# CFD-ACE+

Version 2003

# Micromesh/Heat Tutorial
# 10-03

# Integrated Circuit (IC)
# Thermal Analysis

- **Building a 3D simulation model of an integrated circuit (IC) from a layout using CFD-Micromesh**
- **Setting up a Heat Transfer simulation in CFD-ACE+**
- **Steady-state simulation with surface 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@cfdrc.com, Phone: (256) 726- 4900
Sales: sales@cfdrc.com, www.cfdrc.com

© 2003 by CFD Research Corporation
All rights reserved.

# Tutorial:

# Integrated Circuit (IC) Thermal Analysis

PAGE

**GENERAL INFORMATION**

Problem Description and Objectvies . . . . . . . 3

**CONDENSED TUTORIAL**

Importing Layout and Defining Technology . . . . . . 4
Set Model Gometry Dimensions and Grid Step . . . . . 5
Generate/Edit Process Sequence . . . . . . . 5
Edit Layer Operation, Vertical Position, and Layout . . . . . 5
Save Micromesh Project and Technology Files . . . . . 5
Generating and Viewing 3D Solid Model . . . . . . 5
Automatic Generation of 3D Computational Mesh . . . . . 5
Viewing the Generated 3D Computational Mesh (DTF file) . . . . 6
Setting Up A Thermal Simulation in CFD-ACE+ . . . . . 6
Post-Processing Simulation Results . . . . . . . 6

**DETAILED TUTORIAL**

Importing Layout and Defining Technology . . . . . . 7
Set Model Gometry Dimensions and Grid Step . . . . . 9
Generate/Edit Process Sequence . . . . . . . 10
Edit Layer Operation, Vertical Position, and Layout . . . . . 13
Save Micromesh Project and Technology Files . . . . . 17
Generating and Viewing 3D Solid Model . . . . . . 17
Automatic Generation of 3D Computational Mesh . . . . . 19
Viewing the Generated 3D Computational Mesh (DTF file) . . . . 21
Setting Up A Thermal Simulation in CFD-ACE+ . . . . . 22
Post-Processing Simulation Results . . . . . . . 25

## PROBLEM DESCRIPTION AND OBJECTIVES

Perform thermal analysis of integrated circuit (IC), starting from original electronic layout and process information.

Electronic integrated circuit (IC) designs are very often stored in the form of *layout* (top view) of all the layers composing the design. Similar method is used to store and transfer designs of electronic printed-circuit boards (PCBs), multi-chip modules (MCMs), and micro-electro-mechanical systems (MEMS). To perform a physical simulation of such structures we need to create a 3D computational mesh from the layout, using additional information about particular layer properties (position, thickness, material). The latter information is also called process data, and may be stored as a *technology file*.

In this tutorial we show how to:

- import a layout from a CIF file
- define the process data (properties of layers), and save as a Technology File (which can be used for subsequent fast and automatic building of models from different layouts of the same technological process)
- create and view interactively a 3D model
- generate three-dimensional (3D) mesh
- setup and perform thermal simulation
- visualize thermal results of simulation

As an example, we use an IC device as illustrated in Figure 1. Heat sources present in the silicon device layer of the 3D model are modeled as 2D surface heat sources located exactly at the top surface of the silicon, which corresponds to the location of CMOS transistor channels. The heat sources are located in four groups (inverter clusters), as illustrated in Figure 2. Sample calculated temperature distributions are can also be seen in Figure 2.

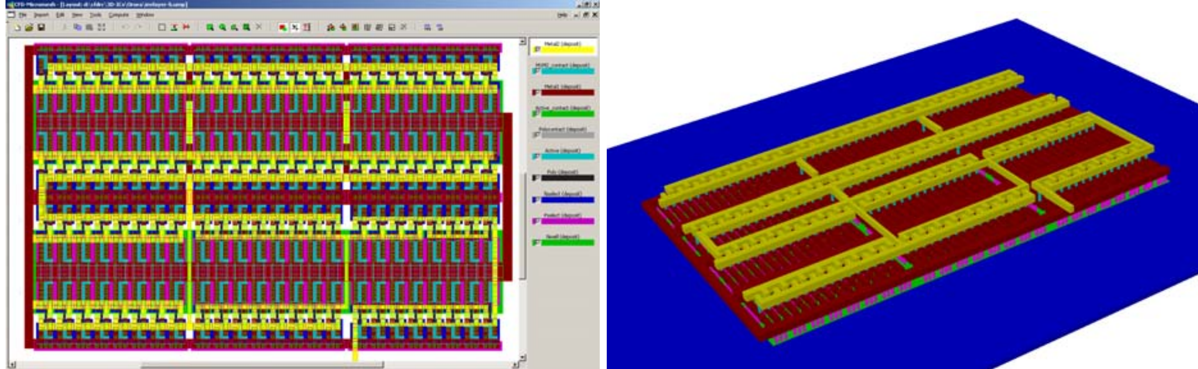


*Figure 1. IC layout in CFD-Micromesh (left) and a three-dimensional solid model built automatically in CFD-Micromesh (right).*

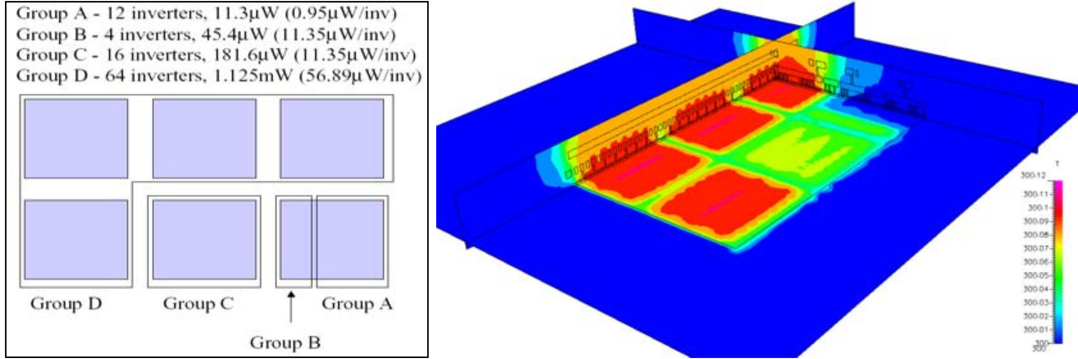



*Figure 2. Location of inverter-clusters used as separate heat sources in the IC thermal model (left) and calculated temperature distribution when all heat source groups (A–D) active (right).*

# CONDENSED TUTORIAL

The following is a summary of the steps in this tutorial. Detailed explanations of each step follows the summary. This summary can be useful for users who are somewhat familiar with the software and the overall process of setting up and running an IC thermal simulation. The summary shown below matches the detailed description which follows, hence allowing the user to use the summary description below and detailed explanations for steps which may be unfamiliar to the user.

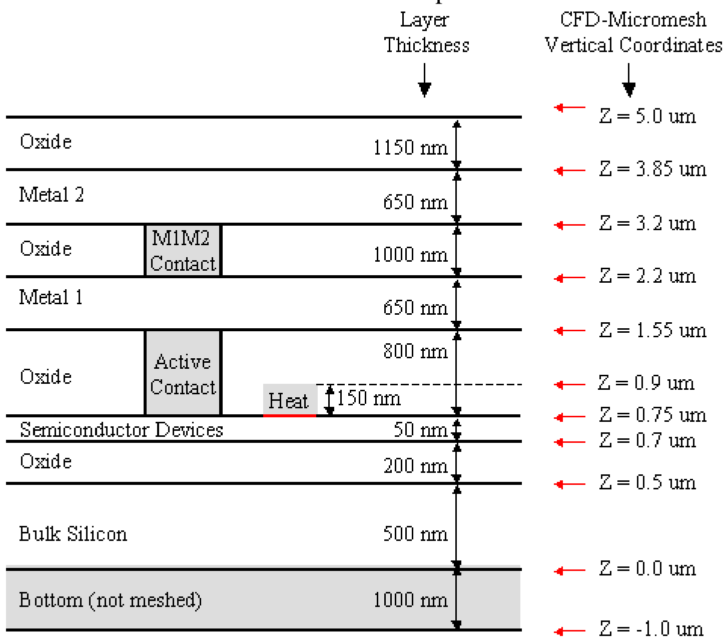
## Importing Layout and Defining Technology

1. Import the IC CIF layout file: "IC_Heat.cif" with no technology file

   a. Obtain layer names from CIF file in text editor:

```
Layer nwell = CIF Layer CWN
Layer nselect = CIF Layer CSN
Layer pselect = CIF Layer CSP
Layer active = CIF Layer CAA
Layer active contact = CIF Layer CCA
Layer poly = CIF Layer CPG
Layer polycontact = CIF Layer CCP
Layer m1m2 contact = CIF Layer CVA
Layer pad = CIF Layer CMP
Layer metal1 = CIF Layer CMF
Layer metal2 = CIF Layer CMS
Layer glass = CIF Layer CFI
```

   b. Obtain vertical dimensions and material composition.




