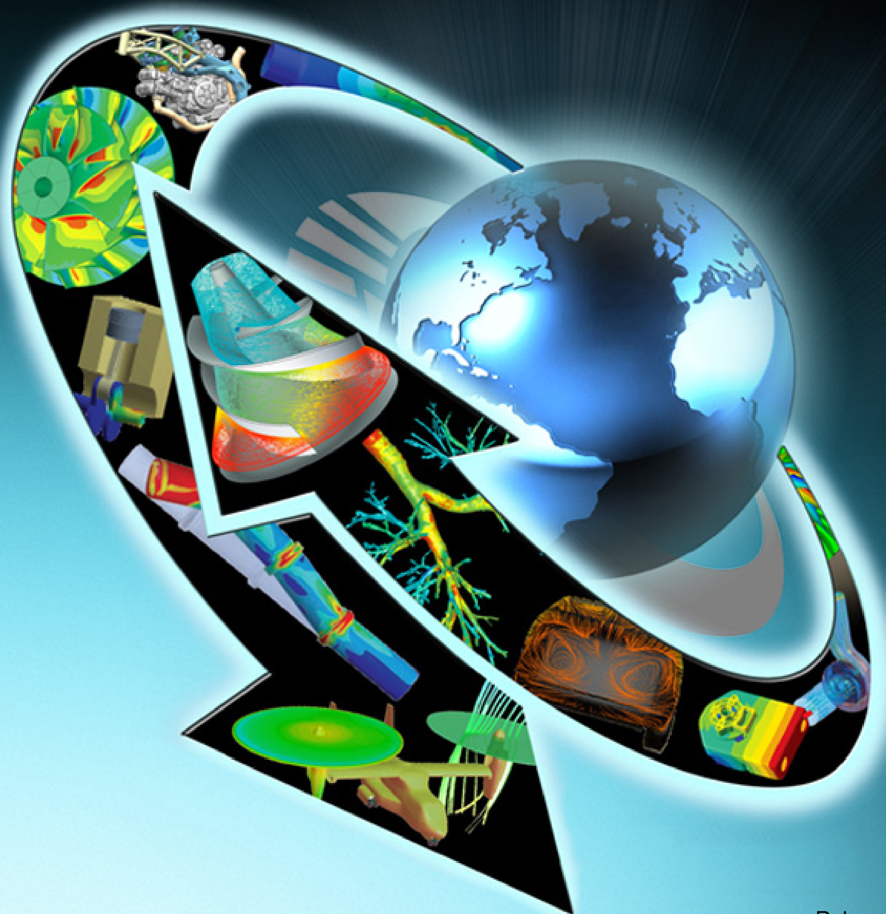
Customer Training Material

Workshop 4

Electronics Cooling with Natural Convection and Radiation

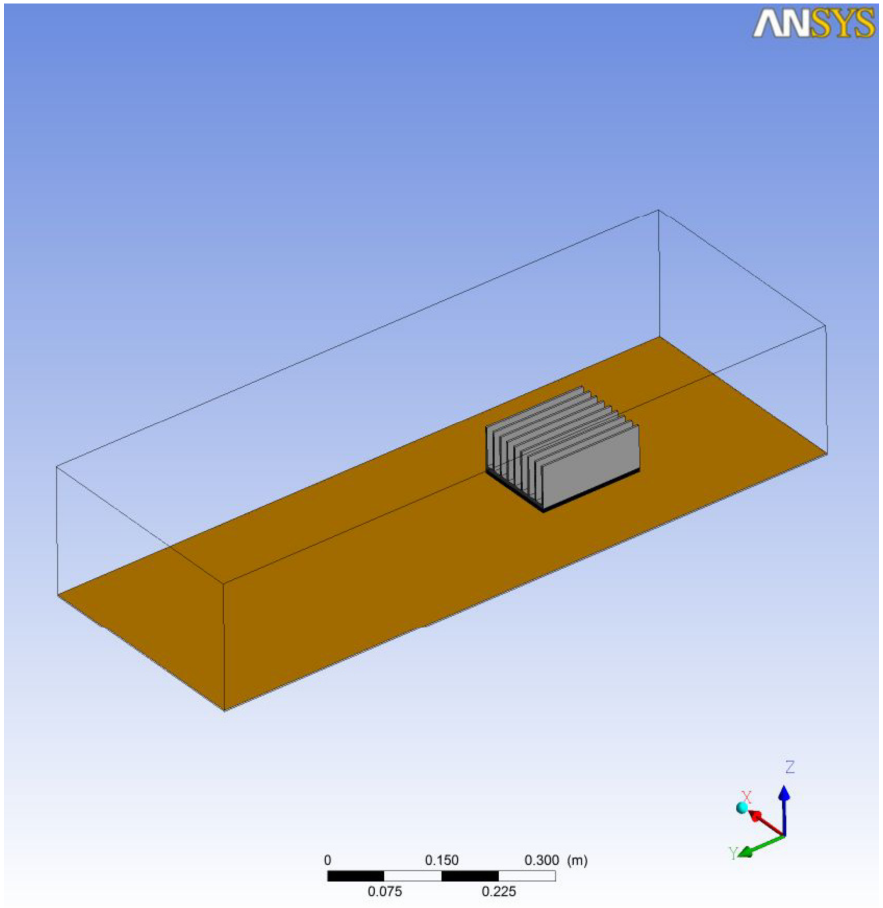


Introduction to ANSYS FLUENT

Release 13.0
December 2010

Goals

- In this workshop, you will model the heat dissipation from a hot electronics component fitted to a printed circuit board (PCB). Heat is dissipated via a finned heat sink
- The PCB is fitted into an enclosure which is open at the top and bottom
- Initially only the heat transfer via convection and conduction will be calculated. The effect of thermal radiation will then be included as a later stage

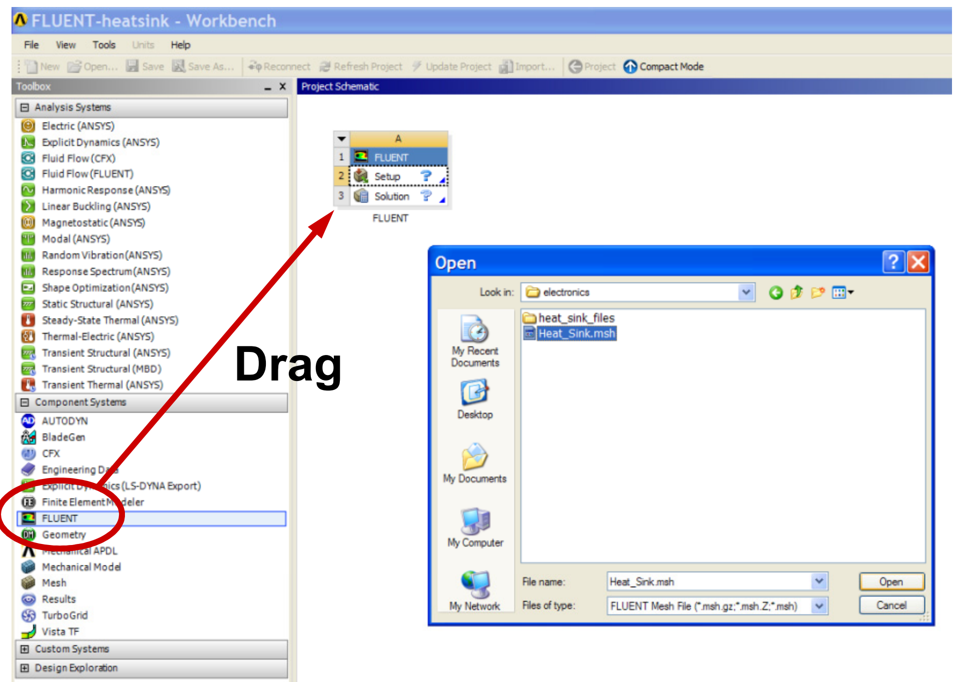


Mesh Import (Workbench)

- **This workshop can be done either inside or outside of ANSYS Workbench.**

If working with FLUENT within Workbench

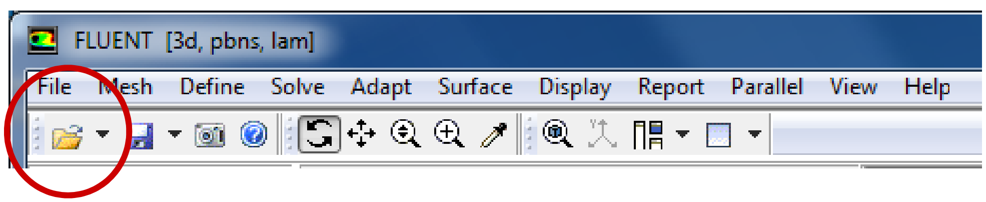
- **Open a new Workbench session and select a new FLUENT session from Component Systems**
- **Use Save As to save the session**
- **Import the mesh file.**
  - **Right-click on the Setup cell**
  - **Click on "Import FLUENT Case"**
  - **Change Files of Type to FLUENT Mesh File**
  - **Select the mesh file heatsink.msh.gz**
  - **Click Open**
- **Launch FLUENT using the default options**



