

LAB EXERCISE 2

EM Basics (Momentum)

Topics: EM simulation in ADS, focusing on Momentum, including substrate and port setups, 3D viewing, visualization, and more.

Audience: Engineers who have a basic working knowledge of AD, have completed the prerequisite course, or who require EM training.

Prerequisites: Completion of the course, *Workspaces and Simulation Tools* or equivalent experience, including ADS layout skills.

Objectives: Be able to set up and run EM simulations from ADS layout, including creating a substrate and defining Technology layers and materials.

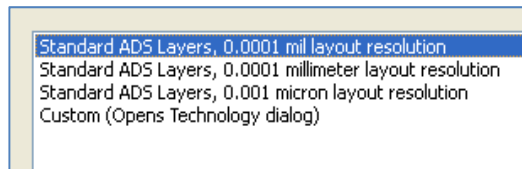
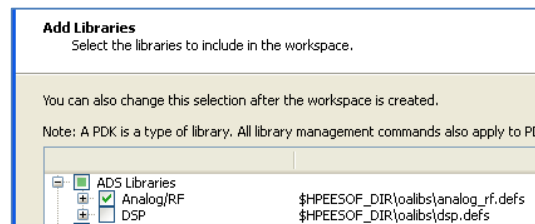
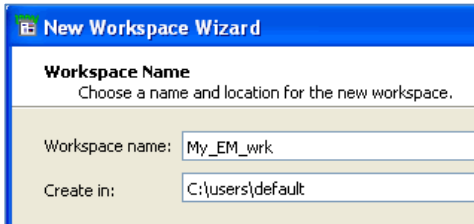


Table of Contents: Lab 2

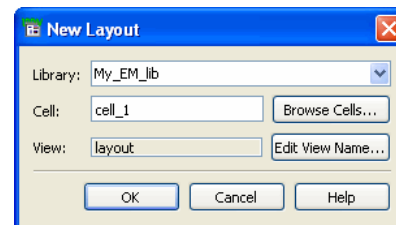
1. Create a New Workspace and Layout	3
2. Define Dielectric Materials in the Technology File	4
3. Draw a layout: 2 metal strips and a via	5
4. Open the EM Simulation Setup	6
5. Add a new Substrate	6
6. Define strip conductors (metal)	8
7. Add a Substrate Layer	9
8. Map a Via and Intruded Conductor	9
9. Importing Substrates	11
10. EM Setup: Yellow Caution Signs	11
11. Selecting Pins and Defining Ports	12
12. Using the 3D EM Preview	14
13. Defining Frequency Plan, Output Plan, and Mesh Options	15
14. EM Simulation, Mesh and Data	16
15. Visualization of Current	18
16. OPTIONAL: Arc Resolution, Edge Mesh, emModel / Symbol	19

1. Create a New Workspace and Layout

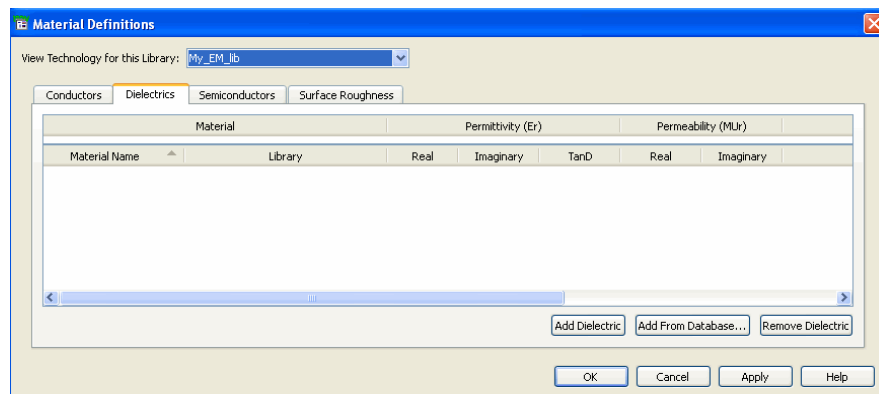
- a. In the ADS Main window, use the wizard to create a new workspace: **File > New > Workspace** or use the icon – and name it: **My_EM_wrk**. Also, name the library **My_EM_lib**. Add only the **Analog/RF** library and mil layers:



- b. Create a new layout using the default cell name: **cell1**, as shown here.

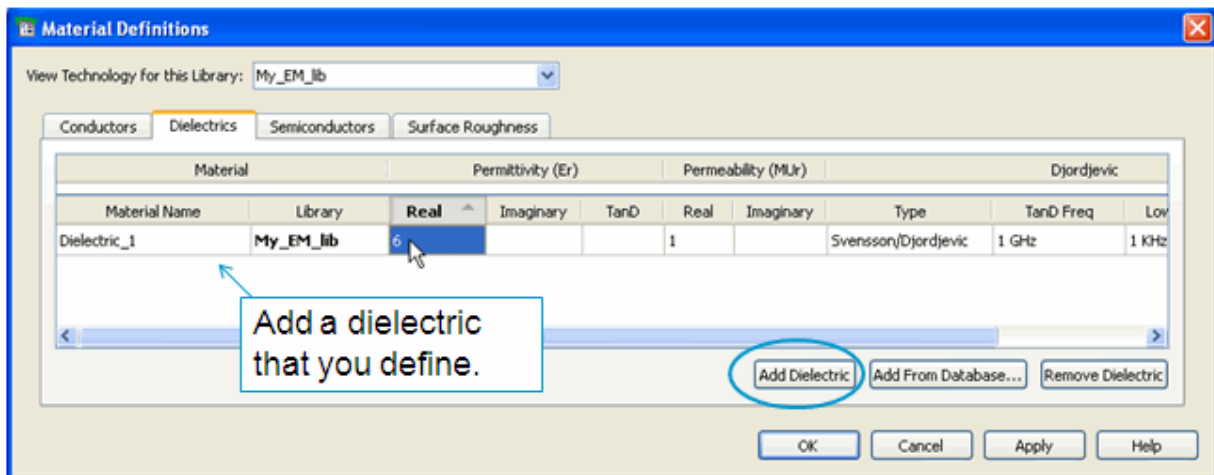


- c. In the Layout window, select **Options > Technology > Material Definitions**. When the dialog appears, select the **Dielectrics** tab – notice it is empty. Of course, if you are using a PDK or library with substrates already setup, then you may see some materials listed here. This lab will show you how to create materials for use with a substrate.



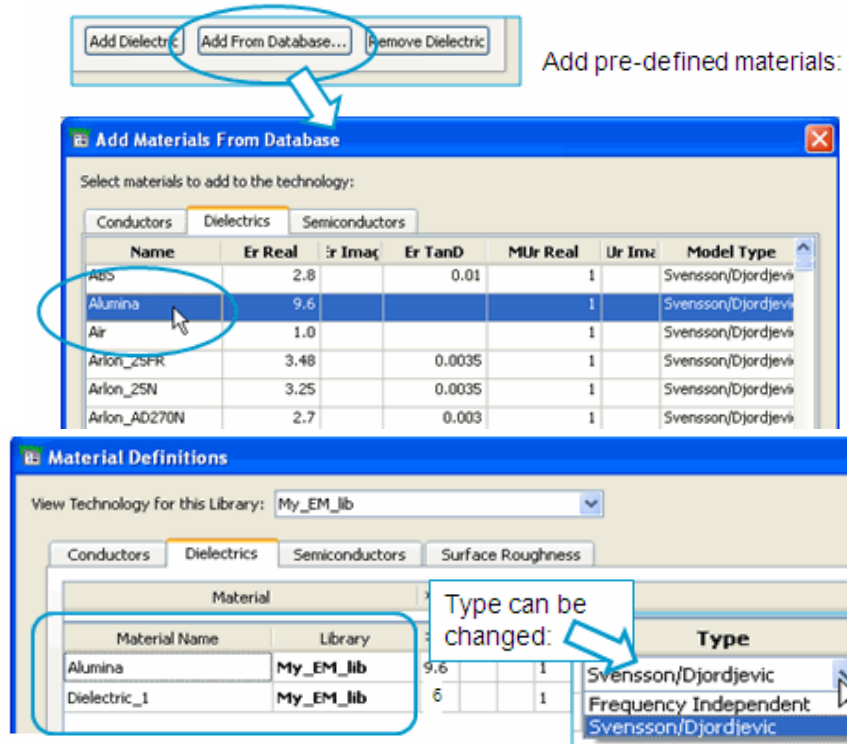
2. Define Dielectric Materials in the Technology File

- a. Click the **Add Dielectric** button and a default name (Dielectric_1) will be added. Next, change the **Real** value = **6** (Permittivity) as shown here and click **Apply**. You could set other values but this is enough at this time.

