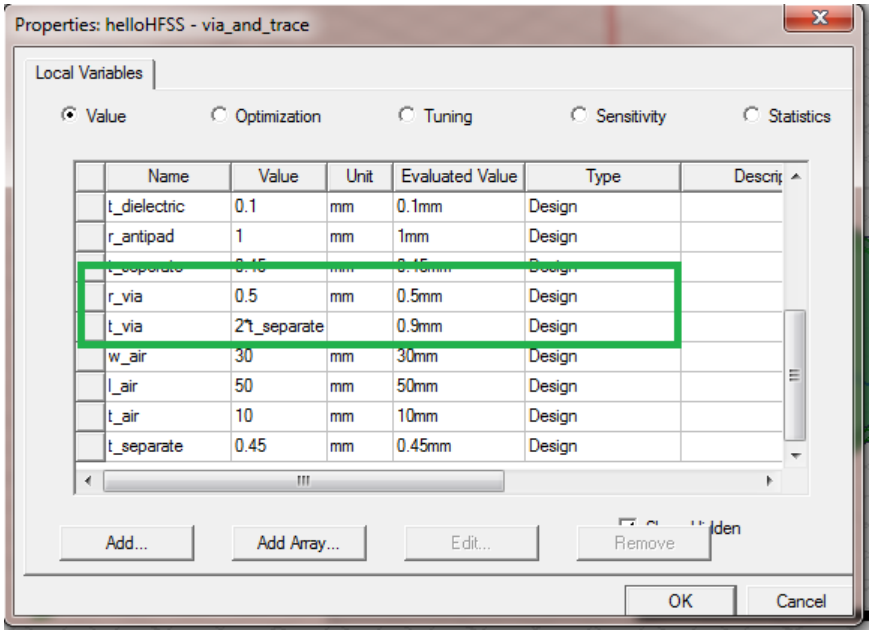
Create Multilayers by Duplication

3D-view of the multilayered structure



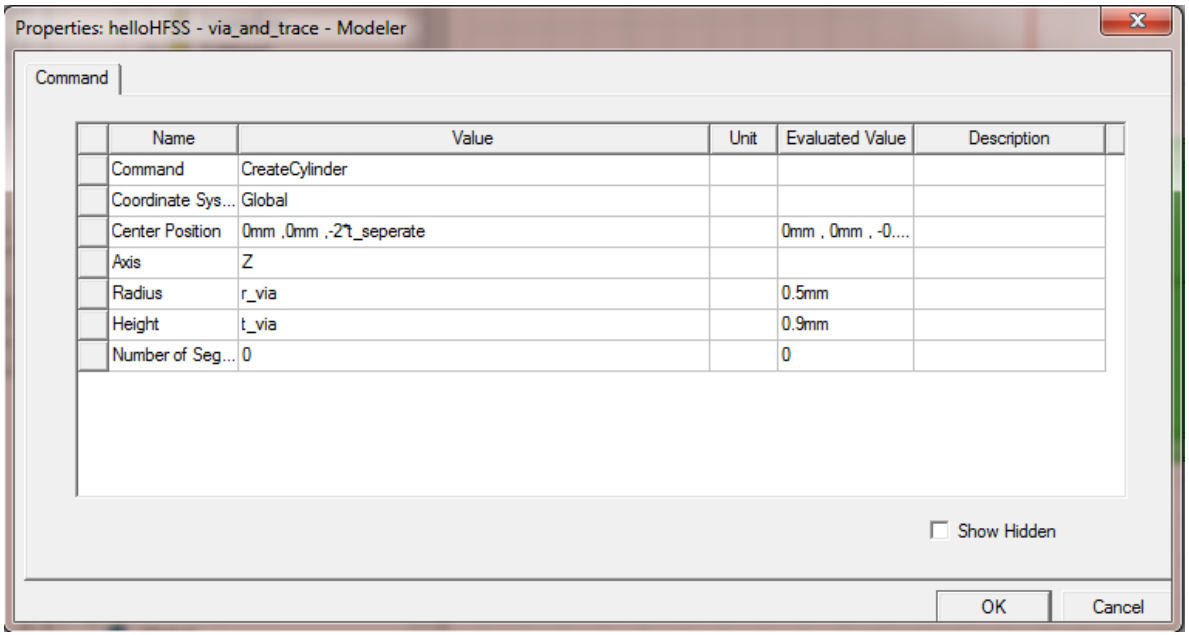
Create Via

1. **Define variables**
   - Modeler ⇒ Coordinate System ⇒ Set Working CS ⇒ RelativeCS1
   - HFSS ⇒ Design Properties ⇒ add $t\_via = 2 * t\_separate$ and $r\_via = 0.5$ mm