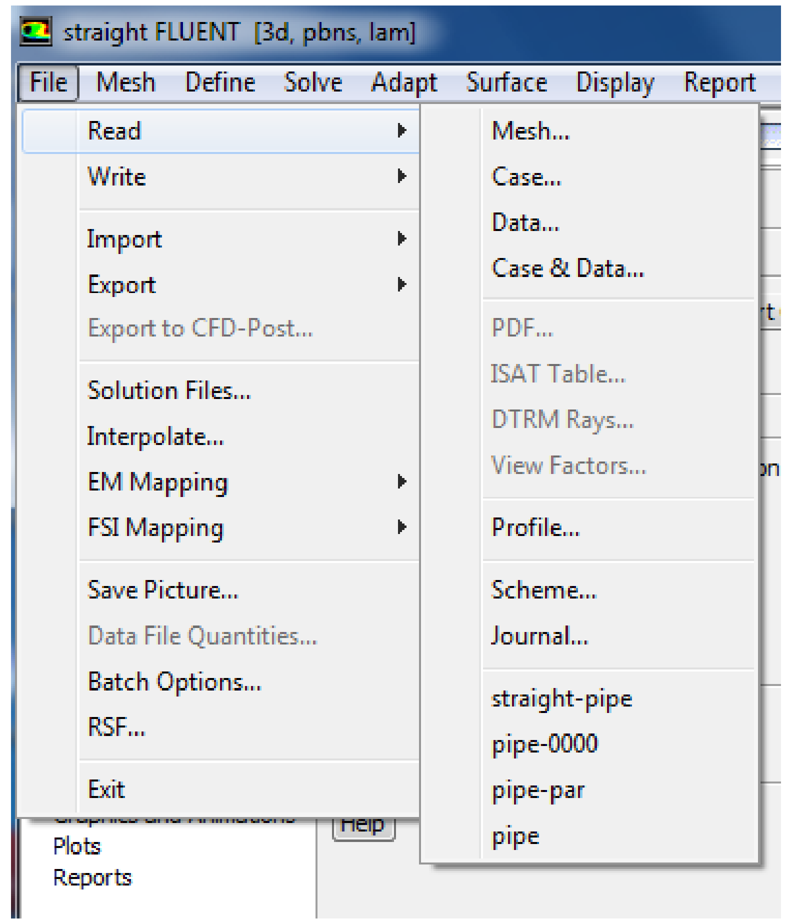
Mesh Import (Stand-alone)

If working with FLUENT as Standalone

- **Start a 3D FLUENT session from the icon or from the Windows Start menu**
- **Select either**
  - **File → Read → Mesh from the top menu**
  - **Open File icon from toolbar and select mesh**

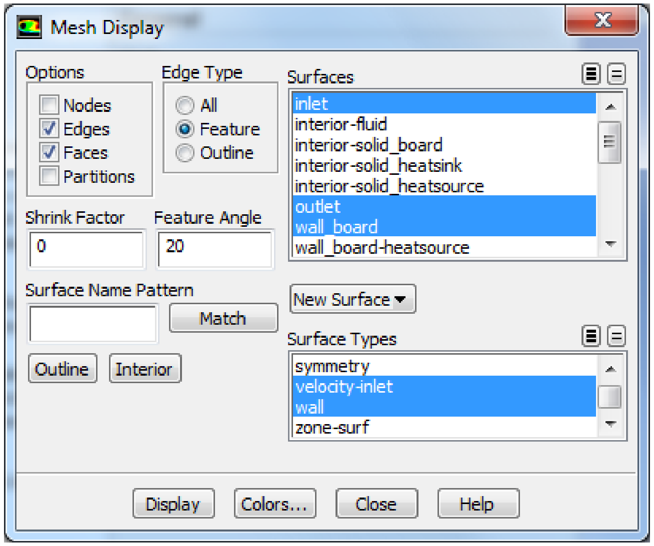


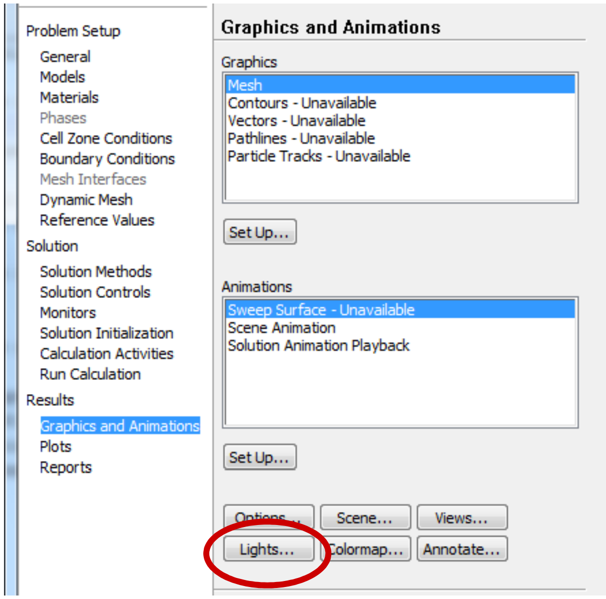
- **Open the file heatsink.msh**



View the model:

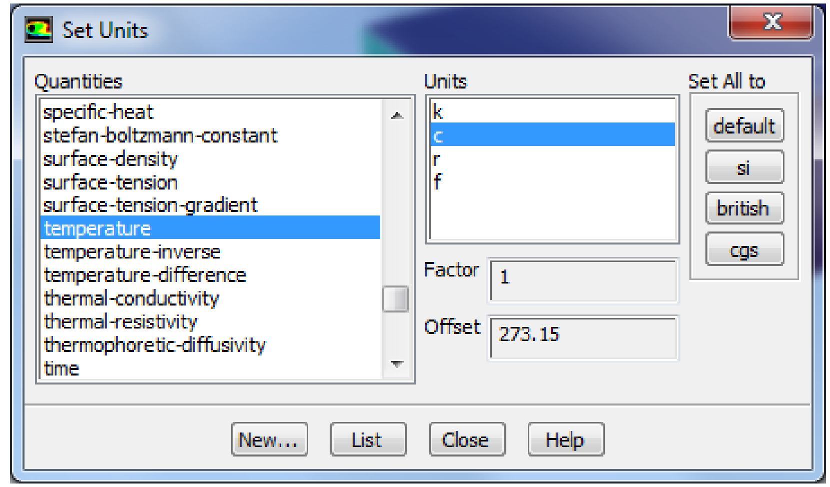
- Check the mesh (from *General* tab).
  - Review the text in the lower FLUENT window and check there are no errors
- Display the mesh and adjust the display settings
  - Under Graphics and Animations select Mesh
  - Set Faces to on, and Edges to ‘Feature’
  - Deselect all currently selected faces
  - Select Surface Types Pressure Outlet, Velocity Inlet and Wall, (note effect on Surfaces list)
  - Select Colors and Color by ID, Close
  - Click Display
- Display Lights and turn on the headlight



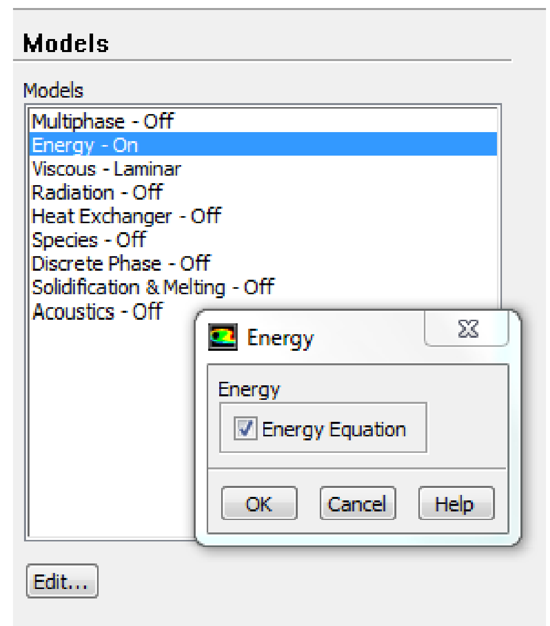


Model setup

- **Change temperature units to °C**
  Define → Units
  - **Select Temperature as a Quantity**
  - **Select “c” as the temperature units**
  - **Close the panel**



- **Enable the energy equation**
  - **Select the Models tree item**
  - **Double-click on Energy and enable the equation**
    OR
  - **Click on Energy and then Edit**



Comments on Model setup

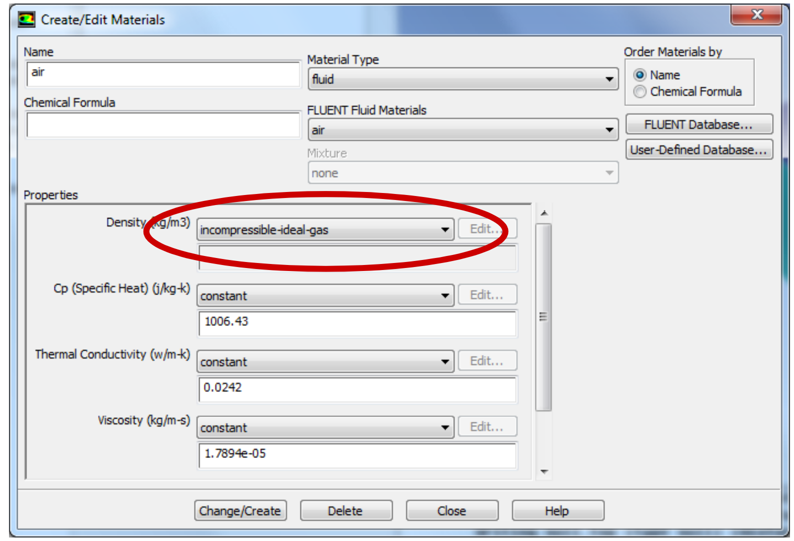
- General
  - It is good practice to display the grid after import to check for any boundary zone miss assignment and that you have opened the correct model
  - Workbench uses SI units (metres, kg etc) but if importing a mesh from another source check the scale and dimensions are correct
  - Check mesh is used to confirm the mesh is suitable for use in a CFD simulation
  - Report Quality is a backup to the quality tools available within the meshing application

- By default the energy equation is not solved to reduce CPU load because many problems are isothermal. In this case, temperature must be calculated so the energy equation needs to be enabled