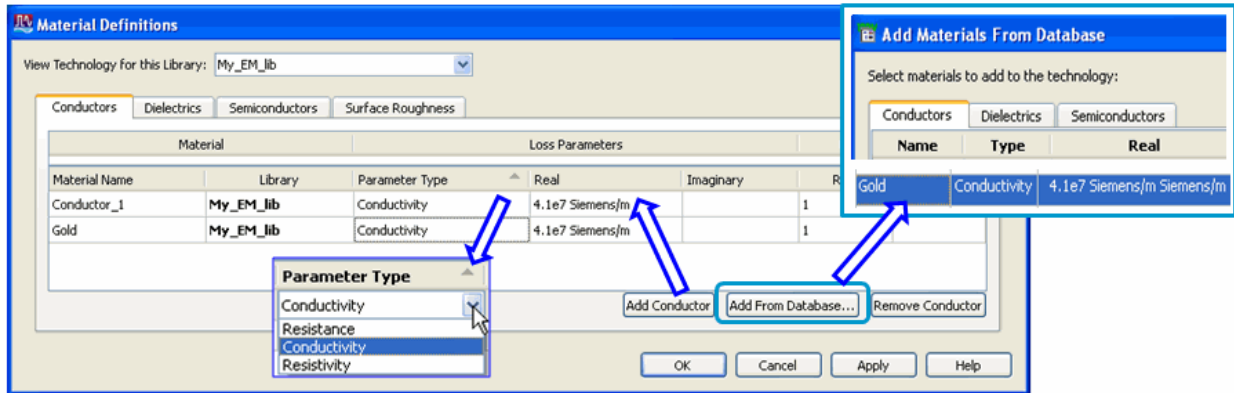


- b. Click the **Add From Database** button and select **Alumina** from the list of pre-defined dielectrics. Click **OK** and now both dielectrics should appear.

Notice that you can change the type, but leave it as Svensson/Djordjevic.

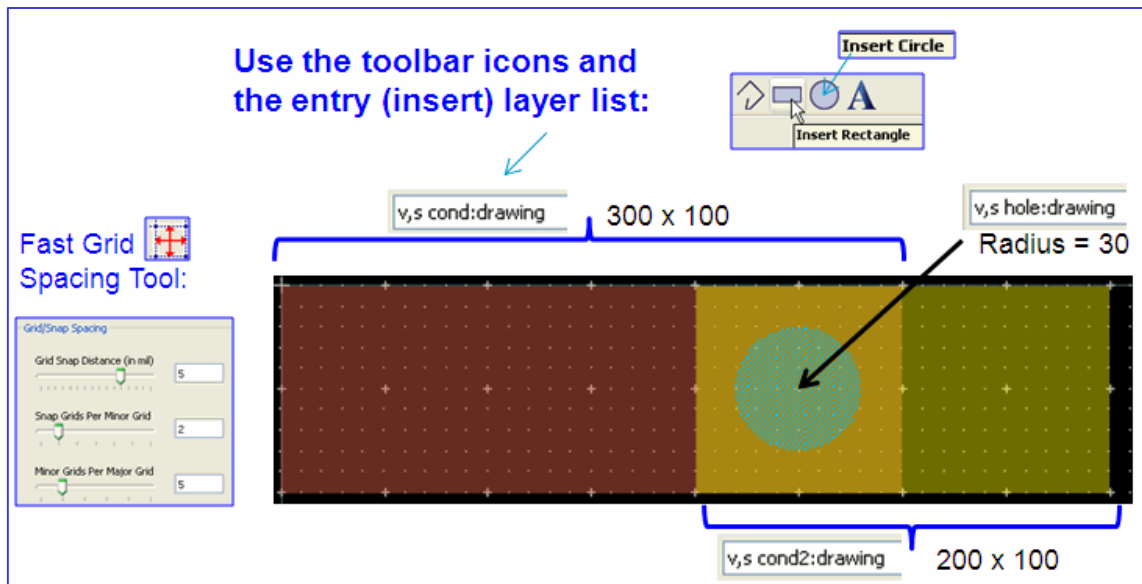


- c. Select the **Conductors** tab and click **Add Conductor**. The default will be added (Conductor_1). Then click **Add From Database** and select **Gold** and click **OK**. With these two added, click **Apply** and **OK**. Notice you can change the Parameter Type – but leave it as Conductivity.



3. Draw a layout: 2 metal strips and a via

- a. In the layout window, **draw this simple design** - the values are in **mils** shown here. Zoom out and change the entry layer as needed to draw two overlapping rectangles: Cond (300 x 100) and Cond2 (200x100), and a centered circle using hole (30 mils radius). NOTE: The default grid spacing and grid snap should be used (5,2,5) as shown here.

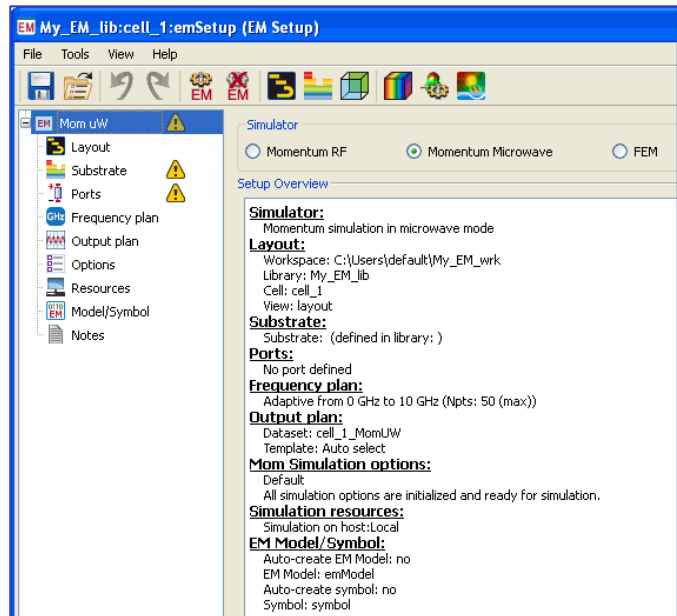


- b. Save the layout when you have finished – you will add pins/ports later.

4. Open the EM Simulation Setup

- Select the EM Simulation Setup icon in layout to open the EM Setup dialog (shown here). Click OK to any prompts.
- Notice the **yellow** caution signs next to the Substrate and Ports icons. This means that these two items have not been specified and you cannot run the simulation yet. Read through the Setup Overview quickly then **close** the EM setup.

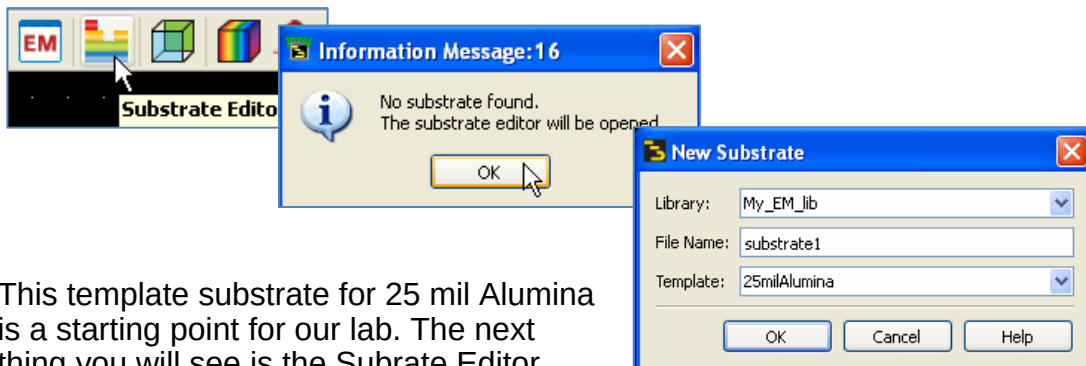
Next, you will open the Substrate editor and define a substrate for your layout.



5. Add a new Substrate

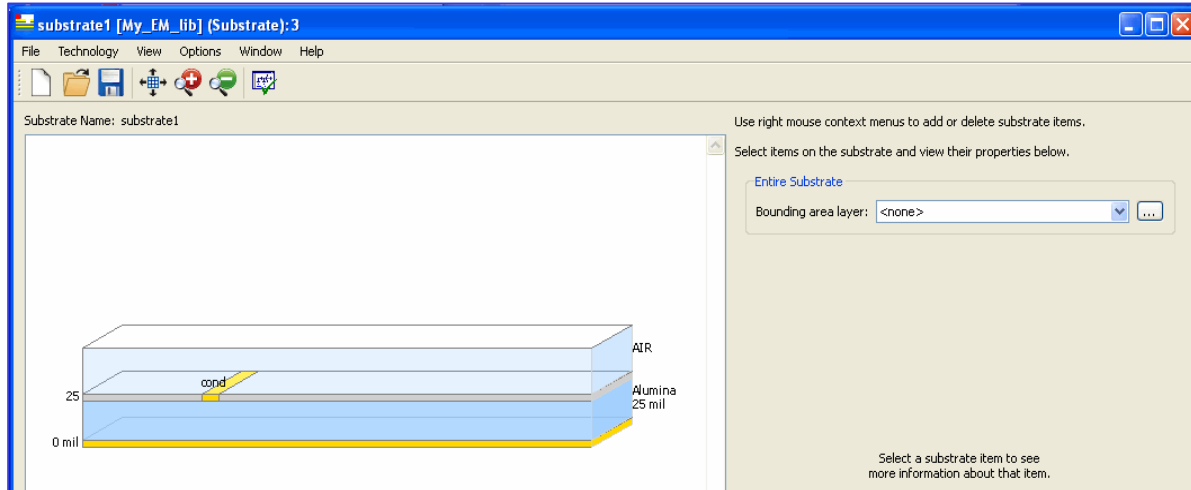
NOTE on substrates: You can use or copy substrates from any library in your workspace. But for this workshop you will create a new substrate so that you are more familiar with how the editor works.

- In the Layout window, click the **Substrate Editor** icon and click OK. Then click **OK** again to use the default name (Substrate1) for your substrate.

