

# **CFD-ACE+**

Version 2003

## **Microelectronics Tutorial**

**3-03**

### **Heat Transfer in Electronic Chip**

---

- **Building a 3D simulation model of an IC chip using CFD-Micromesh**
- **Setting up a Heat Transfer simulation in CFD-ACE+**
- **Steady-state simulations with surface and volumetric heat sources**
- **Post-processing using CFD-VIEW**

#### **CFD Research Corporation**

Cummings Research Park  
215 Wynn Drive, Suite 501, Huntsville, AL 35805  
Phone: (256) 726- 4800, Fax: (256) 726- 4806  
Software Support: software@cfdr.com, Phone: (256) 726- 4900  
Sales: sales@cfdr.com, www.cfdr.com

© 2003 by CFD Research Corporation  
All rights reserved.

**Objective:** To build a three-dimensional (3-D) simulation model of an Integrated Circuit (IC) chip, and run Heat Transfer simulations using surface and volumetric heat sources, in order to study localized heating inside the chip.

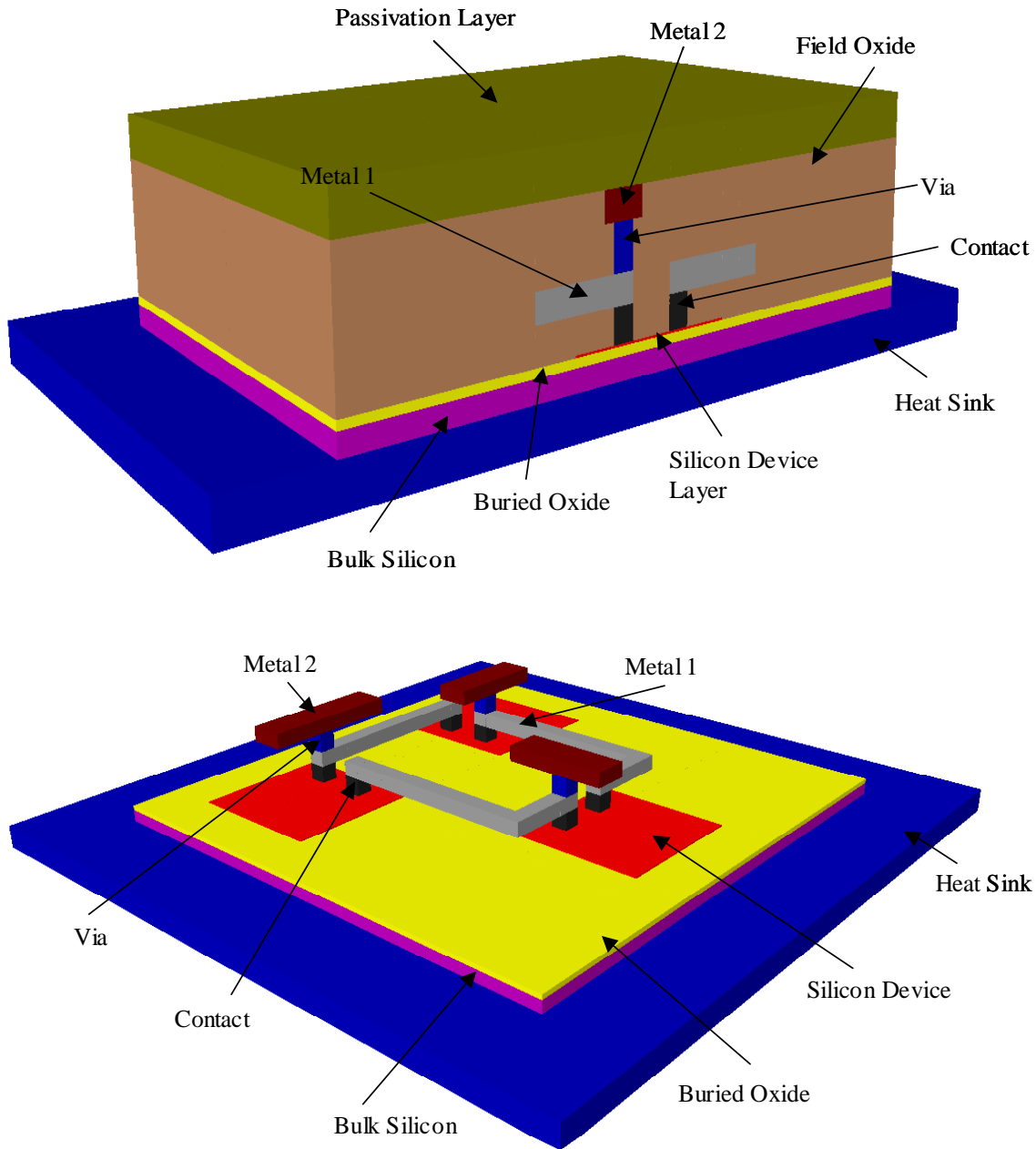


Figure 1. 3D model of the IC chip simulated in this tutorial. Top Figure shows a vertical cross-section through the chip. The vertical dimension has been magnified for better visualization. The bottom Figure the field oxide and passivation layers have been removed to show the interconnects (Metal 1 and 2) clearly.

In this tutorial, we will:

- build a 3D simulation model of an IC chip and automatically generate a computational mesh for it using CFD-Micromesh,
- set the simulation parameters for the Heat Transfer (Heat) module of CFD-ACE+, and run steady-state simulations
- post-process the results using CFD-VIEW to obtain various temperature and heat flux distributions, plots, color contours, cut-planes etc

## **Brief Description of the Problem**

Heat is generated inside an IC chip by active electrical devices. In order to electrically insulate different devices, a dielectric medium (usually silicon dioxide) is used inside the chip. However, silicon dioxide is also a poor thermal conductor, and the heat generated by active devices does not have an efficient path to be carried out. This leads to “localized heat trapping” inside the chip. Since material and electrical properties of semiconductors are temperature dependent, the trapped heat leads to poorer electrical performance of the particular device. If the devices are densely packed inside the chip, this localized heating can affect neighboring devices as well, and adversely affect the electrical behavior of the overall chip.

In this tutorial, we build a 3D model of an IC chip assuming that the manufacturing process is similar to a generic Silicon On Insulator (SOI) process. The buried oxide layer in Figure 1 acts as the insulator over which the thin silicon device layer is present. In the traditional manufacturing process, the buried oxide layer is absent, and the active silicon devices are present as part of the bulk silicon layer.

The heat sink at the bottom is not present in the final simulation model. It is shown in Figure 1 to better illustrate the overall chip structure. In the present simulation, the interface between the bulk silicon and the heat sink is used as a constant temperature boundary condition ( $T=300\text{ K} \Rightarrow$  infinite heat sink). The active devices are assumed to be thin “slices” of silicon as shown in Figure 1. The heat sources are applied either in the entire silicon device volume (volumetric source) or at the upper surface of the silicon devices (surface source), simulating the fact that most of the heat generation occurs in the channel region of transistors. All contacts, vias and metal interconnects are assumed to be copper. The buried oxide and field oxide are both silicon dioxide. The passivation layer at the top of the chip is also assumed to be silicon dioxide.

For Heat Transfer simulations, the following material properties of all constituent materials are required:

- Density
- Specific Heat
- Thermal Conductivity

In the following tutorial these definitions apply:

LB = Left mouse button, MB = Middle mouse button, RB = Right mouse button

### 3D Model Generation and Meshing using CFD-Micromesh

Structures can be built in CFD-Micromesh in two ways – importing layouts already generated by an external layout editor (e.g. CIF or GDSII files), or creating the structure manually. Importing layouts is the better option when the structure is a complicated IC layout with several devices and multiple layers. But if we are interested in building a single device (e.g. transistor or diode), then creating it manually may be more convenient. Both these steps are described in the following sections, though for this Tutorial the structure will be built manually.

#### Importing layouts generated in an external layout editor

1. Start a new project in CFD-Micromesh, by selecting from the top menu: **File -> New**
2. Currently the following layout formats are supported: CIF, GDSII and DXF (optional). Click **Import ->CIF file** (or GDSII file as the case may be) from the top menu. A new window opens up asking if you want to use a “technology file”. Click **No**.
3. A file browsing window appears. Select the layout file to be imported.
4. The layout from the CIF or GDSII file appears in the viewing window. On the right side, a list of layers present in the layout can also be seen. However, the layers in this list do not have their physical names. They have coded layer names (in CIF) or layer numbers (in GDSII). In order to convert these coded names to their physical names, we need to add additional technology information.
5. Convert CIF layer names or GDSII layer numbers into physical layer names, using the process technology information
  - Select (by clicking **LB**) the first layer at the bottom of the list.
  - Click **RB** on the selected layer name  
The “Layer Properties” dialog window opens.
  - Select the **Edit Material** button - the “Materials” dialog window opens
  - Change the CIF “material” names or GDSII layer numbers into new (physical) names, by clicking on **Rename** button in the "Materials" window, and entering the new layer names.
  - Click **Close** in the "Materials" window.
  - Click **Accept** in the "Layer Properties" window.

If you want to see the actual mapping between CIF (or GDSII) layers and CFD-Micromesh layers and operations, that is, the full [Technology Conversion Table](#) for currently active Micromesh project file, select from the menu: **Import -> Tech. Conv. Table...**

6. Click **Import -> Save Technology** from the top menu. A Save Technology window opens up. Save the technology file (i.e. the layer name conversion chart created in the previous step).

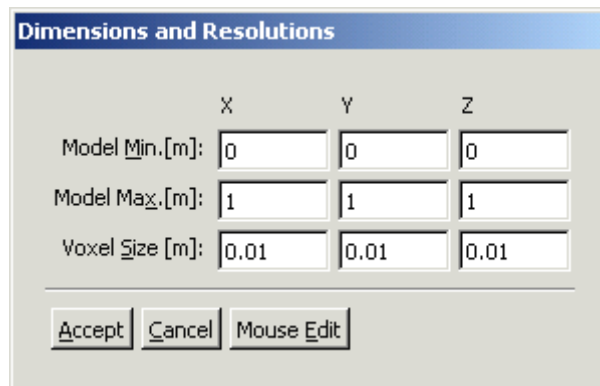
This technology file can be used later to import other CIF or GDSII files associated with the same technology process. In such a case, click **Yes** in response to whether you want to use a Technology File, and select the saved Technology File.