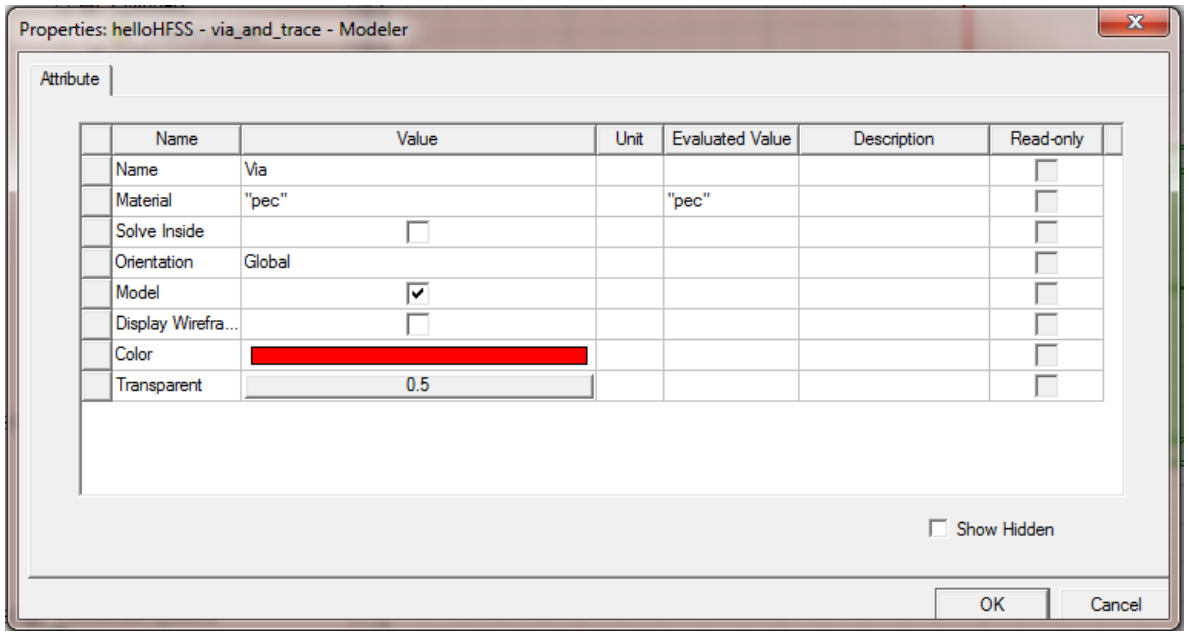
Create Via

2. **Draw a cylinder**
   Draw ⇒ Cylinder and specify the dimensions and position as the figure below