2. Explore and understand CFD-Micromesh environment

3. Convert CIF layer names into physical layer names (using information from step 1a).

   CWN → **Rename** to: "nwell",
   CSN → **Rename** to: "nselect",
   CSP → **Rename** to: "pselect",
   etc. ... (Note that physical layers "pad" and "glass" are not present in the layout)

**Set Model Gometry Dimensions and Grid Step**

4. Set model geometry boundary dimensions (min, max, voxel size) [um]:
   X(0, 100, 0.16), Y(0, 100, 0.16), Z(-1, 5, 0.01)

5. Set size of grid step used in Micromesh for drawing layouts to 1.0 um.

**Generate/Edit Process Sequence**

6. Removal of unnecessary process layers. The following layers may be removed: active, poly contact, poly, nselect, pselect, and nwell

7. Ordering of remaining process layers. For the remaining 4 layers, active contact should be on the bottom, with metal1 above, then m1m2 above, and finally metal2 on top.

8. Generate missing process layers. The following layers (and hence, material names) must be added to the process: Bottom, Bulk_Silicon, Oxide, HeatA, HeatB, HeatC, HeatD, Semi_Device

9. Ordering of process layers. The final ordering should be as follows, from bottom to top: Bottom → Bulk_Silicon → Oxide →Semi_Device → HeatA → HeatB → HeatC → HeatD → Active_Contact → Metal1 → M1M2 → Metal2

**Edit Layer Operation, Vertical Position, and Layout (all units in micrometers)**

| | | | | |
|---|---|---|---|---|
| 10. | layer: Bottom | deposit (no mesh) | 1.0 thick | rectangle: (0, 0) to (100, 100) |
| 11. | layer: Bulk_Silicon | deposit | 0.5 thick | rectangle: (0, 0) to (100 ,100) |
| 12. | layer: Oxide | deposit | 4.5 thick | rectangle: (0, 0) to (100, 100) |
| 13. | layer: Semi_Device | insert | position 0.7, 0.05 thick | rectangle: (22.24, 22.4) to (85.92, 64.0) |
| 14. | layer: HeatA | insert | position 0.75, 0.15 thick | rectangle: (70.56, 23.36) to (85.76, 42.4) |
| 15. | layer: HeatB | insert | position 0.75, 0.15 thick | rectangle: (65.76, 23.36) to (70.4, 42.4) |
| 16. | layer: HeatC | insert | position 0.75, 0.15 thick | rectangle: (44.0, 23.36) to (64.0, 42.4) |
| 17. | layer: HeatD | insert | position 0.75, 0.15 thick | rectangles: (22.56, 23.52) to (42.56, 42.52) : (22.56, 44.0) to (42.56, 63.0) : (44.0, 44.0) to (64.0, 63.0) : (65.4, 44.0) to (85.4, 63.0) |
| 18. | layer: Active_Contact | insert | position 0.75, 0.8 thick | remove rectangles present in four corners |
| 19. | layer: Metal1 | insert | position 1.55, 0.65 thick | remove rectangles present in four corners |
| 20. | layer: M1M2 | insert | position 2.2, 1.0 thick | remove rectangles present in four corners |
| 21. | layer: Metal2 | insert | position 3.2, 0.65 thick | remove rectangles present in four corners |

**Save Micromesh Project and Technology Files**

22. Save Micromesh Technology file : **File → Save Technology …** "IC_Heat.umt"
23. Save Micromesh project file : **File → Save Project As …** "IC_Heat.ump"

**Generating and Viewing 3D Solid Model**

24. Viewing model with 3D quick view. View entire structure or internal components by making layers Oxide and Bulk_Silicon non-meshed.

**Automatic Generation of 3D Computational Mesh**

25. Add mesh cell size layer to process. Set Mesh Cells Size to 10. Draw rectangles with coordinates:
    (0, 0) to (20.6, 100) ; (87.7, 0) to (100, 100) ; (0, 0) to (100.0, 22) ; (0, 65) to (100, 100)

26. Build 3D Prism-Hexa unstructured computational grid. Select "Old prisms-hexahedra mesh (before version 2002)" and set to 25,000 triangles, 1000000 initial z size cell, 6E-6 Z direction threshold, 1 smoothness

tolerance, no Z mesh, "default" mesh → Results in ~145,000 prism and ~41,000 hexa cells are generated, hence ~186,000 total computational cells for the geometry.

**Viewing the Generated 3D Computational Mesh (DTF file)**

27. View 3D computational mesh in CFD-VIEW.

**Setting Up A Thermal Simulation in CFD-ACE+**

28. Open DTF file in CFD-GUI

29. Setup steady simulation for the IC geometry.
    PT: Heat
    MO: Steady
    VC: Properties
        Bulk_Silicon, Semi_Device → Density 2300, Specific Heat 700, Therm Conductivity 155
        Oxide, HeatA, HeatB, HeatC, HeatD → Density 2150, Specific Heat 740, Therm Conductivity 1.3
        Active_Contact, Metal1, M1M2, Metal2 → Density 2700, Specific Heat 938, Therm Conductivity 230
    BC: Bottom_Bulk_Silicon → Isothermal 300K
        Semi_Device_HeatA → Wall Heat Source at 39045 W/m^2
        Semi_Device_HeatB → Wall Heat Source at 513865 W/m^2
        Semi_Device_HeatC → Wall Heat Source at 477895 W/m^2
        Semi_Device_HeatD → Wall Heat Source at 740131.6 W/m^2
    IC: default
    SC: Max Iterations 30, Enthalpy AMG Solver, Enthalpy Relaxation 1E-13
    OUT: Static Temperature and Wall Heat Flux
    **File → Save** (saves file "IC_Heat.DTF")

30. Run the IC thermal simulation.

**Post-Processing Simulation Results**

31. Visualize IC thermal simulation results

## DETAILED TUTORIAL

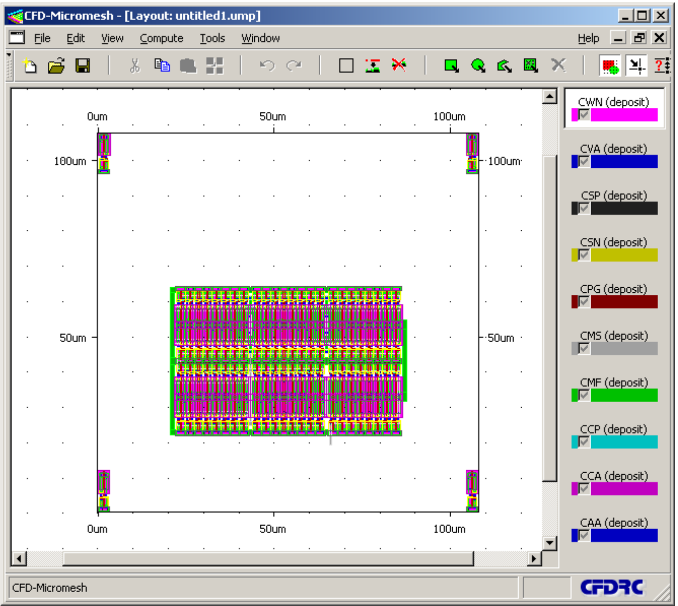
*In the following lesson these definitions apply:*
*LB = Left mouse button,   MB = Middle mouse button,   RB = Right mouse button*

### Importing Layout and Defining Technology

1. Import the IC CIF layout. From the **File** pull down menu, select: **Import → CIF file ...**

   A window appears with the question: "Do you want to select technology file?" At this point, we don't yet have a technology file defined for this project, thus answer No to the question.

   In the "Import CIF Project" window, select the file you want (for this tutorial "IC_Heat.cif"), and click **OK**.




   A layout of the device appears in the "Layout" editing window (shown above), with the layer names coming from the original CIF file. Typically, the layers defined in a CIF file can not be used directly for automatic building of 3D computational models within CFD-Micromesh. Therefore, a special technology file is needed for mapping the layers from a CIF file into physical layers, and a special sequence of operations performed by CFD-Micromesh needs to be defined. A Micromesh Technology File (with extension *.umt*) can be used for subsequent fast and automatic building of models from different layouts of the same technological process.

   In this tutorial we will create such a technology file manually from information provided along with the CIF layout file. In this IC case, we have the following additional technology information available:

   **Layer names**:
   As was seen from the imported CIF file in Micromesh, the coded CIF layer names available are non-descriptive and of little use. Knowledge of what each layer is in the process is required for proper model development. Hence, the coded CIF layer names need to be converted into physical layer names.

   The easiest way to determine the layer name conversion information is to open the CIF file in a text editor. Near the beginning of the file, comments might be present mapping the layer names with the CIF layer names. Whether this information is present or not depends on the layout tool used to generate the CIF file. If this information is not available in the CIF file, direct knowledge of the layer name mapping must be obtained from the CIF layout author.