- Once the layout has been imported and physical layer names assigned, we need to define the vertical dimensions of each layer. The process of assigning the vertical dimensions is the same whether we import layouts or build the structure manually, and is described as part of the next section.

## Building a new structure manually

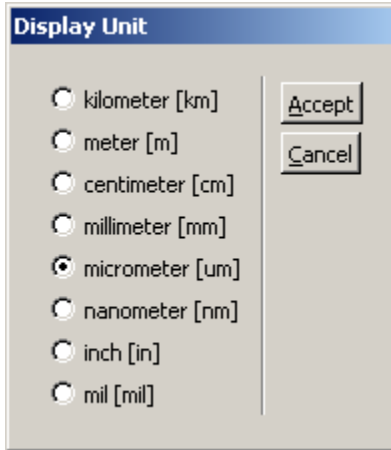
Start a new project in CFD-Micromesh, by selecting from the top menu: **File -> New**

- Setting dimensions and resolutions of your model.
  - Click LB on the Change Boundary button [] in the top toolbar.A "Dimensions and Resolutions" window appears, as below:

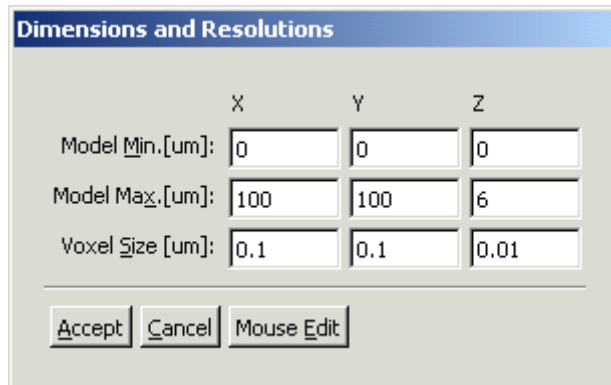


The layout dimensions **Min.[m]** and **Max.[m]** are initially displayed in meters, which may not be convenient in some projects (e.g. microelectronics).

- In such a case, close this window (click **Cancel**)
- Change the units for displaying the project, by clicking in the top menu: **View -> Display Units**. In the "Display Unit" window (see below) you may choose the unit appropriate for your project.
- For our project in this tutorial, let us choose **micrometer** as the unit, and then click **Accept**.




- Re-open the "Dimensions and Resolutions" window by clicking the **Change Boundary** button [□] in the top toolbar. Now type the size of the project, as in the figure below, with dimensions in micrometers.




Note that you have set the three-dimensional resolution (**Voxel Size [um]**) of your model in the above window. The resolution is expressed in voxels, or three-dimensional blocks, analogous to two-dimensional pixels in 2D images. The Voxel Size numbers in this window express the size of each voxel for the three dimensions.

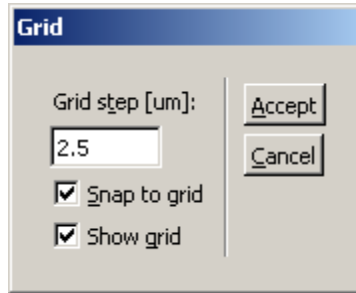
**NOTE:** The voxels must be small enough to enable viewing of the smallest feature desired. The 3D model visualization (ray tracing) procedure in CFD-Micromesh is based on the assumption of cubic voxels. Hence, if their shapes are far from cubic, the viewed proportions of the generated 3D solids will be deformed. However, this does not affect the final model and the computational mesh (DTF file) for the CFD-ACE+ simulation, but only affects the 3D viewing by ray tracing in CFD-Micromesh ("3D Photo").

Decreasing voxel size will result in better accuracy and rendering of shapes of the generated 3D model, but will also increase generation time and the size of intermediate disk files, and at some limits the program may run out of RAM capacity. The minimum voxel size limit depends on the computer configuration (in particular available memory volume). The general rule is: if the model building takes more than 15 minutes (with 700 MHz Pentium or equivalent), the RAM capacity has probably been exceeded and the program is forced to use disk file swapping, which extends the calculation time dramatically. In such a case, stop the current calculation (3D solids generation) by


clicking on "Stop Computation" button [  ] located on the toolbar, re-open the "Dimensions and Resolutions" window (**Tools -> Change Boundary**), and increase the voxel size proportionally in each direction.

9. Setting "Snap to Grid" parameters

Click **LB** on **Change Grid** button [  ] in the top toolbar. A "Grid" window appears, as below:



Change the **Grid step** to 2.5 (micrometers), and click **Accept**.

The grid can be turned on and off by means of the **Show Grid** [  ] button in the top toolbar, or by checking and un-checking the button in the figure above.

10. Creating a layout of the design:



- Click **LB** on the **New Layer** button [  ] in the top toolbar, to create a new layer.

On the right-hand side, you should see a new layer with the name "NEW: AIR (deposit)". At that point, the only existing "material" is "Air" in the empty space of the project area, defined in the previous section.



- Click **RB** on the **NEW: AIR (deposit)** layer.
- In the "Layer Properties" windows that appears, click **Edit Material** button.
- Since we want to create several new layers with different names, click **Add** button in the "Materials" window.
- In the "New Material" window, type the name of the new layers to be added, for example, Name: "Sink", for the Heat Sink at the bottom, and click **Accept**.
- Click **Add** again, type Name: "BulkSi", for the bulk silicon substrate, and click **Accept**.
- In the same manner create names for other layers: BOx (Buried Oxide), FOx (Field Oxide), DevSi (Silicon Device), Contact, Via, Met1 (Metal 1), Met2 (Metal 2) and Pass (Passivation).

**IMPORTANT:** For the Sink layer name above, **UNCHECK** the **Mesh It** box in the "Edit Material" window. This will ensure that the mesh is not generated in the Heat Sink and the heat transfer equations are not solved in these regions (recall that the BulkSi – Sink interface is assumed to be an infinite sink; an isothermal condition of T=300K will be applied there)

- Click **Close** in the "Materials" window.
- Back in the "Layer Properties" window of the first created layer:




- In the **Material**: pull-down list, change the name from AIR to **Sink**.
- Change **Thickness [um]** to 1
- Ensure that **norm n** is 0, **norm beta** is 1 and all other parameters have default values
- Click **Accept**.
  
- Create the Sink block in your model.
  - Draw a layout (top view) of the Sink layer, by selecting the **New Rectangle** button  from the toolbar.
  - While holding left mouse button down and observing the current cursor coordinates in the lower-right corner, draw a rectangle with coordinates (0um, 0um) to (100um, 100um).
  - To check if your Sink layout has the correct dimensions, click **RB** on the red (highlighted) rectangle that you have just drawn. In the "Rectangle Properties" window check if position and dimensions of your rectangle are appropriate. If not, correct them by typing the accurate number (Pos.X = 0, Pos.Y = 0, Size X = 100, Size Y = 100), and click **Accept**.
  
- Create new layer for BulkSi, by clicking LB on **New Layer** button  in the top toolbar.

In the list of layers on the right-hand side, you should see a second layer with the name "NEW: Sink (deposit)" (creating a new layer makes a copy of the currently active layer with all its properties).





  - In the list of layers, click **RB** on the **NEW: Sink (deposit)** layer, and in the "Layer Properties" window:
  - Change the name from Sink to BulkSi in the **Material**: pull-down list
  - Set **Thickness [um]** to 0.5
  - Click **Accept**.
  
- Create the Bulk-Si block in your model.
  - Draw the layout of Bulk-Si by selecting the **New Rectangle** button  from the toolbar.
  - While holding left mouse button down and observing the current cursor coordinates in the lower-right corner, draw a rectangle with coordinates (12.5um, 12.5um) to (87.5um, 87.5um).
  - To check if your Bulk-Si layout has the correct dimensions, click **RB** on the red (highlighted) rectangle that you have just drawn. In the "Rectangle Properties" window check if position and dimensions of your rectangle are correct: Pos.X = 12.5, Pos.Y = 12.5, Size X = 75, Size Y = 75. If they are not correct, type the appropriate numbers, and click **Accept**.
  
- Create new layer for BOx, by clicking LB on **New Layer** button  in the top toolbar.

In the list of layers on the right-hand side, you should see a new layer with the name "NEW: BulkSi (deposit)".







- Click **RB** on the **NEW: BulkSi (deposit)** layer, and in the "Layer Properties" window:
- Change the name from BulkSi to BOx in the **Material:** pull-down list
- Set **Thickness [um]** to 0.2
- Click **Accept**.
  
- Create the BOx block in your model.
  - Draw the layout of BOx, by selecting the **New Rectangle** button  from the toolbar.
  - While holding left mouse button down and observing the current cursor coordinates in the lower-right corner, draw a rectangle with coordinates (12.5um, 12.5um) to (87.5um, 87.5um), i.e. exactly on top of the Bulk-Si layer created earlier.
  - To check if your BOx layout has the correct dimensions, click **RB** on the red (highlighted) rectangle that you have just drawn. In the "Rectangle Properties" window check if position and dimensions of your rectangle are correct: Pos.X = 12.5, Pos.Y = 12.5, Size X = 75, Size Y = 75. If they are not correct, type the appropriate numbers, and click Accept.
  
- Create new layer for FOx, by clicking LB on **New Layer** button  in the top toolbar. In the list of layers on the right-hand side, you should see a new layer with the name "NEW: BOx (deposit)".
  - Click **RB** on the **NEW: BOx (deposit)** layer, and in the "Layer Properties" window:
  - Change the name from BOx to FOx in the **Material:** pull-down list
  - Change the operation type from **Deposit** to **Insert** in the **Operation** pull-down list
  - Set the **Position [um]** to 1.7 and set **Thickness [um]** to 3.1
  - Click **Accept**.
  
- Create the FOx region in your model.
  - Draw the layout of FOx, by selecting the **New Rectangle** button  from the toolbar.
  - While holding left mouse button down and observing the current cursor coordinates in the lower-right corner, draw a rectangle with coordinates (12.5um, 12.5um) to (87.5um, 87.5um).
  - To check if your FOx layout has the correct dimensions, click **RB** on the red (highlighted) rectangle that you have just drawn. In the "Rectangle Properties" window check if position and dimensions of your rectangle are correct: Pos.X = 12.5, Pos.Y = 12.5, Size X = 75, Size Y = 75. If they are not correct, type the appropriate numbers, and click **Accept**.





**NOTE:** The field oxide surrounds all the silicon devices, Metal1 and Metal2 layers. Therefore, we create the field oxide layer first, and then insert the different internal components such as the silicon device layer, Metal1 and Metal2. Also, the Insert operation works better than Deposit when several embedded objects are present.