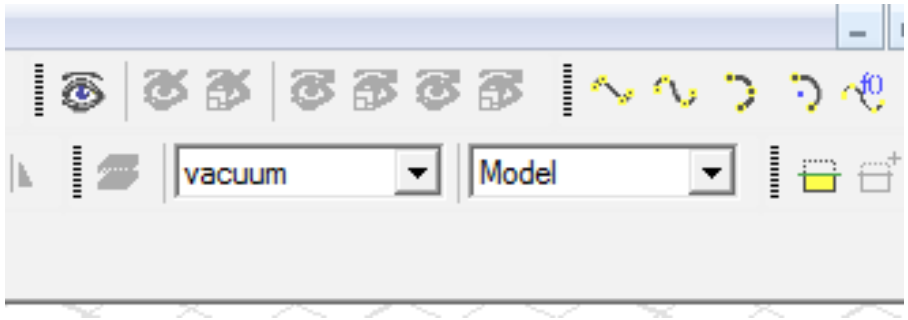
Create Via

3. **Change attributes**
   Double click on Cylinder 1 and rename it as **Via**



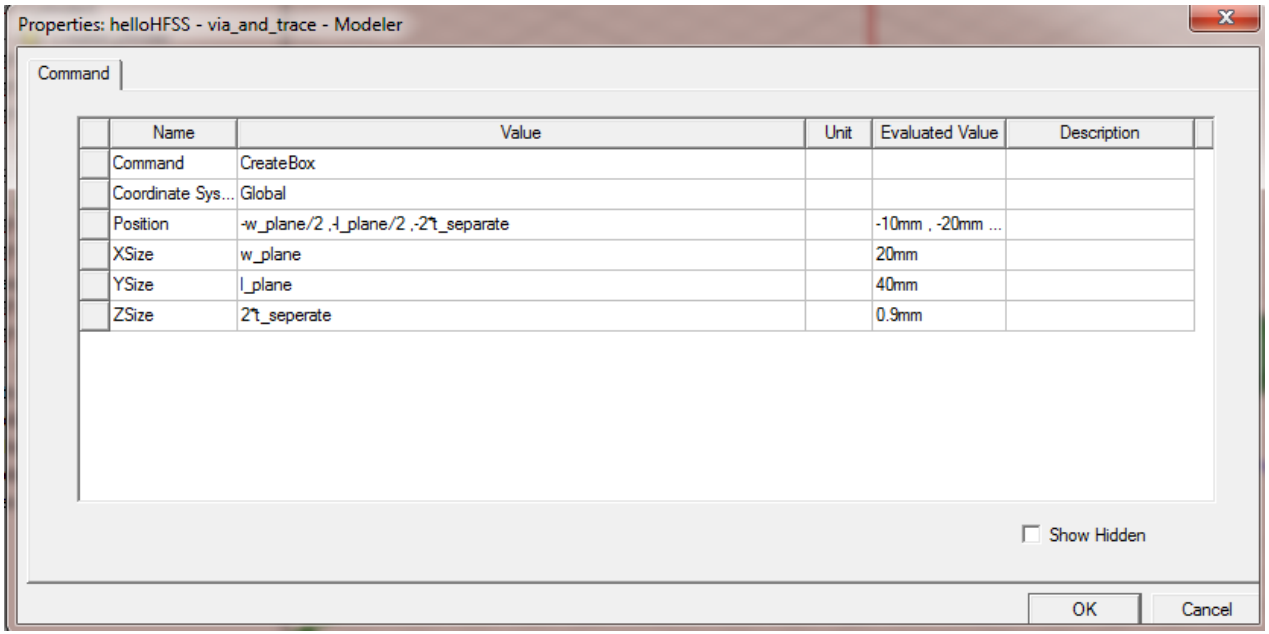
Create Dielectric Layer

1. **Select material**
   Use 3D modeler materials toolbar, choose vacuum



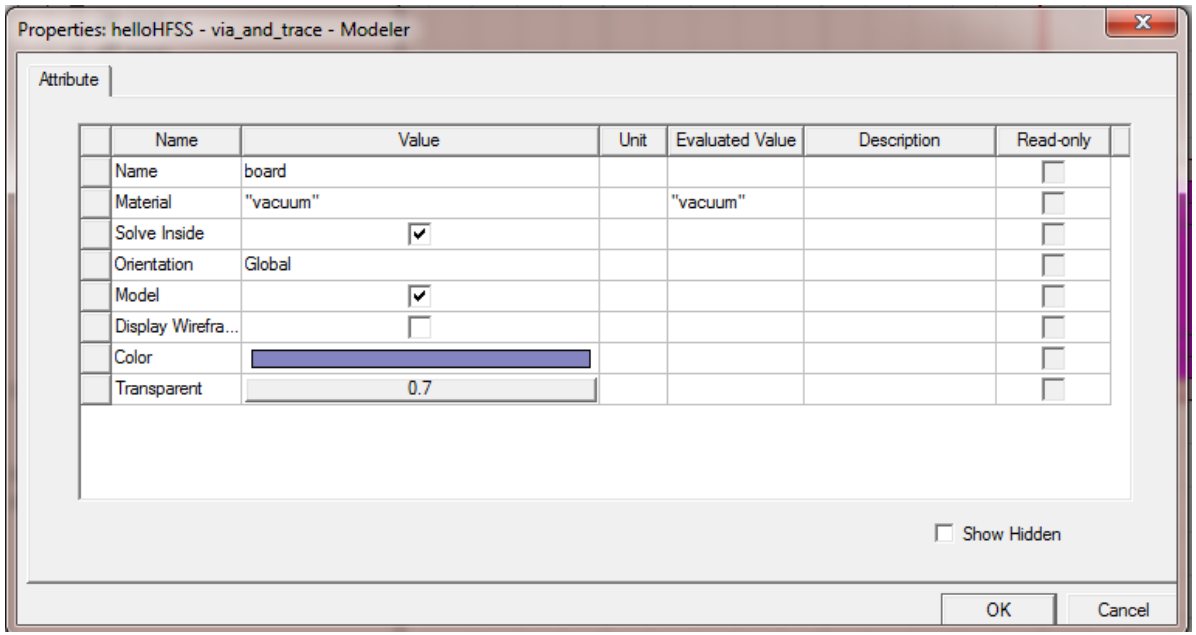
Create Dielectric Layer

2. **Draw Box**
   Draw ⇒ Box and specify the dimensions and position as the figure below