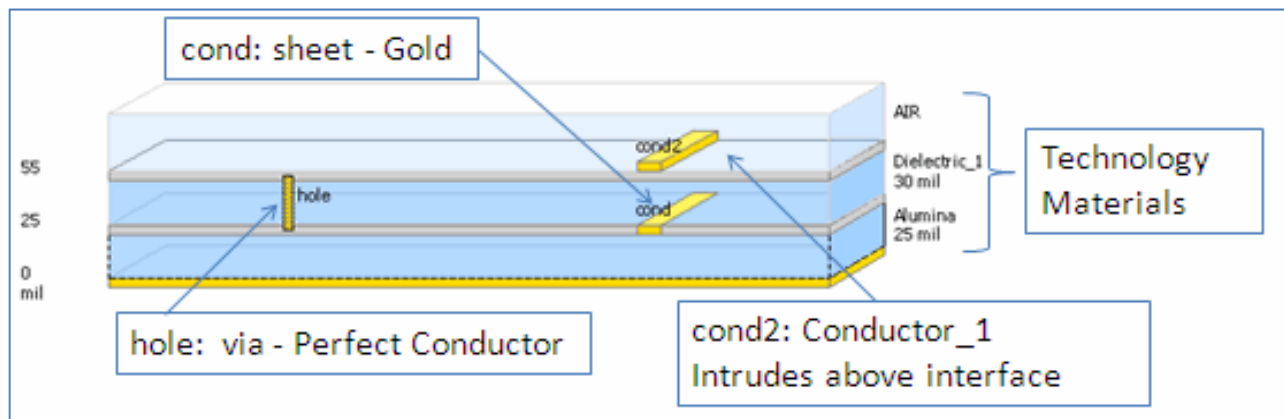


This template substrate for 25 mil Alumina is a starting point for our lab. The next thing you will see is the Substrate Editor.

- b. This is the substrate editor with a 3-D like view of the layers for this template, including a conductor (cond) and the boundary conditions. This is always a starting point for new substrates. Read the information on the right about how to add, delete, or select items but do not do anything yet.



The steps that follow will guide you through the creation of the stack-up and mappings shown here, using library Technology materials – this picture shows what the substrate will look like after you follow the steps.

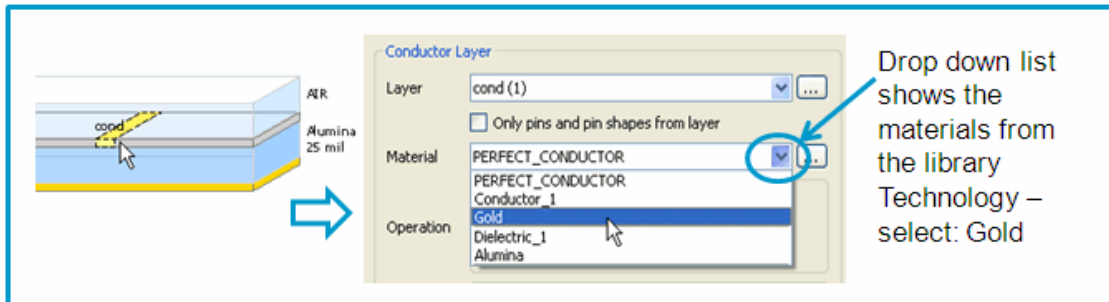


NOTE: You can define materials (dielectrics, conductors, etc.) in the Substrate Editor because it gives you access to the Technology file as you will see. However, defining them first, in the Technology file is recommended.

6. Define strip conductors (metal)

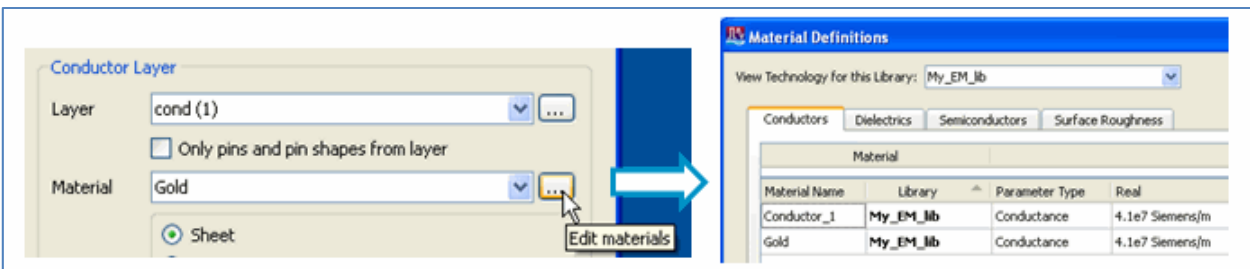
Now you will define the **cond** drawing layer to be a strip conductor: Gold.

- Click on the **cond** strip in the graphical substrate editor on the left - it will become outlined and the properties will appear on the right. You always have to click on the 3D substrate drawing to see the properties. Next, set layer cond **Material = Gold** by using the drop-down arrow button for Material to change cond from perfect conductor to **Gold**.



NOTE: the box for **Only pins and pin shapes from layer** is only checked if you are mapping pins (vias) but no other metals or conductors.

- Next, click the **Edit Materials** button with three dots [...] next to the drop down arrow for Material. This shows the Technology file material.

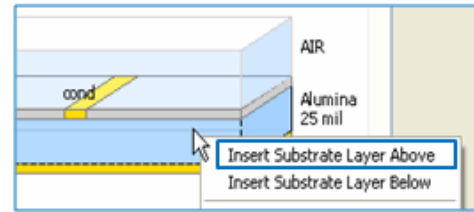


Notice you could add surface roughness for conductors used in Momentum simulations – but do not for this lab. This is only to show how the EM substrate editor works. If you use a library PDK, then all the materials from that PDK would appear.

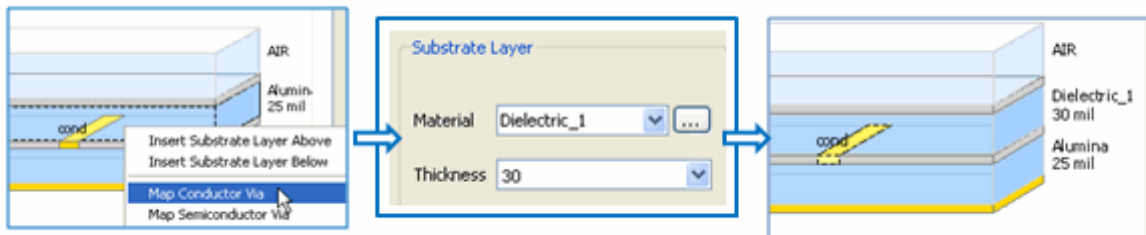
- Cancel** the Material Definitions dialog.

7. Add a Substrate Layer

- a. Use the cursor and **right-click** in the Alumina substrate area as shown here. Then click **Insert Substrate Layer Above** as shown. Immediately, another Alumina substrate is added and you have two Alumina layers.



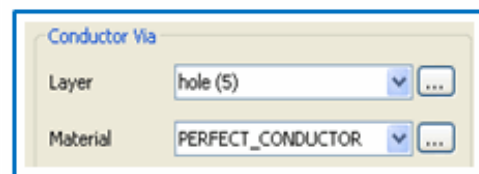
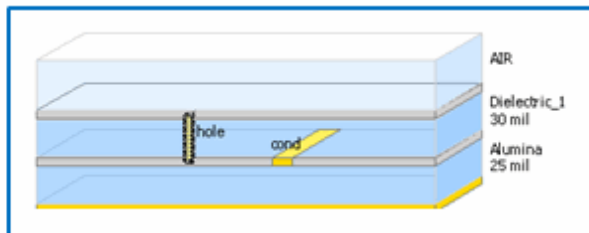
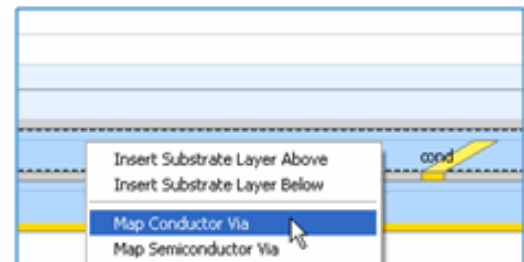
- b. On the right panel, use the Material drop-down arrow button to select the **Dielectric_1** material (you defined this in the library's technology earlier). Set the Thickness to **30** mils – press Enter and you will see it.



8. Map a Via and Intruded Conductor

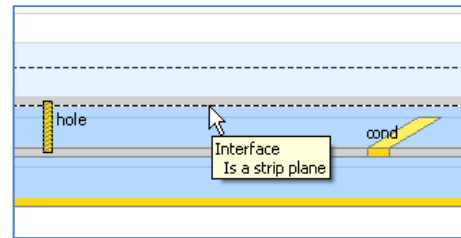
- a. Again, use the cursor and **right-click** on **Dielectric_1** as shown here. When the menu appears, select **Map Conductor Via**.

Immediately, you will see the **hole** layer appear automatically. On the right, set the Conductor Via **Material** to **PERFECT_CONDUCTOR** as shown here. The via is now mapped as the *hole* drawing layer through Dielectric_1.