- Create a new layer for DevSi, by clicking LB on New Layer button  in the top toolbar.  
In the list of layers on the right-hand side, you should see a new layer with the name "NEW: FOx (insert)".
  - In the list of layers, click **RB** on the **NEW: FOx (insert)** layer, and in the "Layer Properties" window:
  - Change the name from FOx to DevSi in the **Material:** pull-down list
  - Set **Position [um]** to 1.7
  - Set **Thickness [um]** to 0.05
  - Click **Accept**.
- Create the DevSi blocks in your model.
  - Draw the layout of DevSi by selecting the **New Rectangle** button  from the toolbar.
  - While holding left mouse button down and observing the current cursor coordinates in the lower-right corner, draw 3 rectangles with coordinates:  
(20um, 60um) to (40um, 80um);  
(60um, 40um) to (80um, 60um); and  
(20um,20um) to (40um,40um)
  - To check if your DevSi layouts have the correct dimensions, click **RB** on the red (highlighted) rectangles that you have just drawn. In the "Rectangle Properties" window check if position and dimensions of your rectangles are correct.
- Create new layer for Contact, by clicking LB on **New Layer** button  in the top toolbar.  
In the list of layers on the right-hand side, you should see a new layer with the name "NEW: DevSi (insert)".
  - In the list of layers, click **RB** on the **NEW: DevSi (insert)** layer, and in the "Layer Properties" window:
  - Change the name from DevSi to Contact in the **Material:** pull-down list
  - Set **Position [um]** to 1.75
  - Set **Thickness [um]** to 0.75 um
  - Click **Accept**.
- Create the Contact blocks in your model.
  - Draw the layout of Contact by selecting the **New Rectangle** button  from the toolbar.
  - While holding left mouse button down and observing the current cursor coordinates in the lower-right corner, draw 6 small rectangles with coordinates:  
(25um, 35um) to (27.5um, 37.5um);  
(32.5um, 35um) to (35um, 37.5um);  
(25um, 62.5um) to (27.5um, 65um);

(32.5um, 62.5um) to (35um, 65um);  
(62.5um, 45um) to (65um, 47.5um);  
(62.5um, 52.5um) to (65um, 55um)

- Create new layer for Met1, by clicking LB on **New Layer** button  in the top toolbar.  
In the list of layers on the right-hand side, you should see a new layer with the name "NEW: Contact (insert)".
  - In the list of layers, click **RB** on the **NEW: Contact (insert)** layer, and in the "Layer Properties" window:
  - Change the name from Via to Met1 in the **Material:** pull-down list
  - Set **Position [um]** to 2.5
  - Set **Thickness [um]** to 0.65 um
  - Click **Accept**
  
- Create the Met1 blocks in your model.
  - Draw the layout of Met1 by selecting the **New Rectangle** button  from the toolbar.
  - While holding left mouse button down and observing the current cursor coordinates in the lower-right corner, draw 5 rectangles with coordinates:  
(25um,35um) to (27.5um,65um);  
(32.5um,35um) to (65um,37.5um);  
(62.5um,35um) to (65um,47.5um);  
(32.5um,62.5um) to (65um,65um);  
(62.5um,52.5um) to (65um,65um)
  
- Create new layer for Via, by clicking LB on **New Layer** button  in the top toolbar.  
In the list of layers on the right-hand side, you should see a new layer with the name "NEW: Met1 (insert)".
  - In the list of layers, click **RB** on the **NEW: Met1 (insert)** layer, and in the "Layer Properties" window:
  - Change the name from Met1 to Via in the **Material:** pull-down list
  - Set **Position [um]** to 3.15
  - Set **Thickness [um]** to 1 um
  - Click **Accept**
  
- Create the Via blocks in your model.
  - Draw the layout of Via by selecting the **New Rectangle** button  from the toolbar.
  - While holding left mouse button down and observing the current cursor coordinates in the lower-right corner, draw 3 rectangles with the coordinates:

(25um, 35um) to (27.5um, 37.5um);  
(32.5um, 62.5um) to (35um, 65um);  
(62.5um, 45um) to (65um, 47.5um)

- Create new layer for Met2, by clicking LB on **New Layer** button  in the top toolbar.  
In the list of layers on the right-hand side, you should see a new layer with the name "NEW: Via (insert)".
  - In the list of layers, click **RB** on the **NEW: Via (insert)** layer, and in the "Layer Properties" window:
  - Change the name from Via to Met2 in the **Material:** pull-down list
  - Set **Position [um]** to 4.15
  - Set **Thickness [um]** to 0.65 um
  - Click **Accept**
  
- Create the Met2 blocks in your model.
  - Draw the layout of Met1 by selecting the **New Rectangle** button  from the toolbar.
  - While holding left mouse button down and observing the current cursor coordinates in the lower-right corner, draw 3 rectangles with coordinates:  
(22.5um,27.5um) to (30um,45um);  
(30um,55um) to (37.5um,72.5um);  
(55um,42.5um) to (72.5um,50um)
  
- Create new layer for Pass, by clicking LB on **New Layer** button  in the top toolbar.  
In the list of layers on the right-hand side, you should see a new layer with the name "NEW: Met2 (insert)".
  - In the list of layers, click **RB** on the **NEW: Met2 (insert)** layer, and in the "Layer Properties" window:
  - Change the name from Met2 to Pass in the **Material:** pull-down list
  - Set **Position [um]** to 4.8
  - Set **Thickness [um]** to 1
  - Click **Accept**
  
- Create the Pass region in your model.
  - Draw the layout of Pass, by selecting the **New Rectangle** button  from the toolbar.
  - While holding left mouse button down and observing the current cursor coordinates in the lower-right corner, draw a rectangle with coordinates (12.5um, 12.5um) to (87.5um, 87.5um).
  - To check if your Pass layout has the correct dimensions, click **RB** on the red (highlighted) rectangle that you have just drawn. In the "Rectangle Properties"

window check if position and dimensions of your rectangle are correct: Pos.X = 12.5, Pos.Y = 12.5, Size X = 75, Size Y = 75. If they are not correct, type the appropriate numbers, and click **Accept**.

**NOTE:** Different layers in CFD-Micromesh should preferably have different colors. This enables easier viewing of the layers while they are being added in the Micromesh Project, and also in the Solid View models generated later. Micromesh provides different colors for the first few layers. To change the color of a layer that has a similar color as another layer:

- click **LB** to select the layer name
  - click **RB** to open the “Layer Properties” window
  - click **Edit Material -> Color**, and select the color of your choice
  - click **Accept**, close the “Materials” window and click **Accept** in the “Layer Properties” window.
  - Save the Project file.
- Save your project, by selecting **Save As...** from File menu (**File -> Save As...**)
    - For example, you may save it under the name *heatchip.ump*.

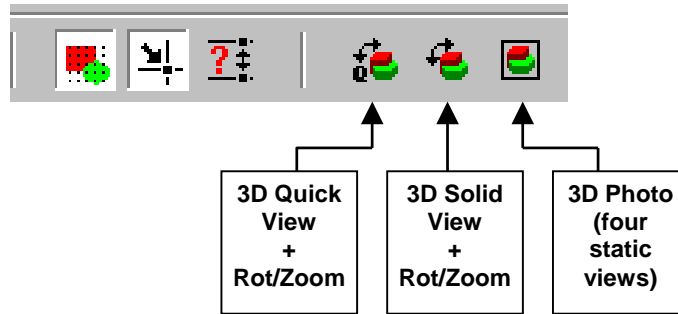
Projects in CFD-Micromesh are saved to the disk in a special format, containing all model data as well as setup parameters for the 3D model and mesh generation for the current project. All these data are saved to a binary file with the extension *.ump* (for *μMesh Project*). Initially, a new project is given a name *untitled1.ump*, to be changed by user during the first saving of the project to disk.

**IMPORTANT:** Before you save files into your working directory, make sure that you can view the complete *file extensions*. From past experience, this is observed to be a source of confusion/errors!


## 11. Generating and viewing a 3D solid model

- There are three methods of generating and viewing a 3D model in CFD-Micromesh:
  - **3D Quick View** – Very fast building of a 3D model from planar elements of defined position and thickness, and viewing the 3D model with zooming and rotation. This option is limited to layouts including rectangular shapes only. Typical model building time: 1-2 seconds (700 MHz Pentium PC).
  - **3D Solid View** – Generating a solid model by geometrical deposition/etching simulation, using voxel (volumetric pixel) technology – this technique allows to generate models of arbitrary shapes, including rounded and oblique side-walls, non-planar conformal layers, etc. Viewing the 3D model allows for zooming and rotation. Typical model building time: 1-5 minutes (700 MHz Pentium PC).
  - **3D Photo** - Generating a solid model by geometrical deposition/etching simulation, using voxel technology (the same as in method 2) – this technique allows to generate models of arbitrary shapes, like rounded side walls, non-planar layers, etc. Visualization procedure uses the *ray tracing* technique that produces up to four different static views (stored as GIF files) of the generated 3D model. Though these are only static snapshots of the 3D model, thanks to ray tracing, they give the highest quality of rendering, with controlled light, shadows, reflections, and anti-aliasing. Typical model and pictures building time: 3-10 minutes (700 MHz Pentium PC).

Each of the above three methods is activated by a separate button in the top toolbar:



- Using the first option “3D Quick View”:  
We will use the “3D Quick View” option since our model consists of rectangular sections only.

➤ Click **LB** on **3D Quick View** button [  ] in the top toolbar.

A new window appears, and it shows the quick view model.