For this tutorial case, the following information was present in the CIF file, mapping the layer names as shown below:

```
Layer nwell = CIF Layer CWN
Layer nselect = CIF Layer CSN
Layer pselect = CIF Layer CSP
Layer active = CIF Layer CAA
Layer active contact = CIF Layer CCA
Layer poly = CIF Layer CPG
Layer polycontact = CIF Layer CCP
Layer m1m2 contact = CIF Layer CVA
Layer pad = CIF Layer CMP
Layer metal1 = CIF Layer CMF
Layer metal2 = CIF Layer CMS
Layer glass = CIF Layer CFI
```

**Vertical Dimensions and Material Composition**:
To build a 3D model from layout, we need also an information about the "3rd dimension" of the structure, that is vertical dimensions of particular layers in the layout, and their material composition. This information must be obtained from some source. For this IC case, this information is illustrated in Figure 3 below.

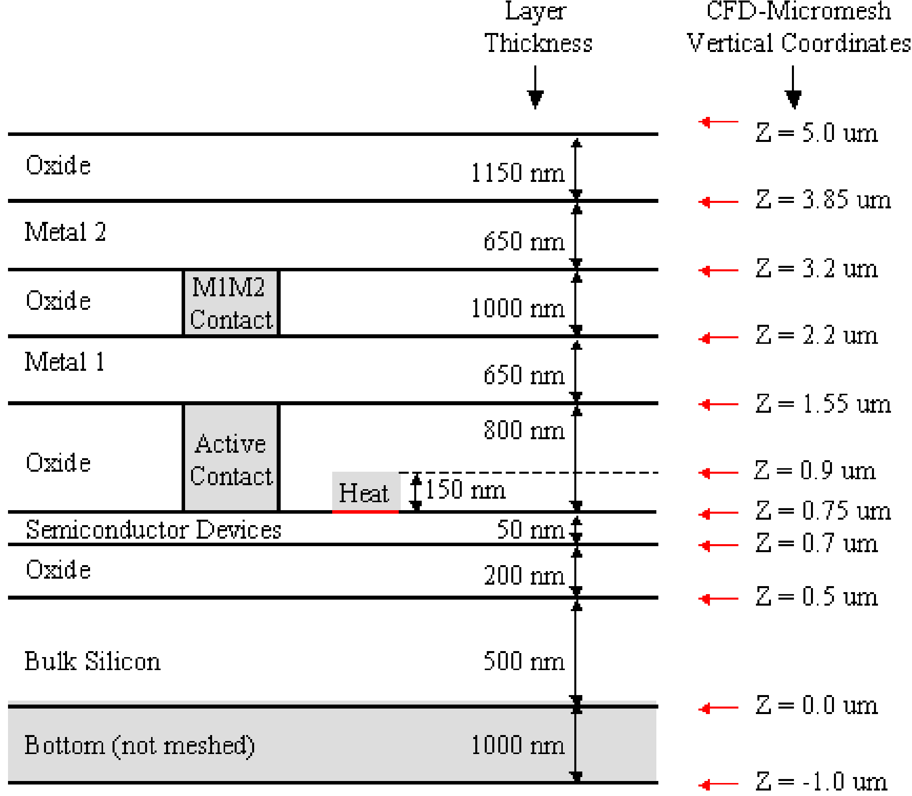



*Figure 3. Vertical dimensions of layers in the chip structure, and their material composition.*

2. If the user is unfamiliar with the Micromesh environment, explore the layout editing window.

   - The mouse buttons work as in the other CFDRC software: LB is used to select an object, MB - to zoom in and out the view, RB - to shift (pan) the view. A context-sensitive help is in the lower-left corner of the main window.
   - Note also the current coordinates (X, Y) of the cursor on the screen, appearing in the lower-right corner of the main window. Initially, the units are set automatically according to the size of the imported file (micrometers in this case), but you can change them (menu: **View** -> **Display Units**).
   - On the right side of the main window, examine all layers constituting your project.
   - Starting from the most bottom layer (GDS_31 in this example), keep clicking LB on subsequent higher layers, to see them appearing sequentially in the layout.
   - Objects belonging to the selected layer are highlighted in the layout.
   - Note that by default only the layers below the currently active layer are displayed.
   - By clicking RB on the highlighted layer name (in the right window), you can view and/or change properties of this layer. The change will affect all the objects belonging to this layer.
   - By clicking LB on an element of layout of the highlighted layer (left part of the window), you can select the specific element in this layer, then move it, stretch, copy, delete, etc.
   - By clicking RB on the selected element of layout, you can view and/or change properties of the particular element. The change will affect only this object.
   - Explore the icon buttons at the top of main window, and their description, by hovering the mouse cursor (without clicking) for a while on each button. A longer explanation of the button function appears also in the bottom-left corner of the main window.

3. Convert CIF layer names into physical layer names, using the technology information presented previously (as obtained from the CIF file as viewed in a text editor).

   - Select (by clicking LB) the first layer at the bottom of the list (in this case: CAA).
   - Click RB on the selected layer name
     - the “Layer Properties” dialog window opens.
   - Select **Edit Material** button - the “Materials” dialog window opens
   - Change the CIF “material” names into new (physical) names, by clicking on **Rename** button in the "Materials" window. For example:
     CWN → **Rename** to: “nwell”,
     CSN → **Rename** to: “nselect”,
     CSP → **Rename** to: “pselect”,
     etc. ...
     Note that physical layers “pad” and “glass” are not present in the layout.
   - Click **Close** in the "Materials" window.
   - Click **Accept** in the "Layer Properties" window.

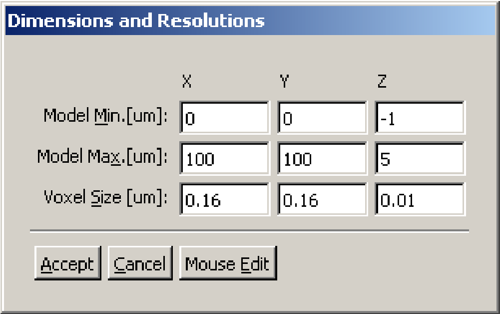
**Set Model Geometry Dimensions and Grid Step**

4. On the basis of the technology (process) description presented in Figure 3 (vertical information), we can now set up the project vertical boundaries at Zmin = -1.0 μm, Zmax = 5.0 μm, and a voxel size to 0.01 μm. At this time, the X and Y range will also be adjusted slightly as shown in the image below.

   - From the toolbar, click the **Change Boundary** button [ ☐ ].

     The "Dimensions and Resolutions" dialog window opens. Change the values of the fields to match the values shown below. Note that voxel size is expressed in *voxels*, or three-dimensional blocks, analogous to two-dimensional pixels in 2D images. The largest voxel size should be chosen that can still capture the dimensions of the elements (or smallest grid element you would like to have). Hence, noting the smallest element resolution in your geometry, including any overlaping elements, is key to allow for the largest voxel size but reduce the required memory allocation for grid generation. For this tutorial, voxel sizes of

0.16 um for the X and Y direction are appropriate. In the Z direction, a voxel size of 0.1 um is chosen to allow for multiple grid cells in the smaller layers if desired (smallest layer thickness is 0.5 um).




- Click the **Accept** button.

5. Change size of grid used in Micromesh for drawing layouts.
   The size of grid step may be changed in Micromesh. This grid is useful in snap-to-grid drawing of 2D layouts. While the grid may not allow for the exact desired dimensions, this smaller grid step will allow for near-exact drawing of layouts. By then editing the drawn layout element, final exact location/dimensions of the elements may be accomplished (RB click on drawn element and edit).
   - From the toolbar, click the **Change Grid** button [ ].
   - The "Grid" dialog window opens. Change the value of the field **Grid Step [um]:** to 1.0
   - Click the **Accept** button.

**Generate/Edit Process Sequence**

6. Removal of Unnecessary Process Layers

   The layout information imported from the CIF layout file contains much detailed information that is not going to be used in this simulation. Specifically, information regarding the manufacturing of the semiconductor devices. For the simulation, a single block layer will be used to represent the silicon semiconductor device layer instead of the multiple layers present in the CIF file. This single layer will be added later, therefore, we may now remove these unnecessary layers from the layout.

   The layers which may be removed from the layout include: active, poly contact, poly, nselect, pselect, and nwell.

   - To remove a layer, click LB on the layer name displayed in the Layer List. Next click on the Delete Layer button [ ]. The layer has now been deleted.
   - Delete the 6 layers as listed above (active, poly contact, poly, nselect, pselect, and nwell)

7. Ordering of Remaining Process Layers

   Model building operations in CFD-Micromesh are performed in the order in which they are displayed in the Layer List on the right-hand side of the main window. The layer displayed at the bottom of the list is inserted into the model first; the others follow in order from the bottom to the top of the list. This is similar to the layers that are processed (deposited, diffused, etched, etc.) one after another, as in the microelectronics manufacturing process.