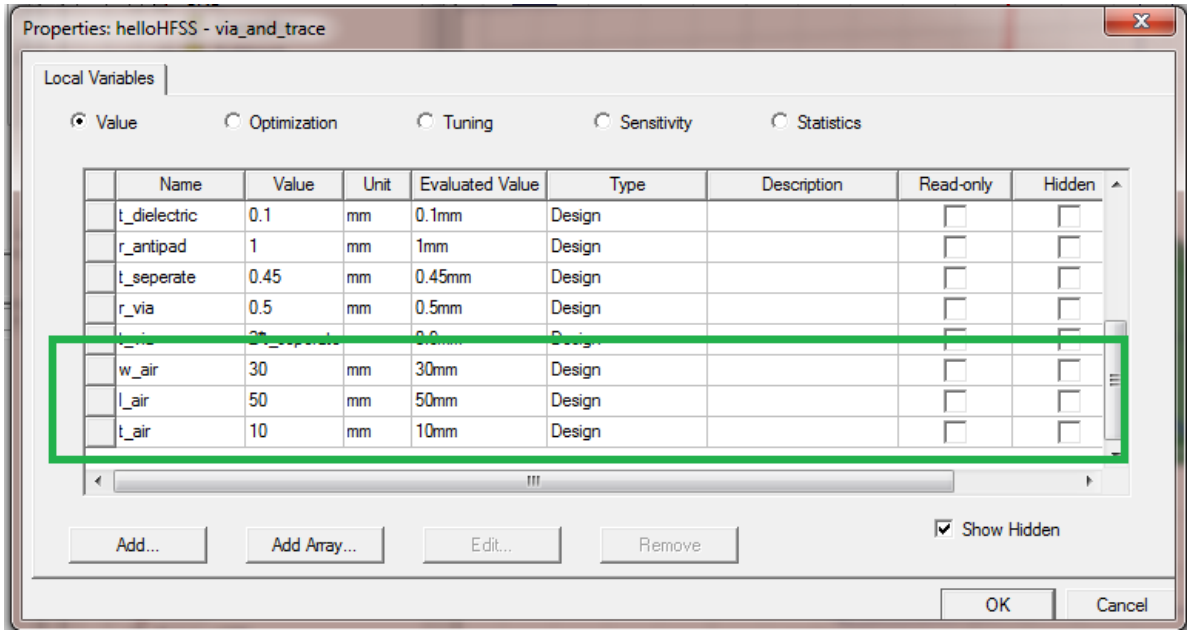
Create Dielectric Layer

3. **Change attributes**
   Rename it as **board**



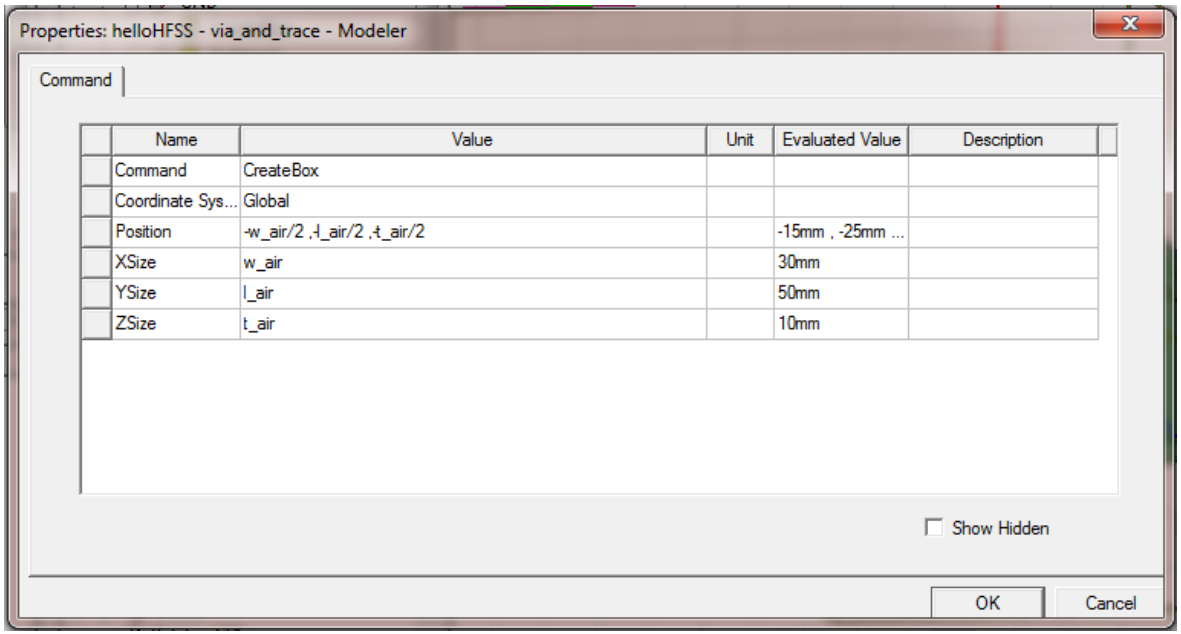
Create Airbox

1. **Define variables**