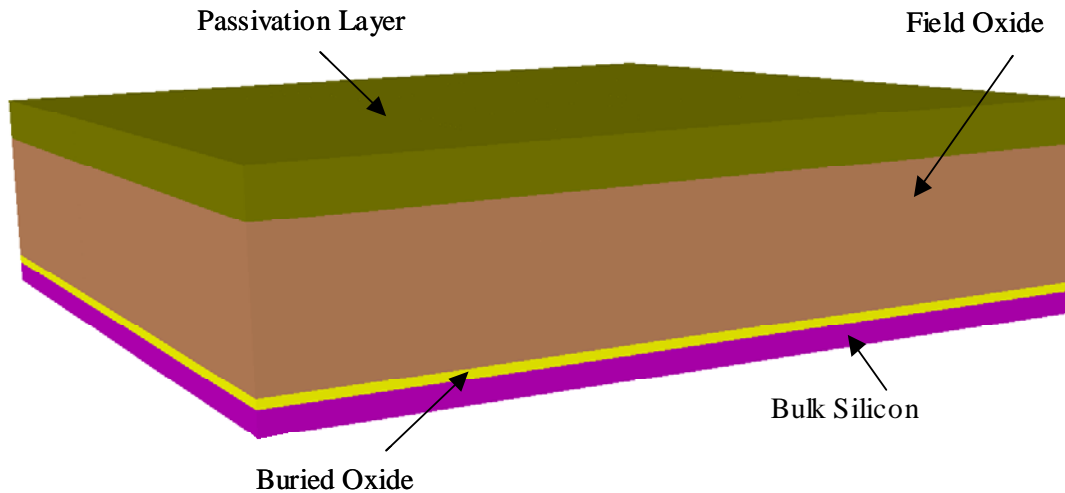

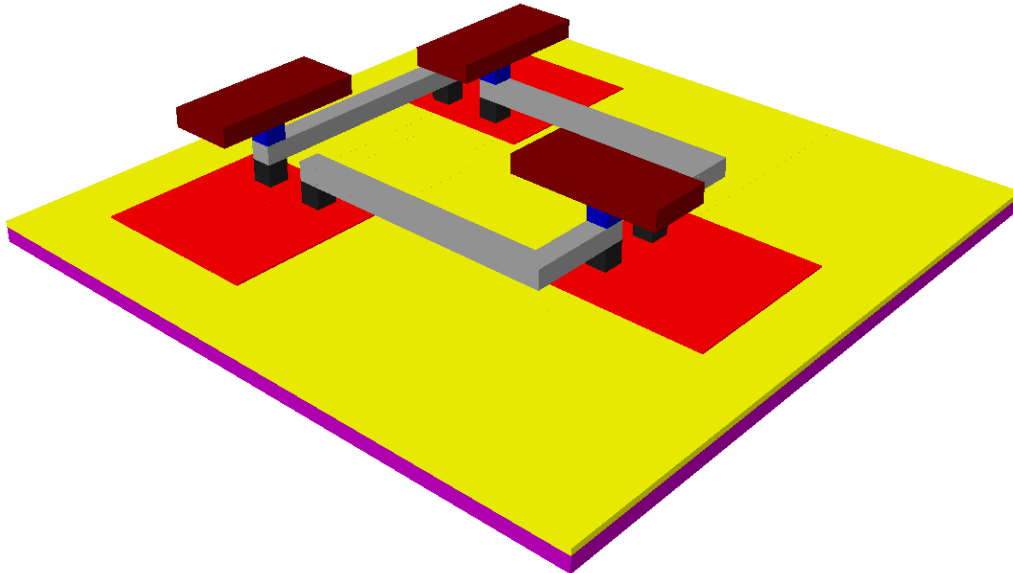


Figure 2. 3D solid model of the IC chip in CFD-Micromesh. Unlike Figure 1, the heat sink does not appear here because we unchecked the “Mesh it” box for the Sink material.


- In the “3D Quick View” window, use **LB** - to rotate the model, **MB** - to zoom in and out the view and **RB** - to shift (pan) the view. Hover the cursor over different layers in the structure to see the layer name.
- To get a better view of the structure, click the **Scale Solid** button [  ] in the top toolbar, enter a value of 4 for the Z direction scaling, and click **Accept**.
- Close the “3D Quick View” window.
- Viewing the “internal” structure, i.e. metal layers, contacts, vias etc.

- Click **LB** on the **FOx** layer name.
- Click **RB** -> **Edit Material**, and uncheck the **Mesh It** box.
- Click **Close**, and then **Accept**.
- Likewise, uncheck the **Mesh It** box for **Pass** layer name.
- Save the Micromesh Project file, and click the **3D Quick View** button in the top toolbar. A solid view of the IC chip, similar to the Figure 1(b), appears in a new window.
- Click the **Scale Solid** button in the top toolbar, enter a value of 3 for Z direction scaling, and click **Accept**. The solid model appears similar to the following figure:



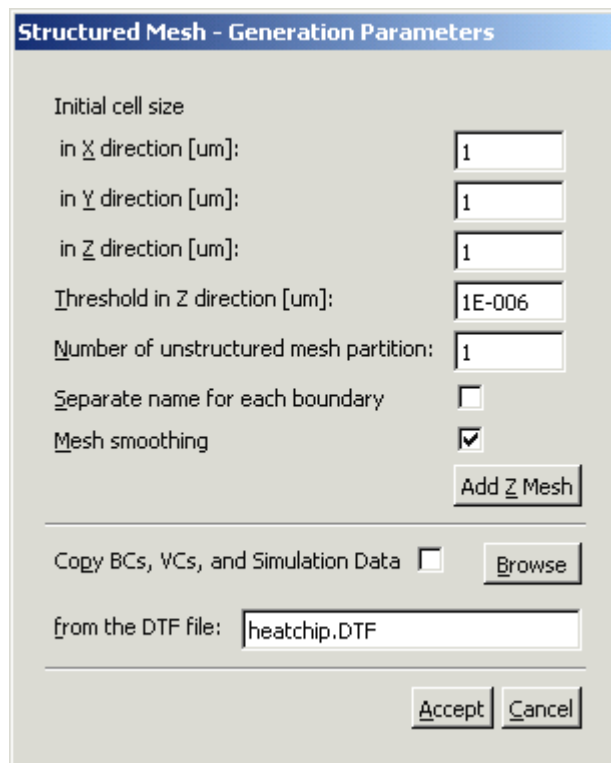
- Close the "3D Quick View" window.
- Re-check the **Mesh It** box for FOx and Pass layer names, and save the Micromesh Project file.

## 12. Automatic building of a 3D computational mesh

- Select the **Structured Mesh** button  from the top toolbar (or from the top menu: **Compute -> Structured Mesh**).

The "Structured Mesh - Generation Parameters" window appears. By setting the numbers in this window, you may control the quality and resolution of the generated 3D mesh that will finally be stored in the DTF file.

- For the mesh of this tutorial, set the parameters as below:




In this tutorial, leave the **Copy BCs, VCs, and Simulation Data** box unchecked. In the first run, boundary conditions (BCs), volume conditions (VCs) and other simulation parameters have to be manually assigned in the DTF file, through the Graphical User Interface (GUI). However, if a change in the device structure or meshing scheme is desired at a later stage, you may copy the simulation parameters (including BCs, VCs) from an existing DTF file by checking the box and specifying the name of the source DTF file.

- Before generation of the model, the user is asked to save the project under a chosen name. Answer **Yes** to the "Save file now?" question.

The progress of the grid generation process is displayed in the "Status" window. The message "COMPUTATION COMPLETED" appears at the bottom when the 3D mesh is ready and has been saved in a DTF file. Note that the DTF file has exactly the same name as the Micromesh Project file we saved earlier (heatchip.DTF in this case)

## Viewing the generated 3D computational mesh (DTF file)


13. Select the **CFD-VIEW** button  from the toolbar.

This button invokes the CFD-VIEW program installed on your machine as part of the CFDRC software. The DTF file just generated by CFD-Micromesh is opened automatically in CFD-VIEW. Please refer to the *CFD-VIEW Manual* from CFDRC for all the details of operation of this program.

14. In CFD-VIEW :

- In the Objects window list in the lower-right corner, select all the items of Type **Wall**.



- From the toolbar above the viewing window, select the following buttons (hover the mouse cursor over the buttons to see their description):
    - **Grid on** and **Flooded on**
  - Select **Scaling** [  ] tab along lower portion of window. Set the Z value to 3. Press Enter on your keyboard.
- Use **LB** to rotate the 3D mesh, **MB** to zoom in or out the figure, **RB** to translate it.  
With a little change in the viewing direction and zooming in on the device, your meshed model in CFD-VIEW should look similar to the one shown below.

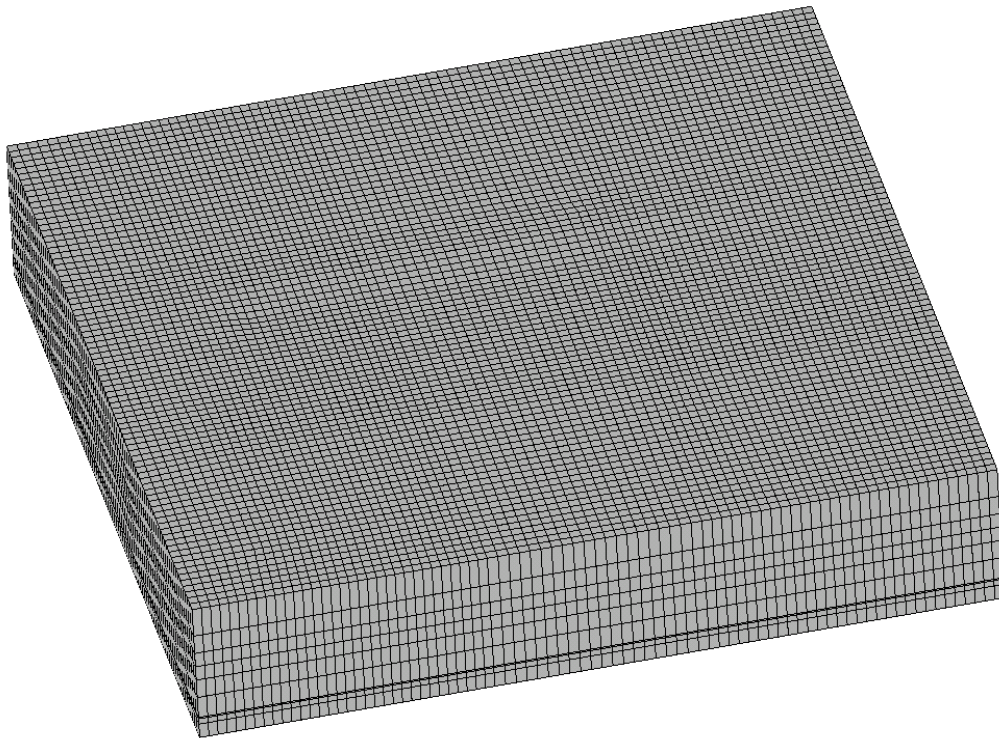



Figure 3. Structured mesh created for the IC chip.

- Close CFD-VIEW by selecting **File -> Quit** and answer **No** to the question “Save current model to file?”