   ➢ On the basis of Figure 3 (vertical description), rearrange the remaining layers into the correct sequence in the layer list. For the remaining 4 layers, active contact should be on the bottom, with metal1 above, then m1m2 above, and finally metal2 on top. The order currently is nearly correct, with only the m1m2 layer needing to be lowered one position in the Layer List. To rearrange:
      - select "m1m2" layer in the list,

- while holding LB, drag the "m1m2" layer name down so that it is between the metal2 and metal1 layers, then releasing the LB

The four layers should now be arranged correctly.

8. Generate Missing Process Layers

If you compare the current layer now with Figure 3, you notice that there are several process layers missing in the layout (Bottom, Bulk_Silicon, Oxide, HeatA, HeatB, HeatC, HeatD, Semi_Device). To build a complete model, we need to add manually those missing layers to our process description in CFD-Micromesh.

➢ Before adding the new layers though, let us add the desired layer name materials to the material list. This can be done by clicking the pull-down menu Tools → Edit Materials…
  - Since we want to create new layers with different names, click the **Add** button in the "Materials" window.
  - In the "New Material" window, type the name "Bottom" and click **Accept**
  - Again click the **Add** button in the "Materials" window
  - In the "New Material" window, type the name "Bulk_Silicon" and click **Accept**
  - Repeat this process to add the remaining desired materials: Oxide, HeatA, HeatB, HeatC, HeatD, and Semi_Device
  - Click **Close** in the "Materials" window.

➢ To add a new "Bottom" layer:
  - click LB on the top Layer metal2 in the Layer List
  - click LB the **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: metal2 (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: metal2 (deposit)" layer.
  - In the "Layer Properties" window which appears, change the **Material** pull-down selection to "Bottom"
  - click **Accept**

➢ To add a new "Bulk_Silicon" layer:
  - click LB the **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: Bottom (deposit)"
  - In the list of layers, click **RB** on the "NEW: Bottom (deposit)" layer.
  - In the "Layer Properties" window which appears, change the **Material** pull-down selection to "Bulk_Silicon"
  - click **Accept**

➢ To add a new "Oxide" layer:
  - click LB the **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: Bulk_Silicon (deposit)"
  - In the list of layers, click **RB** on the "NEW: Bulk_Silicon (deposit)" layer.
  - In the "Layer Properties" window which appears, change the **Material** pull-down selection to "Oxide"
  - click **Accept**

➢ To add a new "HeatA" layer:
  - click LB the **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: Oxide (deposit)"
  - In the list of layers, click **RB** on the "NEW: Oxide (deposit)" layer.
  - In the "Layer Properties" window which appears, change the **Material** pull-down selection to "HeatA"

- click **Accept**

➢ To add a new “HeatB” layer:

- click LB the **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: HeatA (deposit)"
- In the list of layers, click **RB** on the "NEW: HeatA (deposit)" layer.
- In the "Layer Properties" window which appears, change the **Material** pull-down selection to “HeatB”
- click **Accept**

➢ To add a new “HeatC” layer:

- click LB the **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: HeatB (deposit)"
- In the list of layers, click **RB** on the "NEW: HeatB (deposit)" layer.
- In the "Layer Properties" window which appears, change the **Material** pull-down selection to “HeatC”
- click **Accept**

➢ To add a new “HeatD” layer:

- click LB the **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: HeatC (deposit)"
- In the list of layers, click **RB** on the "NEW: HeatC (deposit)" layer.
- In the "Layer Properties" window which appears, change the **Material** pull-down selection to “HeatD”
- click **Accept**

➢ To add a new “Semi_Device” layer:

- click LB the **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: HeatD (deposit)"
- In the list of layers, click **RB** on the "NEW: HeatD (deposit)" layer.
- In the "Layer Properties" window which appears, change the **Material** pull-down selection to “Semi_Device”
- click **Accept**

9. Ordering of Process Layers

All the required process layers are now present in the layout, but the newly added layers need to be moved to appropriate positions in the list. Therefore, based on Figure 3, the process layers should be reorganized. Figure 4 illustrates the order of the layers before and after re-ordering. The left image illustrates the ordering before rearranging, and the right image illustrates the final ordering required. The final ordering should be as follows, from bottom to top: Bottom → Bulk_Silicon → Oxide →Semi_Device → HeatA → HeatB → HeatC → HeatD → Active_Contact → Metal1 → M1M2 → Metal2

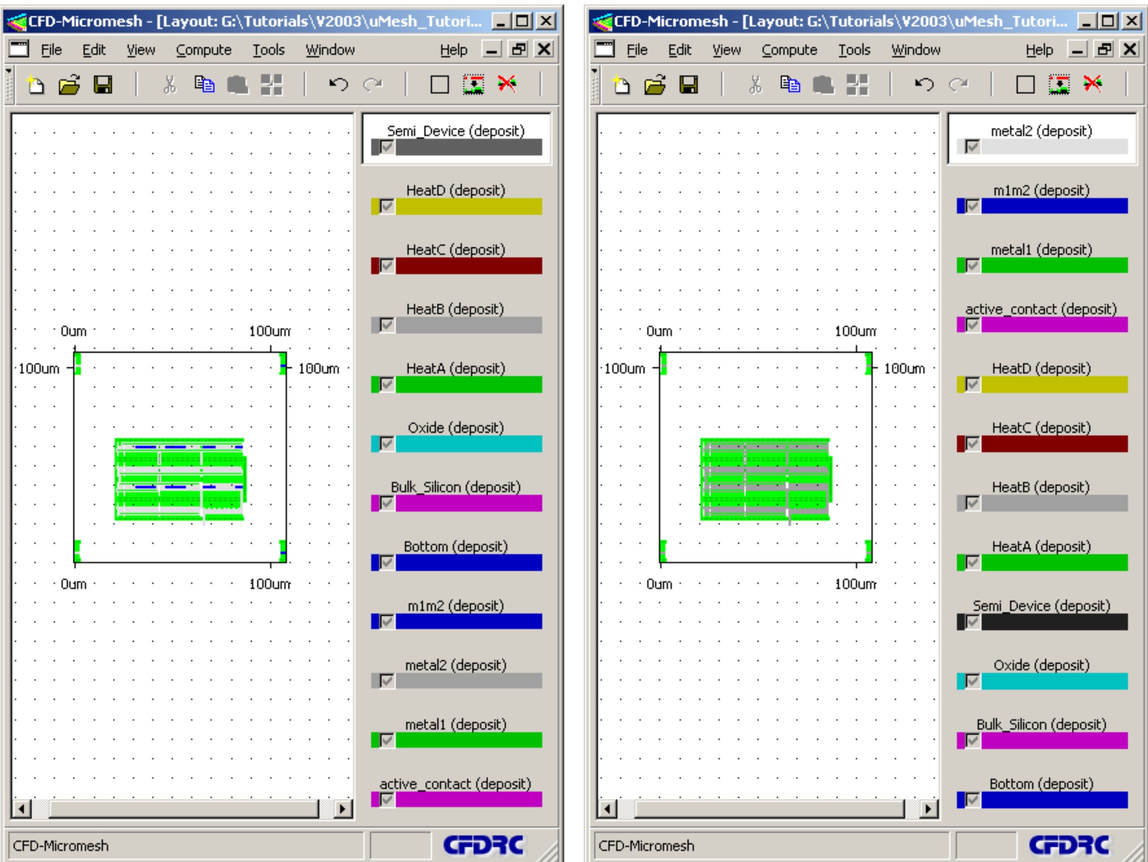



*Figure 4. Layer process order before (left) and after (right) re-ordering.*

**Edit Layer Operation, Vertical Position, and Layout**

10. Edit **Bottom** layer:
    Operation and vertical position:
    - Click LB on the “Bottom” layer from the list of layers on the right-hand side
    - Click RB on the “Bottom” layer
    - In the “Layer Properties” window which appears:
        - set the **Operation:** pull-down menu to “deposit”
        - set the **Thickness [um]:** to 1.0
        - click LB on the **Edit Material** button
            - in the “Materials” window which appears, un-check the **Mesh It** box for the material “Bottom” (hence, this material will not be be meshed in the model)
            - click **Close**, closing the “Materials” window
        - click **Accept**, closing the “Layer Properties” window

    Layout:
    - Draw a layout (top view) of the “Bottom” layer, by selecting the **New Rectangle** button [ ] from the toolbar (or from the top menu: **Tools → New Rectangle**).
    - While holding left mouse button down and observing the current cursor coordinates in the lower-right corner, draw a rectangle with coordinates (0 um, 0 um) to (100 um, 100 um).
    - To check if your “Bottom” 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**.