   Define $w\_air = 30$ mm, $l\_air = 50$ mm, and $t\_air = 10$ mm



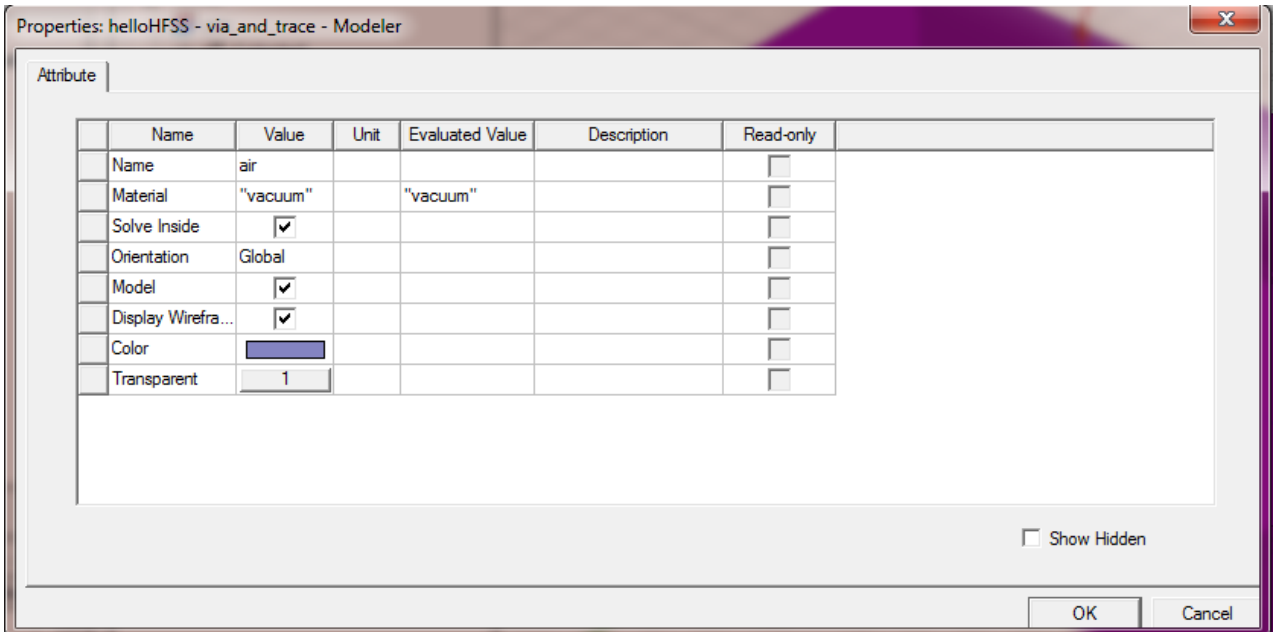
Create Airbox

2. **Draw Box**
   Draw ⇒ Box and specify the dimensions and position as the figure below