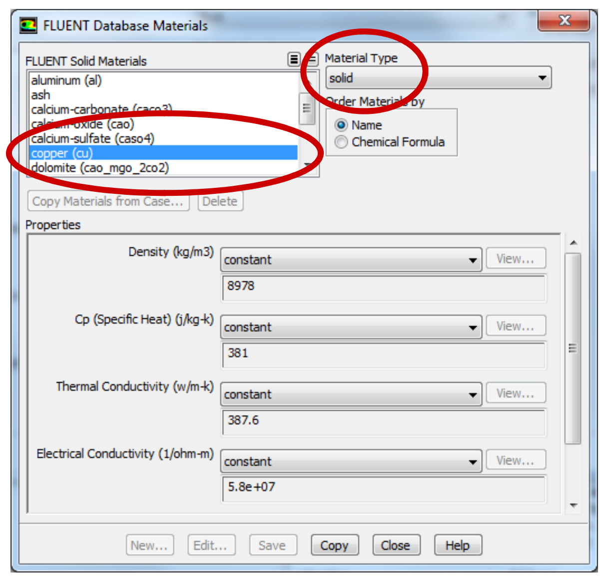
- The onset of turbulence is specified by the Reynolds Number (pipe flow) or Rayleigh Number (natural convection). Calculating these numbers for this geometry and conditions indicates that the flow will be laminar and so no change is required to the viscous model

Material properties

- **The air density needs to change with temperature (but not pressure)**
  - Select Materials → Air → Create/Edit
  - Change density to incompressible ideal gas
  - All other properties remain unchanged
  - Click Change / Create then close the fluid materials window

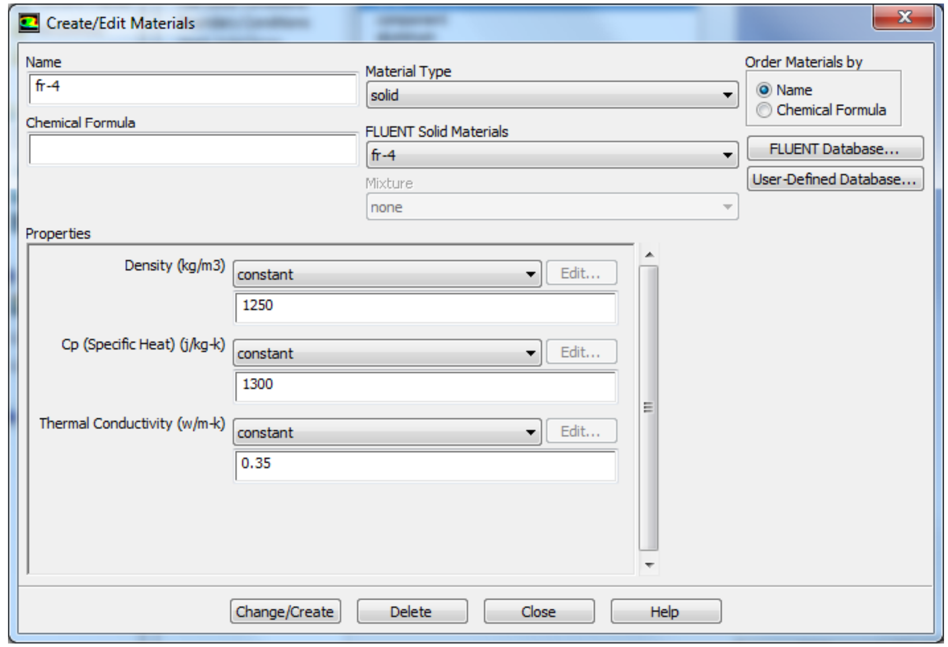


- **Define two additional solid materials (for the board and the heat sink).**
  - Select Materials → Solid → Create/Edit
  - Click the FLUENT Database button.
  - Change Type to Solid
  - Select Copper
  - Copy then close the database window



Material properties

- **Modify the copper material to produce the PCB material**
  - **The PCB is made of material "FR-4"**
    - **Change Name to fr-4**
    - **Delete the chemical formula**
    - **Density = 1250kg/m^3; c$_p$ = 1300 J/kg K; conductivity 0.35 W/m K**
    - **Click Change/Create.**
    - **Click No when prompted to overwrite copper.**
      - **Selecting No will create a new material fr-4, but copper remains in the material list**
      - **Selecting Yes would overwrite the copper material for the current case only (it's still in the database)**



- **Again, modify the copper material to create the "component" material using the steps above**
  - **Density = 1900 kg/m^3; c$_p$ = 795 J/kg K; Thermal Conductivity = 10 W/m K**
- **The heat sink is made of copper**

Comments on Material properties

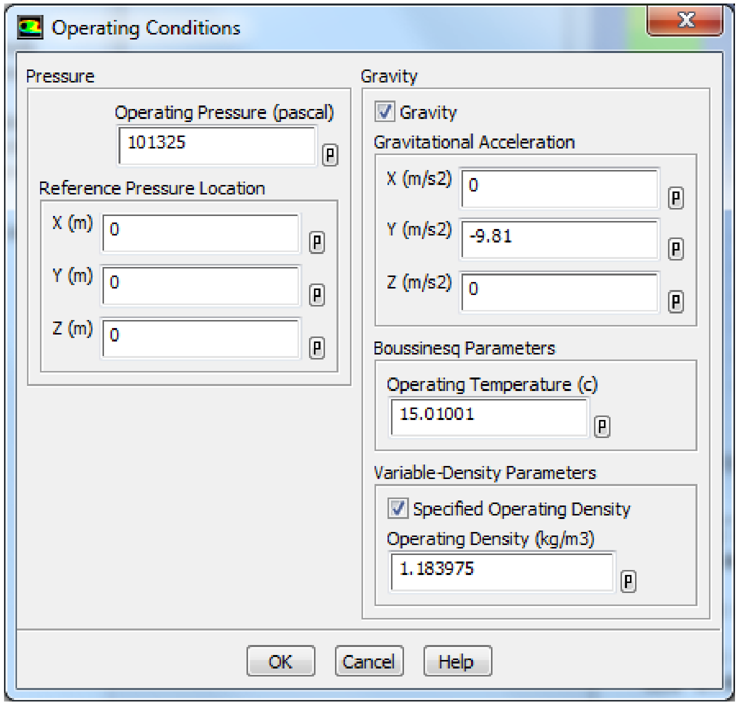
- In most natural convection problems the change of density with temperature drives the flow. The changes in pressure over the domain are minimal, and their effect on density negligible, hence the incompressible ideal gas density formulation can be used instead of the fully the compressible ideal gas model

- The FLUENT database contains basic properties for many materials. These are generally set to the standard STP/RTP values, but always check these are suitable before proceeding

- Additional materials can be added to the database; refer to the user documentation. It is often easier to copy a material from the database and then modify it; alternatively, you can modify the default material of aluminium and then choose not to overwrite

Set Cell Zone Conditions

- **Under Cell Zone Conditions click the Operating Conditions button.**
  - **Enable Gravity, and set the y component to -9.81 m/s^2**
  - **Turn on Specified Operating Density and set to 1.11 kg/m^3**
  - **Click OK**

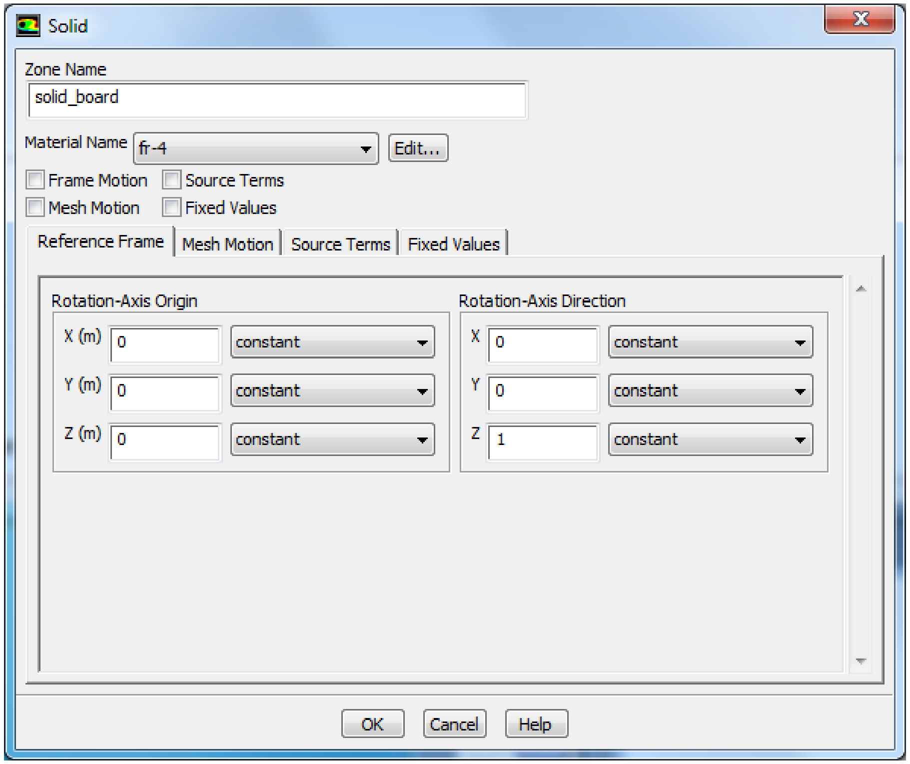


- **There are no changes to the fluid zone**
  - **(Cell Zone → Fluid → Edit)**
  - **Observe that this contains material air which is correct.**
  - **Close the pop-up window**

