

Introduction

This tutorial demonstrates the postprocessing capabilities of **ANSYS FLUENT** using a 3D model of a flat circuit board with a heat generating electronic chip mounted on it. The flow over the chip is laminar and involves conjugate heat transfer.

The heat transfer involves conduction in the chip and conduction and convection in the surrounding fluid. The physics of conjugate heat transfer such as this, is common in many engineering applications, including the design and cooling of electronic components.

In this tutorial, you will read the case and data files (without doing the calculation) and perform a number of postprocessing exercises.

This tutorial demonstrates how to do the following:

- Add lights to the display at multiple locations.
- Create surfaces for the display of 3D data.
- Display filled contours of temperature on several surfaces.
- Display velocity vectors.
- Mirror a display about a symmetry plane.
- Create animations.
- Display results on successive slices of the domain.
- Display pathlines.
- Plot quantitative results.
- Overlay and “explode” a display.
- Annotate the display.

Prerequisites

This tutorial is written with the assumption that you have completed [Tutorial 1](#), and that you are familiar with the **ANSYS FLUENT** navigation pane and menu structure.

Problem Description

The problem considered is shown schematically in Figure 24.1. The configuration consists of a series of side-by-side electronic chips, or modules, mounted on a circuit board. Air flow, confined between the circuit board and an upper wall, cools the modules. To take advantage of the symmetry present in the problem, the model will extend from the middle of one module to the plane of symmetry between it and the next module.

As shown in the figure, each half-module is assumed to generate 2.0 Watts and to have a bulk conductivity of $1.0 \text{ W/m}^2\text{-K}$. The circuit board conductivity is assumed to be one order of magnitude lower: $0.1 \text{ W/m}^2\text{-K}$. The air flow enters the system at 298 K with a velocity of 1 m/s. The Reynolds number of the flow, based on the module height, is about 600. The flow is therefore treated as laminar.

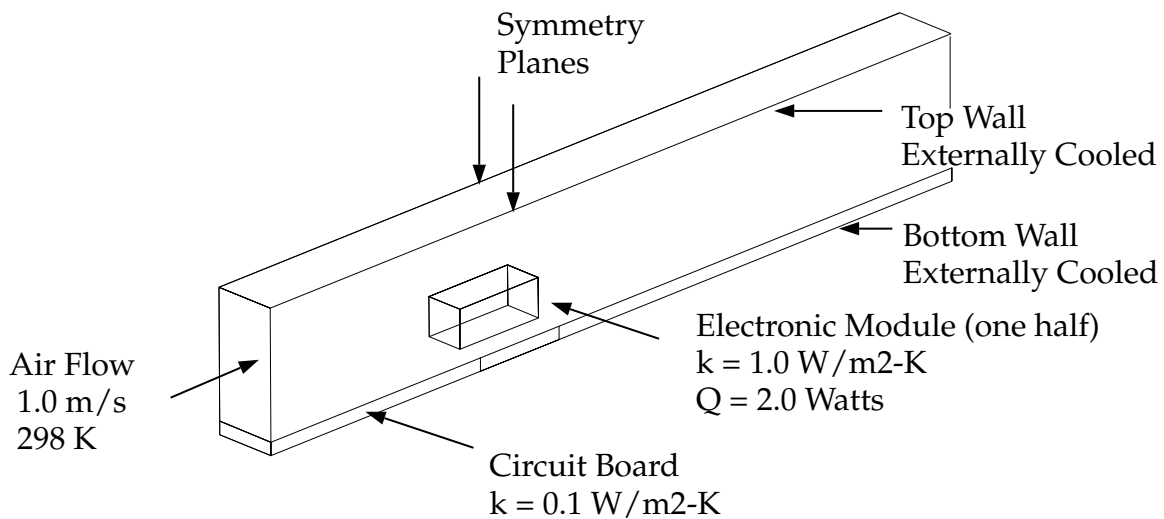


Figure 24.1: Problem Specification

Setup and Solution

Preparation

1. Download `postprocess.zip` from the [User Services Center](#) to your working folder (as described in Tutorial 1).
2. Unzip `postprocess.zip`.