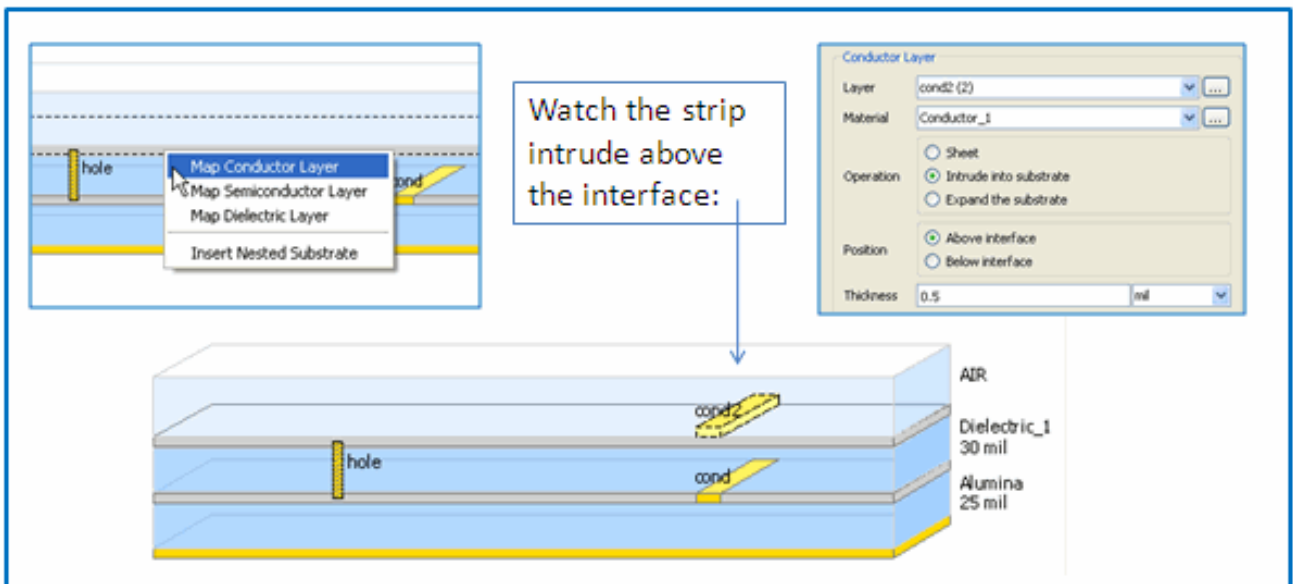


This means that the *hole* drawing layer is now considered a via – you will see this in the 3D preview.

- b. Move your cursor to the **interface** above Dielectric-1 as shown here, and right-click.



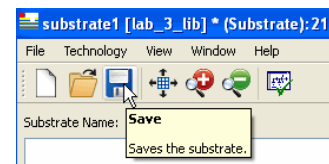
- c. Select the command **Map Conductor Layer** and another strip conductor will appear: cond2. With cond2 selected, go to the panel on the right and set the Material to **Conductor_1**. Also, and click to set the **Operation** to **Intrude into substrate** and set the **Position** to **Above interface**.



- d. Also, set the metal **Thickness** to **0.5** mil as shown – press Enter.

At this point, you have defined the two layer Alumina substrate with a via and 2 metal conductors, including thick metal for Cond2. Now it is ready for a layout to be viewed or solved.

- e. If you have any problems, unmap / delete as needed. Or, close the editor and begin again. But if you have the correct setup, then **Save** it using the icon shown here.



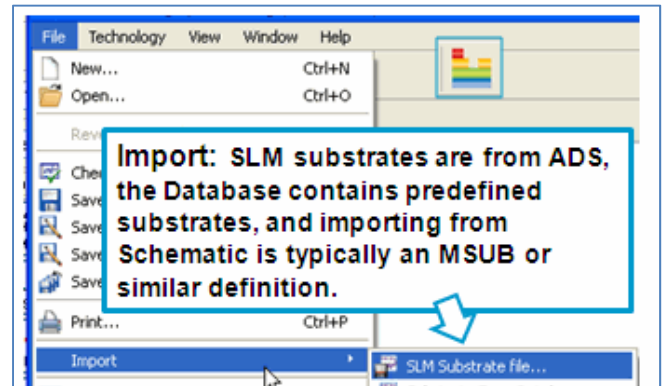
NOTE: Substrates you define and save can be used for other cells or workspaces.

Now it's time to set the ports but first you should see the commands for importing substrate files into ADS from other ADS versions.

9. Importing Substrates

- In the Substrate Editor window, select the command **File > Import** shown here, and look at each dialog that appears for each command.

Do not select anything at this time. This is only to show that you can easily use any other available substrates and you can also modify them and save them in your workspace library or other library.



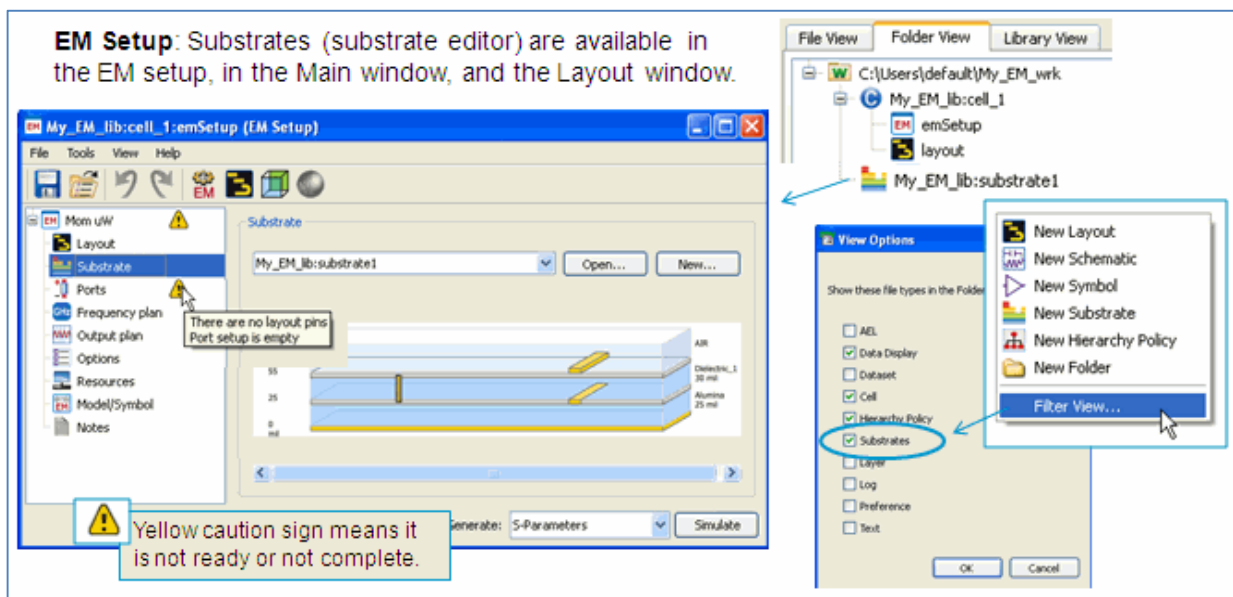
- Cancel the dialogs and substrate editor when finished – it's time for the Ports. Then **close** the Substrate Editor.

10. EM Setup: Yellow Caution Signs

- In layout, click on the **EM Setup** icon. You should see that only the Ports have the yellow caution sign. Also, the Main window should show the Substrate and the EM setup. Right-click and use Filter View if you don't see the substrate.

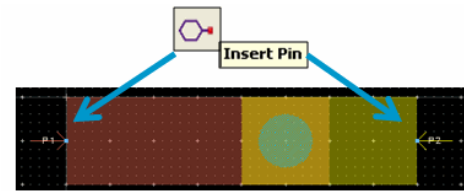


- If your Main window and EM setup window look like this, then you are ready to continue. If not, go back and check your work.



11. Selecting Pins and Defining Ports

NOTE: The EM port feature will automatically recognize layout pins and assign them as the appropriate type of S-parameter ports for simulation for most cases.



- a. In the layout window, select the **Insert Pin** icon and insert two pins (P1 and P2) as shown here – you do NOT need to change layers.

- b. Go to the **EM Setup** window and click the **Ports** icon.

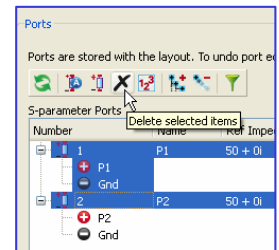


- c. Immediately you will see the pins are assigned as S-parameter ports and the yellow caution sign is gone.

Number	Name	Ref Impedance [Ohm]	Calibration	Ref Offset [mil]
1	P1	50 + 0i	TML	0
	Gnd			
2	P2	50 + 0i	TML	0
	Gnd			

- d. Hold down your keyboard **Shift** key and **select ports 1 and 2** - then click the **Delete** selected ports icon as shown here.

- e. Now the ports (P1 and P2) are shown as Unconnected Layout Pins. This does not mean they are unconnected to your metal, it means they are not connected (assigned) to ports. If a port is not connected to the metal, the EM simulator will recognize it during simulation and inform you.



Ports

Ports are stored with the layout. To undo port edits: use undo from the layout window then refresh.

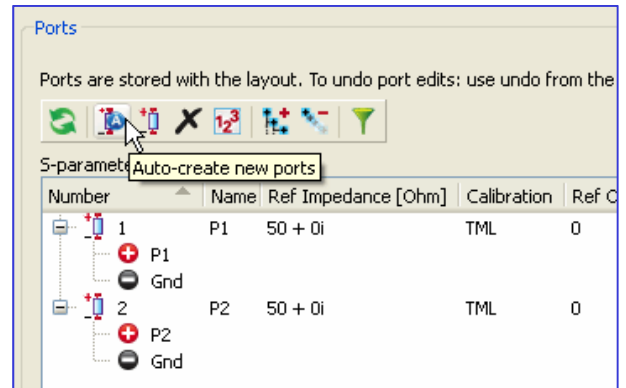
These icons can be used to control the S-parameter ports: refresh, assign, delete, etc.

