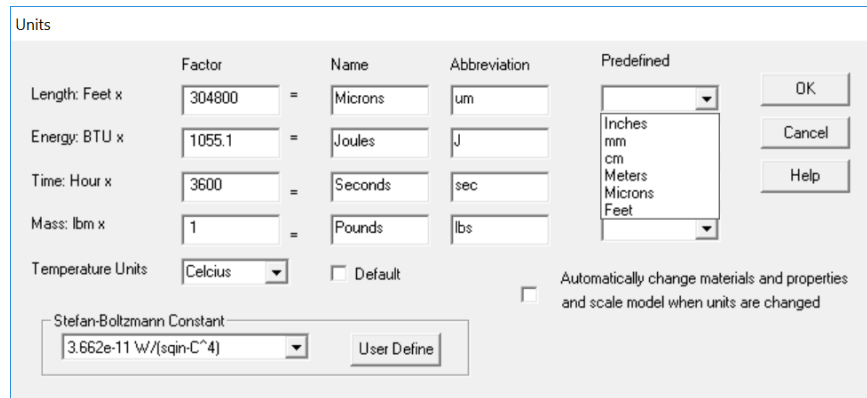


MMIC Front End for Solaria

The MMIC Front End is essentially a preprocessor for Solaria. A group of dialogs are used to define the geometry, materials, heat loads and boundary conditions of a power amplifier. After the data has been defined, a complete ready-to-solve Solaria model is automatically built.

Units

In the process of defining the power amplifier, you will be selecting materials from the material library. The data in the library is in a specific set of units. For the units to be correct when the materials are applied to the model, the values need to be converted to your working units. Therefore, you must first define those units. Select Model>Units. Typically, it is easiest to work in linear units of Microns. Simply select the units from the Predefined drop down. The conversion factors will automatically filled in.




	Factor	Name	Abbreviation	Predefined
Length: Feet x	304800	Microns	um	Inches mm cm Meters Microns Feet
Energy: BTU x	1055.1	Joules	J	
Time: Hour x	3600	Seconds	sec	
Mass: lbm x	1	Pounds	lbs	

Temperature Units: Default

Automatically change materials and properties and scale model when units are changed


Stefan-Boltzmann Constant:

Screen modes

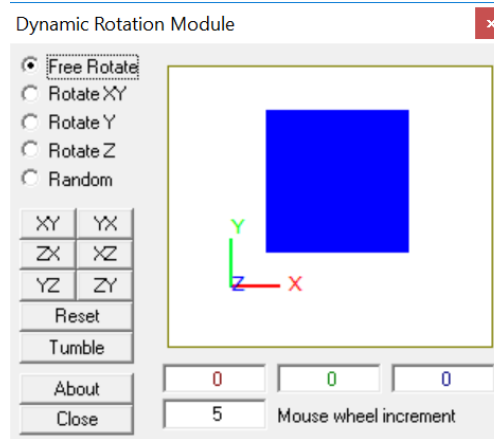
Solaria has several screen modes. When Solaria is launched, you will be in the Solaria Model view mode. You can select models you have generated in the past and solve and view results. SolariaPCB is another preprocessor designed to model printed circuit boards. If you are licensed for SolariaPCB, selecting the  toolbar button will switch to SolariaPCB view mode. Any Solaria models that were on the screen will be cleared. They are not deleted, they are just not viewed. When you select the button again, you will be back in Solaria model mode and you will again see the model.

Likewise, when you select **Add>MMIC Front End**, the view of a model on the screen will be cleared and you will be in MMIC view mode. When you hit **OK** or **Cancel** to the dialog, you will again be in Solaria model view mode.

Navigating in MMIC view mode

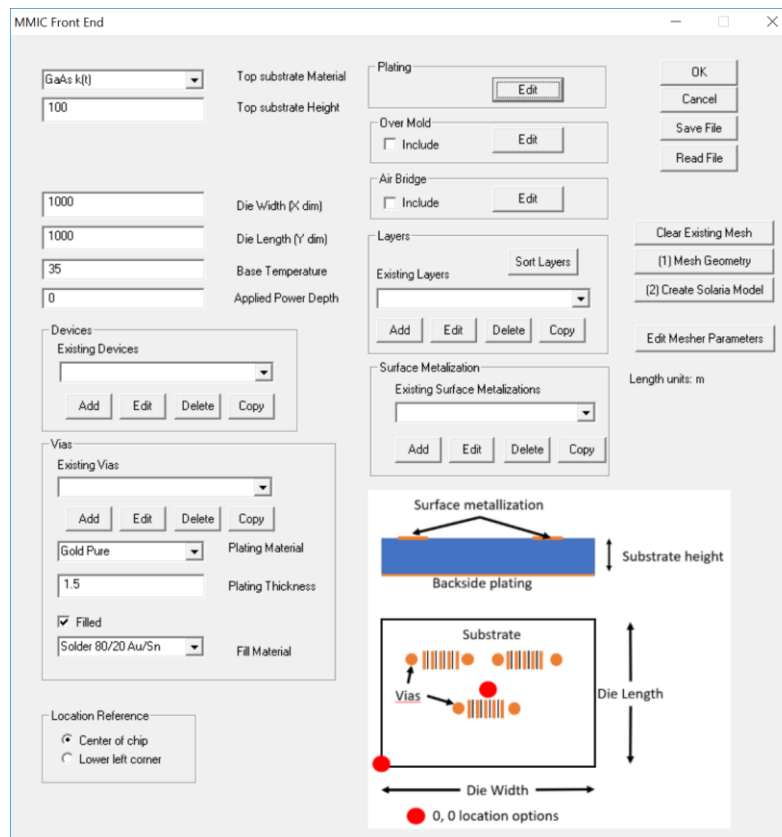
These toolbar buttons can be used to change the view of the screen.  If you hold the left mouse button down and move the mouse, the view will be rotated. Go to **Settings>OpenGL>Display**. There are two modes for this rotation. Check **Rotate freely on XYZ** as this gives you the most control.

If you double-click any place on the screen the following dialog will come up. Here you can select more predefined views.



The Main Dialog

After launching Solaria, select **Add>MMIC Front End**. Below is the main dialog from which all others are launched.



Top substrate Material: The chip material is selected from the Material Library.

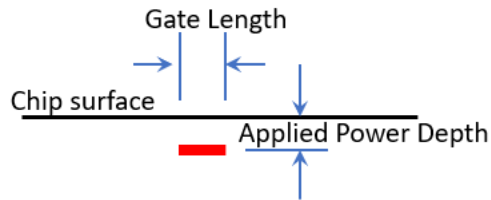
Top substrate Height: This defines the thickness of the amplifier chip.

Die Width (X Dimension): The width of the amplifier chip is defined.

Die Length (Y Dimension): The height of the amplifier chip is defined.

Base Temperature: Boundary Nodes are applied to the bottom of the bottom most Layer at this temperature.

Applied Power Depth: The heat dissipated at the gate can be defined at the surface of the amplifier chip or at a depth below the surface.



Save File button: This is used to save all data to a text based file. The .fet type file is used by default.

Read File button: This is used to read an existing amplifier data file with the .fet extension.

Devices area: Power amplifier devices can be added, edited, deleted and copied. To edit, delete or copy a device, first select the device from the drop-down list.

Via area: Vias in the chip are defined. The via plating material is selected from the Material Library. The via can be filled with the selected material or unfilled. Vias can be added, edited, deleted and copied. To edit, delete or copy a via, first select the via from the drop-down list.

Plating: Plating on the top or bottom of any Layer can be defined. Hit the Edit button to add new plating or edit an existing one.

Over Mold: Over mold can be defined by checking the Include checkbox. Next select the Edit button to define the Material, thickness and select the layer defining the bottom location of the over mold.

Air Bridge: Air bridges connecting Source and Drain metal can be defined. Check the Include checkbox then select the Edit button to define the Material and thickness. Source and Drain metallization must be defined to have air bridges.

Layers: Any number of layers can be added under the amplifier chip and attach. Select the Add button to add a layer. Select the Sort Layers button to define the order of the layers under the amplifier chip. The default order is the order they were generated. Layers can be edited, deleted and copied. To edit, delete or copy a Layer, first select the Layer from the drop-down list.

Surface Metallization: Metallization on the top surface of the amplifier chip can be added. There is no limit to the quantity. Select the Add button to add metallization. Metallization can be edited, deleted and copied. To edit, delete or copy a metallization, first select the metallization from the drop-down list.

Location reference: When defining the location of the amplifier chip, it can be relative to the center or the lower left corner of the chip.

Clear Existing Mesh button: If a mesh had already been defined by selecting the Mesh Geometry button, then this will clear the data. This is used if the mesh parameters had been redefined.

Mesh Geometry button: This is used to create the 2D model mesh as well as define the mesh structure through the thickness.

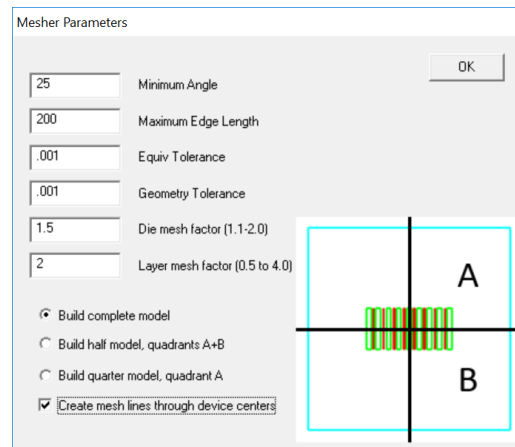
Create Solaria Model button: This button is used after meshing the geometry to create the full 3D Solaria model.

Edit Mesher Parameters button: This button is used to redefine the parameters used to create the 2D mesh as well as the mesh structure through the thickness.

Length Units: The length units defined in Model>Units are defined here. Logical units are Micro-Meters or Microns, um for abbreviation.

Mesh Parameters

Hitting the **Edit Mesher Parameters** brings up the dialog shown below.



Minimum Angle: This defines the minimum internal angle of the triangles created by the mesher. This value should not be changed from 25 degrees as highly skewed triangles can be generated.

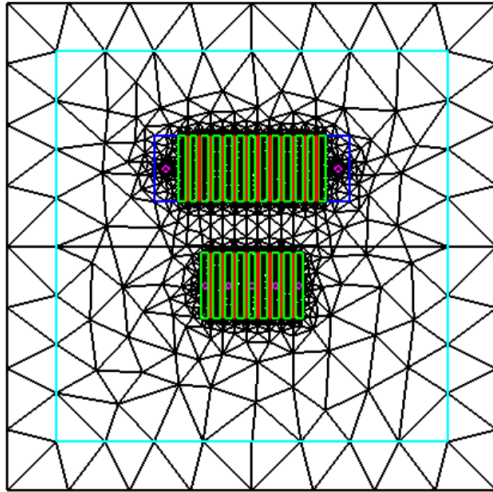
Maximum Edge Length: This defines the maximum edge length of a triangle generated by the mesher. This defines the maximum size of triangles that are generated that are not constrained by geometry. Essentially this defines the overall maximum mesh size. In microns, a value like 100 or 200 is typically good.

Equivalence Tolerance: This is used when cleaning the model, removing Nodes and therefore geometry smaller than this value. Therefore, this value should be at least 10 times smaller than any geometry in the model. For instance, if the Gate Length is 0.25 microns, then this value should be smaller than 0.025. 0.001 is a good value.