The files `chip.cas.gz` and `chip.dat.gz` can be found in the `postprocess` folder created after unzipping the file.

3. Use FLUENT Launcher to start the 3D version of ANSYS FLUENT.

For more information about FLUENT Launcher, see Section 1.1.2 in the separate [User's Guide](#).

Note: *The Display Options are enabled by default. Therefore, after you read in the case and data files, the mesh will be displayed in the embedded graphics window.*

Step 1: Mesh

1. Read in the case and data files `chip.cas.gz` and `chip.dat.gz`.

File → **Read** → Case & Data...

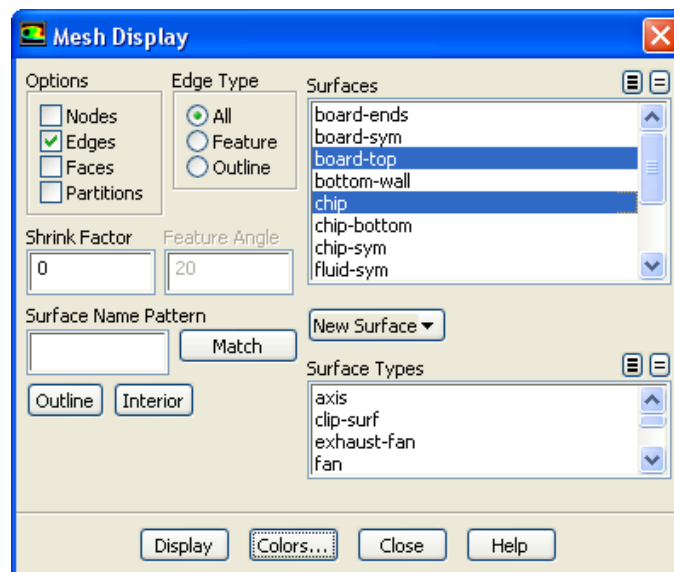
When you select the case file, ANSYS FLUENT will read the data file automatically.

Step 2: General Settings

◆ General

1. Display the mesh surfaces `board-top` and `chip`.

◆ **General** → **Display...**



- (a) Retain the default selection of **Edges** in the **Options** group box.
- (b) Deselect all surfaces and select `board-top` and `chip` from the **Surfaces** selection list.

*To deselect all surfaces click on the far-right unshaded button at the top of the **Surfaces** selection list, and then select the desired surfaces from the **Surfaces** selection list.*

- (c) Click the **Colors...** button to open the **Mesh Colors** dialog box.
 - i. Select **Color by ID** in the **Options** group box.

- ii. Click **Reset Colors** and close the **Mesh Colors** dialog box.
 - (d) Click **Display**.
2. Rotate and zoom the view.

Use the left mouse button to rotate the view. Use the middle mouse button to zoom the view until you obtain an enlarged display of the circuit board in the region of the chip, as shown in Figure 24.2.