Unconnected Layout Pins

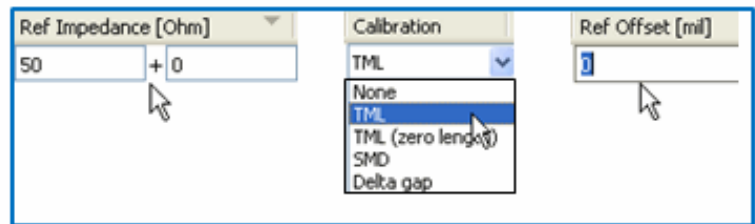
Name	Connected to	Layer Num	Layer	Purpose Num	Purpose	X
P1		1	cond	-1	drawing	0
P2		2	cond2	-1	drawing	40

When the ports were deleted, they became unconnected layout pins = unconnected to a port.

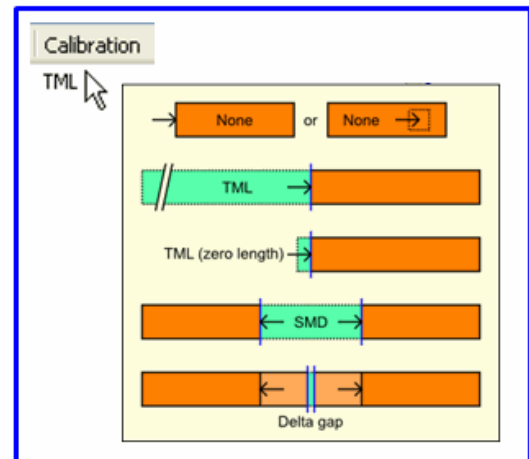
- f. Click the icon, **Auto-create new ports**, and immediately pins 1 and 2 will be created as ports. Also, these ports are automatically set to TML calibration (Transmission Line Calibration) – this is the default type, with 50 Ohms as the reference impedance.



- g. Use your cursor to click in the fields shown here. This is how you can change port settings for your designs. But do not make any changes for this lab.



- h. Use your cursor: hover over the calibration type (TML) and you should see a pop-up that gives a visual description of the calibration types as shown here. You can use this to determine which type of port calibration you want. For this lab, TML (transmission line) will be used.



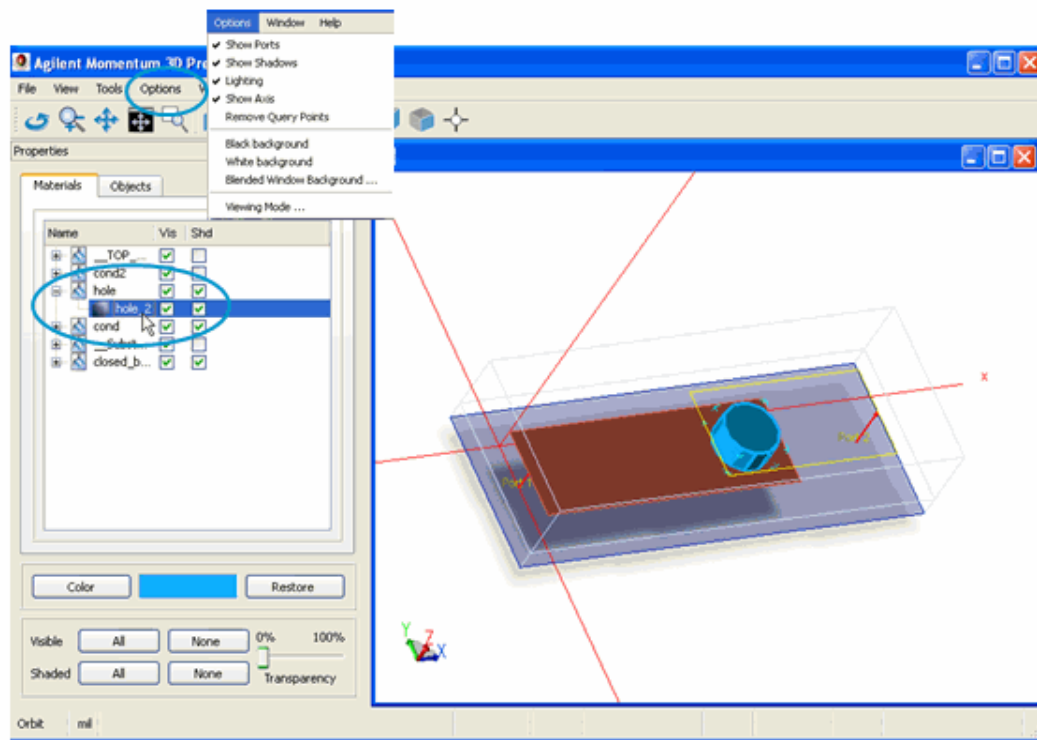
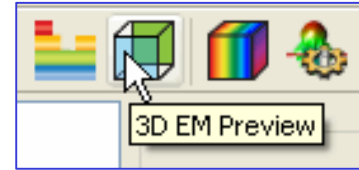
- i. Click the Save icon to save this EM setup.



NOTE on Ports: There are many different ways to set up ports for EM analysis. For more information, click the Help button (Topics and Index) and look for Ports.

12. Using the 3D EM Preview

- With the substrate and the ports now complete, you can preview the layout in the 3D viewer. Also, there are no more yellow caution signs.
- In the EM Setup, click the **3D EM Preview** icon shown here – this is the same 3D viewer that can be launched from layout. It will take only a minute or two to appear.
- Take a few minutes to look at the object – try rotating it with your cursor, click on layer names, change colors, and try changing various **Options**.

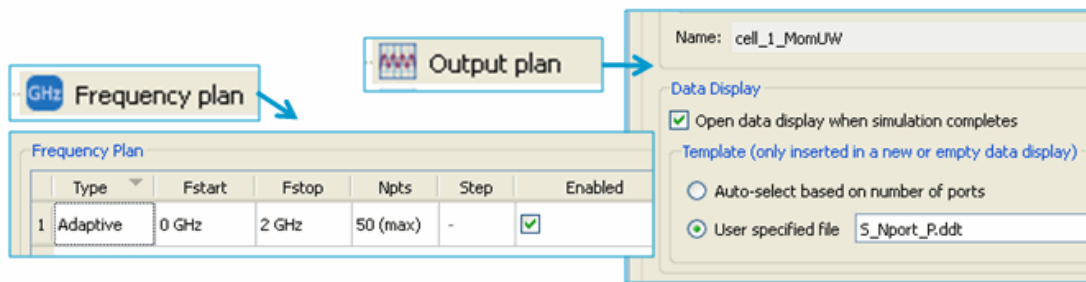


NOTE: A substrate definition is required for a 3D view.

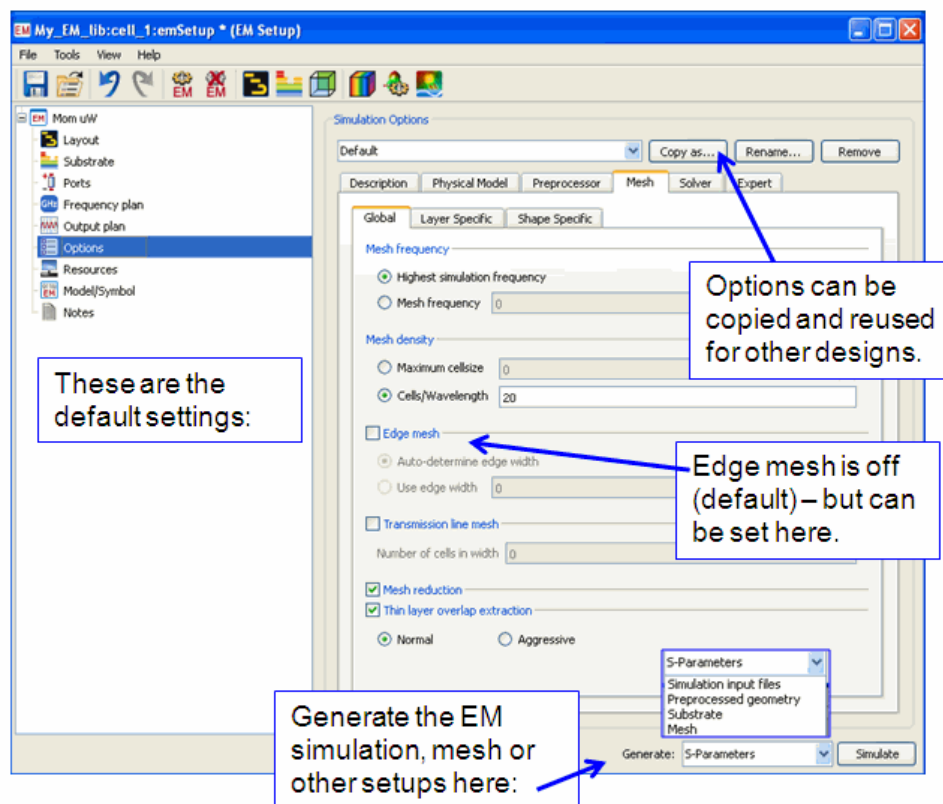
- After the simulation, you will use Visualization in this same viewer to see the currents. **Close** the viewer when you have finished.