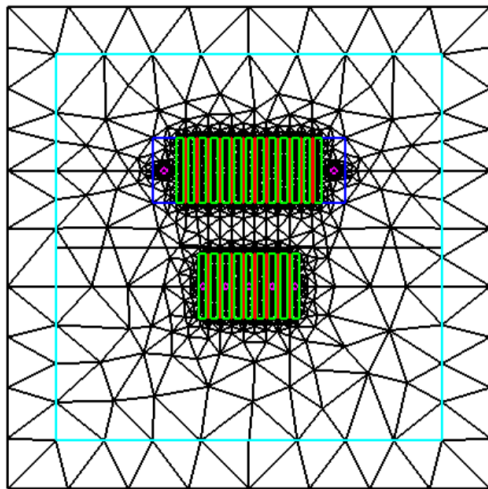
Geometry Tolerance: This is used by the mesher to clean very small geometry. It again should be at least ten times smaller than any geometry in the model, like Gate Length.

Create mesh lines through device centers: Checking this will create mesh lines through the center of each device. This will make it possible to plot the device temperatures versus location in the Solaria model.

In the main dialog, hitting the **Mesh Geometry** button creates the 2D mesh as well as the mesh through the thickness. It is visualized on the screen as shown below.

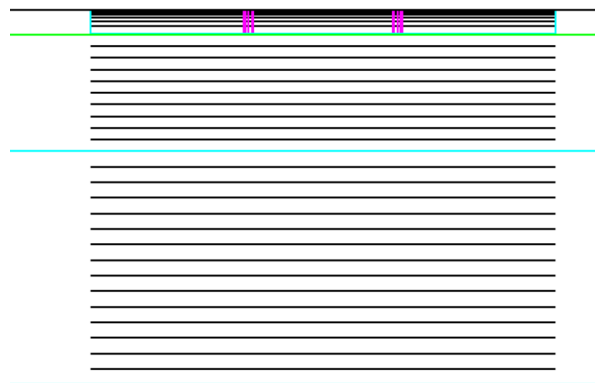


Create mesh lines through device centers was not checked.



Create mesh lines through device centers has been checked.

If you double click on the screen you can select the **XZ** button to view a side view showing the mesh through the thickness, shown below. If a value is changed, hit the **Clear Existing Mesh** then again hit the **Mesh Geometry** button.



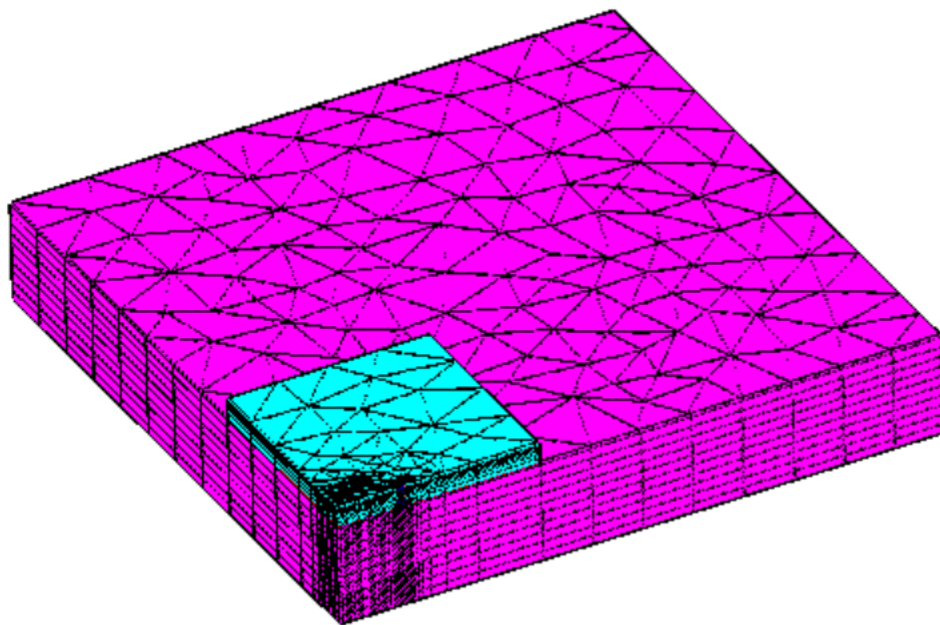
Display of the mesh structure through the thickness.

The following two settings will affect the mesh through the thickness of the device.

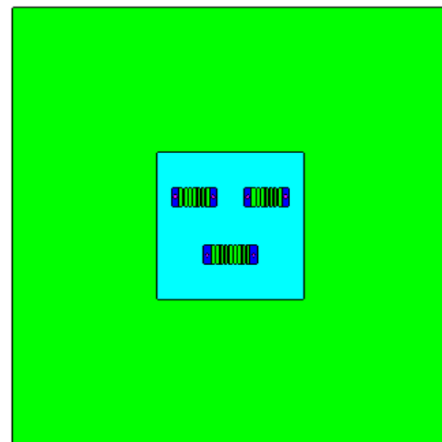
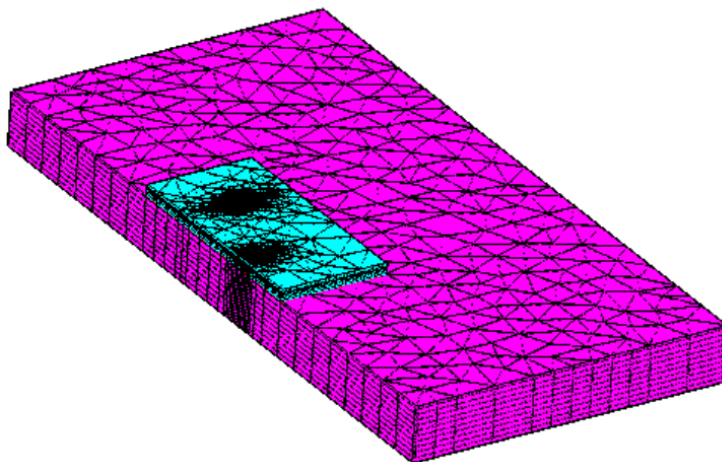
Die mesh factor: This is a factor you can change that affects the mesh of the die through its depth. You can see that the mesh is finer at the die surface. The factor must be between 1.1 and 2.0.

Layer mesh factor: This is a factor you can define that affects the mesh of the layers through their depth. The factor must be between 0.5 and 4.0.

The option is provided to either build the entire model, half the model or a quarter model. Many times there is thermal symmetry therefore not requiring a complete model for accurate results.



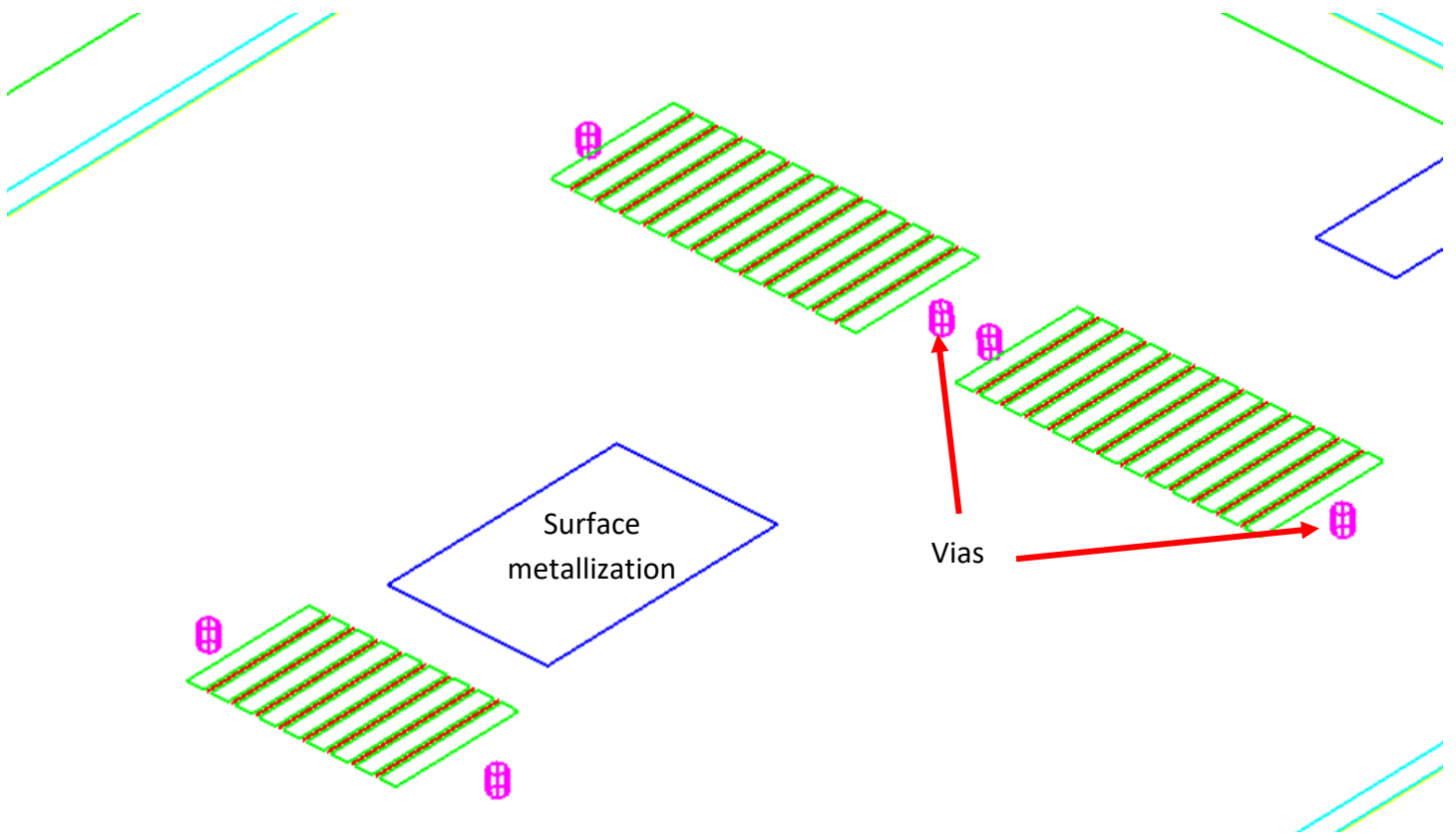
Quarter model showing the chip and one Layer.



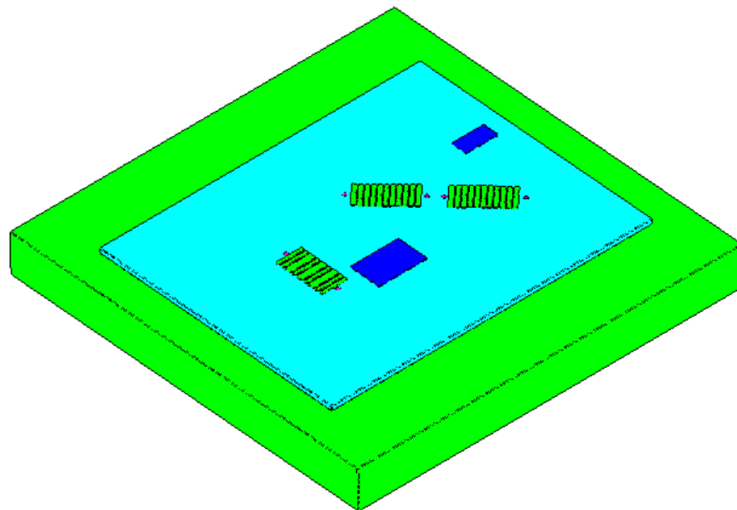
Half model of geometry shown on the right.