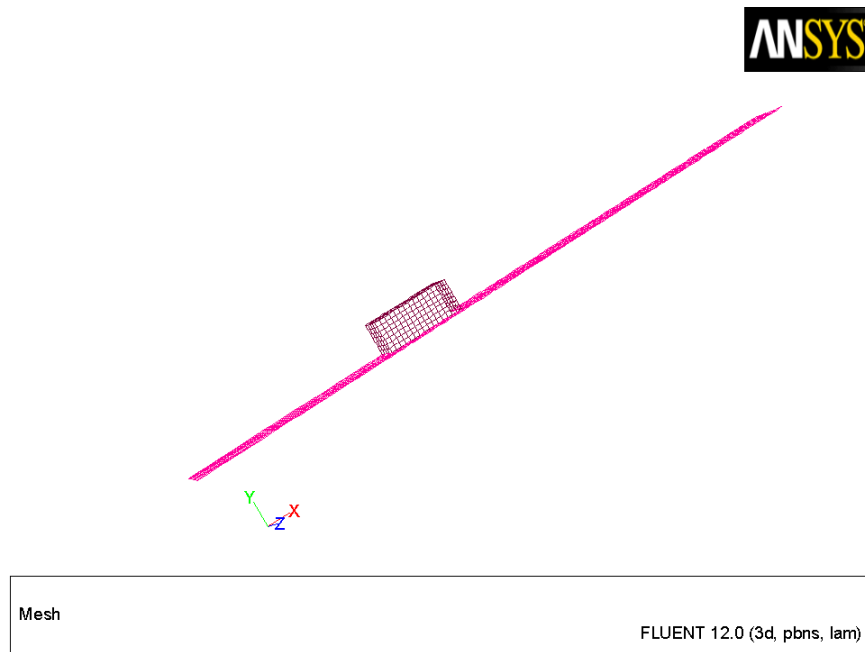


Figure 24.2: Mesh Display of the Chip and Board Surfaces

Extra: *You can click the right mouse button on one of the boundaries displayed in the graphics window and its zone number, name, and type will be printed in the console. This feature is especially useful when you have several zones of the same type and you want to distinguish between them.*

3. Display the mesh faces.



- (a) Disable **Edges** and enable **Faces** in the **Options** group box.
- (b) Click **Display** and close the **Mesh Display** dialog box.

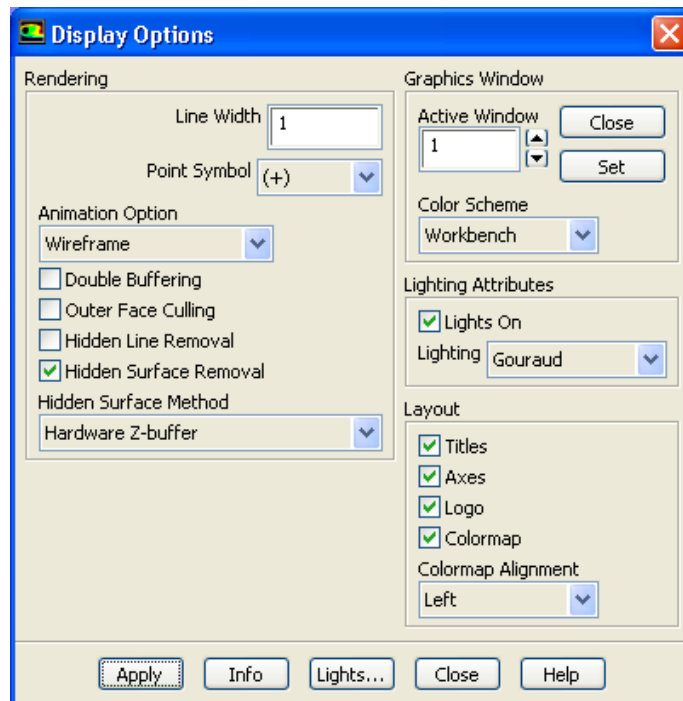
The surfaces run together with no shading to separate the chip from the board.

Step 3: Adding Lights

1. Add lighting effects.

✦ **Graphics and Animations** → Options...

The default light settings add a white light at the position (1, 1, 1). The default light is defined in the Lights dialog box by the Light ID 0 with Direction vectors (X, Y, Z) as (1, 1, 1).



- (a) Enable Lights On in the Lighting Attributes group box.
- (b) Select Gouraud from the Lighting drop-down list.

Flat is the most basic lighting whereas Gouraud gives better color gradiation.

- (c) Click Apply and close the Display Options dialog box.

Shading will be added to the surface mesh display (Figure 24.3).