## Setting up a Heat Transfer Simulation

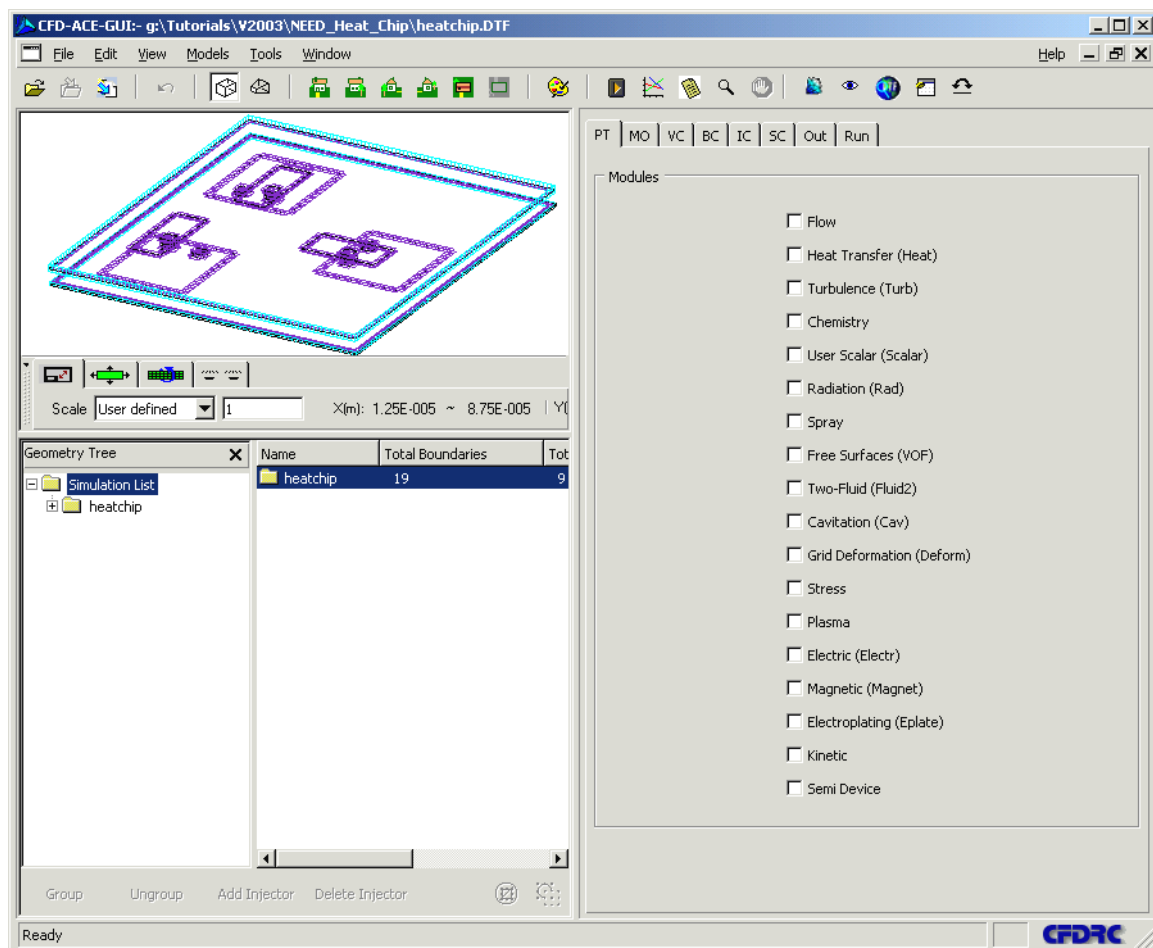
15. Open CFD-ACE-GUI with the generated 3D computational mesh (DTF file)

- Select the **CFD-ACE-GUI** button [  ] from the toolbar in CFD-Micromesh.  
This button invokes the CFD-ACE-GUI program installed on your machine in the CFDRC package. Within CFD-ACE-GUI you can apply boundary conditions appropriate to your

problem, set up analysis parameters, and start simulations. Please refer to the *CFD-ACE(U) User Manual* from CFDRC for all the details of operation of CFD-ACE-GUI.

- When started from Micromesh, GUI should automatically open the DTF file you just created (heatchip.DTF). If it does not open the DTF file automatically, open the DTF file using the menu command: File -> Open ... heatchip.DTF.

After adjusting the view of the 3D meshed model in the GUI viewing window, the device should appear similar to the illustration below.



## 16. Steady-state simulation with a surface source

- **PT** (Problem Type)
  - Check the **Heat Transfer** (Heat) module.
- **MO** (Model Options)
  - In the **Shared** and **Heat** panels leave all parameters at their default settings.
- **VC** (Volume Conditions)

- Check the volume name list at the bottom of the GUI to make sure that all volumes have the **Solid VC** type. If for some reason this is not the case, click **Ctrl-A**. In the **Properties** drop-down list at right, select **Solid**, and click **Apply**.
- Select the BulkSi volume name by clicking **LB** on it. While keeping **Ctrl** pressed, also click **LB** on the DevSi volume name. Both of them should be highlighted. Click the **Group** button at the bottom. (You may be prompted that they already belong to a group. Simply click the **Ungroup** button THEN click on the **Group** button.)
  - In the **Phys** panel:
    - Set the **Density** as  $2330 \text{ kg/m}^3$ .
  - In the **Heat** panel:
    - Set the **Specific Heat** at  $710 \text{ J/kg.K}$ .
    - Set the **Thermal Conductivity** at  $148 \text{ W/m.K}$
  - Click **Apply**.
- Select the BOx volume name by clicking **LB** on it. While keeping **Ctrl** pressed, also click **LB** on the FOx and Pass volume names. All three of them should be highlighted. Click the **Group** button at the bottom. (You may be prompted that they already belong to a group. Simply click the **Ungroup** button THEN click on the **Group** button.)
  - In the **Phys** panel:
    - Set the **Density** as  $2270 \text{ kg/m}^3$ .
  - In the **Heat** panel:
    - Set the **Specific Heat** at  $1000 \text{ J/kg.K}$ .
    - Set the **Thermal Conductivity** at  $1.4 \text{ W/m.K}$
  - Click **Apply**.
- Select the Contact volume name by clicking **LB** on it. While keeping **Ctrl** pressed, also click **LB** on the Via, Met1 and Met2 volume names. All of them should be highlighted. Click the **Group** button at the bottom. (You may be prompted that they already belong to a group. Simply click the **Ungroup** button THEN click on the **Group** button.)
  - In the **Phys** panel:
    - Set the **Density** as  $8920 \text{ kg/m}^3$ .
  - In the **Heat** panel:
    - Set the **Specific Heat** at  $386 \text{ J/kg.K}$ .
    - Set the **Thermal Conductivity** at  $410 \text{ W/m.K}$
  - Click **Apply**.
- **BC** (Boundary Conditions)

You should observe that only four BCs have the Wall BC Type, and the others have the Interface BC type.

- Select the Sink\_BulkSi wall BC name:
  - In the **Heat** panel at right, set the sub-type to **Isothermal** with a constant temperature value of 300 K.
  - Click **Apply**.
- Select the FOx\_DevSi interface BC name:
  - In the **Heat** panel at right, check the **Wall Heat Source** box.
  - Enter a constant **Wall Heat Source** value of  $1E+7$  W/m<sup>2</sup>.
  - Click **Apply**.
- **IC** (Initial Conditions)
  - Leave all parameters at their default values.
- **SC** (Solver Control)
  - In the **Iter** panel, set **Max. Iterations** to 30, and **Convergence Crit.** to  $1e-7$
  - In the **Solvers** panel, change **Enthalpy** solver to AMG
  - In the **Relax** panel, change the **Inertial Relaxation** parameter for **Enthalpy** to  $1E-13$

- **Out** (Output Parameters)

- Under the **Print** sub-panel, check the **Heat Trans. Sum** Box.
- Under the **Graphic** sub-panel, set the output variables you want to see in CFD-VIEW. You may turn off the variables that you do not need.

For example, for the Heat Transfer simulation, you may turn on:

Static Temperature  
Wall Heat Flux

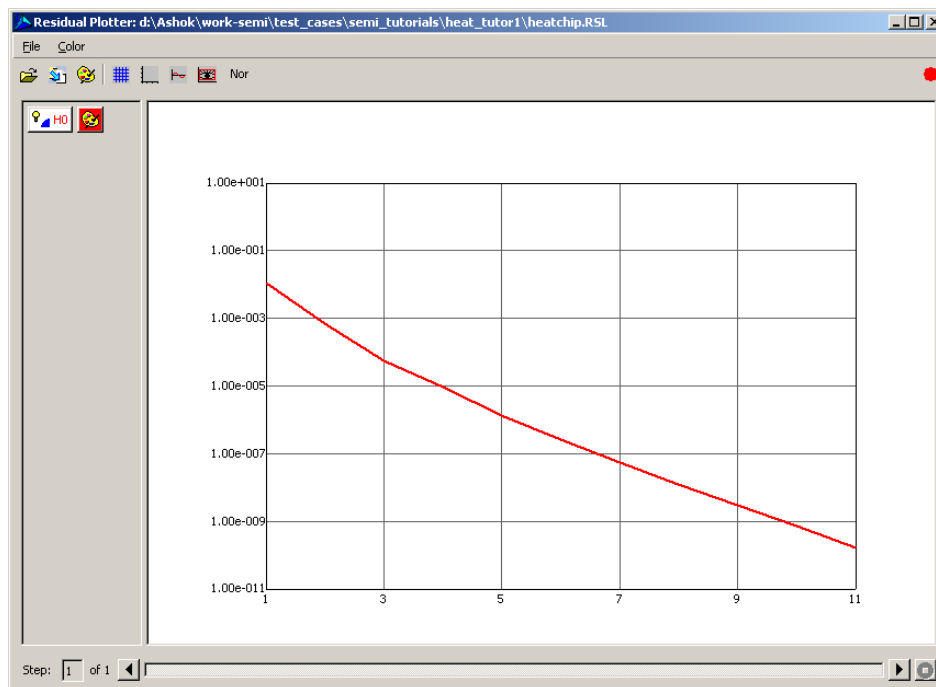
- **Run**

- Click **Submit to Solver**.

A new window opens up asking you whether you want to submit the simulation under the same name or a different name.

- Click the **Submit Job Under Current Name** button.
- Click **View Residual**.

If the values of the residuals eventually go down by 5-6 orders of magnitude, it typically means the calculations converge properly, as illustrated below.