General graphics

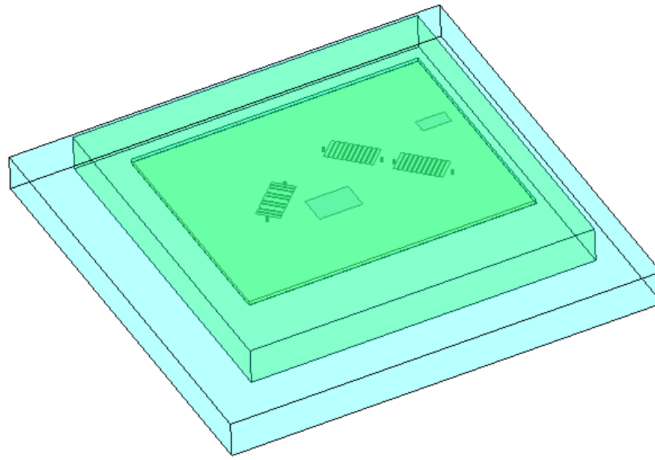
As geometry is defined, the 3D geometry is automatically displayed. The geometry can be revolved and zoomed dynamically.



Rotate dynamically in wireframe or solid view.

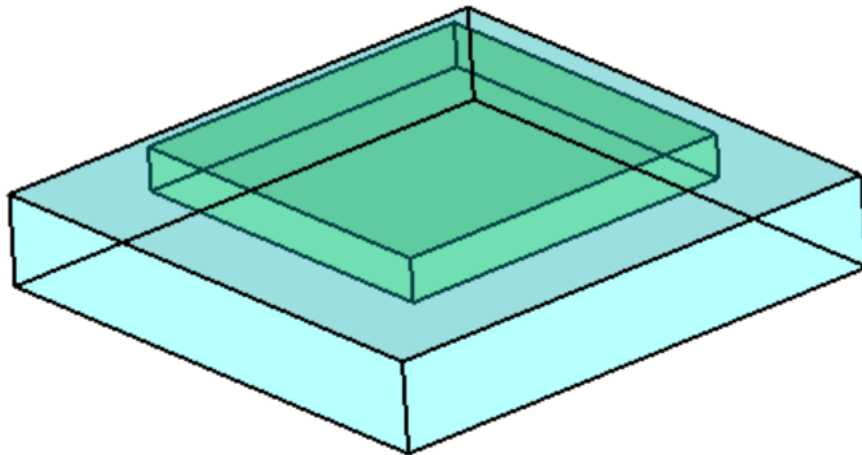


Devices can be any orientation. Vias and Surface Metal can be associated with a Device so they are moved and rotated with the Device.



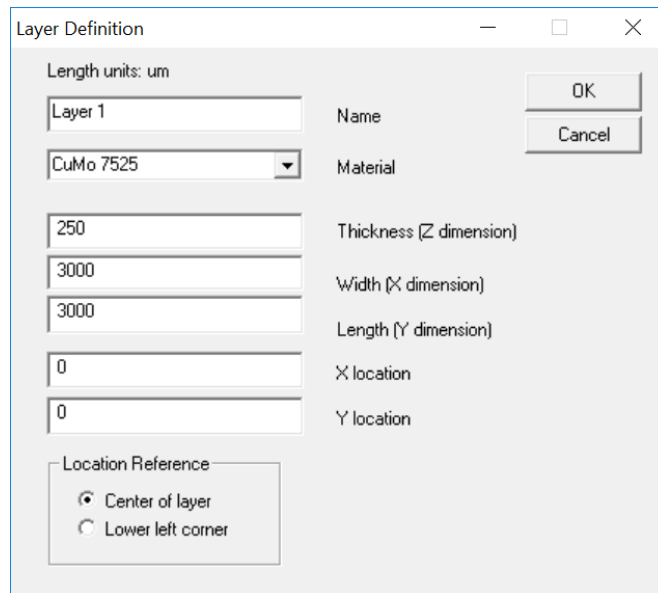
In transparent mode, View>Toggle>Transparent, you can see into the geometry. If you can't see through the materials, go to **Settings>OpenGL>Display** and change the value for **GL Transparency**. A value of 0.15 usually works well.

If you have defined plating on any Layers, they can be viewed in transparent mode. In the figure below, the top of both Layers are gray indicating Plating.



Define Layers

Layers can be added to the model, though they are not required. Any number of layers can be added. Layers are uniform over the area defined. Hitting the Add button in the Layer area brings up the following dialog.



The image shows a 'Layer Definition' dialog box with the following fields and options:

- Length units: um
- Name: Layer 1
- Material: CuMo 7525
- Thickness (Z dimension): 250
- Width (X dimension): 3000
- Length (Y dimension): 3000
- X location: 0
- Y location: 0
- Location Reference: Center of layer, Lower left corner
- Buttons: OK, Cancel

Name: A name is defined for the Layer.

Material: A material is selected from a library.

Thickness (Z dimension): The thickness of the Layer is defined.

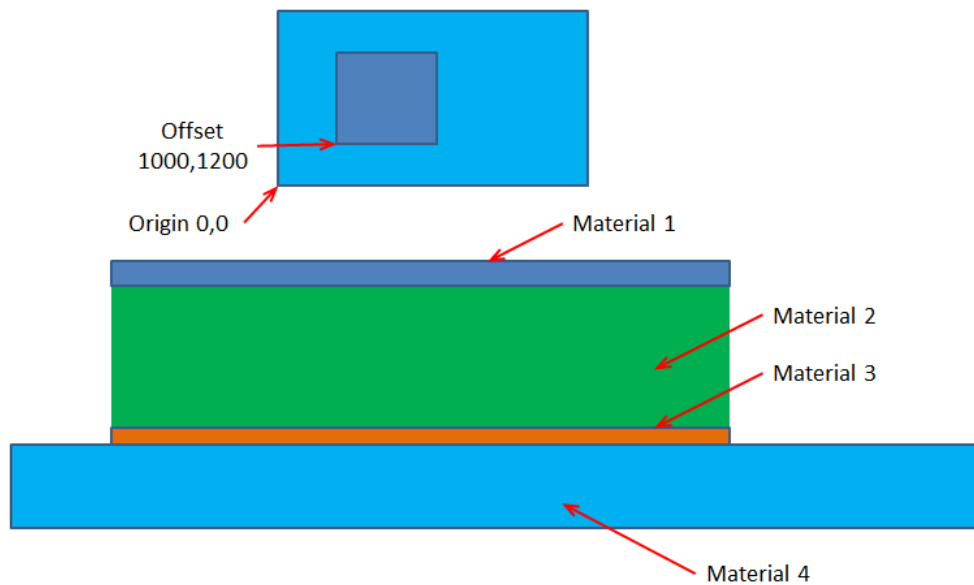
Width (X dimension): The width or the X dimension of the Layer is defined.

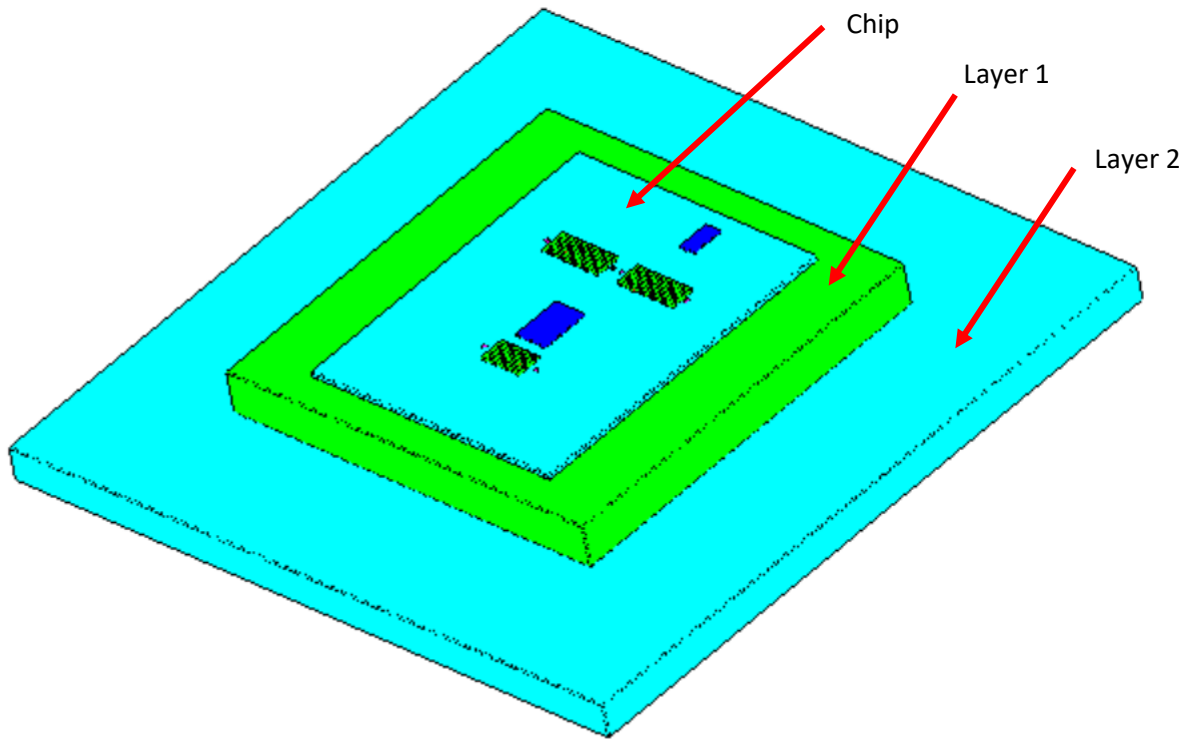
Length (Y dimension): The length or the Y dimension of the Layer is defined.

X location: The offset in the X direction from the origin is defined.

Y location: The offset in the Y direction from the origin is defined.

Location Reference: This defines whether the X location and Y location is respect to the center of the Layer or the lower left corner of the Layer.





Adding Plating to a Layer

To add plating to the top or bottom surface of a Layer, first hit the Add Plating button. Select the material for the plating from the Material drop down list. Enter the plating thickness in the defined units. Select the Layer from the Layer list then define whether the plating is on the top or bottom side of the layer.

Plating Definition

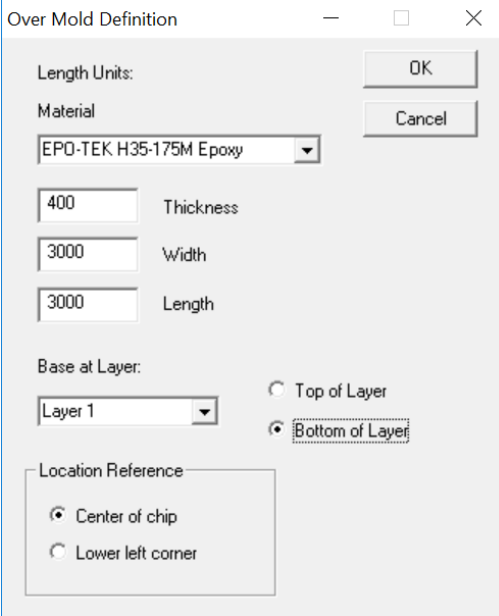
1	Plating to Edit	OK
Gold Pure	Material	Add Plating
10	Thickness (Z dimension)	Cancel
bond	Layer	

Top of Layer
 Bottom of Layer

Length units: um

Over mold

Over mold can be defined by checking the **Include** checkbox in the **Over Mold** area of the main dialog. Hitting the **Edit** button brings up the following dialog.



The image shows a software dialog box titled "Over Mold Definition". It contains several input fields and options:

- Length Units:** A field with an "OK" button next to it.
- Material:** A dropdown menu showing "EPO-TEK H35-175M Epoxy" with a "Cancel" button next to it.
- Thickness:** A text input field containing "400".
- Width:** A text input field containing "3000".
- Length:** A text input field containing "3000".
- Base at Layer:** A dropdown menu showing "Layer 1".
- Location Reference:** A group box containing two radio buttons: "Center of chip" (which is selected) and "Lower left corner".
- At the bottom right, there are two radio buttons: "Top of Layer" and "Bottom of Layer" (which is selected).