Create Airbox

3. **Change attributes**
   Rename it as **air**

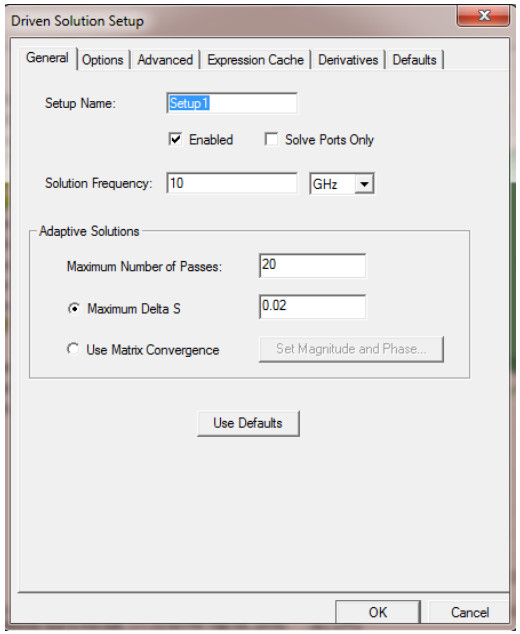


Create Airbox

4. **Assign perfect-H boundary**
   - Edit ⇒ Select ⇒ By Name ⇒ air ⇒ Click OK
   - HFSS ⇒ Boundaries ⇒ Assign ⇒ Perfect H ⇒ Click OK
5. **Override Material**
   - HFSS ⇒ Design Settings ⇒ Enable Material Override
   - check ✓Enable Material Override
   - click OK

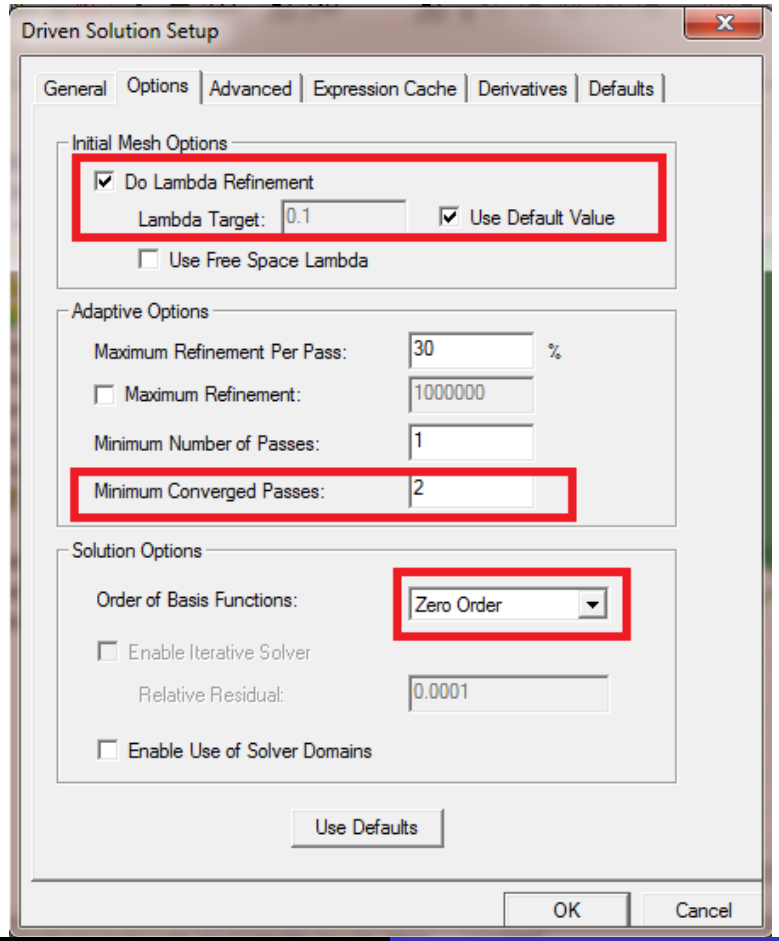
Create an Analysis Setup

1. **Create an Analysis**
   HFSS ⇒ Analysis Setup ⇒ Add Solution Setup ⇒ General tab
   - Solution Frequency: 10 GHz
   - Maximum number of passes: 20
   - maximum Delta S: 0.02