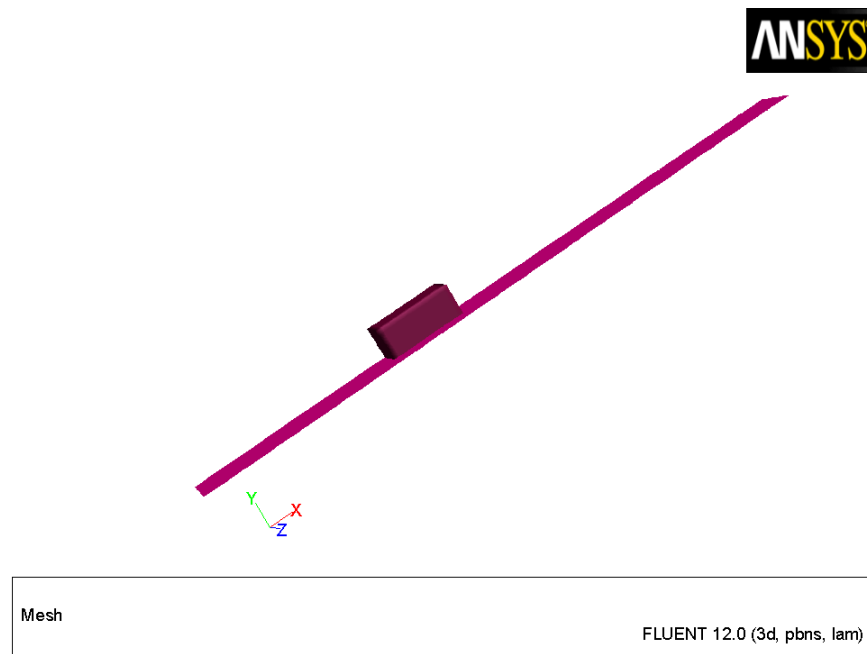
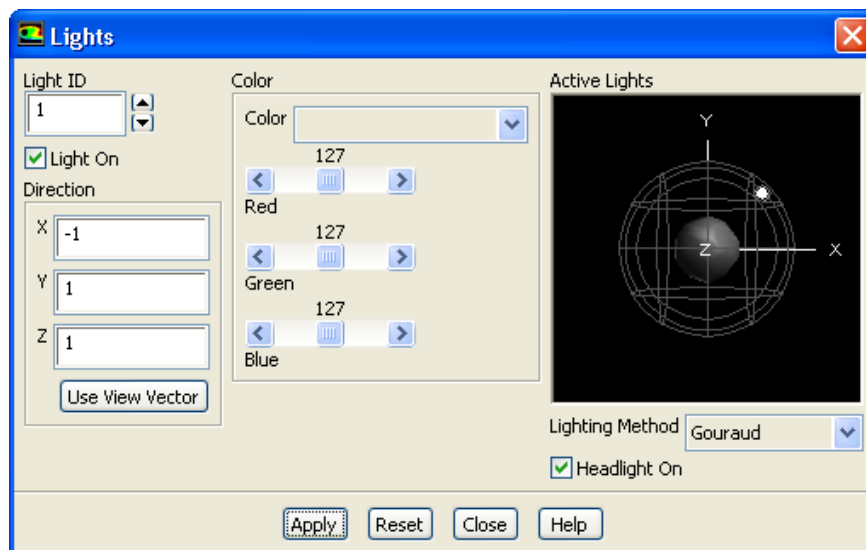


Figure 24.3: Graphics Display with Default Lighting

2. Add lights in two directions, $(-1, 1, 1)$ and $(-1, 1, -1)$.

◆ **Graphics and Animations** → **Lights...**



You can also open the Lights dialog box by clicking the Lights... button in the Display Options dialog box.

- (a) Set Light ID to 1.

- (b) Enable **Light On**.
- (c) Enter -1, 1, and 1 for X, Y, and Z respectively in the **Direction** group box.
- (d) Retain the selection of **Gouraud** in the **Lighting Method** drop-down list.
- (e) Enable **Headlight On**.