Material: A material is selected from a library.

Thickness: This defines the total thickness of the over mold.

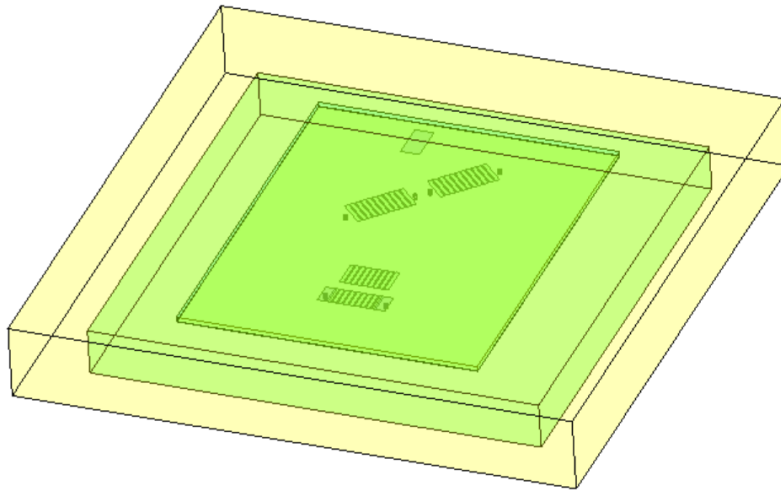
Width: This defines the width in the X direction.

Length: This defines the length in the Y direction.

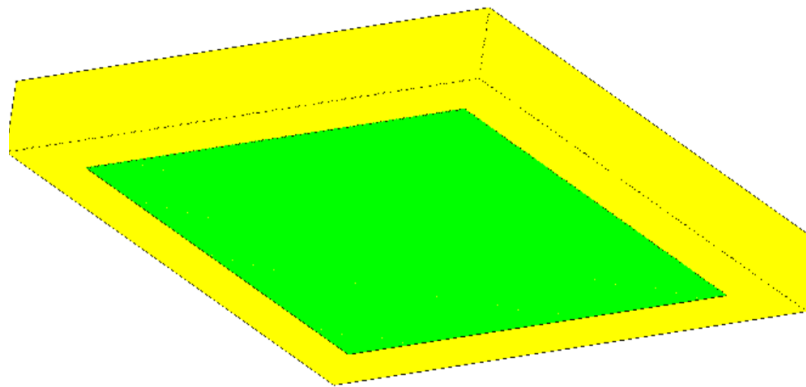
Base at Layer: This defines where the bottom of the over mold goes to. It is defined relative to a defined layer. The Z location can be defined by the top or the bottom of the layer.

Location Reference: This defines whether the over mold location is centered on the chip or the lower left corner of the chip.

When changes are made, the screen graphics are automatically updated. What you see on the screen will be exactly as in the Solaria model built.

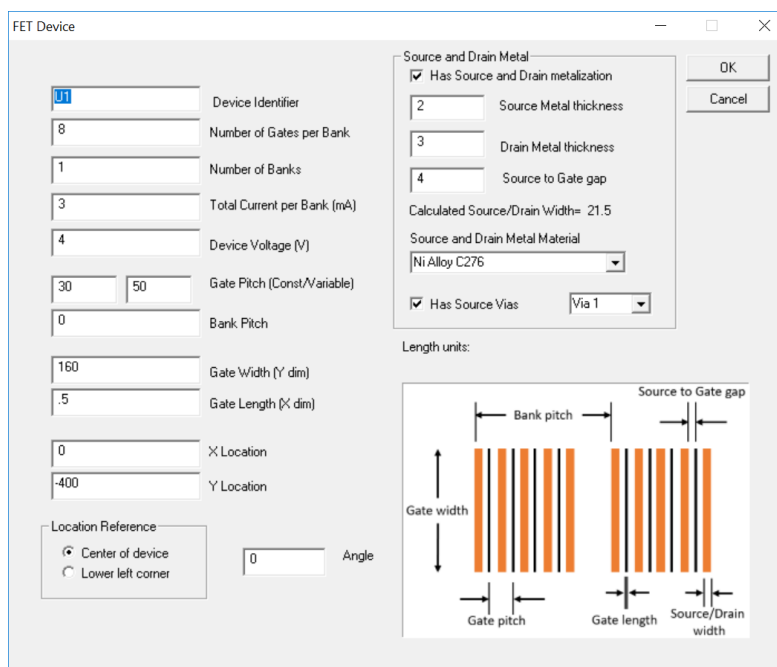


Thermally conductive pad under the chip, exposed to the bottom.



Define amplifier stage (devices)

Hitting the **Add** button in the **Devices** area brings up the dialog shown below.



Device Identifier: This is a name given to the device. It should be unique.

Number of Gates per Bank: This defines how many Gates there are for each Bank defined.

Number of Banks: This defines how many Banks are to be generated. Each Bank will be exactly the same. If Banks are different then they should be created individually.

Total Current per Bank (mA): This is the Current draw per bank in milliamps.

Device Voltage (V): This is the supplied Voltage.

Gate Pitch (Const/Variable): This defined the pitch of the Gates. If they are constant, then the same value should be entered for both. If they alternate, then the two pitches should be entered.

Bank Pitch: This defined the pitch of the Banks. This is not used if there is only one Bank.

Gate Width (Y dim): This defines the width of the Gates. This is the long, Y dimension.

Gate Length (X dim): This defines the length of the Gate. This is the short, X dimension.

X Location: This defines the X or horizontal location of the device.

Y Location: This defines the Y or vertical location of the device.

Location Reference: This defines whether the X location and Y location is respect to the center of the chip or the lower left corner of the chip.

Angle: The Device can be rotated to any angle.

Source and Drain metal:

If the Device has Source and Drain metallization, then check **Has Source and Drain metallization**.

Source Metal thickness: This defines the thickness of the Source metal.

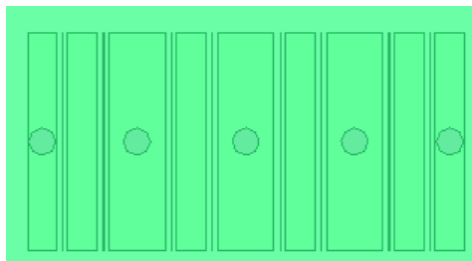
Drain Metal thickness: This defines the thickness of the Drain metal.

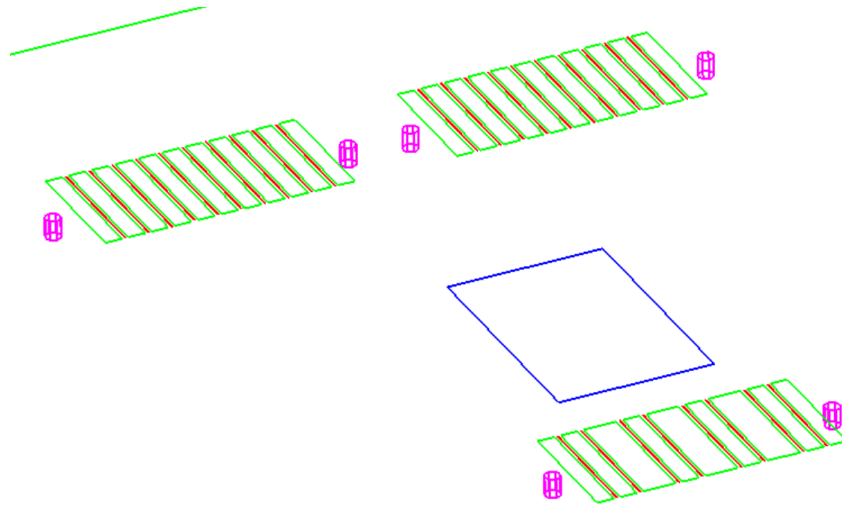
Source to Gate gap: This defines the space between the Source and the Gate.

Calculated Source/Drain Width: This value is automatically calculated based on the data entered.

Source and Drain Metal Material: A material is selected from a library.

Has Source Vias: Checking this checkbox will automatically create vias under the X-Y center of each Source Metal. The geometry of each via is based on the existing via selected from the drop down to the right. This is shown below. Also shown are alternating widths of the Source and Drains. To the right of this checkbox is a dropdown list of existing Vias. A Via must be referenced that defines the geometry of the Via. If there are only Source Vias on the chip, then in the Via definition dialog, check Disable Via. When disabled, the via will not be drawn to the screen and it will only be generated at the Source Vias.

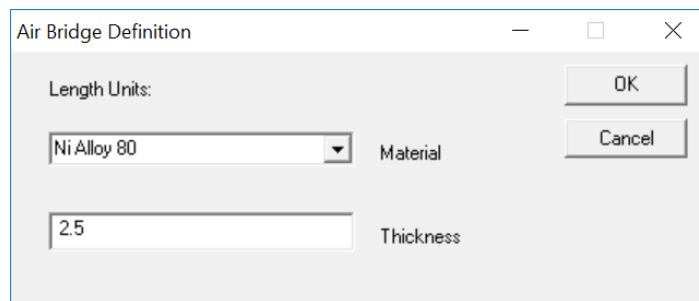




Uniform and non-uniform gate pitch.

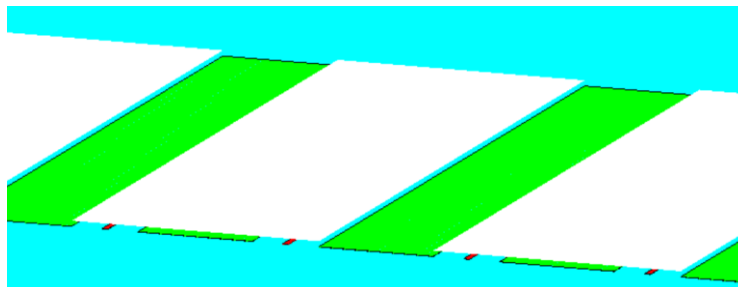
Air bridges

Air bridges can be created between the source metallization. If there are air bridges, check the Include checkbox in the **Air Bridge** area of the main dialog. Hitting the **Edit** button brings up the dialog shown below.



Material: The air bridge material is selected from the Material library drop down.

Thickness: This defines the thickness of the air bridges.



Note that the graphics show the air bridges floating over the gates. In the model they are attached to the Sources and Drains.

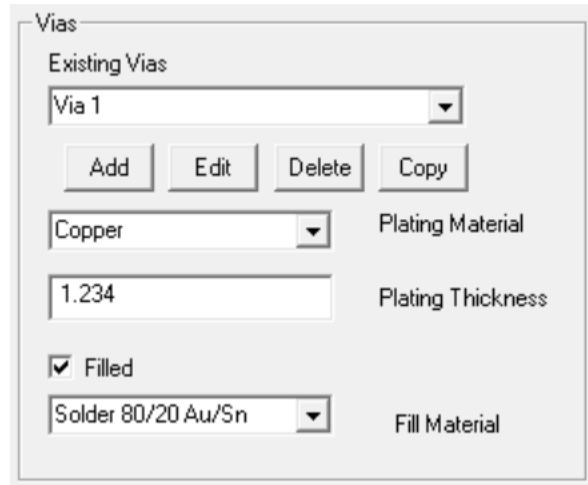
Vias

Any number of vias can be defined. They are defined in the **Vias** area of the main dialog.

Plating Material: The plating material is selected from the Material Library drop down.