Set Cell Zone Conditions

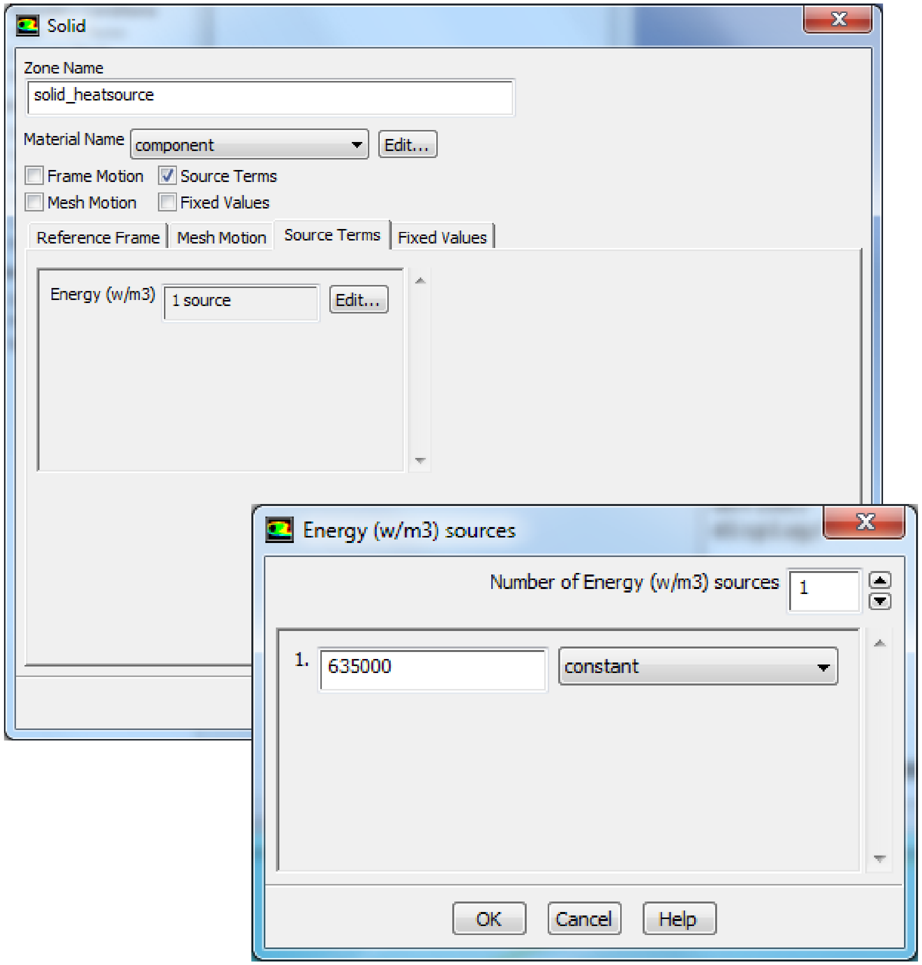
- **Set the material properties for the PCB:**
  - **Select cell zone solid_board, and Edit**
  - **Change the material to fr-4**
  - **OK to close the window**

- **Set the material properties for the heat sink:**
  - **Heatsink is made of copper**
  - **Select the cell zone solid_heatsink**
  - **Change the material to copper**



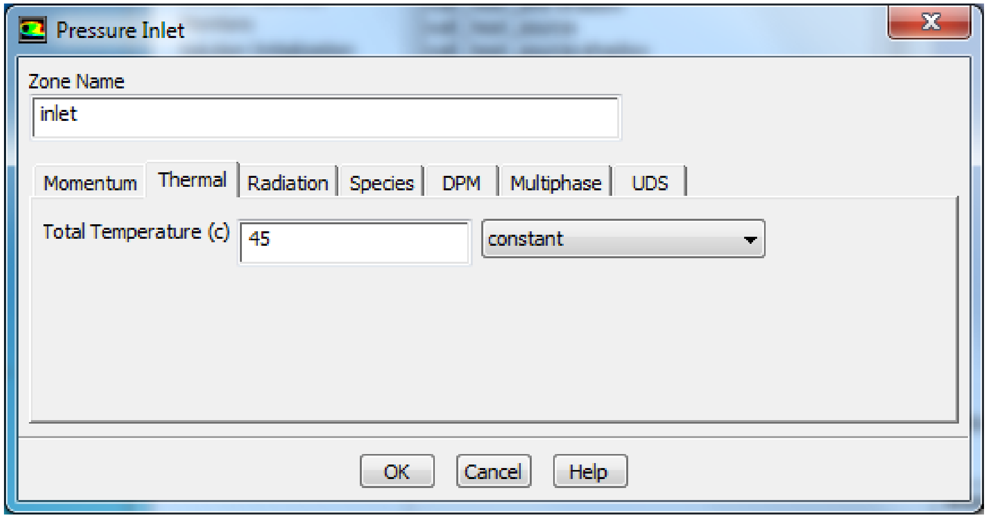
Set Cell Zone Conditions

- **Set the conditions for the component:**
  - **We need to set both the material, AND the thermal power (75W) dissipated by this component**
  - **Select zone solid_heatsource**
  - **Change the material to component**
  - **Check Source Terms then go to the Source Terms tab**
  - **The component volume is 0.11808x10^{-3} m^3. Hence, the volumetric source is 635,000 W/m^3**
  - **Create 1 constant energy source with the above value**



Set Boundary Conditions

- **For the Inlet (Boundary Conditions → Inlet)**
  - **Change type to Pressure Inlet**
  - **Keep the pressure at 0 Pa**
  - **Under the Thermal tab set the temperature to 45 °C, then OK**



- **For the Outlet:**
  - **Keep type as Pressure Outlet**
  - **Set 0 Pa Gauge Pressure**
  - **Change Direction to "From Neighbouring Cell"**
  - **Under the Thermal tab set Backflow Total Temperature to 45°C, then OK**

Set Boundary Conditions

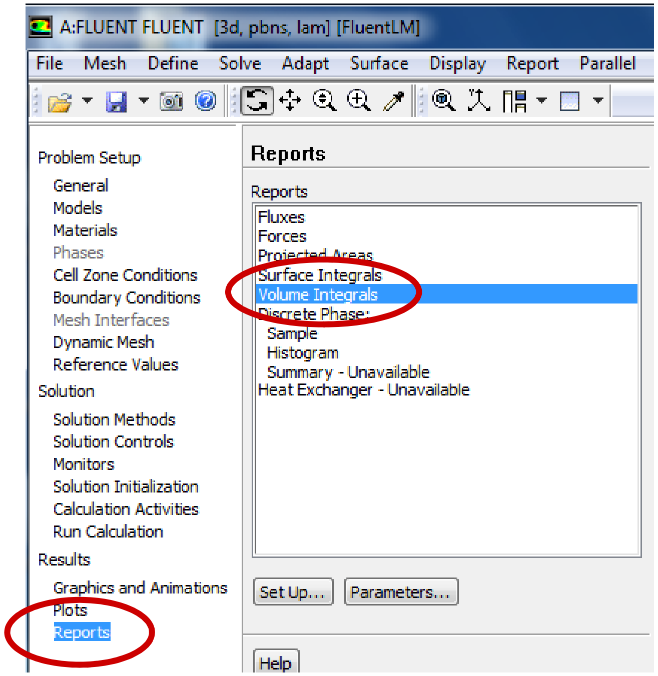
- Set the casing walls to be adiabatic:
  - Select boundary zone wall_left
  - Under the thermal tab, check this is set to zero heat flux
  - Repeat for wall_right and wall_top

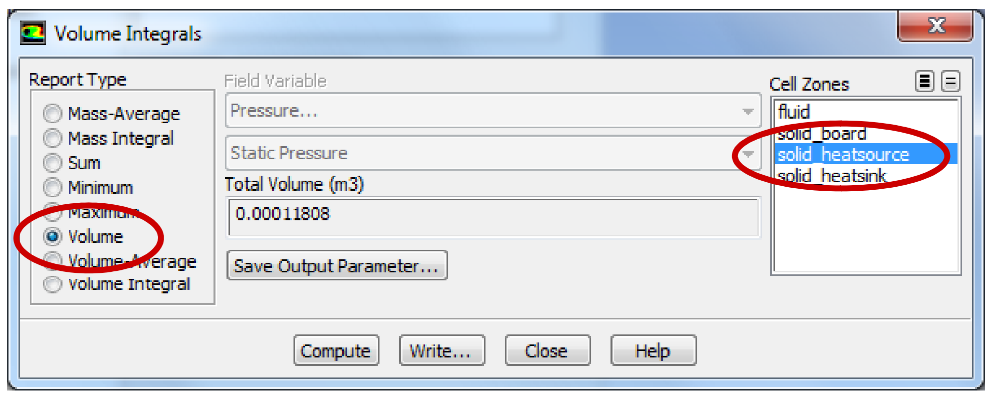
- Set the PCB outer surface thermal properties to be adiabatic.
  - These are the external surfaces of the model
  - The surfaces to set are wall_board_bottom and wall_board_side
  - Set these to zero heat flux as above.