13. Defining Frequency Plan, Output Plan, and Mesh Options

- In the EM Setup window, click the **Frequency plan** icon and set the **Fstop** to **2 GHz** using your cursor in the field. This is how you set values in these fields.
- Then click the **Output plan** icon and look at the Data Display field. You could change names but for this lab, use the defaults, including the template. After this course, you can change these easily for your designs.

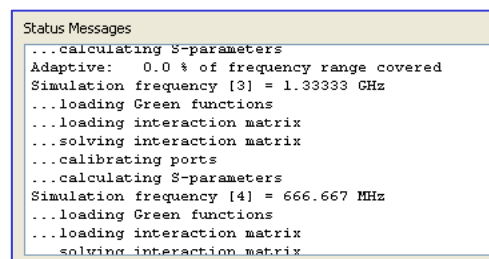
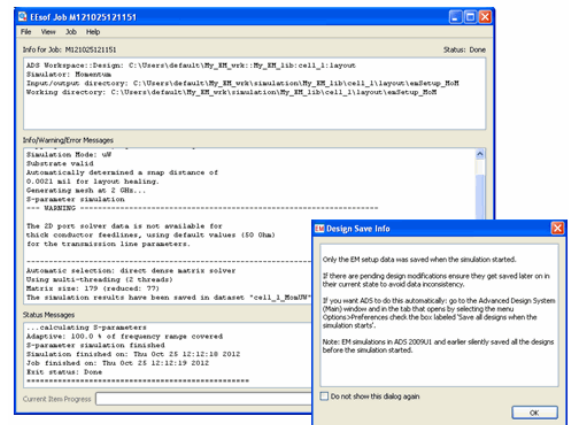


- Before running the simulation, you can look at the Options in the EM setup window where the Mesh controls are located. Most of the time, you will not have to adjust the settings, but some designs may require it. Click the **Options** icon and go to the **Mesh** tab as shown here – but do **NOT** make any changes for this lab. Also, do NOT simulate yet.

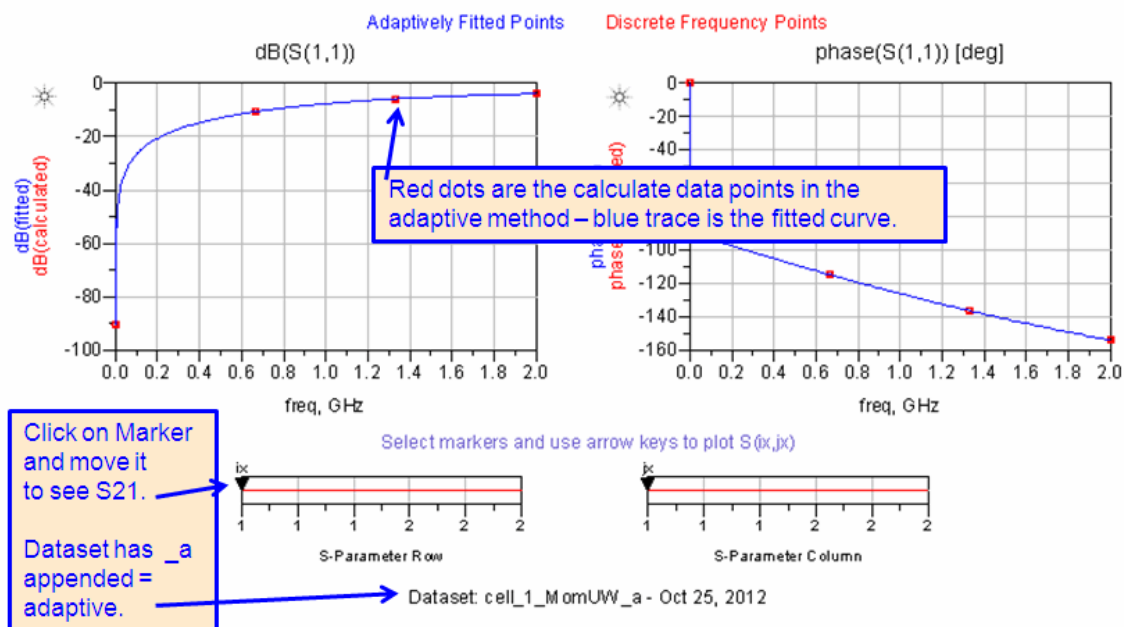


14. EM Simulation, Mesh and Data

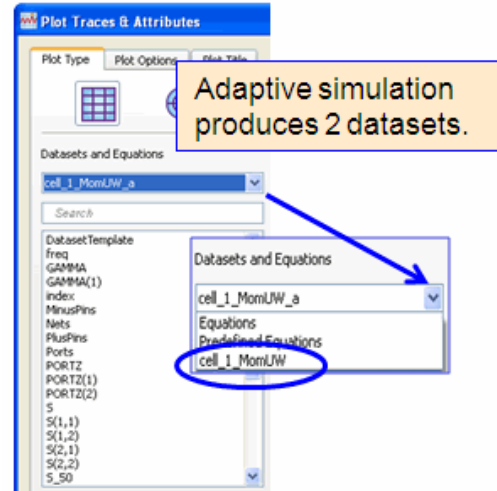
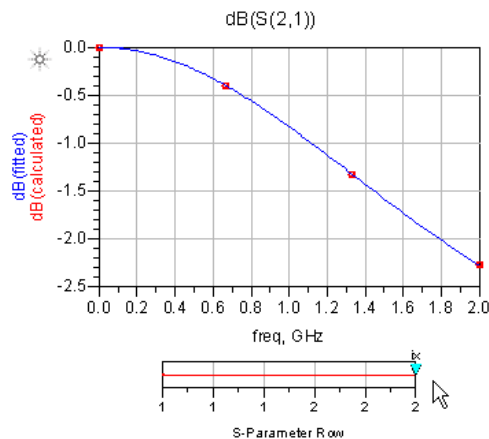
- Now, click the **Simulate** button at the bottom of the Model/Symbol setup. The simulation will start and you can watch the **Job** status window as the simulator reports the information in the Job window. Click OK to any prompts or messages that appear.
- When finished, look in the *Status Message field* of the Job window. You should see the first and last frequency points - then it chose frequencies in-between. This is the adaptive technique (a type of curve fitting) that shortens simulation time.
- Look at the **Data Display**. It uses the dataset adaptive data: the cell name ends with: **_a**. This is correct and this is the one you want to use. Also, this is the template selected by default in the EM Setup Output Plan. Now, **select the marker on the slider** (S-Parameter Row) and move it to the right to the number 2 – this will give you the S-21 data.



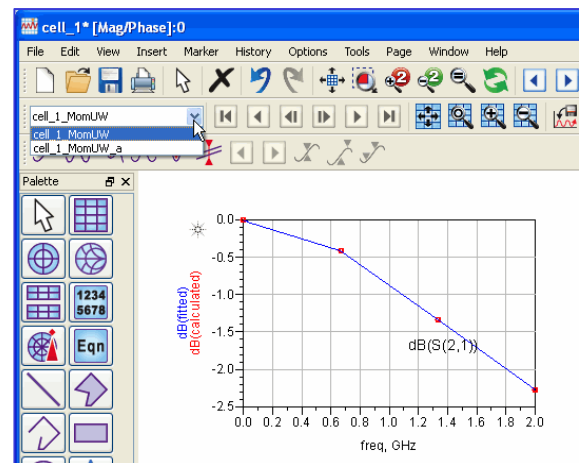
Mag/Phase of S(1,1)



- d. Notice the S-21 data. As you can see, there is some increasing loss through the structure. **Edit (double-click) the S-21plot** and you will see the Plot Traces and Attributes dialog. As shown here, click on the **Dataset and Equations** drop down arrow and you can see 2 datasets. One is the adaptive (a) that is used to produce the smooth fitted curve and the other is the data points only. You always get two datasets with the adaptive method. Cancel the Plot Traces and Attributes.

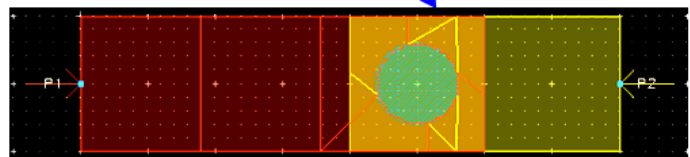


- e. In the Data Display, change the default dataset as shown here. It has the same name but without the appended a. As you can see, the plot now shows only the calculated points.