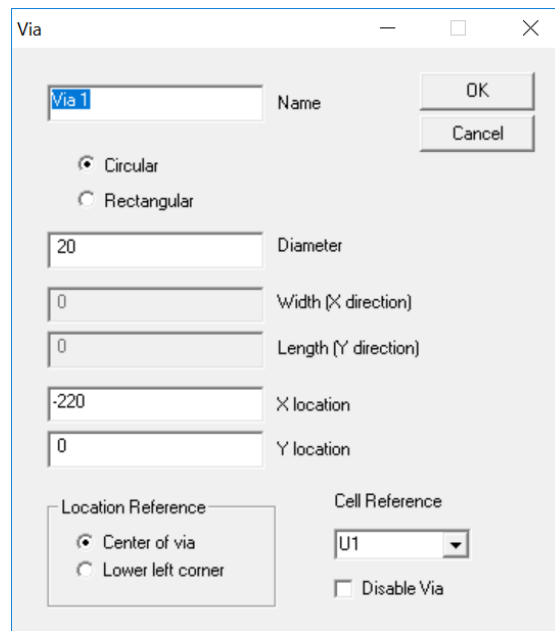
Plating Thickness: The thickness of the vias are defined.

Filled: If the vias are filled, the Filled is checked and the fill material is selected.



The screenshot shows a dialog box titled "Vias". It contains a list of existing vias with "Via 1" selected. Below the list are buttons for "Add", "Edit", "Delete", and "Copy". The "Add" button is highlighted. Below the buttons are fields for "Plating Material" (set to "Copper"), "Plating Thickness" (set to "1.234"), a checked "Filled" checkbox, and "Fill Material" (set to "Solder 80/20 Au/Sn").

Hitting the Add button in the Vias area of the main dialog brings up the dialog shown below.



The screenshot shows a dialog box titled "Via". It has a "Name" field containing "Via 1". There are "OK" and "Cancel" buttons. The "Circular" radio button is selected, and the "Diameter" field is set to "20". The "Rectangular" radio button is unselected, and the "Width (X direction)" and "Length (Y direction)" fields are both set to "0". The "X location" field is set to "-220" and the "Y location" field is set to "0". There are "Location Reference" and "Cell Reference" sections. The "Location Reference" section has "Center of via" selected and "Lower left corner" unselected. The "Cell Reference" section has "U1" selected in a dropdown menu and a "Disable Via" checkbox which is unselected.

Name: This should be a unique name given to the dialog.

Circular/Rectangular: select whether the via is circular or rectangular.

Diameter: If Circular is selected then the via diameter is entered.

Width (X direction): If Rectangular is selected, enter the width of the via.

Length (Y direction): If Rectangular is selected, enter the length of the via.

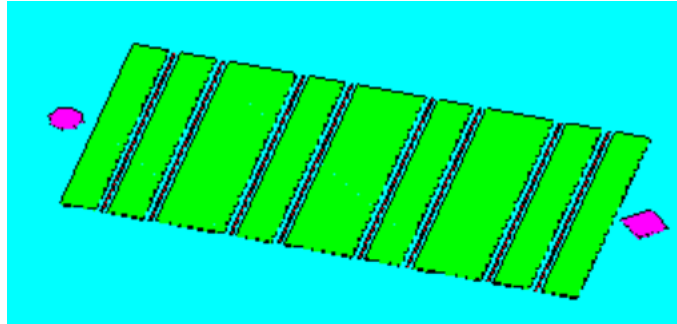
X location: Enter the X location of the via.

Y location: Enter the Y location of the via.

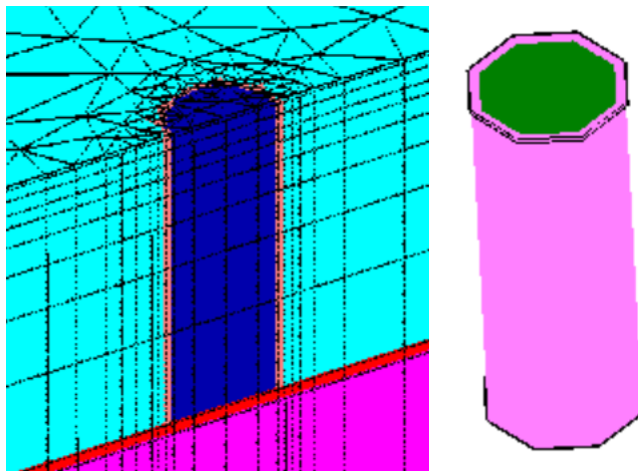
Location Reference: This defines whether the X location and Y location is respect to the center of the via or the lower left corner of the via to the chip.

Cell Reference: The via location can be defined with respect to a device or cell, not the chip. If a cell or Device is selected then if the device is moved or rotated, the via will also.

Disable Via: If the only Vias on the chip are Source Vias, then this Via should be checked as Disabled. The only purpose for this via is to define the geometry for the Source vias.



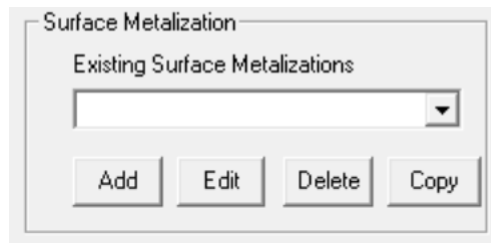
Vias can be round and square.



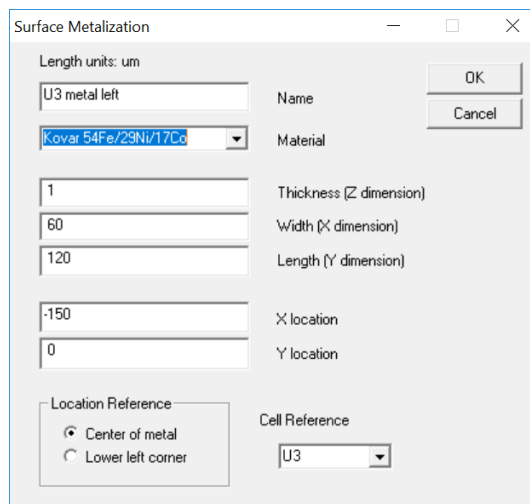
Model of a filled via. The fill material is dark blue and the plating material is orange. The light blue is the chip and the red is the die attach.

Surface metallization

Surface metallization, other than Source and Drain, is defined in the **Surface Metallization** area of the main dialog.



Hitting the **Add** button will bring up the dialog shown.



Name: A unique name should be given to the surface metallization.

Material: A material is selected from the Material library.

Thickness (Z dimension): The thickness of the surface metallization is defined.

Width (X dimension): The width or X dimension is defined.

Length (Y dimension): The length or Y dimension is defined.

X location: Enter the X location of the surface metallization.

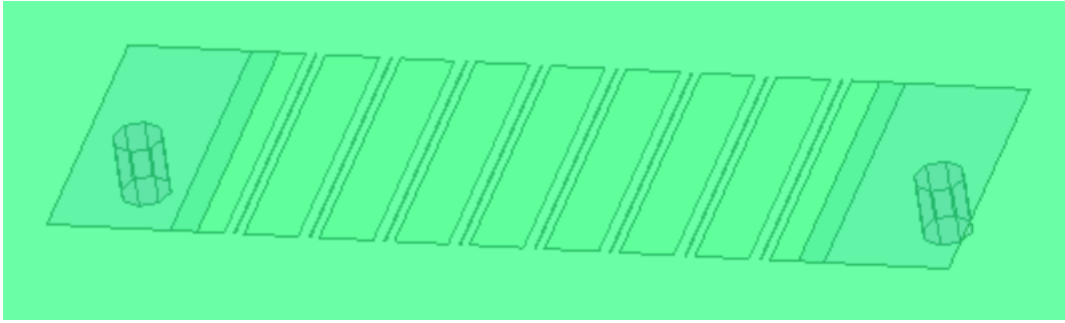
Y location: Enter the Y location of the surface metallization.

Location Reference: This defines whether the X location and Y location is respect to the center of the chip or the lower left corner of the chip.

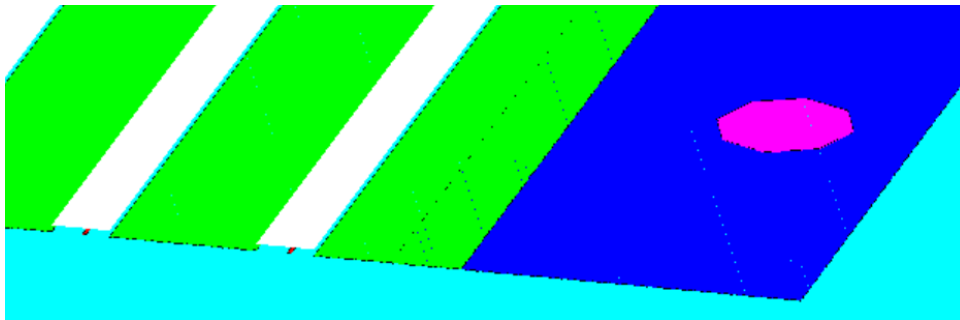
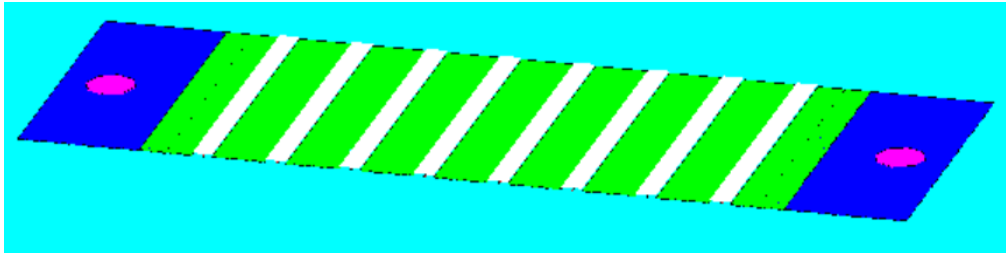
Cell Reference: The surface metallization location can be defined with respect to a device or cell, not the chip. If a cell or Device is selected then if the device is moved or rotated, the surface metallization will also.

Examples

2 Source vias with Air Bridges

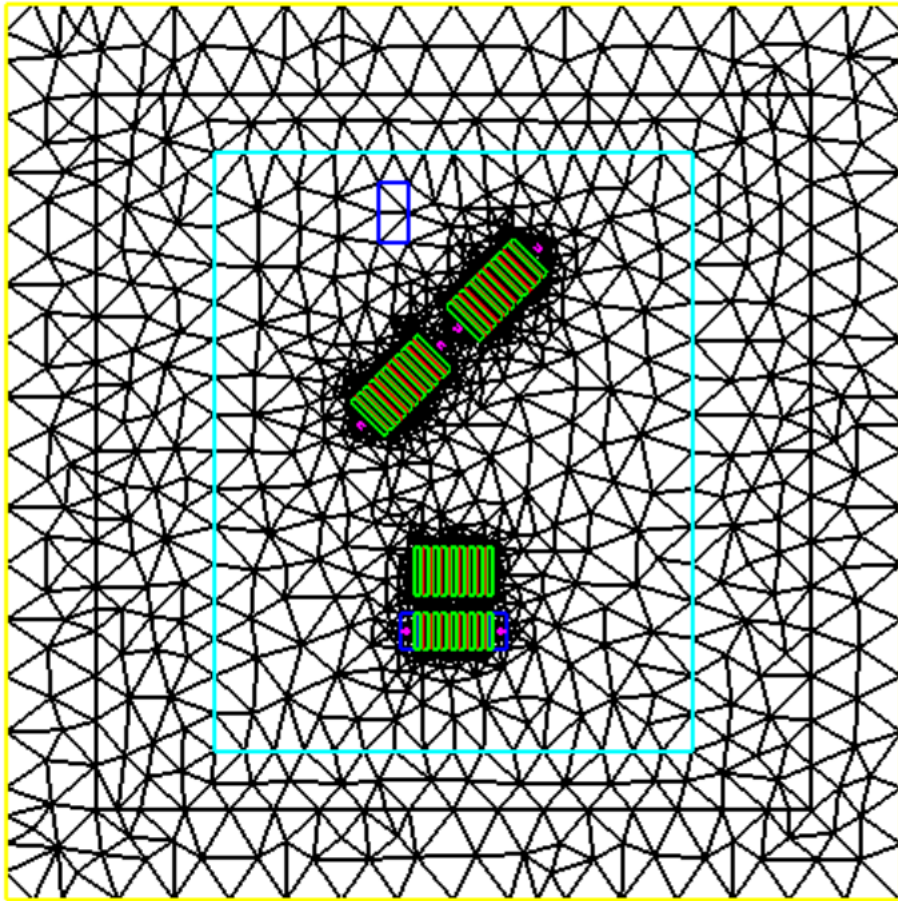


Dark blue surface metallization used to connect the top of the via with the Sources.