  - Note that the surface wall_board is the surface of the PCB that borders the fluid air region, and so is not an exterior boundary

Comments on Cell and Zone Conditions

- **Operating conditions**
  - Gravity is required for natural convection. In many cases there are no gravity effects, and the option can remain inactive
  - Operating density is critical in natural convection problems, and should be set to the density at the far field temperature (i.e. inlet temperature)
  - Operating pressure position is related to the inlet/outlet pressure settings. In this example, it is positioned at the inlet
- **Boundary zones**
  - Inlet and Outlet pressure set to 0 Pa gauge, as it is at the operating pressure position
- **Volume data**
  - The volume of an entity can be reported from the Volume Integrals panel. Note that the solution must be initialized before the volume integrals are enabled





Comments on Cell and Zone Conditions

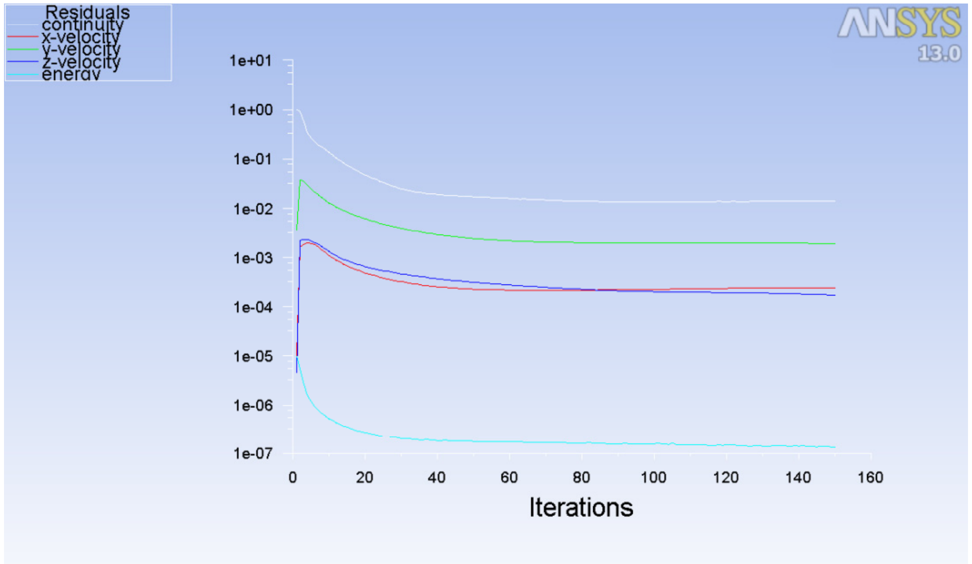
- **External heat loss:**
  - **Here, adiabatic conditions have been set for the external walls.**
  - **However if needed, FLUENT can model the external boundary in several different ways, including:**
    - **Known heat flux (W/m^2)**
    - **Known heat transfer coefficient (eg for natural convection from a vertical plate)**
    - **Heat transfer through radiation (by setting emissivity and temperature). Note this is not related to radiation models available within FLUENT**
    - **Shell conduction in the outer wall (hence fields for material and wall thickness)**

- **Interior boundaries:**
  - **In this example, there are four cell zones; including fluid and solid_board**
  - **Just one label wall_board was created at the pre-processing stage for the surface common to these two zones**
  - **When loading the mesh, FLUENT creates a clone: wall_board_shadow**
  - **Open up the boundary conditions for these surfaces and observe that:**
    - **wall_board has the adjacent cell zone fluid**
    - **wall_board_shadow has the adjacent cell zone solid_board**
  - **If setting emissivity or roughness, make sure the correct face is selected!**

Solver controls

- **Solution Methods → Spatial Discretization → Pressure**
  - **Change the pressure scheme to Body Force Weighted**
  - **Leave the remaining variables set to First Order Upwind.**

- **Solution Controls → Momentum**
  - **Adjust the Momentum under-relaxation factor to 0.3**



- **Solution Initialization**
  - **Compute from inlet**
  - **Set y-velocity to 0.1 m/s**
  - **Click the Initialize button**

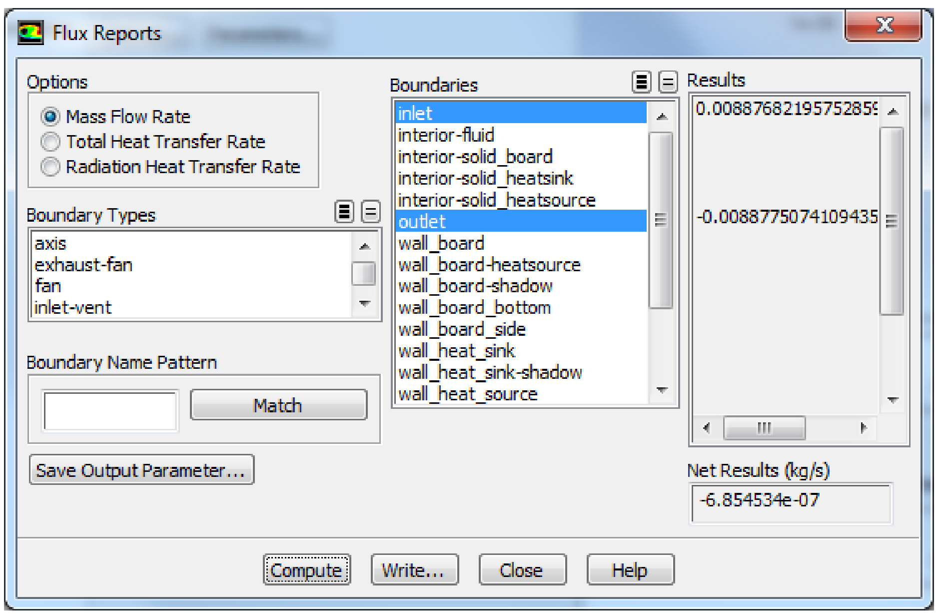
- **Run calculation**
  - **Set for 150 iterations and click Calculate.**
  - **The solution should only 5-10 minutes to complete**
  - **Note, a fully converged cas & dat are in the tutorial folder (ws4_no-radiation.*.gz)**

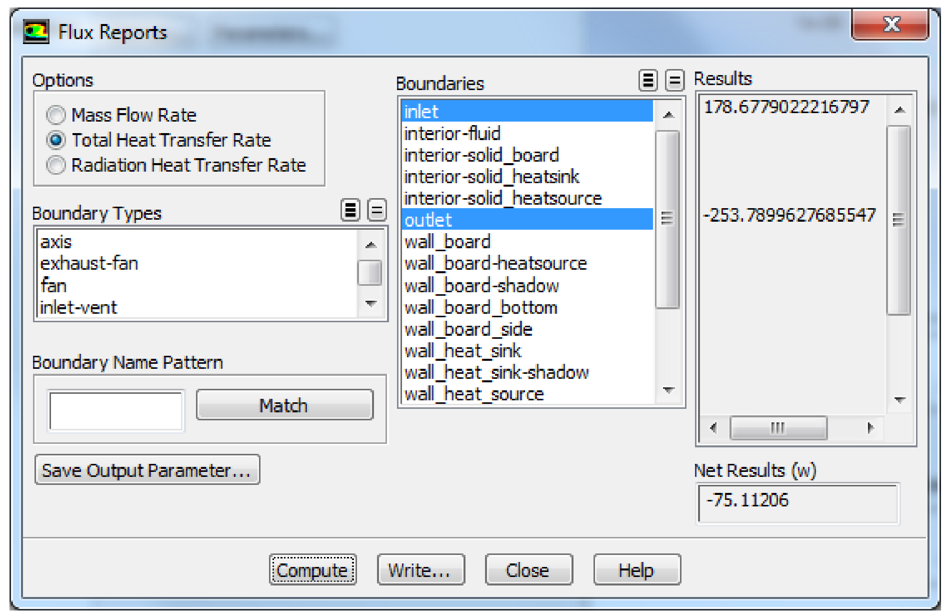
Comments on Solver controls

- **The solver settings are tuned for an overall robust solution of most situations. In this model we require the Body Force Weighted pressure scheme to account for the natural convection effects**
- **Once the solution has begun to, or has converged, the momentum and energy equations should be switched to second order to improve the accuracy of the solution. You may wish to do this later if time permits**
- **The momentum within the model is relatively low so reducing the momentum under-relaxation factor is recommended**
- **Solution initialization is used to provide the first guess prior to the first solver iteration; and it should be as close to the final solution as practical**
- **In this model convergence cannot easily be measured using the residuals. Typically mass conservation and monitor points would be used to ensure the solution has stopped changing**

Quick post-processing

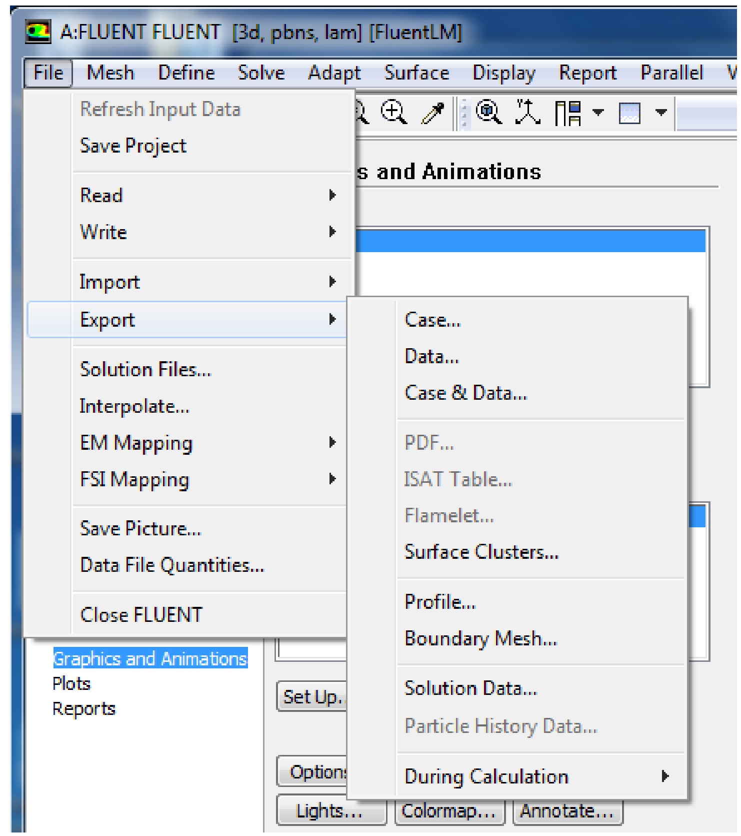
- **Check overall heat and mass balances.**
  - **Reports → Fluxes → Set Up**
  - **Select the Inlet and Outlet surfaces, then click Compute.**
    - **The net imbalance mass flux is shown under Net Results**
    - **Note that the net imbalance is very small relative to the boundary mass flow**
  - **Switch to Total Heat Transfer Rate. Select all walls, the inlet, and the outlet and click Compute.**
    - **Note that the difference equals the energy source input to the package (75W)**