11. Edit **Bulk_Silicon** layer:
    Operation and vertical position:
    - Click LB on the “Bulk_Silicon” layer from the list of layers on the right-hand side
    - Click RB on the “Bulk_Silicon” layer
    - In the “Layer Properties” window which appears:
        - set the **Operation:** pull-down menu to “deposit”
        - set the **Thickness [um]:** to 0.5
        - click **Accept**

    Layout:
    - Draw a layout (top view) of the “Bulk_Silicon” layer, by selecting the **New Rectangle** button []
    - Draw a rectangle with coordinates (0 um, 0 um) to (100 um, 100 um).
    - To check dimensions, click **RB** on the red (highlighted) rectangle: correct values should be (Pos.X = 0, Pos.Y = 0, Size X = 100, Size Y = 100)

12. Edit **Oxide** layer:
    Operation and vertical position:
    - Click LB on the “Oxide” layer from the list of layers on the right-hand side
    - Click RB on the “Oxide” layer
    - In the “Layer Properties” window which appears:
        - set the **Operation:** pull-down menu to “deposit”
        - set the **Thickness [um]:** to 4.5
        - click **Accept**, closing the “Layer Properties” window

    Layout:
    - Draw a layout (top view) of the “Oxide” layer, by selecting the **New Rectangle** button []
    - Draw a rectangle with coordinates (0 um, 0 um) to (100 um, 100 um).
    - To check dimensions, click **RB** on the red (highlighted) rectangle: correct values should be (Pos.X = 0, Pos.Y = 0, Size X = 100, Size Y = 100)

13. Edit **Semi_Device** layer:
    Operation and vertical position:
    - Click LB on the “Semi_Device” layer from the list of layers on the right-hand side
    - Click RB on the “Semi_Device” layer
    - In the “Layer Properties” window which appears:
        - set the **Operation:** pull-down menu to “insert”
        - set the **Position [um]:** to 0.7
        - set the **Thickness [um]:** to 0.05
        - click **Accept**, closing the “Layer Properties” window

    Layout:
    - Draw a layout (top view) of the “Semi_Device” layer, by selecting the **New Rectangle** button []
    - Draw a rectangle with coordinates (22.24 um, 22.4 um) to (85.92 um, 64 um).
    - To check dimensions, click **RB** on the red (highlighted) rectangle: correct values should be (Pos.X = 22.24, Pos.Y = 22.4, Size X = 63.68, Size Y = 41.6)

14. Edit **HeatA** layer:
    Operation and vertical position:
    - Click LB on the “HeatA” layer from the list of layers on the right-hand side
    - Click RB on the “HeatA” layer
    - In the “Layer Properties” window which appears:
        - set the **Operation:** pull-down menu to “insert”
        - set the **Position [um]:** to 0.75
        - set the **Thickness [um]:** to 0.15

  - click **Accept**, closing the "Layer Properties" window

Layout:

- Draw a layout (top view) of the "HeatA" layer, by selecting the **New Rectangle** button [ ]
- Draw a rectangle with coordinates (70.56 um, 23.36 um) to (85.76 um, 42.4 um).
- To check dimensions, click **RB** on the red (highlighted) rectangle: correct values should be (Pos.X = 70.56, Pos.Y = 23.36, Size X = 15.2, Size Y = 19.04)

15. Edit **HeatB** layer:

Operation and vertical position:

- Click LB on the "HeatB" layer from the list of layers on the right-hand side
- Click RB on the "HeatB" layer
- In the "Layer Properties" window which appears:
  - set the **Operation:** pull-down menu to "insert"
  - set the **Position [um]:** to 0.75
  - set the **Thickness [um]:** to 0.15
  - click **Accept**, closing the "Layer Properties" window

Layout:

- Draw a layout (top view) of the "HeatB" layer, by selecting the **New Rectangle** button [ ]
- Draw a rectangle with coordinates (65.76 um, 23.36 um) to (70.4 um, 42.4 um).
- To check dimensions, click **RB** on the red (highlighted) rectangle: correct values should be (Pos.X = 65.76, Pos.Y = 23.36, Size X = 4.64, Size Y = 19.04)

16. Edit **HeatC** layer:

Operation and vertical position:

- Click LB on the "HeatC" layer from the list of layers on the right-hand side
- Click RB on the "HeatC" layer
- In the "Layer Properties" window which appears:
  - set the **Operation:** pull-down menu to "insert"
  - set the **Position [um]:** to 0.75
  - set the **Thickness [um]:** to 0.15
  - click **Accept**, closing the "Layer Properties" window

Layout:

- Draw a layout (top view) of the "HeatC" layer, by selecting the **New Rectangle** button [ ]
- Draw a rectangle with coordinates (44.0 um, 23.36 um) to (64.0 um, 42.4 um).
- To check dimensions, click **RB** on the red (highlighted) rectangle: correct values should be (Pos.X = 44.0, Pos.Y = 23.36, Size X = 20.0, Size Y = 19.04)

17. Edit **HeatD** layer:

Operation and vertical position:

- Click LB on the "HeatD" layer from the list of layers on the right-hand side
- Click RB on the "HeatD" layer
- In the "Layer Properties" window which appears:
  - set the **Operation:** pull-down menu to "insert"
  - set the **Position [um]:** to 0.75
  - set the **Thickness [um]:** to 0.15
  - click **Accept**, closing the "Layer Properties" window

Layout:

- Draw a layout (top view) of the "HeatD" layer, which includes four rectangles
- Draw first rectangle by selecting the **New Rectangle** button [ ]
  - Draw a rectangle with coordinates (22.56 um, 23.52 um) to (42.56 um, 42.52 um).
  - To check dimensions, click **RB** on the red (highlighted) rectangle: correct values should be (Pos.X = 22.56, Pos.Y = 23.52, Size X = 20.0, Size Y = 19.0)
- Draw second rectangle by selecting the **New Rectangle** button [ ]
  - Draw a rectangle with coordinates (22.56 um, 44.0 um) to (42.56 um, 63.0 um).

    - To check dimensions, click **RB** on the red (highlighted) rectangle: correct values should be (Pos.X = 22.56, Pos.Y = 44.0, Size X = 20.0, Size Y = 19.0)
- Draw third rectangle by selecting the **New Rectangle** button [ ]
    - Draw a rectangle with coordinates (44.0 um, 44.0 um) to (64.0 um, 63.0 um).
    - To check dimensions, click **RB** on the red (highlighted) rectangle: correct values should be (Pos.X = 44.0, Pos.Y = 44.0, Size X = 20.0, Size Y = 19.0)
- Draw fourth rectangle by selecting the **New Rectangle** button [ ]
    - Draw a rectangle with coordinates (65.4 um, 44.0 um) to (85.4 um, 63.0 um).
    - To check dimensions, click **RB** on the red (highlighted) rectangle: correct values should be (Pos.X = 65.4, Pos.Y = 44.0, Size X = 20.0, Size Y = 19.0)

18. Edit **Active_Contact** layer:

Operation and vertical position:
- Click LB on the "Active_Contact" layer from the list of layers on the right-hand side
- Click RB on the "Active_Contact" layer
- In the "Layer Properties" window which appears:
    - set the **Operation:** pull-down menu to "insert"
    - set the **Position [um]:** to 0.75
    - set the **Thickness [um]:** to 0.8
    - click **Accept**, closing the "Layer Properties" window

Layout:
- The layout for this layer already exists, though some clean-up is of use. Specifically, there are elements of the contacts in the four corners of the model which are not needed.
- These elements may be deleted by highlighting them and hitting the delete button on your keyboard
- To highlight these elements, simply click LB near the elements and drag and hold the mouse to create a square enclosing the elements. The highlighted elements will appear in red. Simply hit delete to remove these elements.

19. Edit **Metal1** layer:

Operation and vertical position:
- Click LB on the "Metal1" layer from the list of layers on the right-hand side
- Click RB on the "Metal1" layer
- In the "Layer Properties" window which appears:
    - set the **Operation:** pull-down menu to "insert"
    - set the **Position [um]:** to 1.55
    - set the **Thickness [um]:** to 0.65
    - click **Accept**, closing the "Layer Properties" window

Layout:
- The layout for this layer already exists, though some clean-up is of use. Specifically, there are elements of the contacts in the four corners of the model which are not needed.
- These elements may be deleted by highlighting them and hitting the delete button on your keyboard

20. Edit **M1M2** layer:

Operation and vertical position:
- Click LB on the "M1M2" layer from the list of layers on the right-hand side
- Click RB on the "M1M2" layer
- In the "Layer Properties" window which appears:
    - set the **Operation:** pull-down menu to "insert"
    - set the **Position [um]:** to 2.2
    - set the **Thickness [um]:** to 1.0
    - click **Accept**, closing the "Layer Properties" window

Layout:
- The layout for this layer already exists, though some clean-up is of use. Specifically, there are elements of the contacts in the four corners of the model which are not needed.
- These elements may be deleted by highlighting them and hitting the delete button on your keyboard

21. Edit **Metal2** layer:
    Operation and vertical position:
    - Click LB on the “Metal2” layer from the list of layers on the right-hand side
    - Click RB on the “Metal2” layer
    - In the “Layer Properties” window which appears:
        - set the **Operation:** pull-down menu to “insert”
        - set the **Position [um]:** to 3.2
        - set the **Thickness [um]:** to 0.65
        - click **Accept**, closing the “Layer Properties” window

    Layout:
    - The layout for this layer already exists, though some clean-up is of use. Specifically, there are elements of the contacts in the four corners of the model which are not needed.
    - These elements may be deleted by highlighting them and hitting the delete button on your keyboard

**Save Micromesh Project and Technology Files**

22. Save your current settings as a *Micromesh Technology File*:
    - Click **File → Save Technology** from the top menu.
      The Save Technology window opens up.
    - In **File name**, type “IC_Heat.umt” as the name of the file, and click **OK**.