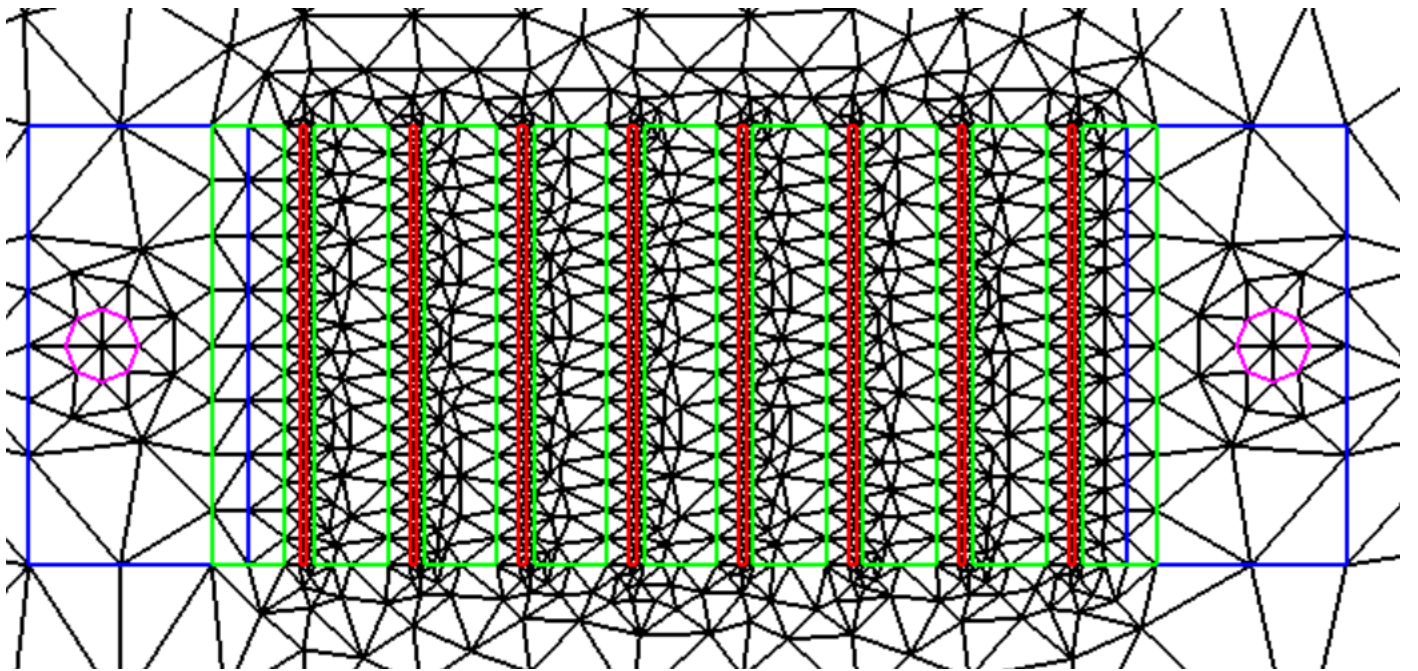


Building the Solaria model

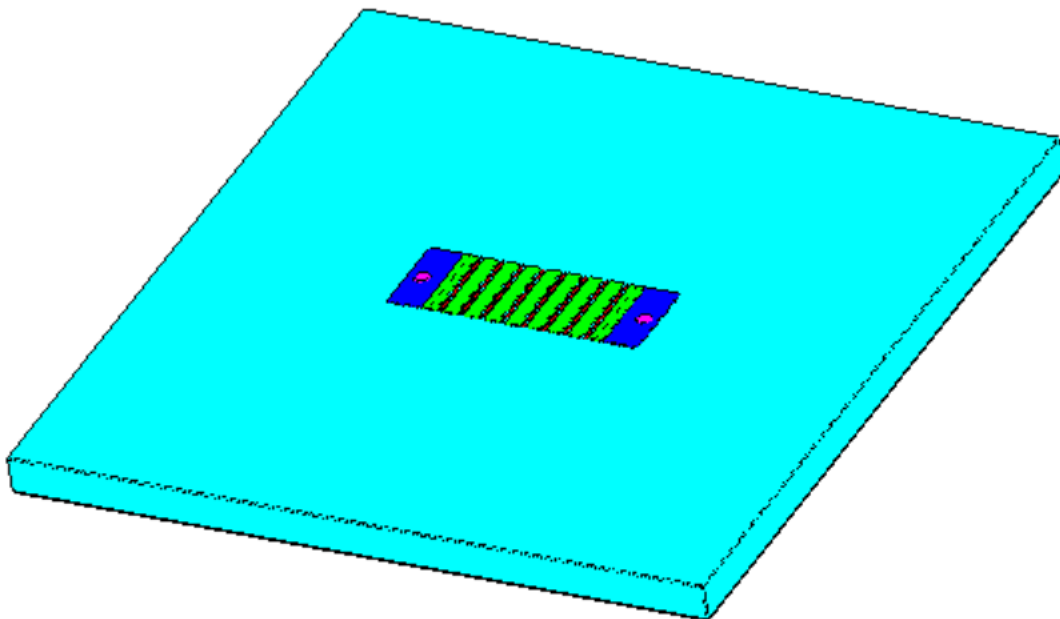
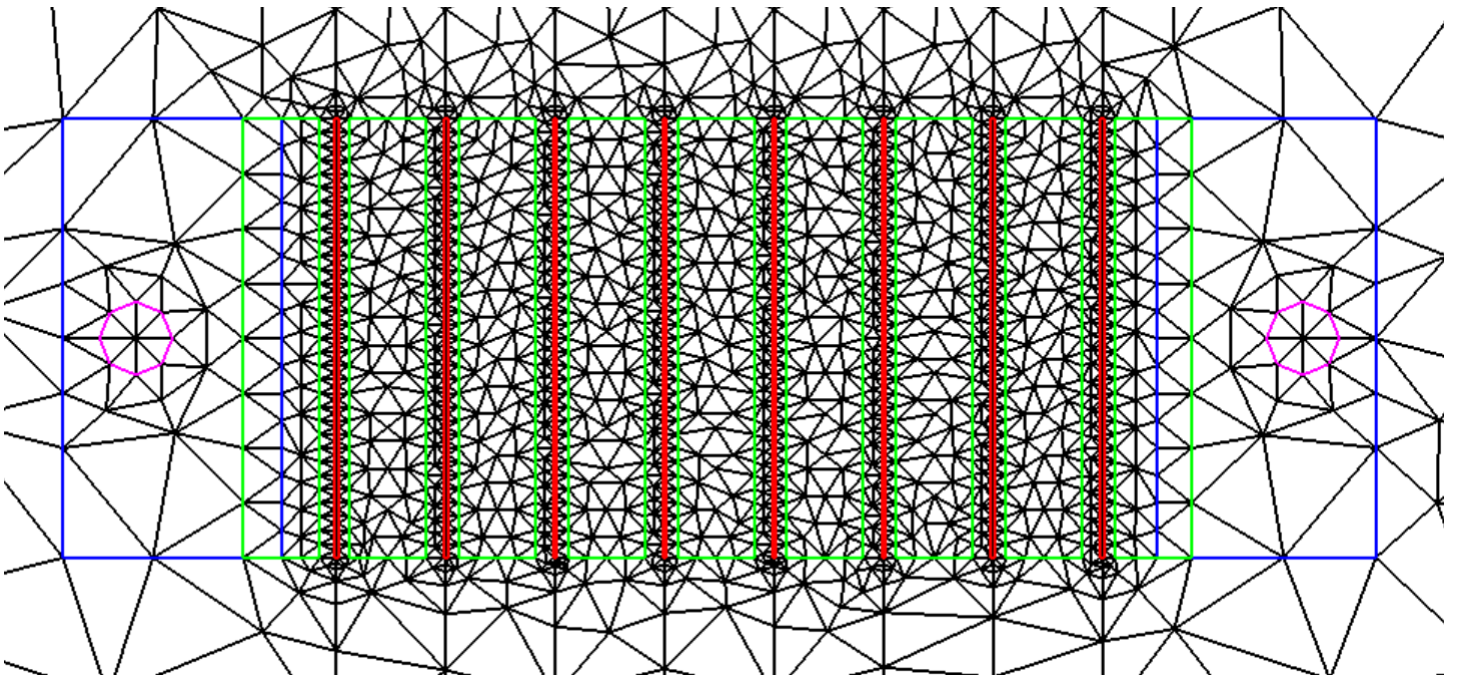
First hit the **Edit Mesher Parameters** button to review the settings. Next hit the **(1) Mesh Geometry** button. You will be notified when it is done. The 2D triangular mesh will be drawn to the screen, as shown below.