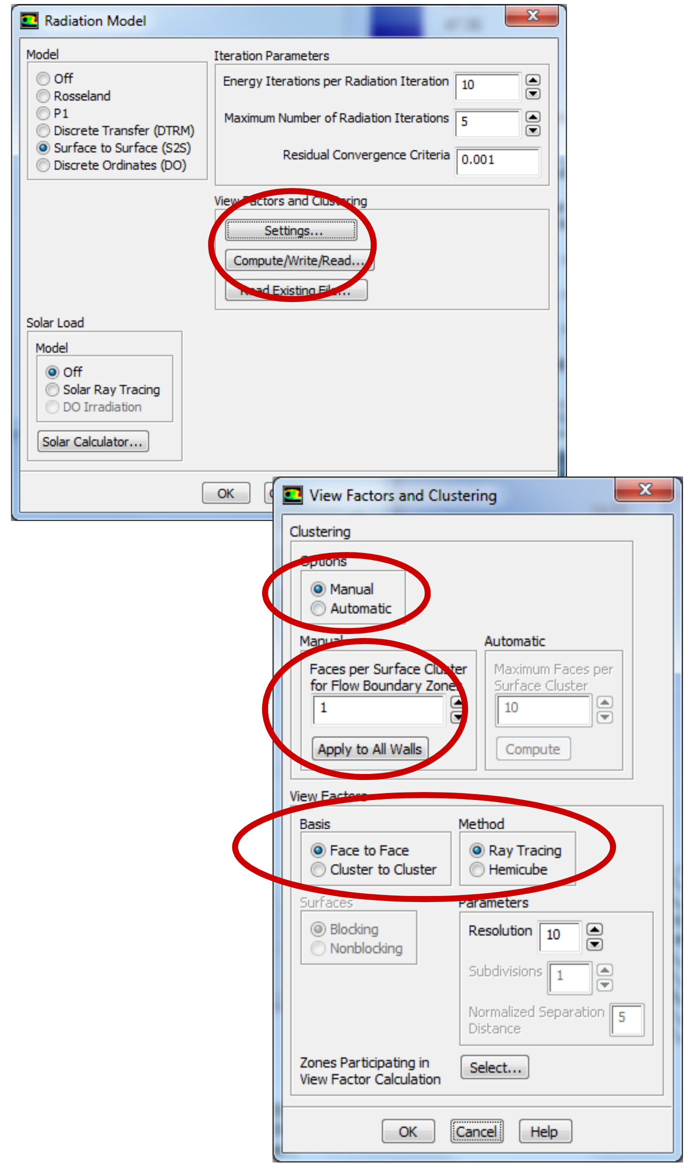
Save Case and Data Files for Later Use

- **The purpose of the Workbench structure is to simplify the file structure, and reduce the risk of stray files on the hard drive**
- **Part of this workshop is to allow the comparison of the results both with and without radiation active. To retain this set of results use the top menu to:**
  - **If using FLUENT standalone**
    - **File →Write → Case & Data…**
  - **If using FLUENT under Workbench**
    - **File →Export → Case & Data…**
  - **Change to the working directory and label the file logically**
- **TIP**
  - **Adding the .gz extension will compress the case and data files, reducing hard disk usage**
  - **You do not need to uncompress the files when opening them later**



Setup Radiation Model

- The temperature difference across the air space is notable, and therefore heat transfer via thermal radiation may be significant
- Go to the Models tree item and select radiation.
  - Enable the Surface-to-Surface (S2S) model
  - Click the Settings button to define model parameters
    - Manual Options
    - 1 Face Per Cluster and Apply to All Walls
    - Select Face to Face as the Basis and the Ray Tracing Method
    - OK
  - Either click on the Compute/Write/Read button
    - Enter a filename. Calculating and writing the view factors will take a few minutes
  - Or click on Read Existing File and load ws4_viewfactor.s2s.gz (recommended)
- Click OK when the S2S view factor step is complete



Comments on S2S Radiation model

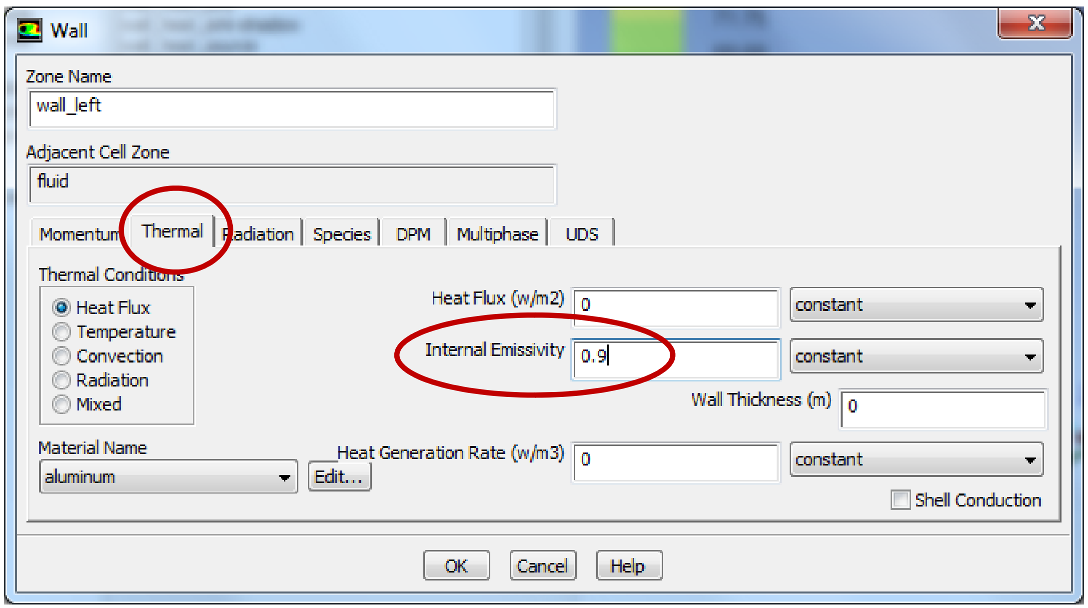
- **The model uses a ray tracing method, and this calculation is completed prior to the main solution. Thus, whilst the set up time is slightly longer due to the calculation of the view factors; the overall solution time is lower when compared to alternative radiation models available within ANSYS FLUENT**

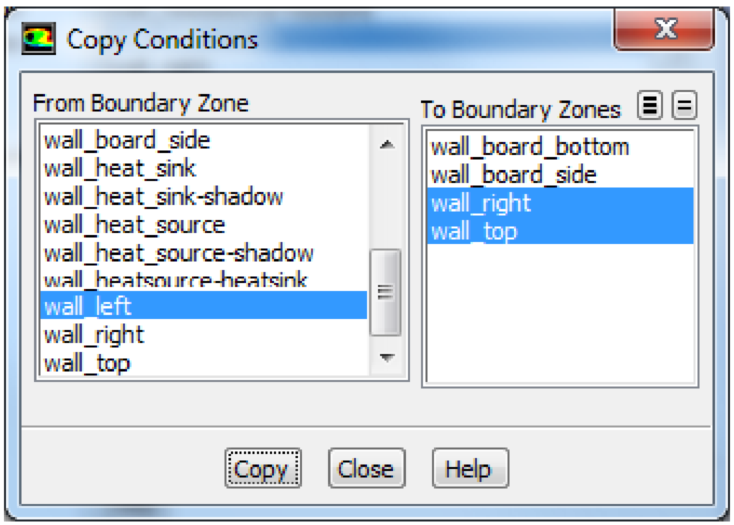
- **The method determines the view factor from each wall (or boundary) surface facet (mesh cell) to every other facet. Clustering is used to reduce the number of facet calculations needed in the actual solution. Neighbouring facets are grouped together based on the set values and other geometric factors**

- **The User documentation contains full details of the model**

Revise Boundary Conditions

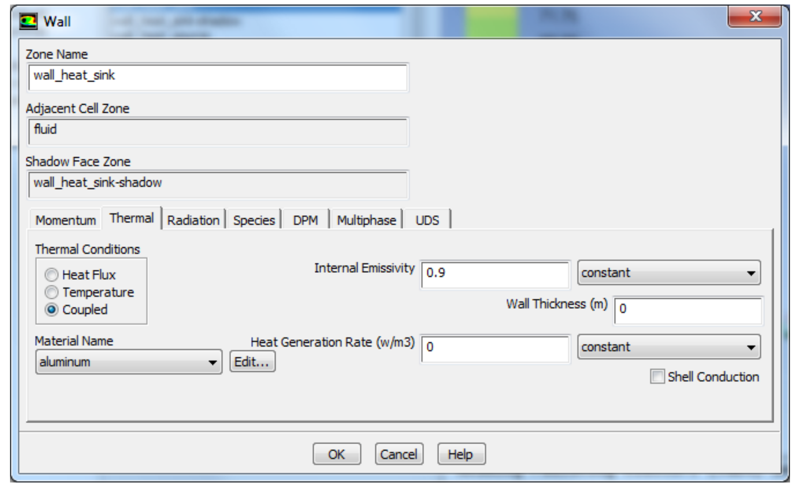
- The surfaces surrounding the fluid region now also require an emissivity value for the radiation model
- Open the wall_left boundary condition and under the thermal tab change the Internal Emissivity to 0.9
- Click the Copy button and copy the boundary conditions to wall_right and wall_top