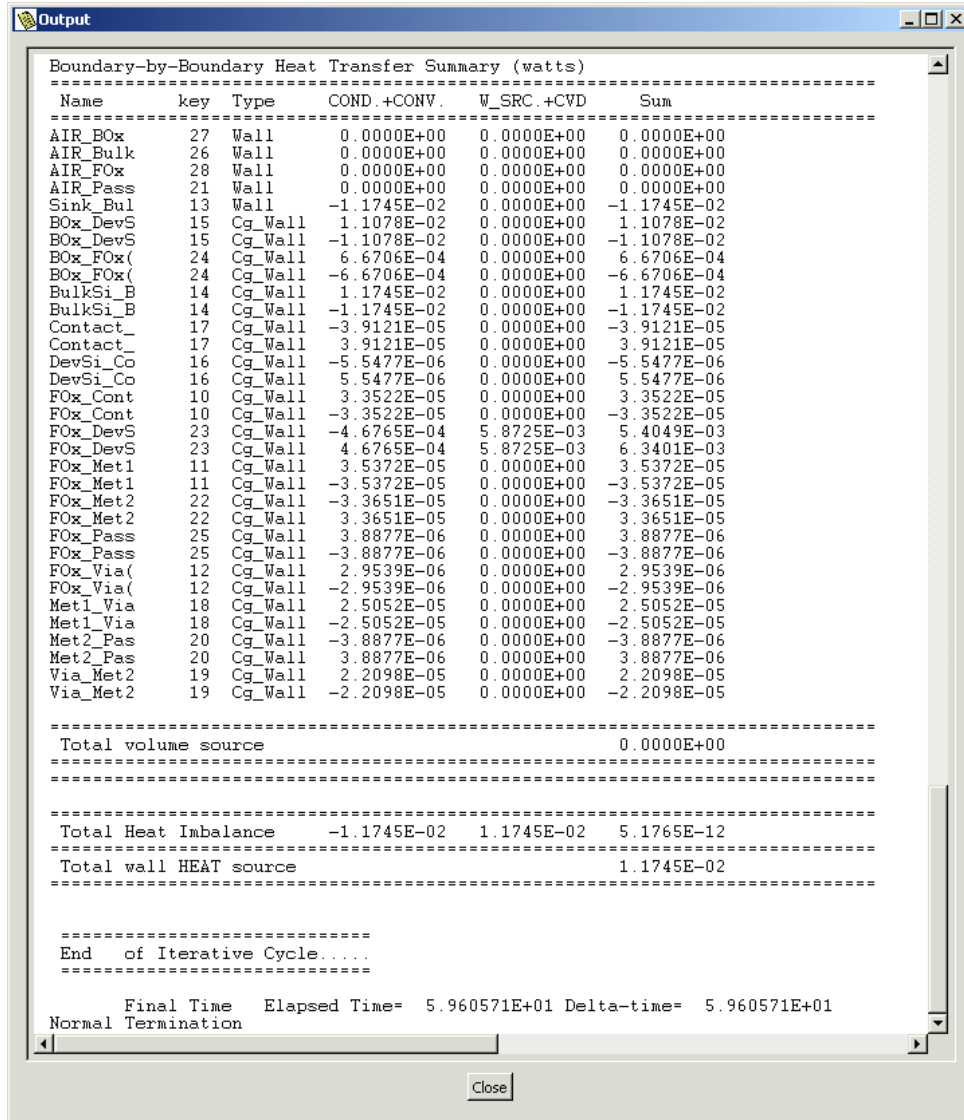


**NOTE:** There is a **colored dot** in the upper right corner of the residual window. This dot is an indicator of whether the simulation is running (green) or has stopped (red). Also, you might have to wait for a few seconds after clicking the Submit button, for the variable names and residual curves to appear in the window. This is because variable initialization, memory allocation, and even the solution itself can sometimes take a while (e.g. 3D cases with a large number of cells).

➤ Click **View Output**

The output file contains a Heat Transfer Summary at the end. A good heat balance in the Heat Transfer Summary may also be treated as an indicator of a properly converged solution. At steady state, the heat entering a boundary should be equal to the heat leaving the boundary (except in the case of a heat source or sink). This is clearly observed in the boundary-by-boundary heat transfer summary in the following figure.

The "Elapsed Time" value at the bottom of the Output file shows the CPU time for that simulation, in seconds.





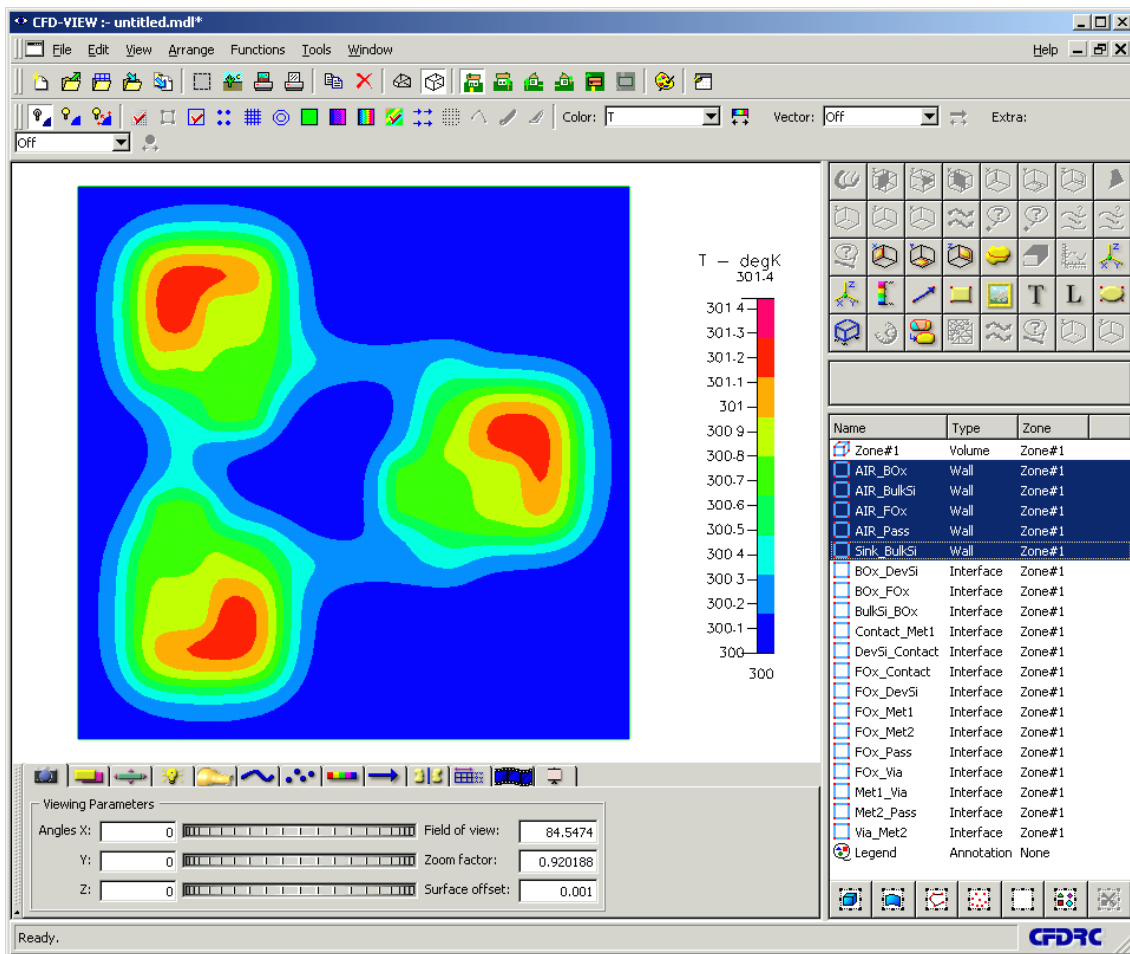
## Post-processing using CFD-VIEW


This section shows how to use CFD-VIEW to obtain color contours for temperature, wall heat flux etc. It also describes how to obtain line plots for temperature variations in a cross-section. For more details on how to use VIEW refer to the *CFD-VIEW User Manual*.

- As earlier, click the Launch **CFD-VIEW** button [  ] in the top toolbar of GUI. A new CFD-VIEW window displaying the IC chip structure of heatchip.DTF should open. **CFD-VIEW** may also be launched on it's own, then Click the **Import DTF or Plot 3D** button [  ] and open the file heatchip.DTF.


## Color Contour Plots Of Full Geometry


- In the Objects window in the lower-right corner, select all the objects of type **Wall**
- From the top toolbar, select **Flooded On** [  ]
- From the **Color:** drop-down list along the top toolbar, select T
- Use **LB** to rotate the 3D mesh, **MB** to zoom in or out the figure and **RB** to pan the figure. Make sure you have a convenient view of the IC chip to begin with.
- Add a legend to the screen by clicking on the **Add Legend** button (  ) located in the upper right panel of objects
- The temperature color contours should look similar to the following:



- To turn off previous contour plot: In the Objects window in the lower-right corner, select all the objects of type **Wall** and select **Flooded Off** [  ] from top tool-bar.