*The **Headlight On** option provides constant lighting effect from a light source directly in front of the model, in the direction of the view. You can turn off the headlight by disabling the **Headlight On** option (Figure 24.4).*

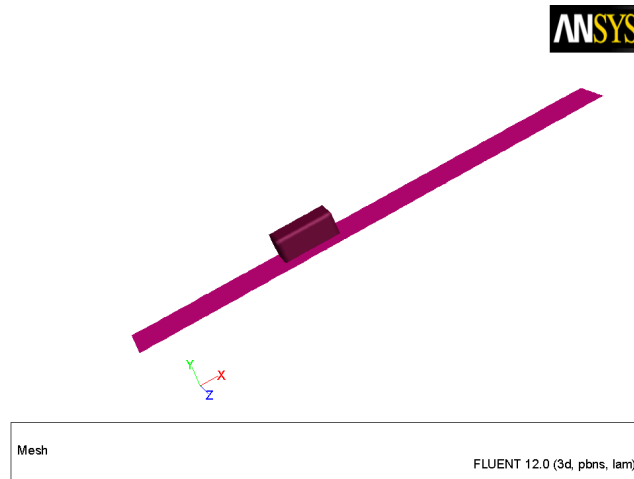


Figure 24.4: Display with Additional Lighting: - Headlight Off

- (f) Click **Apply**.
- (g) Similarly, add a second light (Light ID=2) at (-1, 1, -1).

The result will be more softly shaded display (Figure 24.5).

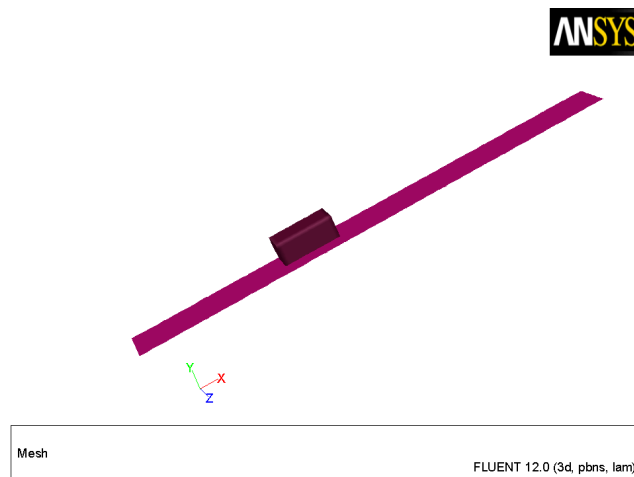


Figure 24.5: Display with Additional Lighting

- (h) Close the Lights dialog box.

Extra: You can use the left mouse button to rotate the ball in the Active Lights window to gain a perspective view on the relative locations of the lights that are currently active, and see the shading effect on the ball at the center.

You can also change the color of one or more of the lights by selecting the color from the Color drop-down list or by moving the Red, Green, and Blue sliders.