    Micromesh Technology File is saved with extension *.umt* (for µ*Mesh Technology*).This file can be used later for fast and automatic building of 3D models and meshes from different layouts imported from GDSII or CIF files, associated with the same technological process.
    In such a case, click Yes in the Technology File window appearing during importing GDSII or CIF layout, and select the saved Micromesh Technology File (.umt)

23. At this point, you may save your existing Micromesh project to a disk file, to secure the work you have done so far. Projects in CFD-Micromesh are saved to 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 disk file with extension *.ump* (for µ*Mesh Project*).
    - From the toolbar, click **Save** **.** The “Save Project As” dialog opens.
    - In **File name**, type “IC_Heat.ump” as the name of the file, and click **OK**.

**Generating and Viewing 3D Solid Model**

24. There are three methods of generating and viewing a 3D model in CFD-Micromesh:

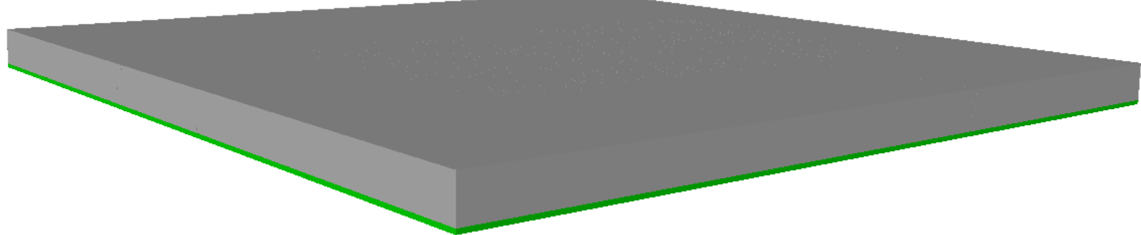
    - **3D Quick View** – Very fast building of a 3D model from planar elements of defined position and thickness, and viewing the 3D model with zooming and rotation. This option is limited to layouts including <u>rectangular shapes only</u>. Non-rectangular shapes will appear as rectangular in any 3D quick view representation. Typical model building time: <u>1-2 seconds</u> (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 <u>arbitrary shapes</u>, including rounded and oblique side-walls, non-planar conformal layers, etc. Viewing the 3D model allows for zooming and rotation. Typical model building time: <u>1-5 minutes</u> (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 <u>arbitrary shapes,</u> 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 (or from top menu: **Compute -> Quick View**).

  A new window appears, and it shows the quick view model, similar to below. Note that all that can be seen is the deposited Oxide and Bulk_Silicon. The other layers are internal to the geometry and are not visible here.



  - 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.
  - Scaling of the model, if desired, is available by clicking the **Scale Solid** button [ ] in the top toolbar
  - Close the "3D Quick View" window.

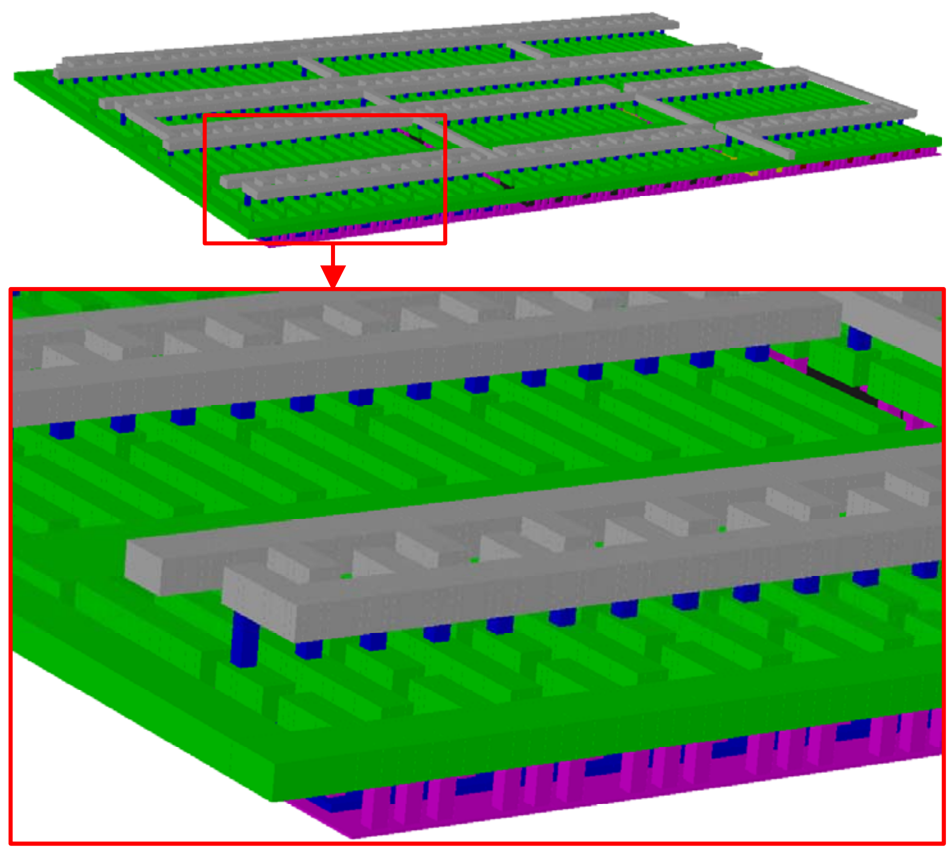
- Viewing the "internal" structure, i.e. interconnects (metal1, metal2), vias, heat sources, etc. This will be accomplished by making the Oxide and Bulk_Silicon layers non-visible by temporarily turning off meshing of these layers.

  - Click **LB** on the **Oxide** layer name.
  - Click **RB** on **Oxide** layer
  - In the **Layer Properties** window which opens, click on **Edit Material** button
  - In the **Materials** window which opens, uncheck the **Mesh It** box for **Oxide**
  - Click **Close**, in the **Materials** window and click **Accept** in the **Layer Properties** window

  - Click **LB** on the **Bulk_Silicon** layer name.
  - Click **RB** on **Bulk_Silicon** layer
  - In the **Layer Properties** window which opens, click on **Edit Material** button
  - In the **Materials** window which opens, uncheck the **Mesh It** box for **Bulk_Silicon**
  - Click **Close**, in the **Materials** window and click **Accept** in the **Layer Properties** window

  - Click **LB** on "3D Quick View" button [ ] in the top toolbar (or from top menu: **Compute → Quick View**).

  A new window appears, and it shows the quick view model, similar to below. Note that the deposited Oxide and Bulk_Silicon layers are no longer seen. Now, the remaining internal layers are visible.



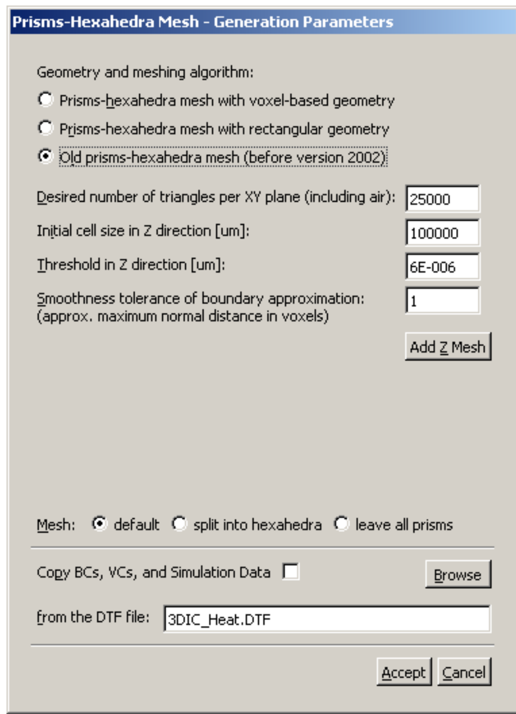
- Close the "3D Quick View" window.
- Re-check the **Mesh It** box for the **Oxide** and **Bulk_Silicon** layers layer, and save the Micromesh Project file.