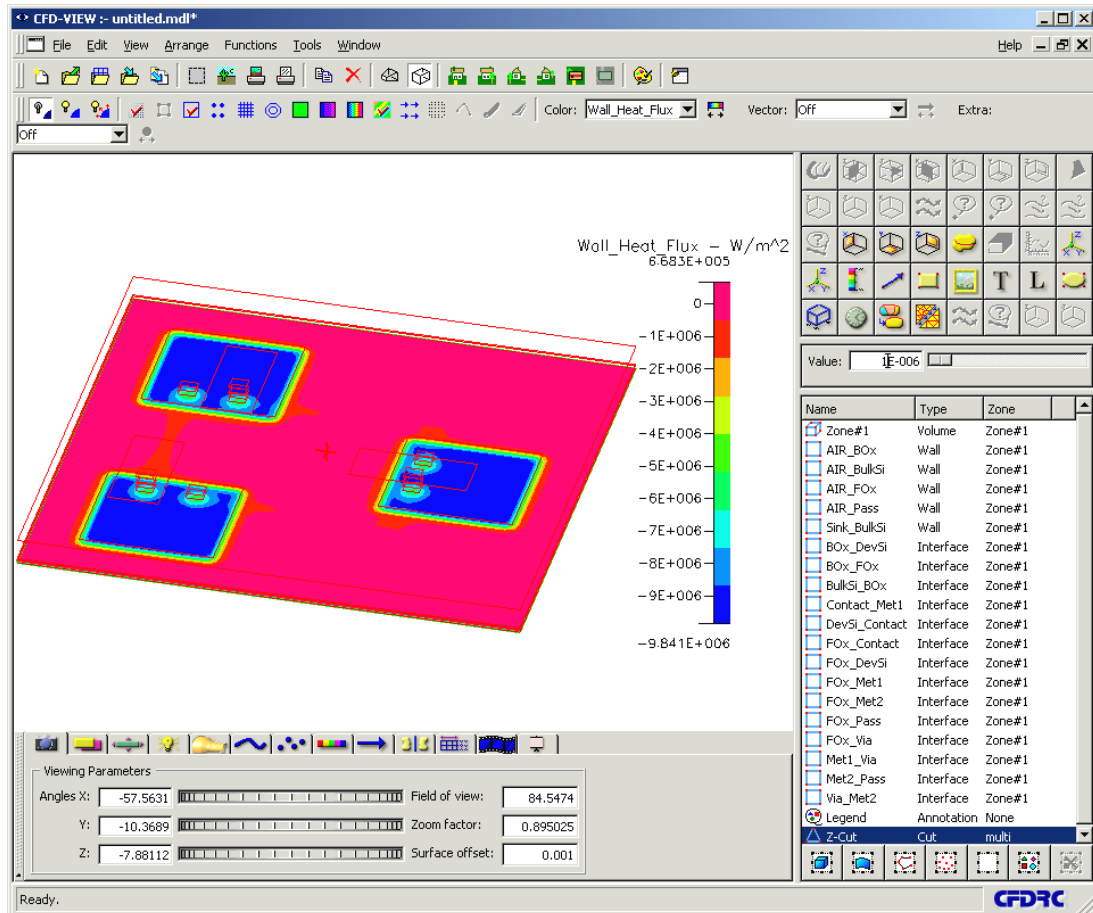
## Color Cut-Plane Contour Plots

- Another useful feature of CFD-VIEW is the ability to view variable distributions in various cross-sections (or cut-planes).
- In the Objects window in the lower-right corner, select the **Zone** object named Zone#1
- In the upper right hand section of the screen, click the **Z-cut** button (  ) (Note: A new object called Z-Cut is added to the object list and now highlighted)

- From the top toolbar, select **Flooded On** (  )
- From the **Color:** drop-down list along the top toolbar, select Wall Heat Flux

By default, the various cut planes are positioned at the center of the structure. The position can be changed using the **slider** located above the object list on the right.

- Using the **slider bar**, set the location of the Z-cut at 1e-6
- The following wall heat flux distribution should be seen:





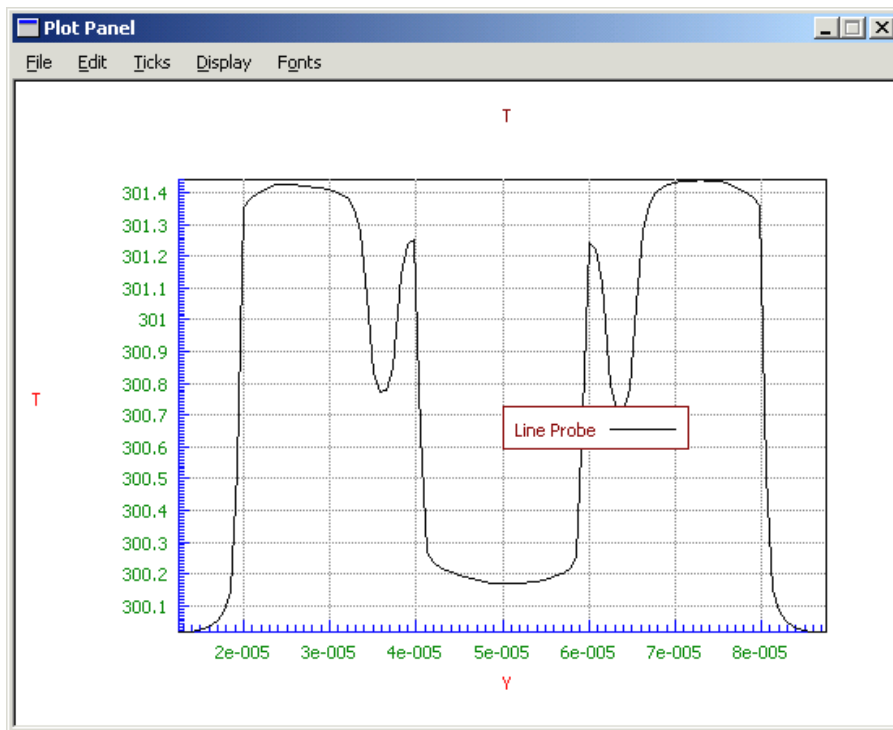
Such cut-planes can be created in any of the 3 directions, and can be used in conjunction with each other as well. (To remove the Z-Cut object, select Z-Cut from Object list window and hit Delete key)



## Line Plots

In some cases, simulation results are better represented by means of a line plot in a cross-section of the device.

- In the Objects window in the lower-right corner, select the **Zone** object named Zone#1
  - In the upper right hand region, select the **Line Probe** button (  ), which creates and highlights a new Line Probe object in the lower right object list
  - Above the object list, set the following values for **Point 1** (start point of the plot-line) and **Point 2** (end point of plot-line):
    - Point 1: X: 2.62e-5, Y: 1.25e-5, Z: 1.74e-6
    - Point 2: X: 2.62e-5, Y: 8.75e-5, Z: 1.74e-6
  - In the upper right hand region, select the **Plot** button (  ), which creates and highlights a new Plot object in the lower right object list
  - Above the object list, set **Plot X-Axis** to variable Y and **Plot Y-Axis** to variable T
  - Check **External Plot** box to generate plot as seen below



We can clearly observe the variations in temperature across the cross-section of the IC chip. The silicon devices are the hottest regions, while the surrounding areas have a relatively lower temperature. The sudden dips in temperature are due to the presence of metallic contacts. Being good thermal conductors, they conduct heat away efficiently and their temperatures are lower.

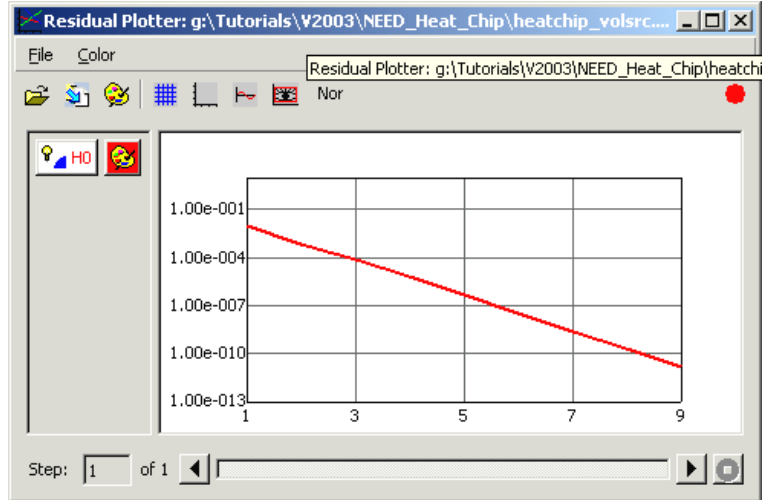
## Steady State Solution with Volumetric Heat Source

The procedure for setting a volumetric heat source in CFD-GUI is similar to setting a surface heat source. The *heatchip.DTF* file created earlier may be used for this simulation too.

17. Assuming the *heatchip.DTF* file is still open in the CFD-ACE-GUI window, click **File -> Save As...**, and save the DTF file as "heatchip\_volsrc.DTF". This will prevent overwriting of the earlier DTF file with the results of our surface heat source simulations. We will work with the *heatchip\_volsrc.DTF* file for this simulation.

18. Set-up and run a new simulation in the following manner:

- **PT** (Problem Type) and **MO** (Model Options)
  - Follow the same procedure as in the surface heat source case.
  
- **VC** (Volume Conditions)
  - Select the DevSi volume name. BulkSi also gets highlighted since the two are grouped. Click the **Ungroup** button at the bottom to separate the two volume names.
  - Change the **VC setting mode** drop-down list on the right from **Properties** to **Heat**
  - Select the DevSi volume name
    - Check the **Heat Source** button
    - In the **Heat Source** drop-down list, select **Fixed Source (Volumetric)**, and enter a value of  $2E14 \text{ W/m}^3$ .
    - Click **Apply**.
  - You may now change the **VC setting mode** drop-down list on the right from **Heat** back to **Properties** if you desire, as this is the default mode.
  
- **BC** (Boundary Conditions)
  - Ensure that the Sink\_BulkSi face is isothermal at 300K
  - Select the Fox\_DevSi interface:
    - Uncheck the **Wall Heat Source** button
    - Click **Apply**
  
- **IC** (Initial Conditions) and **SC** (Solver Control) and **Out**
  - Leave the earlier settings as set for the previous surface heat source simulation.
  