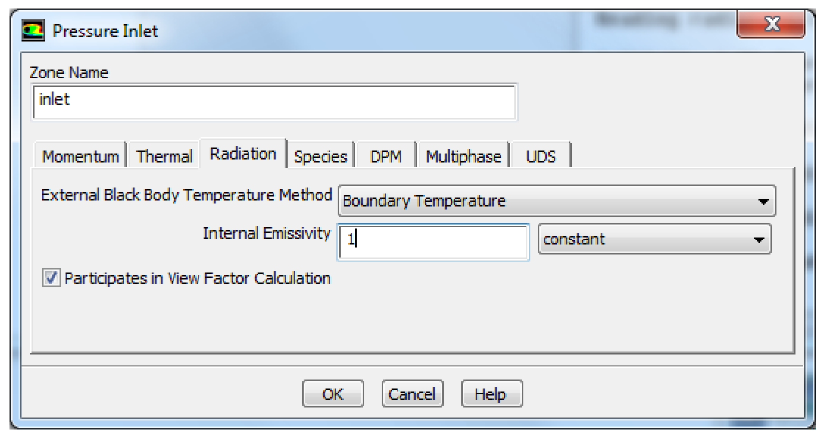
Revise Boundary Conditions

- **Change some of the other internally facing walls (wall_heat_sink and wall_board) to have an emissivity of 0.9**



- **Change wall_heat_source-shadow to have an emissivity of 0.3**
  - **Note these panels are slightly different to those in the previous slide as these are coupled walls – that is they do not make up the outer boundary of the domain**

- **The flow boundaries (inlet and outlet) can be left with an emissivity of 1**

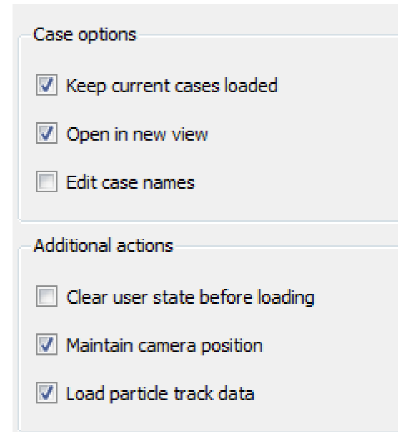


- **Solve the flow field again (this time with radiation enabled) for a further 500 iterations. If you are short of time you may want to stop the solution prematurely and proceed to the post-processing**

Post-processing using CFD Post

- **Post-processing can be done either in FLUENT or CFD Post**
  - **Both methods have their advantages**
  - **In this tutorial we are going to take advantage of CFD Post's ability to compare two models simultaneously**
- **Workbench**
  - **Choose a Results component from the Component Systems in Workbench**
  - **Connect the FLUENT solution panel to Results**
  - **Open the Results; this opens CFD Post with the result of the radiation calculation as labelled as FLU**
- **Standalone**
  - **Start CFD Post from the menu system/icon**
- **Fully converged cas & dat files are available within the Workshop 4 folder**
  - **If using the Workbench route, drag a Results icon onto the worksheet**
  - **Right click and edit the cell and follow the standalone notes**

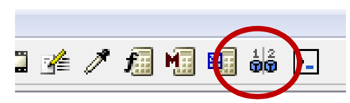
Case Comparison – Within Workbench



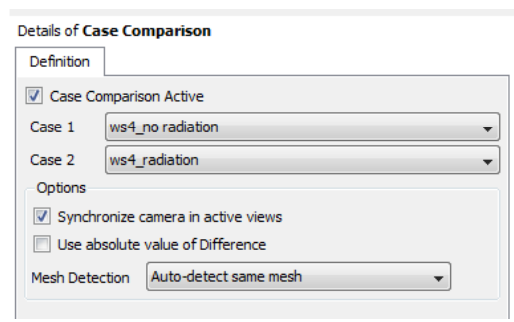
- **The case comparison tool allows two different setups to be shown side by side and any differences between the two cases identified**

- **We will compare the latest results (with radiation) with those saved earlier (no radiation)**

- **File → Load Results and select the non-radiation case saved earlier. Select Keep Current Cases Loaded, then Open**
- **From the toolbar, select the Case Comparison Icon:**



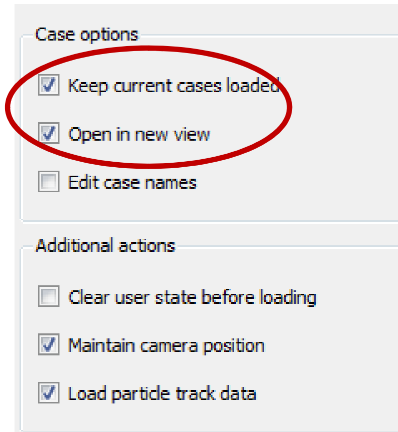
- **In the Case Comparison details**
  - **Check the box "Case Comparison Active" and check that:**
  - **Case 1 is the most recent results FLU (with radiation)**
  - **Case 2 is the file saved earlier (no radiation)**
  - **Apply.**



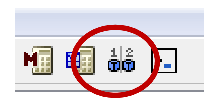
- **Press Apply to enter comparison mode**

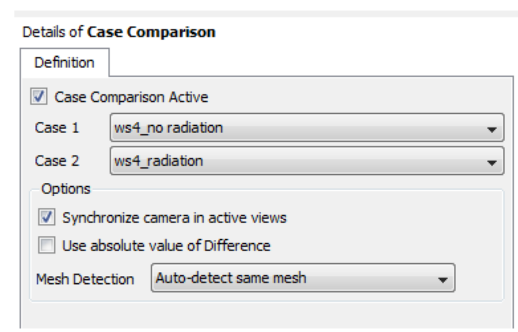
Case Comparison – Outside Workbench

- The case comparison tool allows two different setups to be shown side by side and any differences between the two cases identified
- We will compare the latest results (with radiation) with those saved earlier (no radiation)



- File → Load Results and select the non-radiation .dat file saved earlier
- Repeat the Load Results but read in the radiation results and "Keep current cases loaded"
- From the toolbar, select the Case Comparison Icon



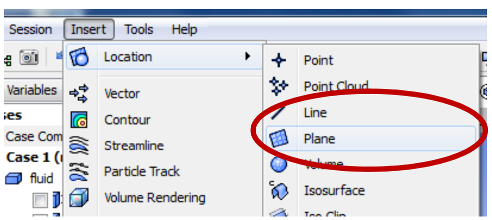


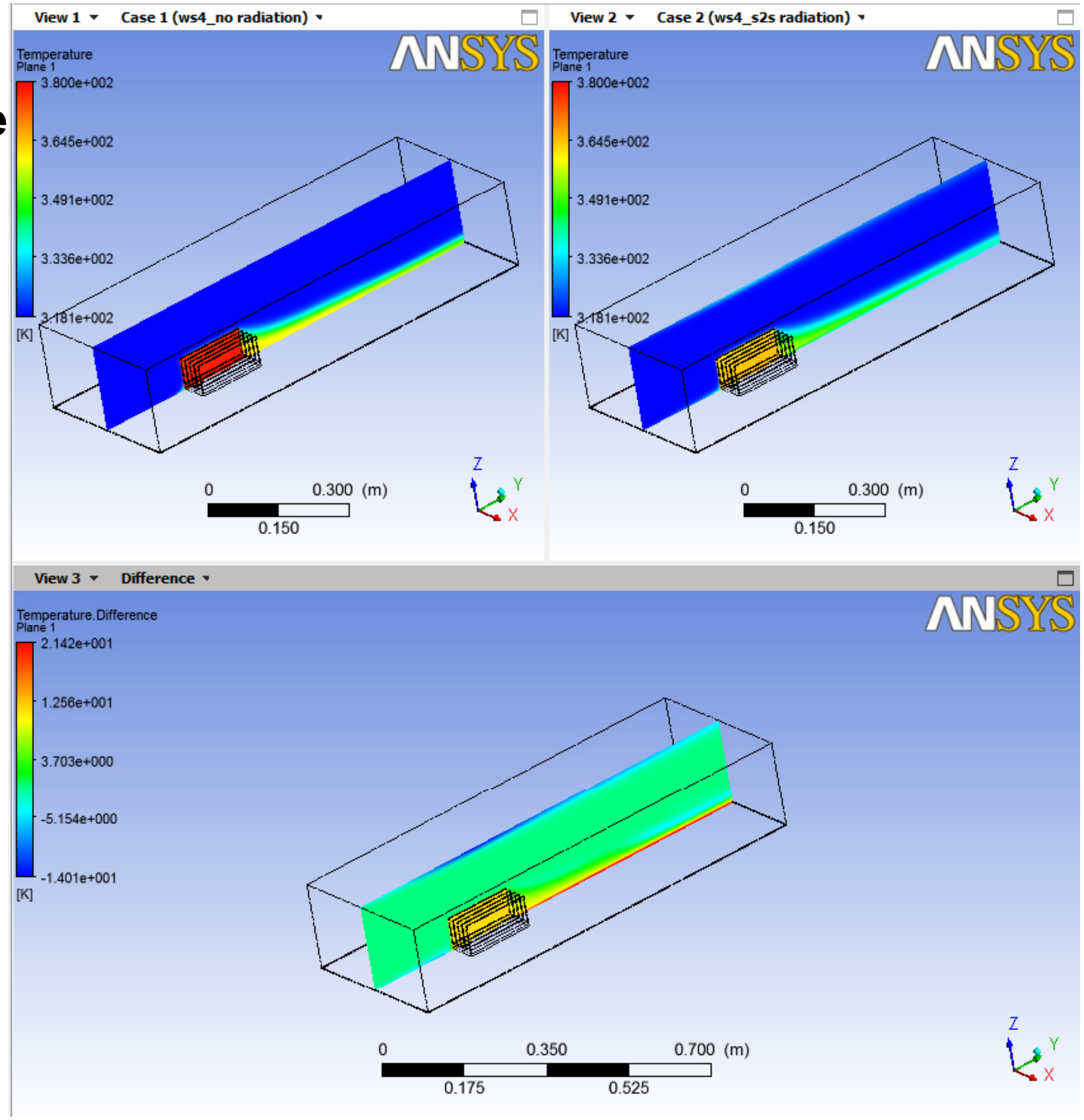
- Press Apply to enter comparison mode

Displaying Temperature

- **Temperature will be a key variable for any electronics cooling application so it will be displayed in several locations, such as within the flow, on the surfaces of the solid region and by extracting the maximum temperature within the component**