**Automatic Generation of 3D Computational Mesh**

25. Add layer to enable control of mesh cell size.
    - Add mesh cell size layer.
        - Click LB on the top-most layer in the right side panel, which should be **metal2**
        - click LB the **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: metal2 (deposit)"
        - In the list of layers, click **RB** on the "NEW: metal2 (deposit)" layer.
        - In the "Layer Properties" window which appears, change the **Operation** pull-down selection to "mesh cells size"
        - In the **Mesh Cells Size:** field enter 10
        - click **Accept**
    - Add mesh cell size layer layout, consisting of four rectangles surrounding the IC component layout.
        - Draw a layout (top view) of the "mesh cell size" layer, by selecting the **New Rectangle** button [ ]
        - Draw a rectangle with coordinates (0.0 um, 0.0 um) to (20.6 um, 100.0 um).
        - To check dimensions, click **RB** on the red (highlighted) rectangle: correct values should be (Pos.X = 0.0, Pos.Y = 0.0, Size X = 20.6, Size Y = 100.0)
        - Repeat this process for the following three additional layout rectangles:
            - coordinates (87.7 um, 0.0 um) to (100.0 um, 100.0 um) verified with values be (Pos.X = 87.7, Pos.Y = 0.0, Size X = 12.3, Size Y = 100.0)

  - coordinates (0.0 um, 0.0 um) to (100.0 um, 22.0 um) verified with values be (Pos.X = 0.0, Pos.Y = 0.0, Size X = 100.0, Size Y = 22.0)
  - coordinates (0.0 um, 65.0 um) to (100.0 um, 100.0 um) verified with values be (Pos.X = 0.0, Pos.Y = 65.0, Size X = 100.0, Size Y = 35.0)

26. Build 3D unstructured computational grid.
    - Select the “Prism-Hex Mesh” button [ ] from the top toolbar (or from the top menu: **Compute → Prism-Hex Mesh**).
    - A **Prisms-Hexahedra Mesh** window will open (as below)
    - Check the “Old prisms-hexahedra mesh (before version 2002)” circle
    - Set the field values as shown in the image below.

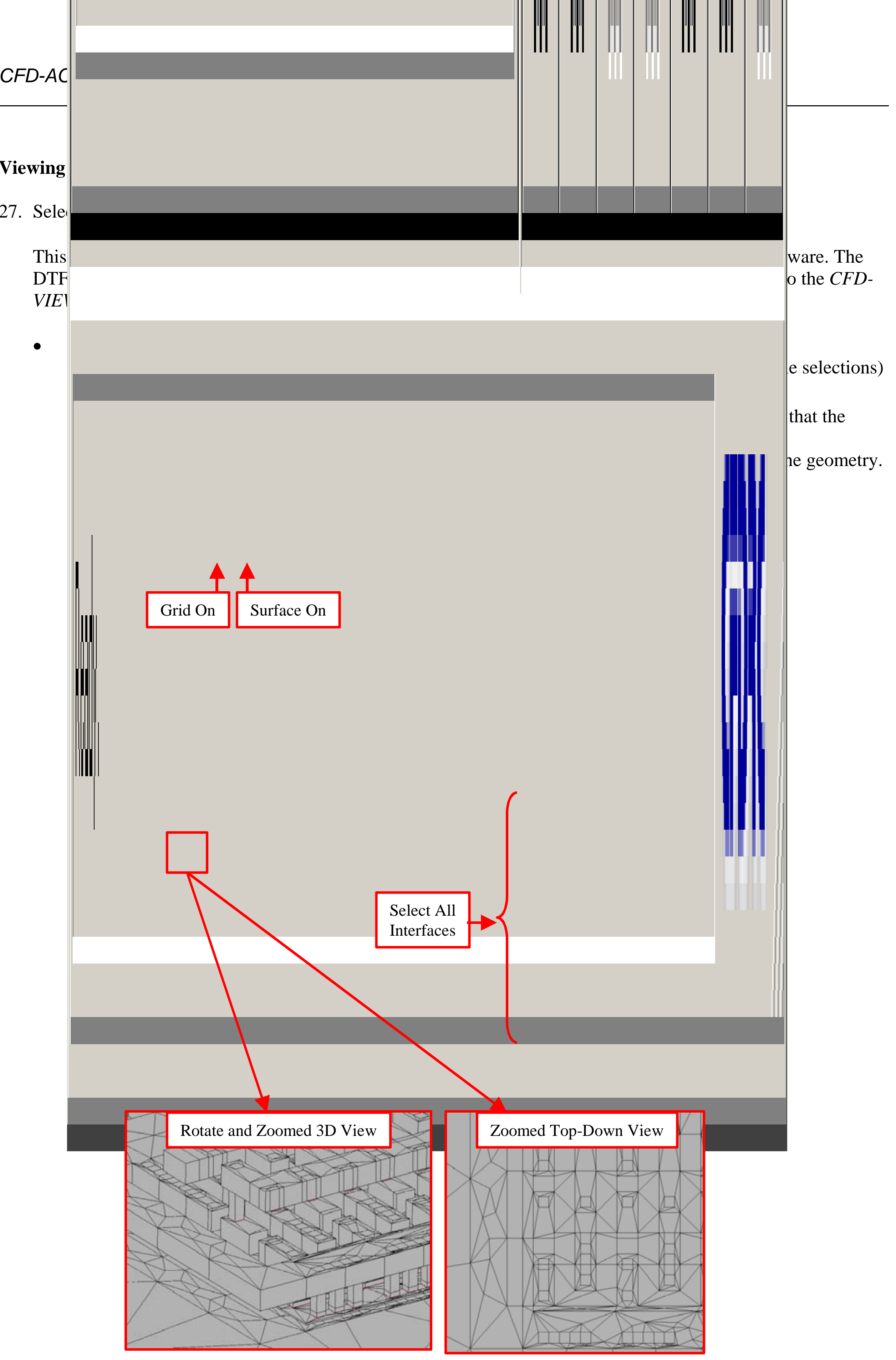



In this tutorial, leave the **Copy BCs, VCs, and Simulation Data** box unchecked. In the first run, boundary conditions (BCs), volume conditions (VCs) and other simulation parameters have to be manually assigned in the DTF file, through the Graphical User Interface (GUI). However, if a change in the 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.

- Click **Accept** to build the 3D computational grid. 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 (microstrip.DTF in this case).

- In the “Status” window just before the “Computation Completed” statement, the number of grid cells generated is provided. Specifically, both prism and hex cell counts are given. For this tutorial, ~145,000 prism and ~41,000 hexa cells are generated, hence ~186,000 total computational cells for the geometry.


Grid On
Surface On
Select All Interfaces
Rotate and Zoomed 3D View
Zoomed Top-Down View

**Setting Up A Thermal Simulation in CFD-ACE+**

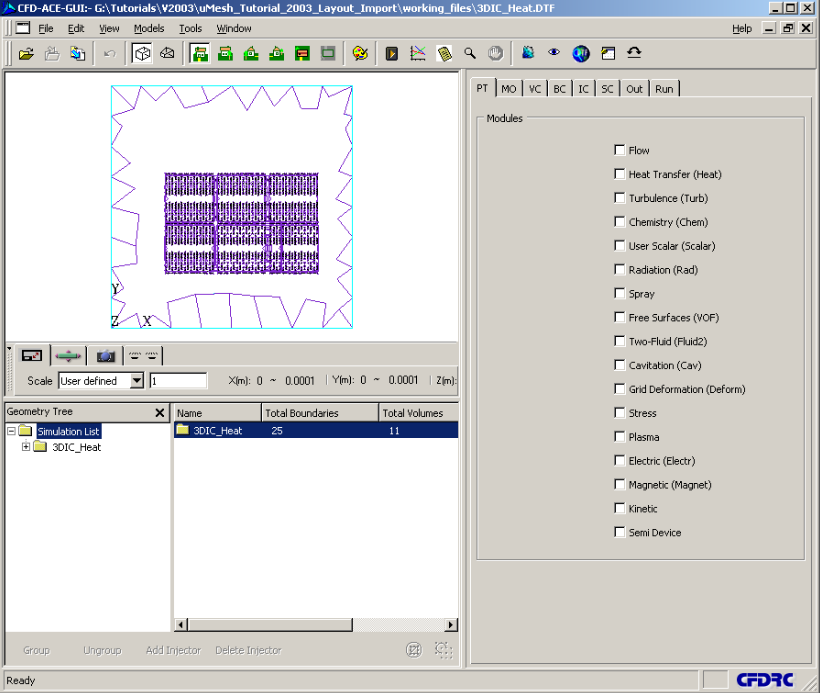
28. Open CFD-ACE-GUI with the generated 3D computational mesh (DTF file).

- Select the **CFD-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 and volume 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, CFD-ACE-GUI should automatically open the DTF file you just created (IC_Heat.DTF). If it does not open the DTF file automatically, open the DTF file using the menu command: **File → Open ...** IC_Heat.DTF.

  The image below illustrates the IC_Heat.DTF opened in CFD-ACE-GUI.




29. Setup steady simulation for the IC geometry.

- **PT** (Problem Type)
  - ➢ Check the **Heat Transfer (Heat)** module.

- **MO** (Model Options)
  - ➢ In **Shared** panel, verify that Transient Conditions is set to **Steady**

- **VC** (Volume Conditions)

  - ➢ Check the list of volumes (in lower left window) to make sure that all volumes have the **VC Type** Solid. If for some reason this is not the case, click **Ctrl-A** to select all the Volumes. In the **Properties** drop-down list at right, select **Solid**, and click **Apply**.