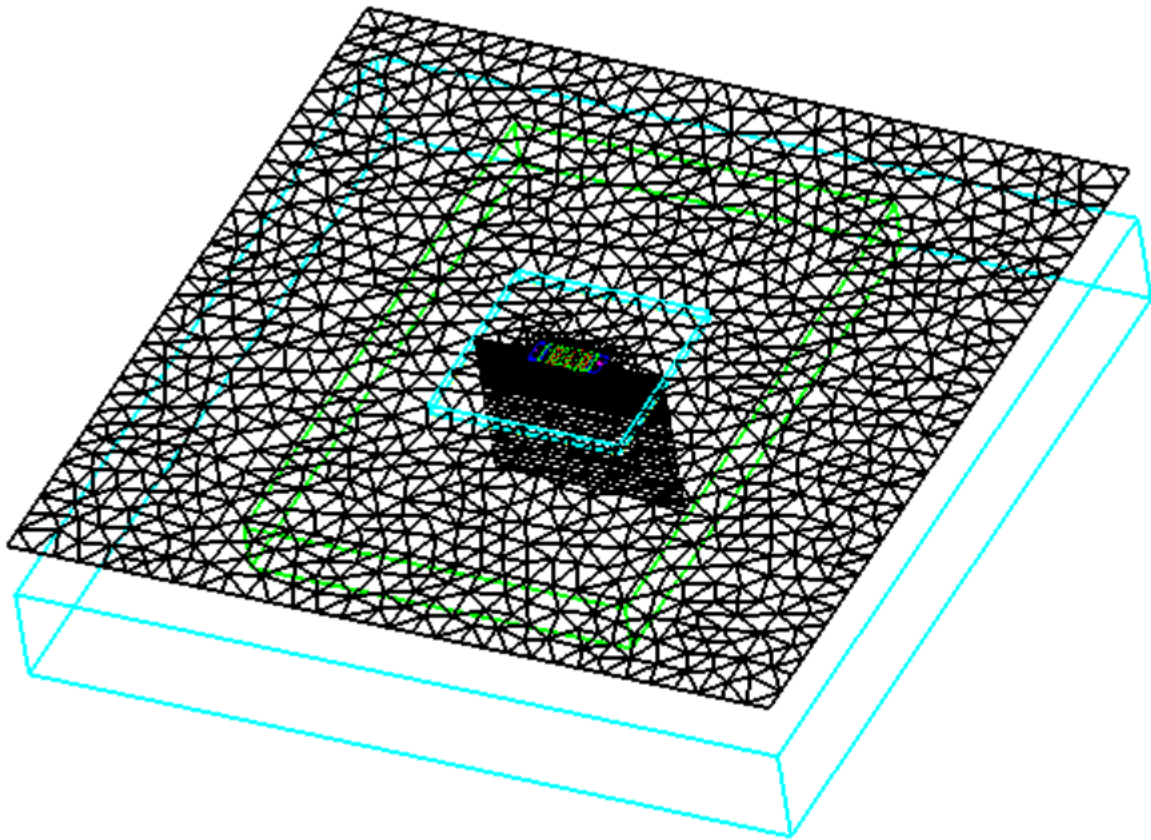
2 micron gate length



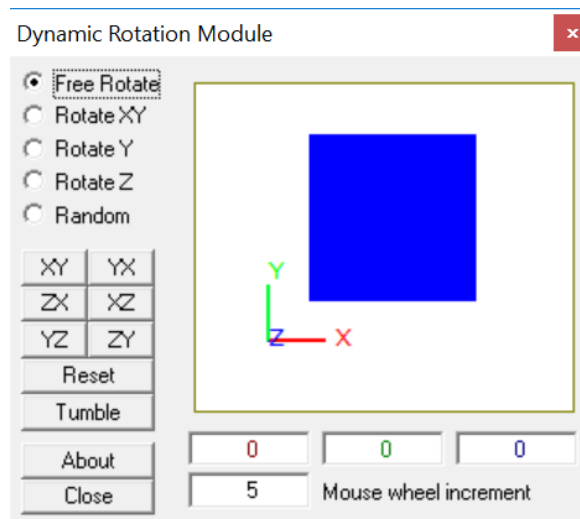
1 micron gate length



After hitting the **Mesh Geometry** button, you can view the 2D mesh and change a mesh parameter if it isn't acceptable. Hit the **Clear Existing Mesh** button to remove the existing mesh.

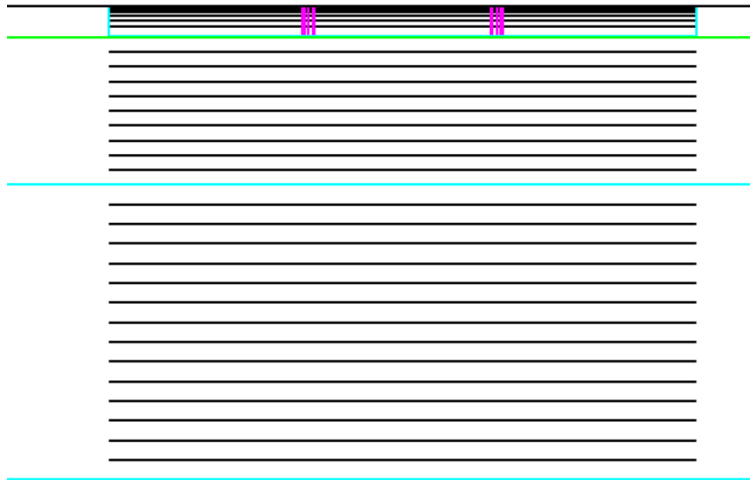


You can also view the mesh through the thickness before building the Solaria model. Double click on the screen to bring up the following dialog. Hit the **XZ** button to see a side view of the device then hit Close.



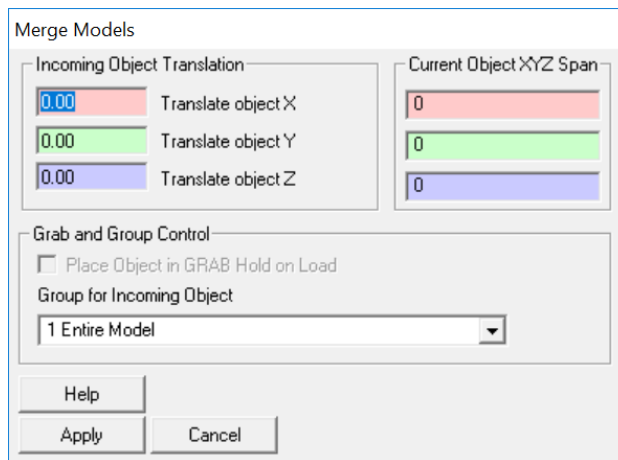
Die mesh factor (1.1-2.0): This value can be changed to change the mesh through the thickness of the chip. It must be between 1.1 and 2.0.

Layer mesh factor (0.5-4.0): This value can be changed to change the mesh through the thickness of the Layer, if there are any defined.





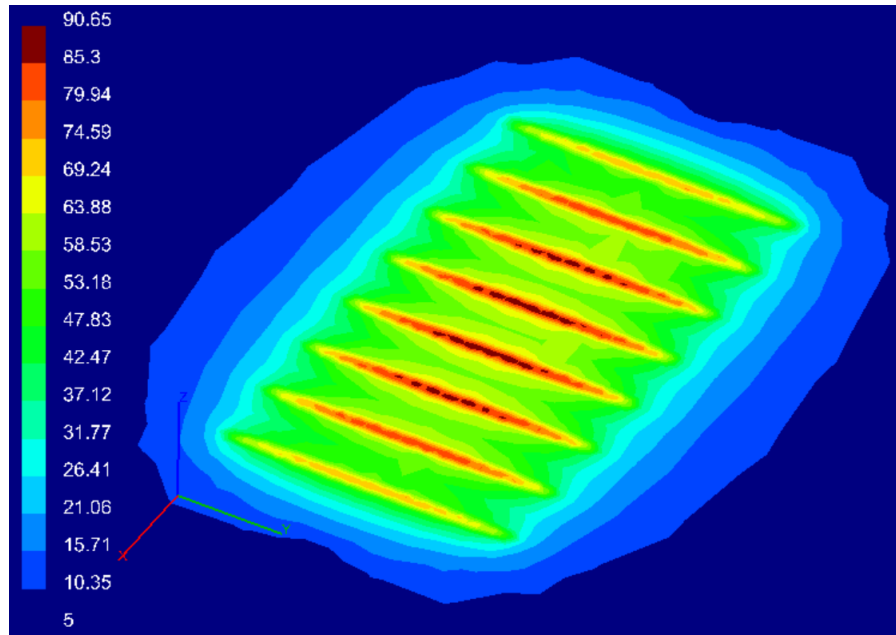
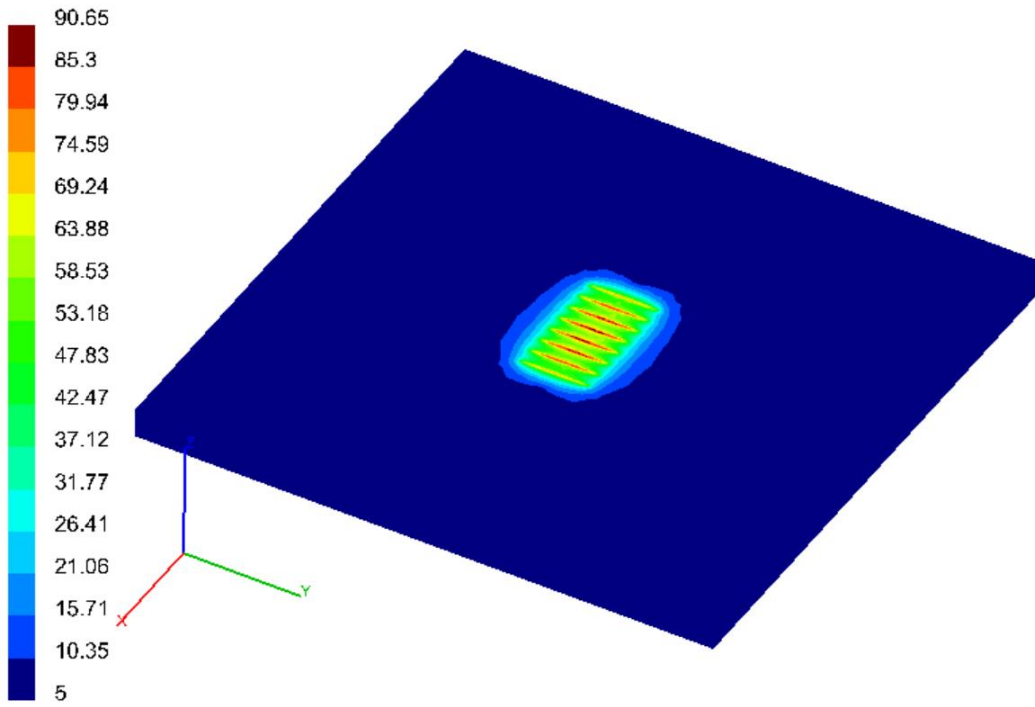
Create the Solaria model

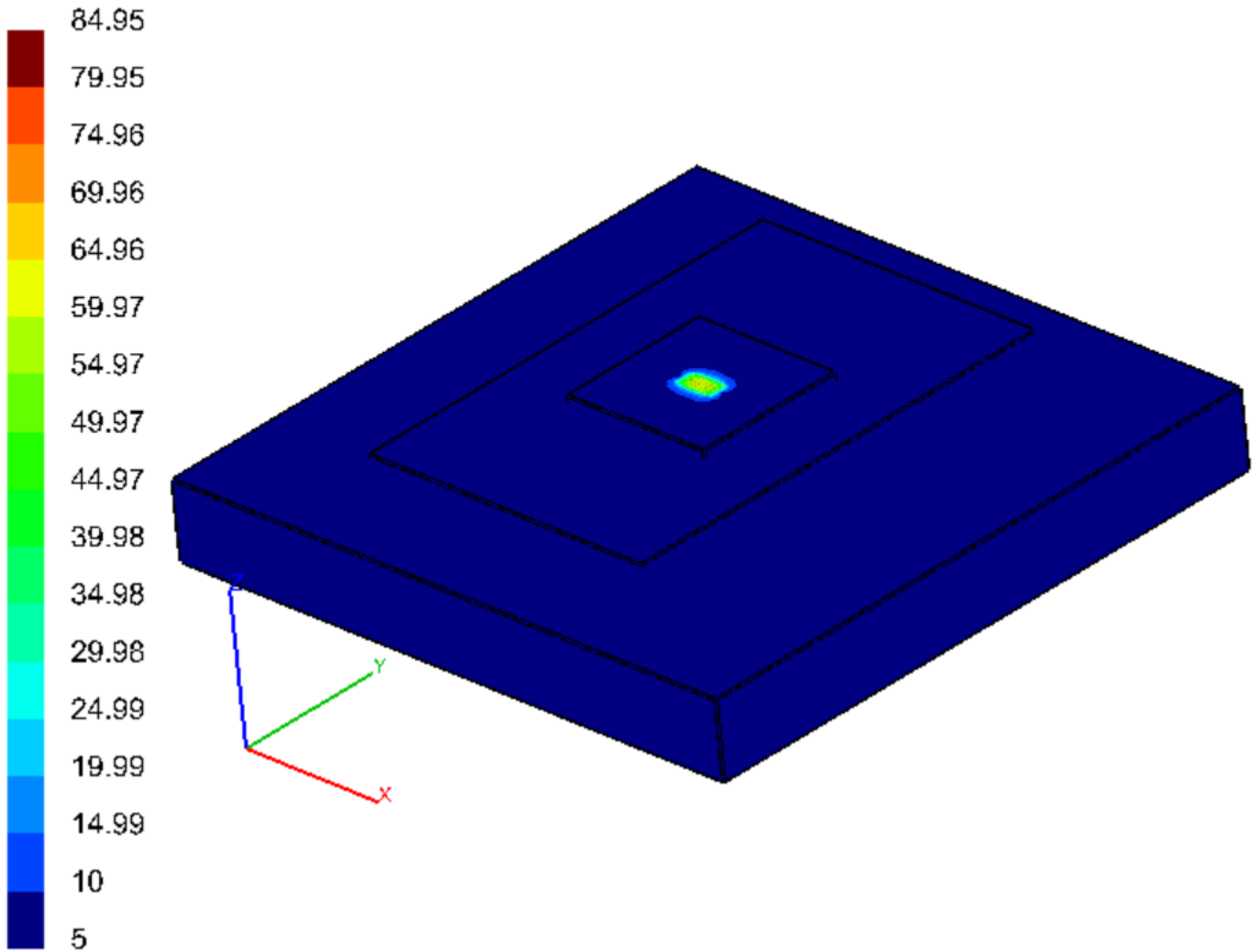
When everything has been defined and the X-Y mesh and the Z mesh are acceptable, hit the **(2) Create Solaria Model** button. The dialog below will come up. If you had a Solaria model on the screen and wish to merge the amplifier model with it, then this dialog is used to define an offset to the amplifier model. Otherwise simply hit the **Apply** button with no changes.