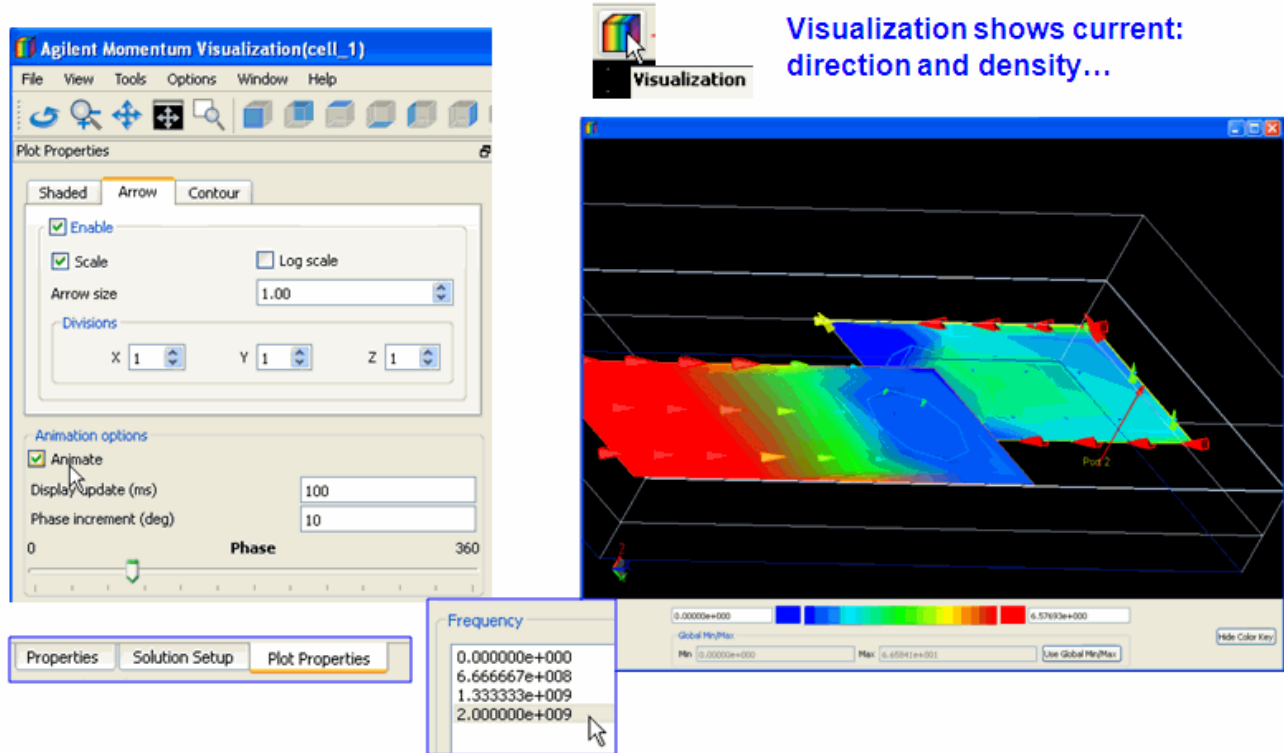
- f. Look at the layout window and you will see the mesh. Notice the circular via produces a triangular mesh – this is expected because the EM mesh engine chooses a reasonable arc resolution value for circles and curves. But these settings can be adjusted in the Options.

Circular Via mesh has triangles.



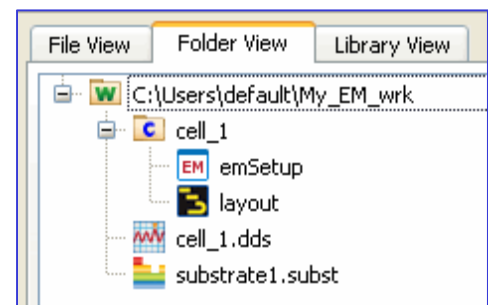
15. Visualization of Current

- a. Close the Data Display to save space and also make the layout window small - then click the **Visualization** icon in layout as shown here.



- b. When the window opens, go to the bottom left and click **Solution Setup** and, under Frequency, select **2 GHz**. Then go to the **Plot Properties** tab and the **Arrow** tab and **Enable** the arrows and click the **Animate** box. You should see the animated surface currents – rotate the object as desired.
- c. Experiment with the controls and when finished, **close** the Visualization window.
- d. Save the layout window and the EM setup. Your Main window should now show the cell contents: the emSetup, layout, data display and the substrate which are all in your library.

You should now have a good basic understanding of: how to set up an EM substrate, assign the ports to the pins in your layout, run a Simulation, examine the mesh and data, and plot the currents.

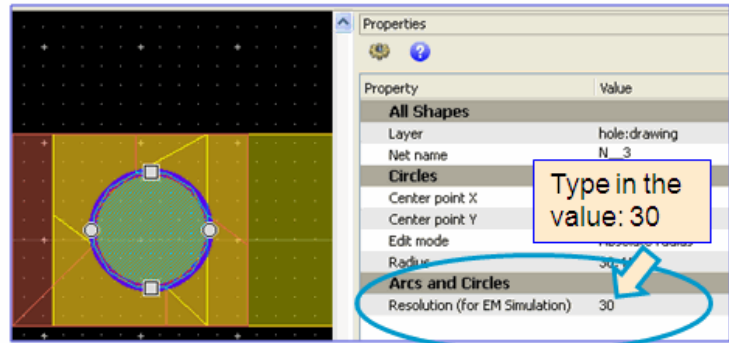


- e. The next step optional only if you have time. Otherwise, go to the Main window and use the command **File > Close All** and save all your work.

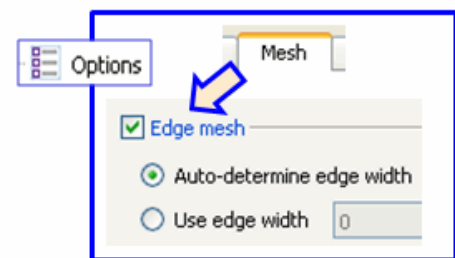
16. OPTIONAL: Arc Resolution, Edge Mesh, emModel / Symbol

This step demonstrates how the simulation results can be slightly improved with edge mesh (increases simulation time). Also, it shows how to easily specify arc resolution the via (hole), and how the EM model is created for use with the layout look-alike symbol.

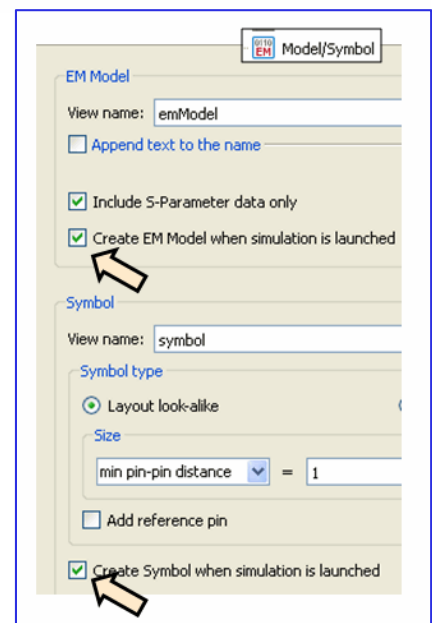
- a. **Arc Resolution:** In layout, select the circle as shown here. In the Properties docking window, change the Arcs and Circles resolution for this circle to 30 as shown here. This only changes for the EM simulation and not your drawing.



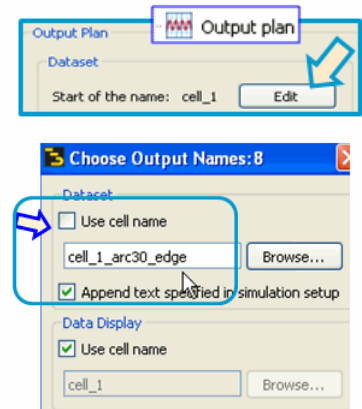
- b. **Edge Mesh:** In the EM Setup, click the Options icons and then go to the Mesh tab. In the **Global** tab, check the box for Edge Mesh only. This is usually only used for structures where close coupling effects can be critical if current is very dense on the edges; this also increases simulation time.



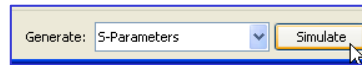
- c. **emModel and Symbol:** In the EM Setup, select the **Model/Symbol** icon and check the boxes shown here to create the emModel and Symbol. This will create the emModel you can use for other simulations along with the symbol that will look like the layout structure.



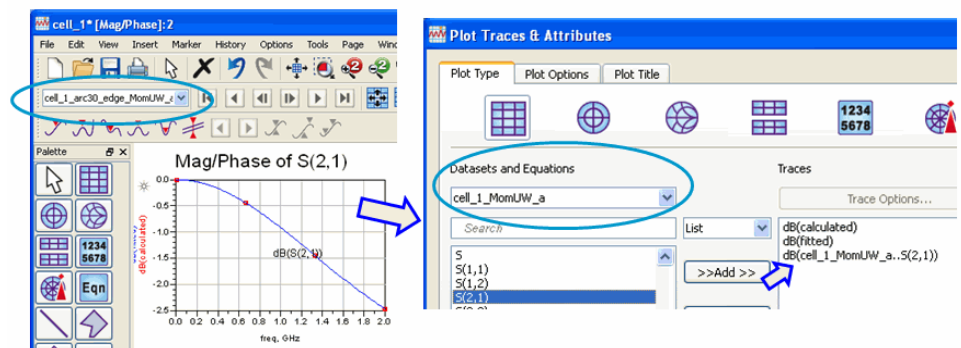
- d. **Specify a new dataset name:** In the EM Setup, select the **Output Plan** icon and click the Edit button shown here. Then uncheck the box for Use cell name and type in the dataset name: **cell_1_arc30_edge**. Click **OK** - this name describes the simulation setup and, after the simulation, you will compare this data to the previous data using all the default settings.



- e. **Simulate:** At the bottom of the EM Setup, click the Simulate button and wait until it is finished.