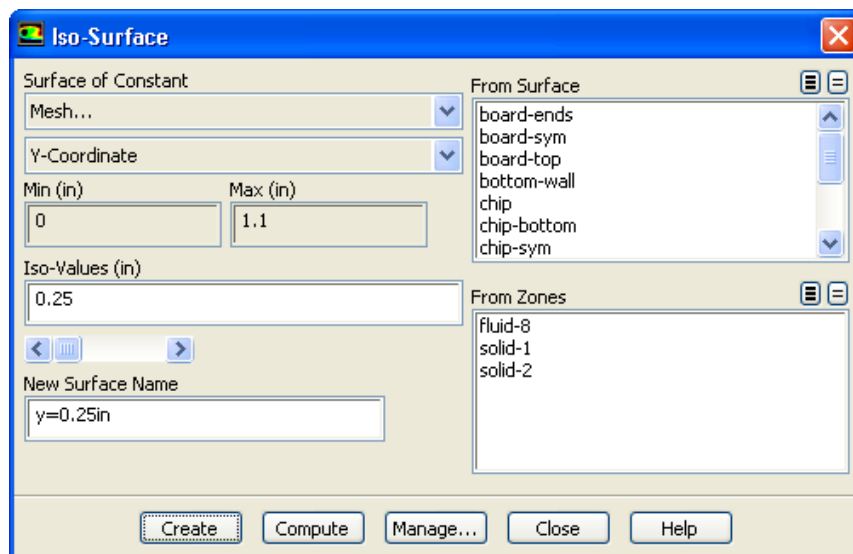
Step 4: Creating Isosurfaces

To display results in a 3D model, you will need surfaces on which the data can be displayed. ANSYS FLUENT creates surfaces for all boundary zones automatically. Several surfaces have been renamed after reading the case file. Examples are board-sym and board-ends, which correspond to the side and end faces of the circuit board.

You can define additional surfaces for viewing the results, for example, a plane in Cartesian space. In this exercise, you will create a horizontal plane cutting through the middle of the module with a y value of 0.25 inches. You can use this surface to display the temperature and velocity fields.

1. Create a surface of constant y coordinate.

Surface → Iso-Surface...

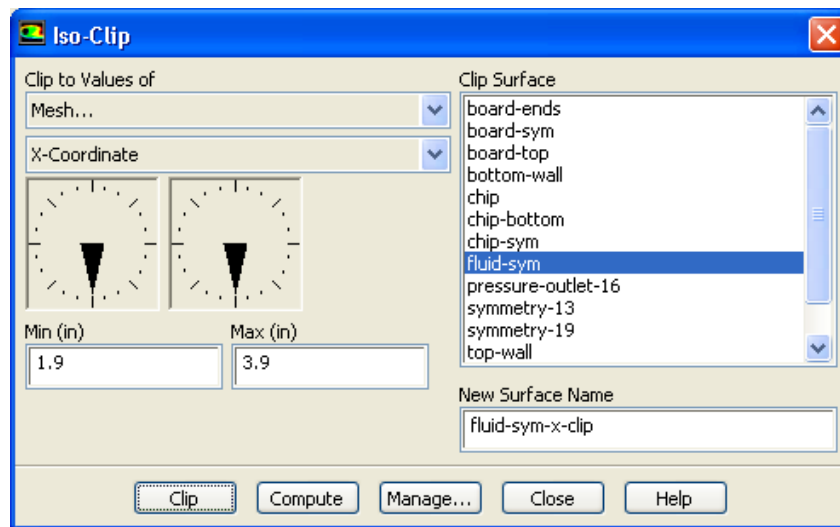


- (a) Select Mesh... and Y-Coordinate from the Surface of Constant drop-down lists.
- (b) Click Compute.

The Min and Max fields will display the y extents of the domain.

- (c) Enter 0.25 for Iso-Values.
 - (d) Enter $y=0.25\text{in}$ for New Surface Name.
 - (e) Click **Create** and close the Iso-Surface dialog box.
2. Create a clipped surface for the X-coordinate of the fluid (fluid-sym).

Surface → Iso-Clip...



- (a) Select **Mesh...** and **X-Coordinate** from the **Clip to Values of** drop-down lists.
- (b) Select **fluid-sym** from the **Clip Surface** selection list.
- (c) Click **Compute**.

The values will be displayed in the Min and Max fields.

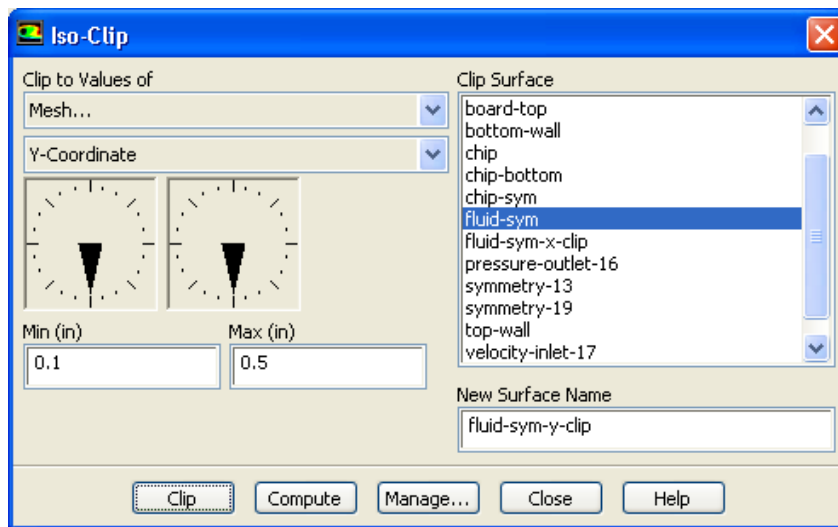
- (d) Enter 1.9 and 3.9 for Min and Max respectively.