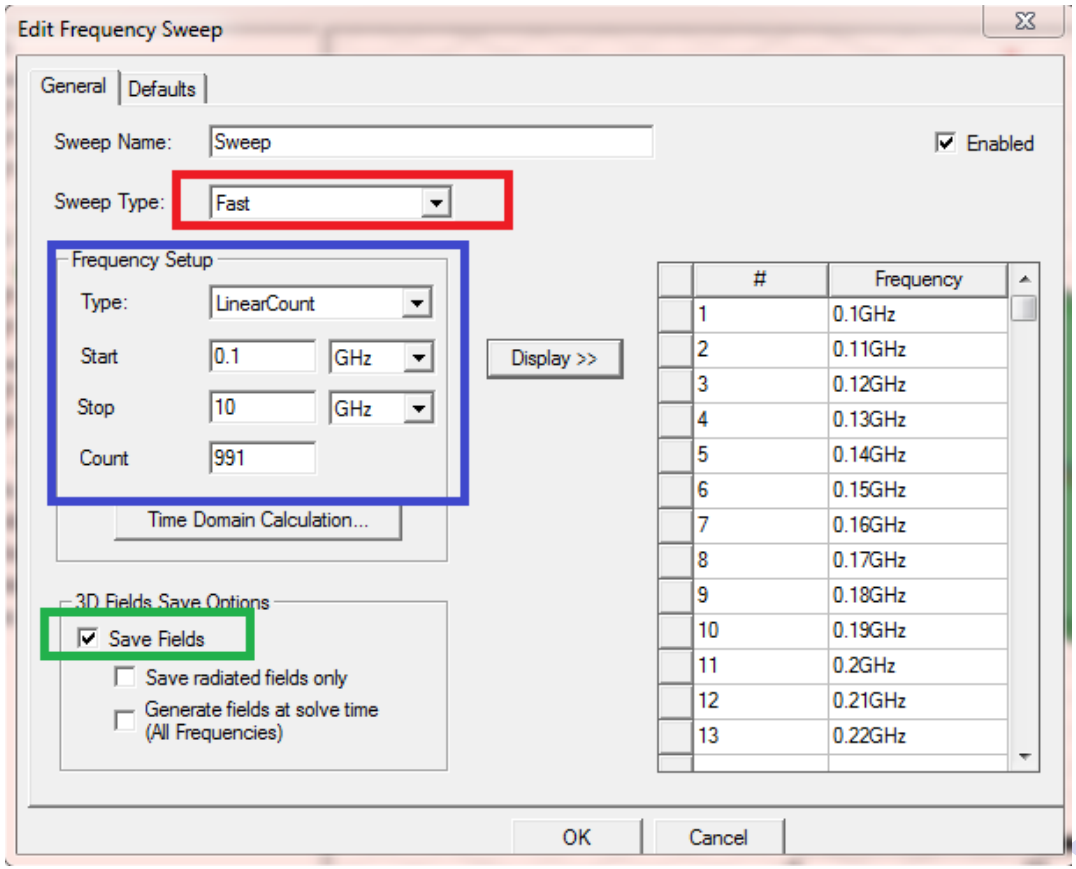
Create an Analysis Setup

2. **Options tab**



Create an Frequency Sweep

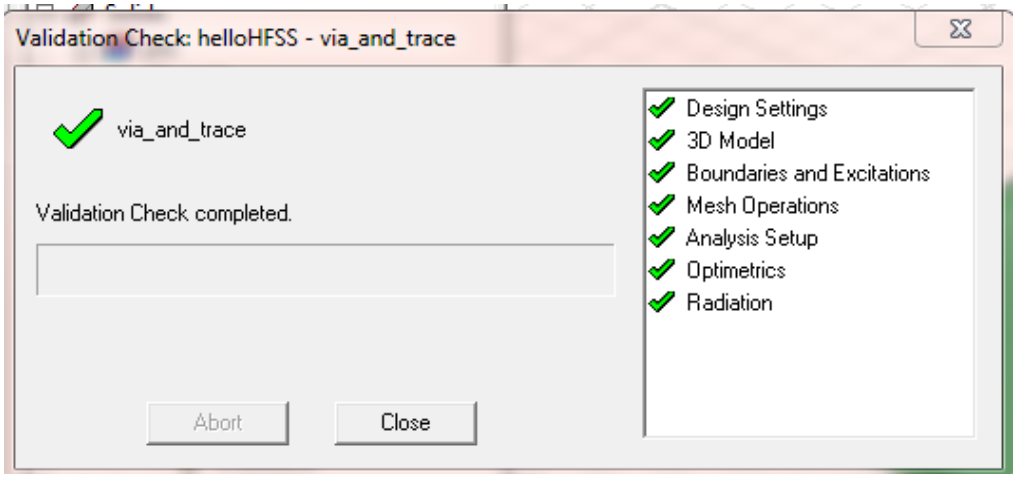
3. HFSS ⇒ Analysis Setup ⇒ Add Frequency Sweep ⇒ Edit the sweep as the figure below ⇒ click OK



#	Frequency
1	0.1GHz
2	0.11GHz
3	0.12GHz
4	0.13GHz
5	0.14GHz
6	0.15GHz
7	0.16GHz
8	0.17GHz
9	0.18GHz
10	0.19GHz
11	0.2GHz
12	0.21GHz
13	0.22GHz