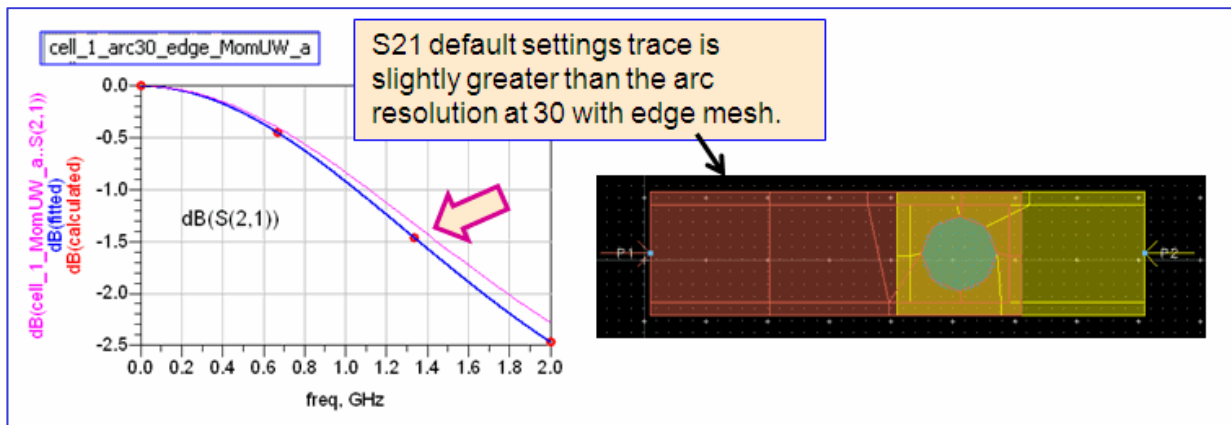


- f. Notice that the same Data Display (cell_1) appear with the new dataset plotted.



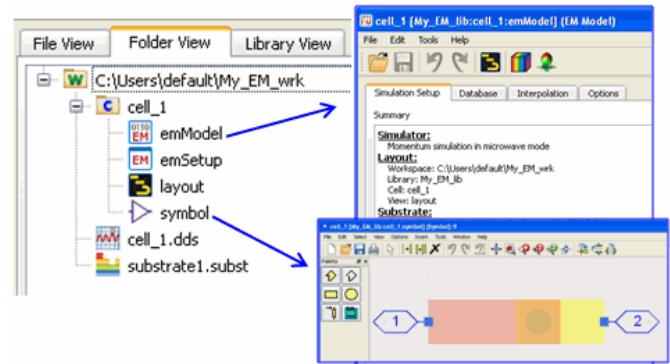
- g. Add the previous data by editing (double-clicking) the S-21 plot. Then, in the Traces and Attributes dialog, select the **cell_1_MomUW_a** dataset (your original data) and Add the 2(2,1) data as shown here.

You should see that the original (default) trace is slightly greater than the customized setup. This is expected because you used edge mesh and a denser mesh was created for the via. However, there is very little difference.

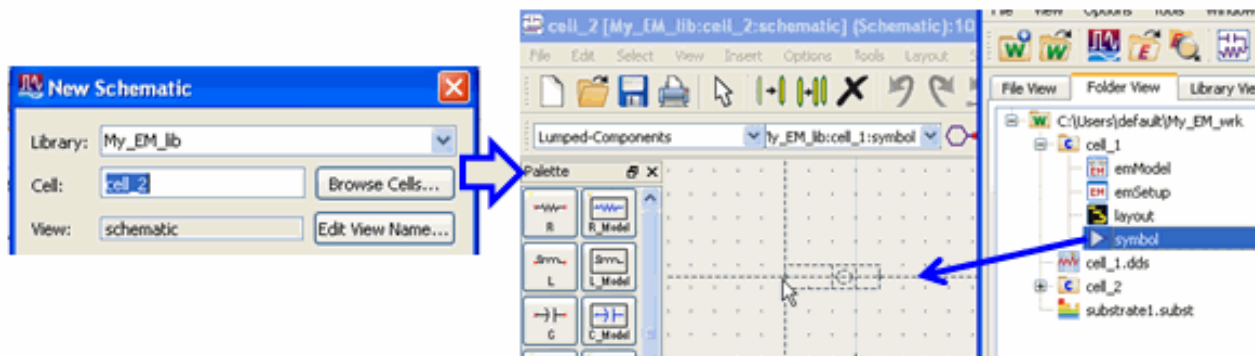


- h. **emModel and Smbol:** Look at your Main window Folder View) and you should see the em Model and Symbol created by EM. Open each one and verify that you have them as shown here. IF not, go back and check your work. The symbol should look like your layout and the emModel looks like the EM Setup but it also contains the EM data.

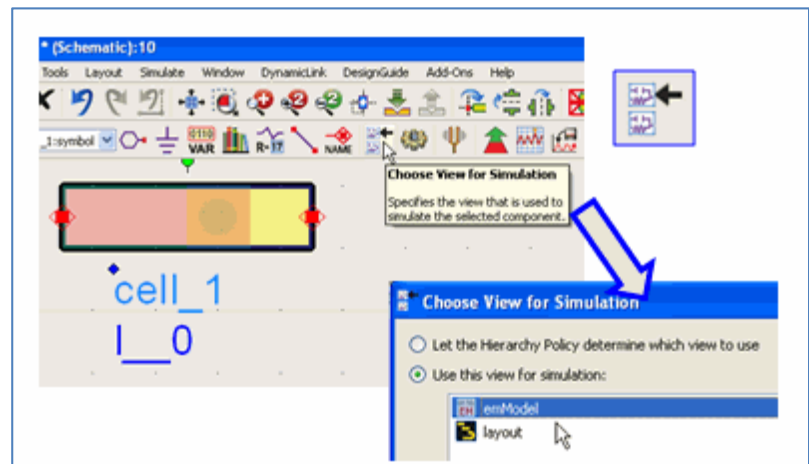
emModel and Symbol:



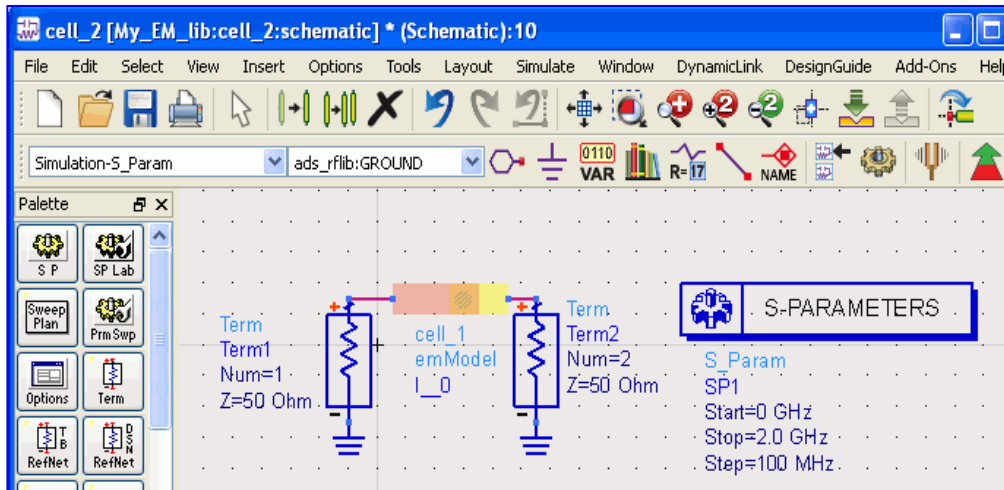
- Now, create a new schematic and drag and drop the cell 1 smybol into the schematic (from the Main window) as shown here.



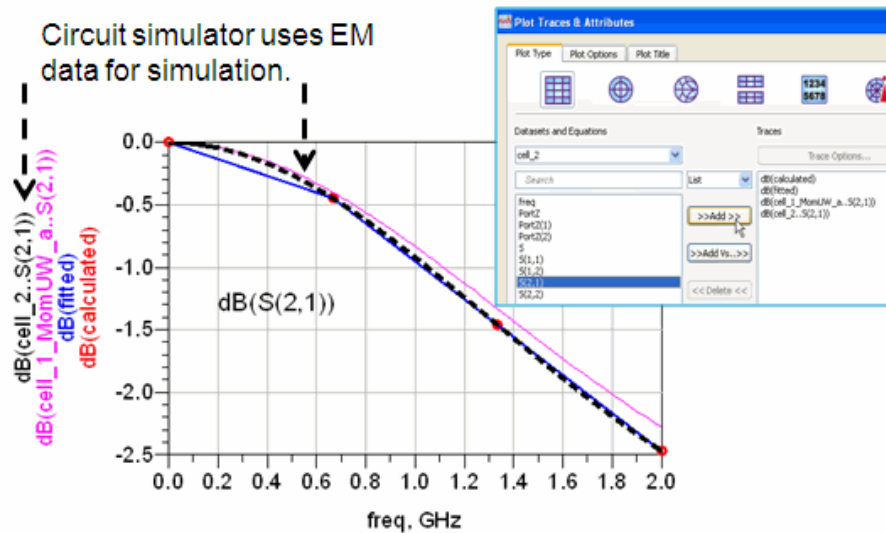
- With your look-alike symbol inserted in schematic, select it and then click on the icon, Choose View for Simulation and select the emModel as shown here.



- Set up and S-parameter simulation as shown here, 0 – 2 GHz, and run the schematic simulation.



- When the simulation completes, close the blank Data Display (cell_2) if it opens and go to your cell_1 Data Display. Edit the S-21 plot and add the cell_2 data S-21 in dB. As you should see, this is how the emModel is used with the look-alike symbol for circuit simulation. Of course, you can also use other circuit simulators (AC, Harmonic Balance, etc.).



- If finished, same and close all the ADS windows..

END OF LAB EXERCISE