- In upper right panel in the **VC Setting Mode,** set the top pull-down menu to **Properties**
- Select the **Bulk_Silicon** and **Semi_Device** volume names by clicking **LB** on them (hold shift or ctrl key to select multiple names at once)
    - Click the **Group** button in the lower left corner to group these volumes together.
    - In the **Phys** panel at right:
        - Set the **Density** to 2300 kg/m^3
    - In the **Therm** panel at right:
        - Set the **Specific Heat** to 700 J/kg-K
        - Set the **Thermal Conductivity** to 155 W/m-K
    - Click **Apply**
- Select the volumes **Oxide**, **HeatA, HeatB, HeatC, HeatD** by clicking **LB** on them (hold shift or ctrl key to select multiple names at once).
    - Click the **Group** button in the lower left corner to group these volumes together.
    - In the **Phys** panel at right:
        - Set the **Density** to 2150 kg/m^3
    - In the **Therm** panel at right:
        - Set the **Specific Heat** to 740 J/kg-K
        - Set the **Thermal Conductivity** to 1.3 W/m-K
    - Click **Apply**
- Select the volumes **Active_Contact, Metal1, M1M2, Metal2** by clicking **LB** on them (hold shift or ctrl key to select multiple names at once).
    - Click the **Group** button in the lower left corner to group these volumes together.
    - In the **Phys** panel at right:
        - Set the **Density** to 2700 kg/m^3
    - In the **Therm** panel at right:
        - Set the **Specific Heat** to 938 J/kg-K
        - Set the **Thermal Conductivity** to 230 W/m-K
    - Click **Apply**

- **BC** (Boundary Conditions)
    - Select the **Bottom_Bulk_Silicon** (wall) name.
        - In the **Heat** panel at right
            - Set the **Sub Type** to **Isothermal**
            - Set the **Temperature** to a Constant value of 300 K
        - Click **Apply**
    - Select the **Semi_Device_HeatA** (interface) name.
        - In the **Heat** panel at right
            - Check the **Wall Heat Source** box
            - Set the **Wall Heat Source** to a Constant value of 39045 W/m^2
        - Click **Apply**
    - Select the **Semi_Device_HeatB** (interface) name.
        - In the **Heat** panel at right
            - Check the **Wall Heat Source** (interface) box
            - Set the **Wall Heat Source** to a Constant value of 513865 W/m^2
        - Click **Apply**
    - Select the **Semi_Device_HeatC** (interface) name.
        - In the **Heat** panel at right
            - Check the **Wall Heat Source** box
            - Set the **Wall Heat Source** to a Constant value of 477895 W/m^2
        - Click **Apply**
    - Select the **Semi_Device_HeatD** name.
        - In the **Heat** panel at right
            - Check the **Wall Heat Source** box
            - Set the **Wall Heat Source** to a Constant value of 740131.6 W/m^2
        - Click **Apply**

- **IC** (Initial Conditions)
  - ➢ Leave all parameters at their default values.

- **SC** (Solver Control)
  - ➢ In the **Iter** panel, set **Max. Iterations** to 30
  - ➢ In the **Solvers** panel, change the **Enthalpy** pull-down menu to AMG
  - ➢ In the **Relax** panel, set the **Enthalpy** value to 1E-13

- **Out** (Output Parameters)
  - ➢ Under the **Print** sub-panel, check the **Heat Flux Summary** 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 this tutorial simulation, you may turn on: Static Temperature and Wall Heat Flux

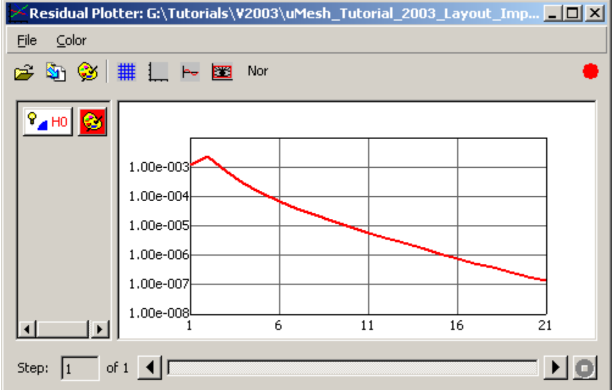
- Save the simulation file from the top menu by: **File → Save** (saves file "IC_Heat.DTF")

30. Run the IC thermal simulation.

- From the **Run** panel:

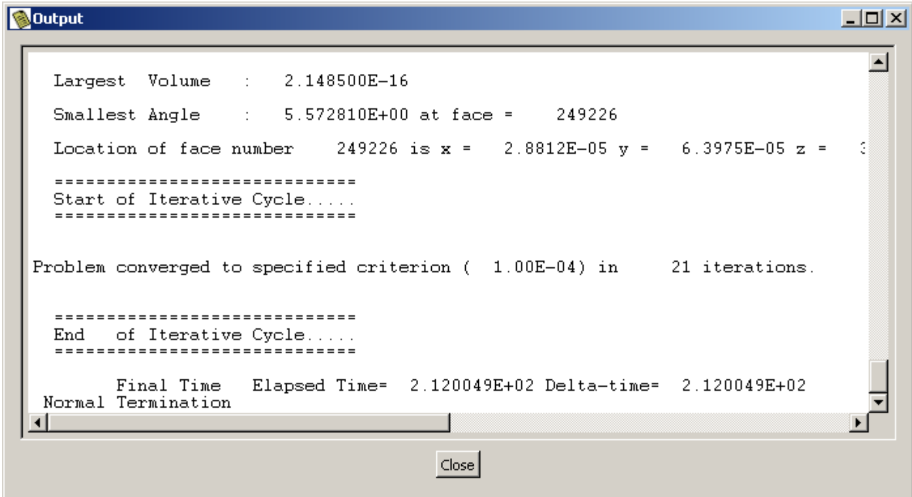
  - ➢ Click **Submit to Solver**.
    (If any changes are made to the file after opening, a new window will open up asking you whether you want to submit the simulation under the same name or a different name. If you have made any changes, click the **Submit Job Under Current Name** button.)

  - ➢ Click **View Residual**.
    If the values of the residuals eventually go down by 3 or more 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 various information with regard to the simulation and the state of the simulation. In addition, the "Elapsed Time" value at the bottom of the Output file shows the CPU time for the completed simulation, in seconds.


```
Largest  Volume    :   2.148500E-16
Smallest Angle     :   5.572810E+00 at face =     249226
Location of face number     249226 is x =   2.8812E-05 y =   6.3975E-05 z =

=============================
Start of Iterative Cycle.....
=============================

Problem converged to specified criterion (  1.00E-04) in     21 iterations.

=============================
End   of Iterative Cycle.....
=============================

        Final Time   Elapsed Time=  2.120049E+02 Delta-time=  2.120049E+02
 Normal Termination
```

**Post-Processing Simulation Results**

31. Visualize IC thermal simulation results.

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*.

- Click the Launch **CFD-VIEW** button [ ] in the top toolbar of CFD-GUI. A new CFD-VIEW window displaying the IC structure of "IC_Heat.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 "IC_Heat.DTF"

- In CFD-VIEW, perform the following steps to obtain a temperature image as illustrated below:

  - In the image window, use the **LB**, **MB**, and **RB** to rotate, zoom, and pan the image as desired.

  - Create Z-Cut Temperature Plane:
    - Select the **Bulk_Silicon volume** from the bottom right panel list of objects
    - Click on the **Z-Cut** button in the upper right panel of buttons
    - Set the Z-Cut location to **0.75e-6** (in meters) in the text-field in the middle right panel
    - Click the **Flooded On** button along the top tool bar
    - In the pull-down **Color:** menu, select **T** (temperature)

  - Create X-Cut Temperature Plane:
    - Select the **Bulk_Silicon volume** from the bottom right panel list of objects
    - Click on the **X-Cut** button in the upper right panel of buttons
    - Set the X-Cut location to **75.0e-6** (in meters) in the text-field in the middle right panel
    - Click the **Flooded On** button along the top tool bar
    - In the pull-down **Color:** menu, select **T** (temperature)

  - Create Y-Cut Temperature Plane:
    - Select the **Bulk_Silicon volume** from the bottom right panel list of objects
    - Click on the **Y-Cut** button in the upper right panel of buttons
    - Set the Y-Cut location to **55.0e-6** (in meters) in the text-field in the middle right panel
    - Click the **Flooded On** button along the top tool bar
    - In the pull-down **Color:** menu, select **T** (temperature)

- Generate temperature legend:
  - Select the **Bulk_Silicon volume** from the bottom right panel list of objects
  - Click on the **Legend** button in the upper right panel of buttons
  - You may edit the legend by **RB** clicking on the legend generated in the main window and selecting **properties**

- (Optional) Click the **Perspective** button along the top tool bar to visualize the geometry from a perspective point of view