Validation Check and Analyze All

4. HFSS ⇒ Validation Check
   - click **Close** if all pass
   - Use Message Manager to view any warning or error message

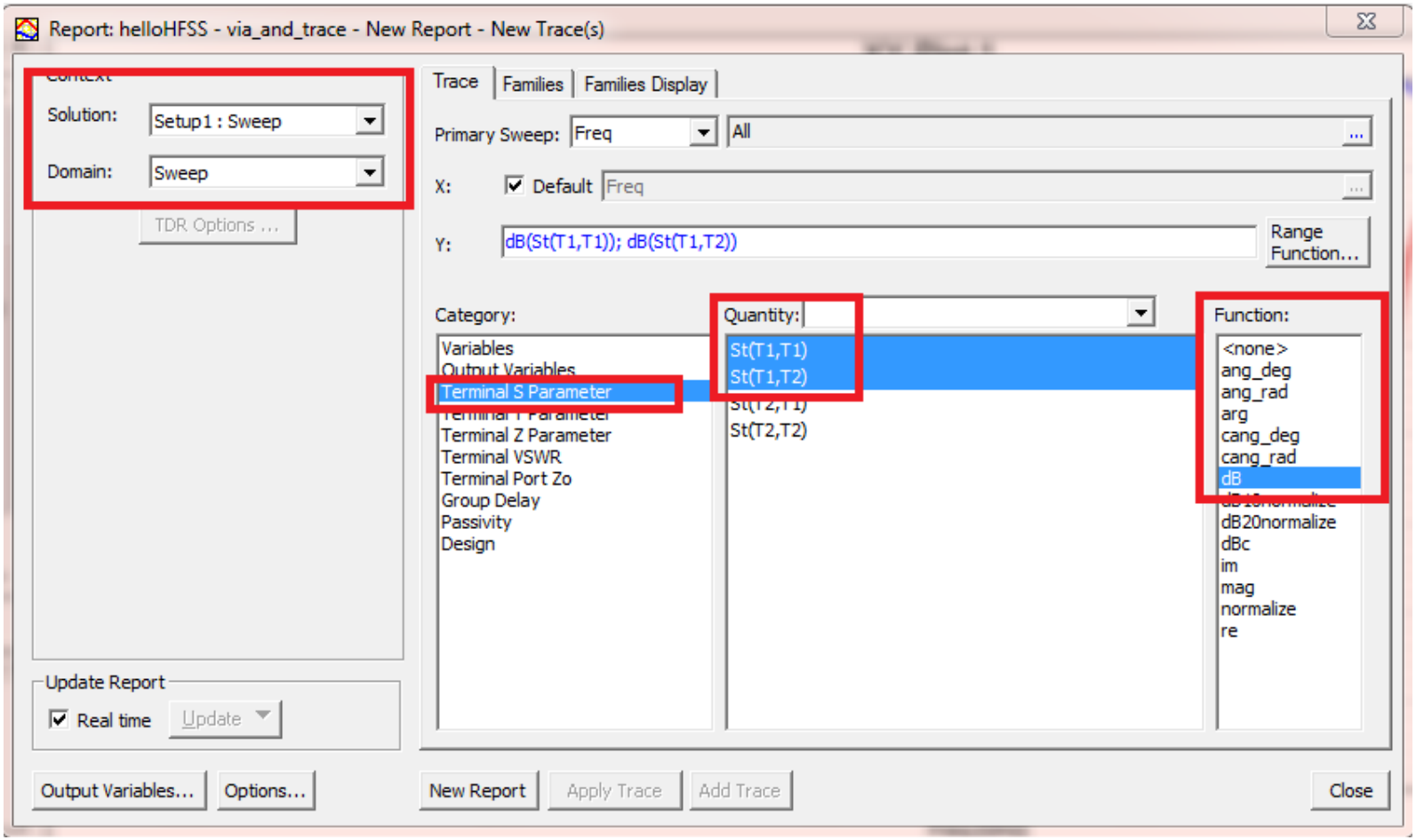


5. HFSS ⇒ Analyze all

Plot S-parameters vs. Frequency

6. HFSS $\Rightarrow$ Results $\Rightarrow$ Create Terminal Solution Data Report $\Rightarrow$ Rectangular Plot
   - Solution: Setup1:Sweep1
   - Domain: Sweep
   - Category: Terminal S Parameters
   - Quantity: St(T1,T1), St(T1,T2)
   - Function: dB

Plot S-parameters vs. Frequency



Plot S-parameters vs. Frequency