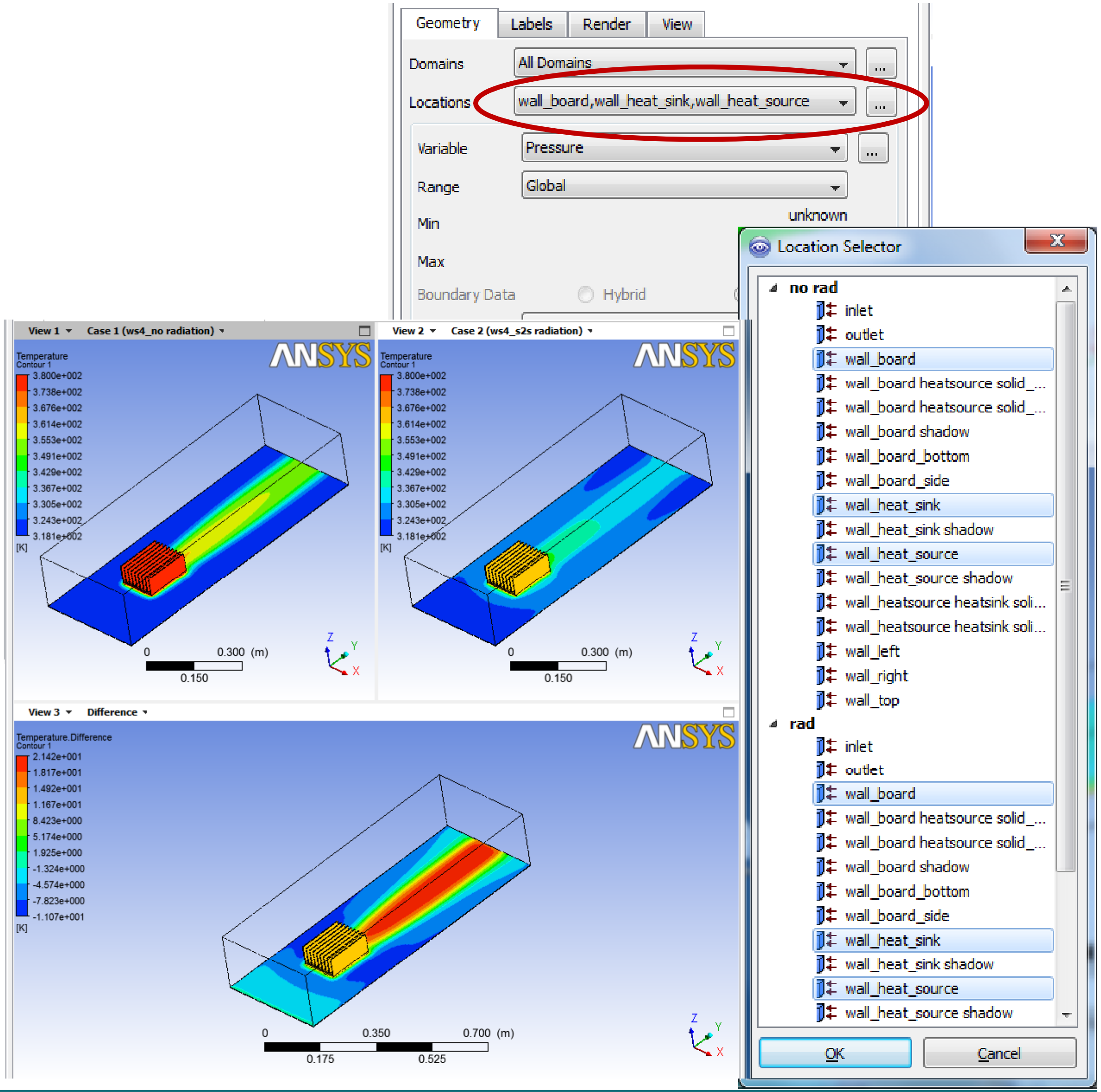
- **Create a YZ plane using Insert→Location→Plane:**
  - **Call it Centre**
  - **Located at X=0**
  - **Colour it using the variable Temperature.**





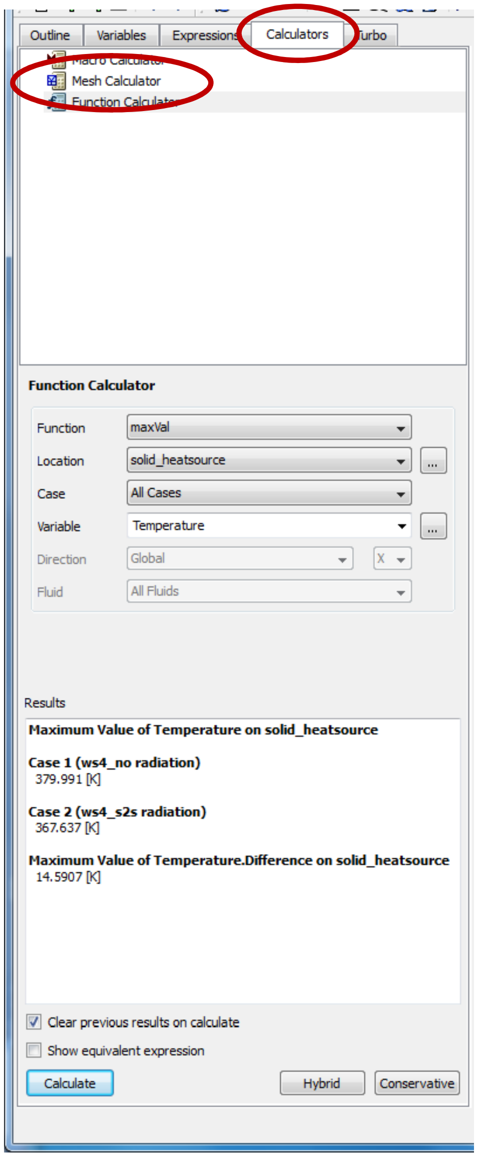
Displaying Temperature

- **Create a contour plot:**
  - **Use either Insert → Contour or click on**
  - **Use the fluid-solid interfaces as the location (wall_board , wall_heat_sink and wall_heat_source). Multiple locations can be selected by clicking ... and holding CTRL (select surfaces from both sets)**
  - **Set the variable to Temperature using the Global Range, and Apply**



Reporting Temperature

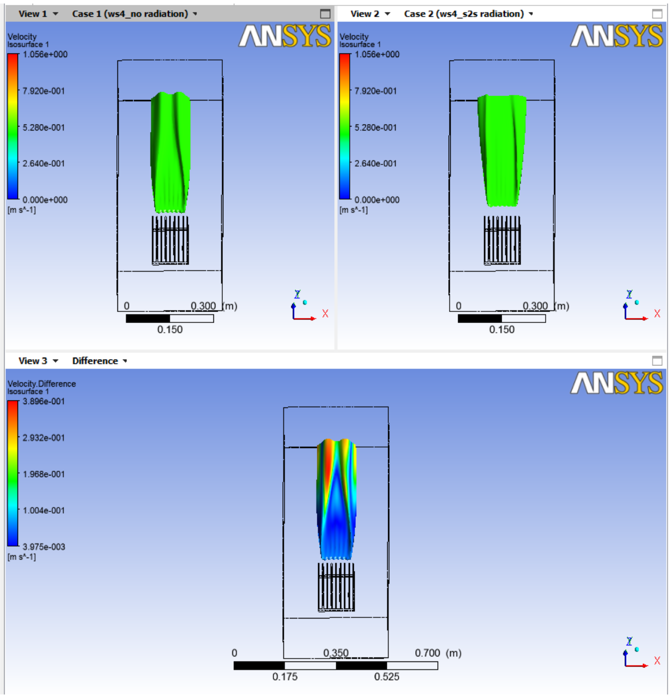
- Use the the function calculator using the icon on the toolbar.
  - Set the options to *maxVal*, *solid_heatsource*, *All Cases*, *Temperature*
- Note that with radiation, the temperature in the solid is lower than when radiation was not included
- The cooling of the component is mirrored with an increase in the temperature of the walls around the fluid zone. This can be seen if you plot the temperature on the walls or use the Function Calculator with the areaAve function



Flow Displays

- **To show the flow patterns a range of methods can be used including streamlines, vector plots and isosurfaces**

- **Insert an isosurface using Insert→ Location→ Isosurface and set the Variable to Velocity with a value of 0.5m/s. Gradually reduce the value to 0.2 and notice that for the radiation case higher speed flow can be observed close to the fluid walls as well as the PCB**



- **Insert a vector plot using Insert-Vector or click on . Set the location to Centre. Change the Sampling to Equally spaced with 1000 points. If you wish to see the pattern in the slow speed sections try going to the Symbol tab and select Normalize Symbols**

Summary

- **In this Workshop you have:**
  - **Seen how FLUENT can be used within and outside of the Workbench environment**
  - **Modelled a natural convection flow with conjugate heat transfer and source terms**
  - **Modified/created and used multiple solid materials**
  - **Used the S2S radiation model**
  - **Post processed two results simultaneously and compared the differences**