When the model is built, you will be notified. In the main dialog, hit the **OK** button to go back to the Solaria model view. It may be necessary to hit the **View>Reset OpenGL Viewport** to resize the model on the screen.

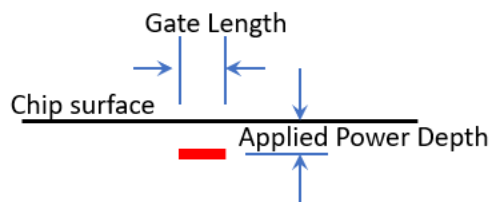
You are now looking at the ready to solve Solaria model. Hit the  button to solve using the Finite Difference solver. It should solve very quickly, seconds. If for some reason it takes some time, you can hit the  button to use the GCG solver.

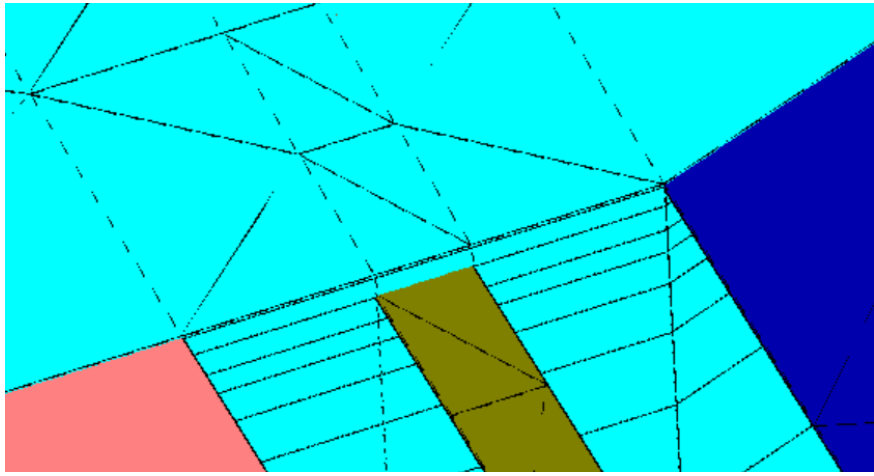




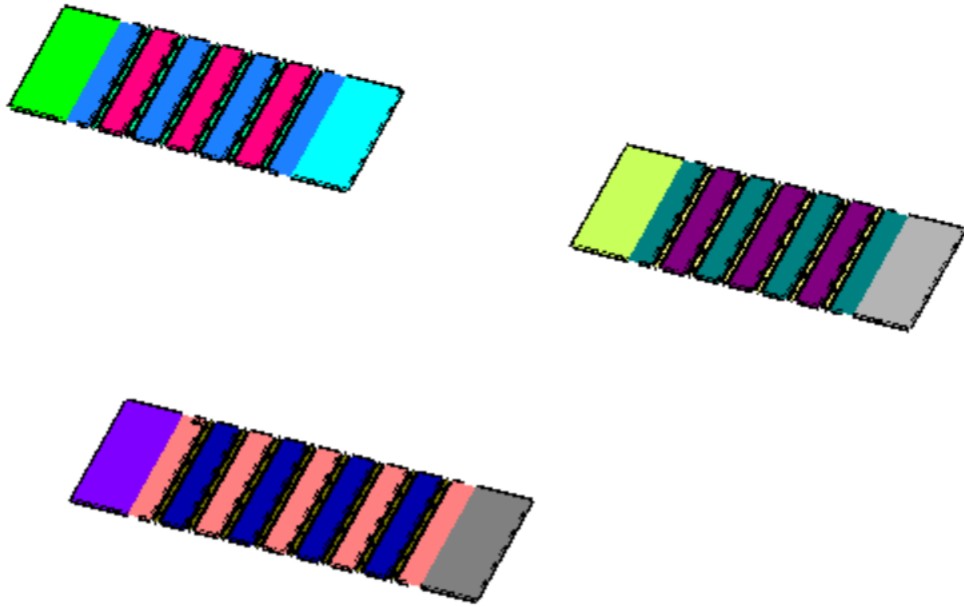
Applied Power Depth

In the example below, the **Applied Power Depth** was defined at 0.5 microns. In the picture, the light blue is the chip, the dark blue and peach are the Source and Drain metal. The green is the heat source which you can see is 0.5 microns below the surface.

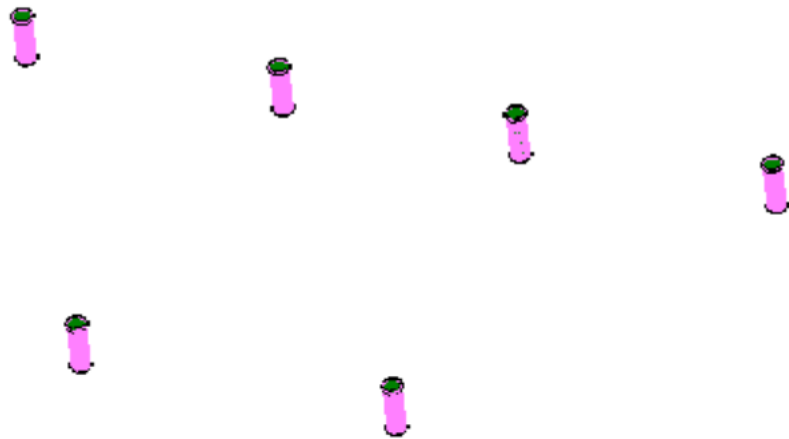




Gate heat below the chip surface



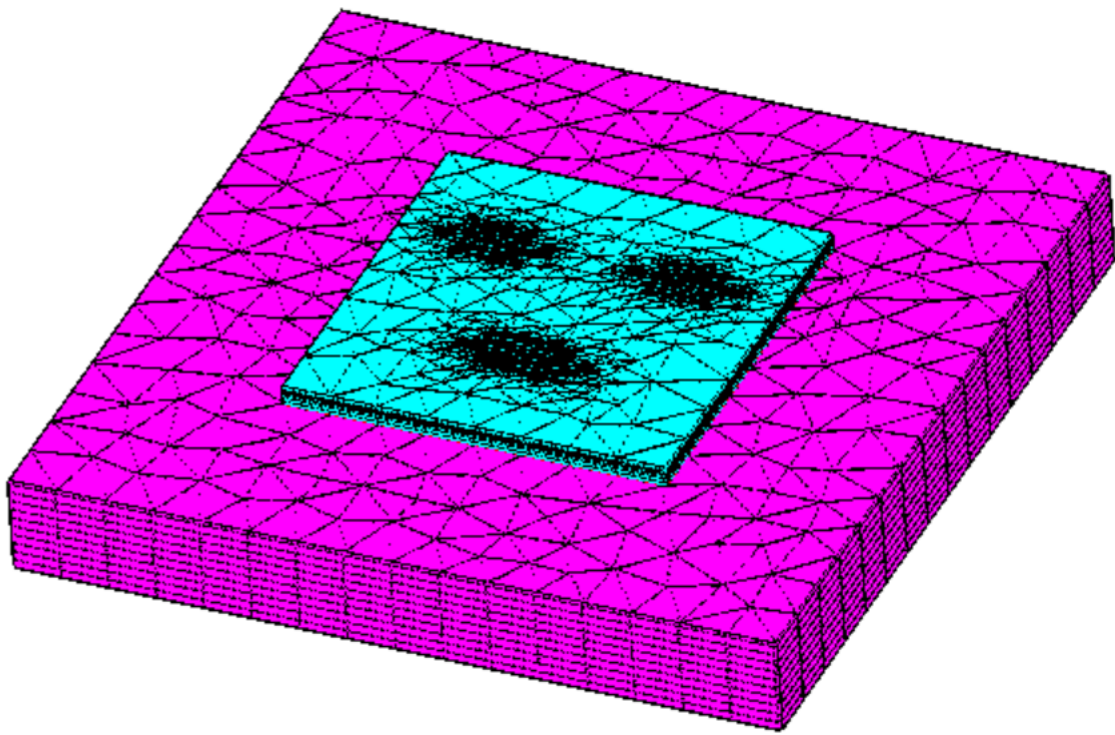
Source and drain metallization with Surface Metallization at the left and right sides connecting the Source vias to the Source metal.

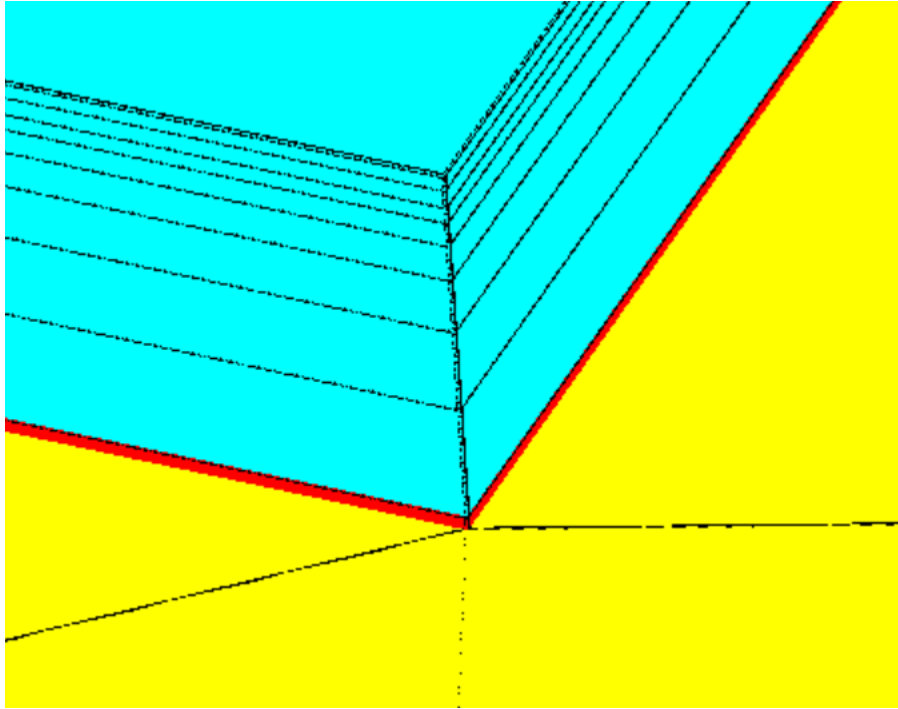


Vias



A single via





Zoomed in showing the chip attach material.

Air Bridge representation in the Solaria model

The picture below shows only Source metal (green) and air bridges (yellow). The air bridge elements actually lie on the same plane as the Source and Drain metallization. But because the air bridges are only one element, they only make thermal contact to the Source metal.