- **Run**
  - Click **Submit to Solver**. In the new window that opens, click **Submit Job Under Current Name**.
  - The Residual and Output files after simulation should look similar to the following:



Output

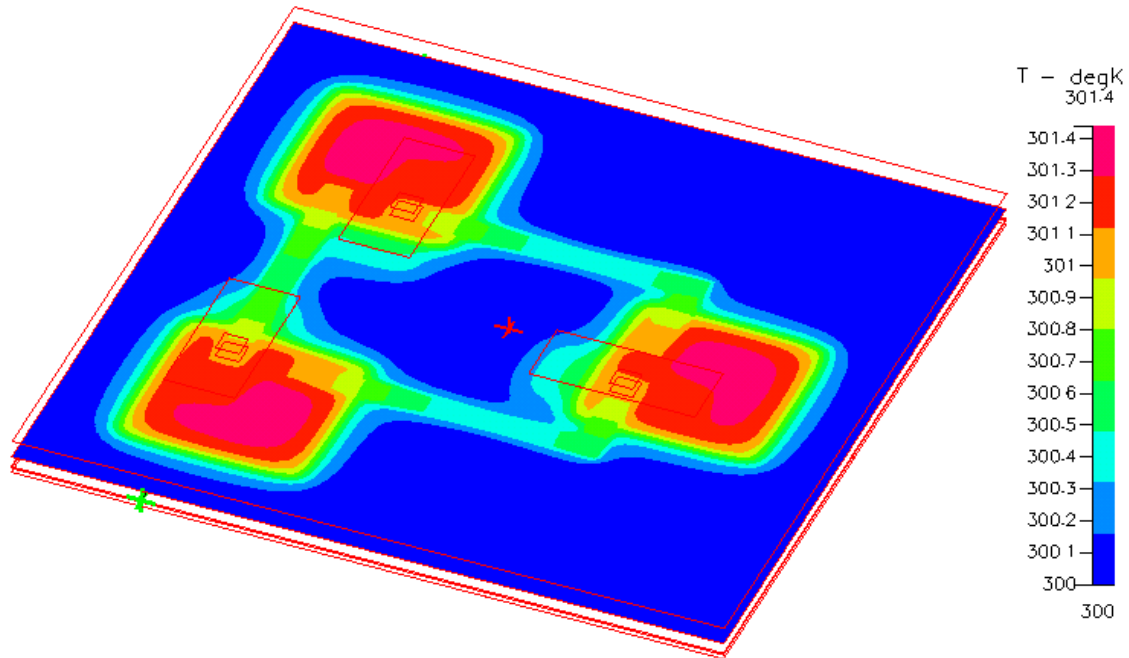
Boundary-by-Boundary Heat Transfer Summary (watts)

Name	key	Type	COND. +CONV.	W_SRC. +CVD	Sum
Sink_Bul	13	Wall	-1.2000E-02	0.0000E+00	-1.2000E-02
AIR_Pass	21	Wall	0.0000E+00	0.0000E+00	0.0000E+00
AIR_Bulk	26	Wall	0.0000E+00	0.0000E+00	0.0000E+00
AIR_Box	27	Wall	0.0000E+00	0.0000E+00	0.0000E+00
AIR_Fox	28	Wall	0.0000E+00	0.0000E+00	0.0000E+00
FOx_Cont	10	Cg_Wall	1.5504E-05	0.0000E+00	1.5504E-05
FOx_Cont	10	Cg_Wall	-1.5504E-05	0.0000E+00	-1.5504E-05
FOx_Met1	11	Cg_Wall	4.7804E-05	0.0000E+00	4.7804E-05
FOx_Met1	11	Cg_Wall	-4.7804E-05	0.0000E+00	-4.7804E-05
FOx_Via(	12	Cg_Wall	8.9014E-07	0.0000E+00	8.9014E-07
FOx_Via(	12	Cg_Wall	-8.9014E-07	0.0000E+00	-8.9014E-07
BulkSi_B	14	Cg_Wall	1.2000E-02	0.0000E+00	1.2000E-02
BulkSi_B	14	Cg_Wall	-1.2000E-02	0.0000E+00	-1.2000E-02
BOx_DevS	15	Cg_Wall	1.1254E-02	0.0000E+00	1.1254E-02
BOx_DevS	15	Cg_Wall	-1.1254E-02	0.0000E+00	-1.1254E-02
DevSi_Co	16	Cg_Wall	-2.0555E-04	0.0000E+00	-2.0555E-04
DevSi_Co	16	Cg_Wall	2.0555E-04	0.0000E+00	2.0555E-04
Contact_	17	Cg_Wall	-2.2111E-04	0.0000E+00	-2.2111E-04
Contact_	17	Cg_Wall	2.2111E-04	0.0000E+00	2.2111E-04
Met1_Via	18	Cg_Wall	-4.3924E-05	0.0000E+00	-4.3924E-05
Met1_Via	18	Cg_Wall	4.3924E-05	0.0000E+00	4.3924E-05
Via_Met2	19	Cg_Wall	-4.4814E-05	0.0000E+00	-4.4814E-05
Via_Met2	19	Cg_Wall	4.4814E-05	0.0000E+00	4.4814E-05
Met2_Pas	20	Cg_Wall	-1.1912E-05	0.0000E+00	-1.1912E-05
Met2_Pas	20	Cg_Wall	1.1912E-05	0.0000E+00	1.1912E-05
FOx_Met2	22	Cg_Wall	-5.9120E-06	0.0000E+00	-5.9120E-06
FOx_Met2	22	Cg_Wall	5.9120E-06	0.0000E+00	5.9120E-06
FOx_DevS	23	Cg_Wall	-4.5878E-04	0.0000E+00	-4.5878E-04
FOx_DevS	23	Cg_Wall	4.5878E-04	0.0000E+00	4.5878E-04
BOx_Fox(	24	Cg_Wall	7.4582E-04	0.0000E+00	7.4582E-04
BOx_Fox(	24	Cg_Wall	-7.4582E-04	0.0000E+00	-7.4582E-04
FOx_Pass	25	Cg_Wall	1.1912E-05	0.0000E+00	1.1912E-05
FOx_Pass	25	Cg_Wall	-1.1912E-05	0.0000E+00	-1.1912E-05
Total volume source					1.2000E-02
Total Heat Imbalance					-1.2000E-02 0.0000E+00 6.0831E-12
Total wall HEAT source					0.0000E+00
End of Iterative Cycle.....					
Final Time Elapsed Time= 6.212934E+01 Delta-time= 6.212934E+01					
Normal Termination					

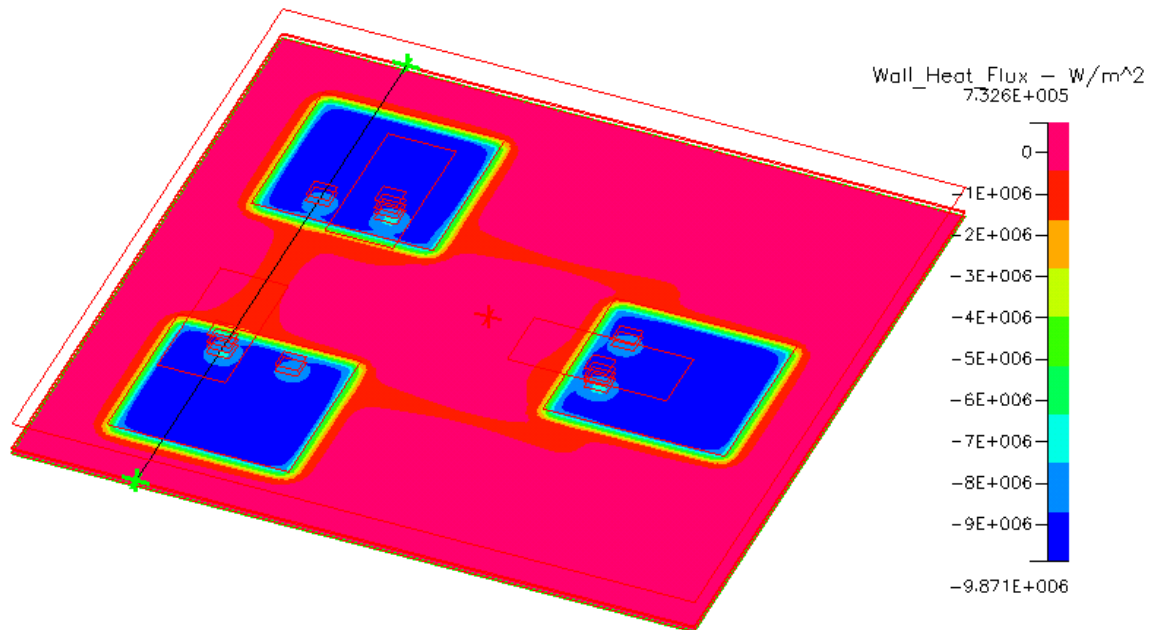
Close

## Viewing the results

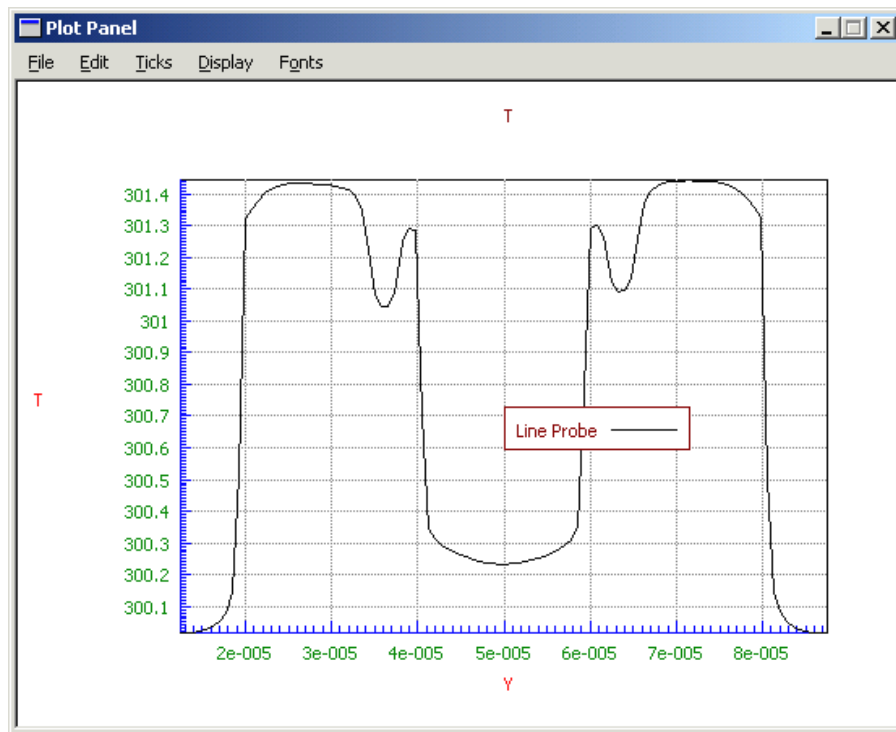
The temperature and wall heat flux distributions in CFD-VIEW should look similar to the following figures. The temperature distribution is shown below in a Z-cut at  $Z = 2.9E-6$  m from the bottom.



The wall heat flux is shown below in a Z-cut at  $Z = 1E-6$  (i.e. at the bottom of the chip).



A plot of temperature, along the dotted line shown in the above figure, should look similar to the following plot. (This line plot end points are identical to the points described for the previous simulation VIEW results).



We can clearly observe the variations in temperature across the cross-section of the IC chip. The silicon devices are the hottest regions, while the surrounding areas have a relatively lower temperature. The slight decrease in temperature at the ends of the silicon devices are due to the metallic contacts, as in the surface source case earlier.