Note: *This will isolate the area around the chip.*

- (e) Enter **fluid-sym-x-clip** for New Surface Name.
- (f) Click **Clip**.

3. Create a clipped surface for the Y-coordinate of the fluid (fluid-sym).

Surface → Iso-Clip...



- (a) Select Mesh... and Y-Coordinate from the Clip to Values of drop-down lists.
- (b) Retain the selection of fluid-sym from the Clip Surface selection list.
- (c) Click Compute.

The values will be displayed in the Min and Max fields.

- (d) Enter 0.1 and 0.5 for Min and Max respectively.

Note: *This will isolate the area around the chip.*

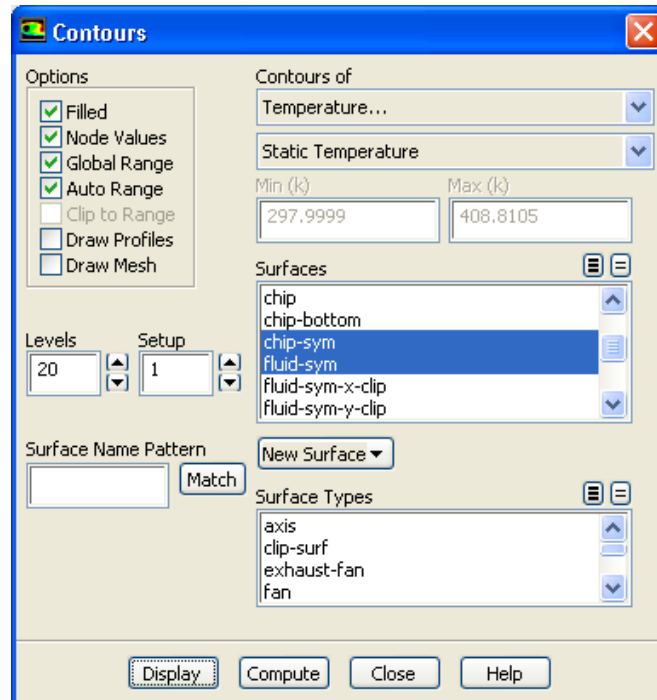
- (e) Enter fluid-sym-y-clip for New Surface Name.
- (f) Click Clip and close the Iso-Clip dialog box.

Step 5: Contours

❖ Graphics and Animations

1. Display filled contours of temperature on the symmetry plane (Figure 24.6).

❖ Graphics and Animations →  Contours → Set Up...



- (a) Retain the default settings in the Options group box.
- (b) Select Temperature... and Static Temperature from the Contours of drop-down lists.
- (c) Select board-sym, chip-sym, and fluid-sym from the Surfaces selection list.
- (d) Click Display.
- (e) Rotate and zoom the display using the left and middle mouse buttons, respectively, to obtain the view as shown in Figure 24.6.

Hint: If the display disappears from the screen at any time, or if you are having difficulty manipulating it with the mouse, you can open the Views dialog box by clicking the Views... button in Graphics and Animations task page and use the Default button to reset the view. Alternatively, you can revert to a previous graphics display using the keyboard shortcut <Ctrl>-L.

The peak temperatures in the chip appear where the heat is generated, along with the higher temperatures in the wake where the flow is recirculating.

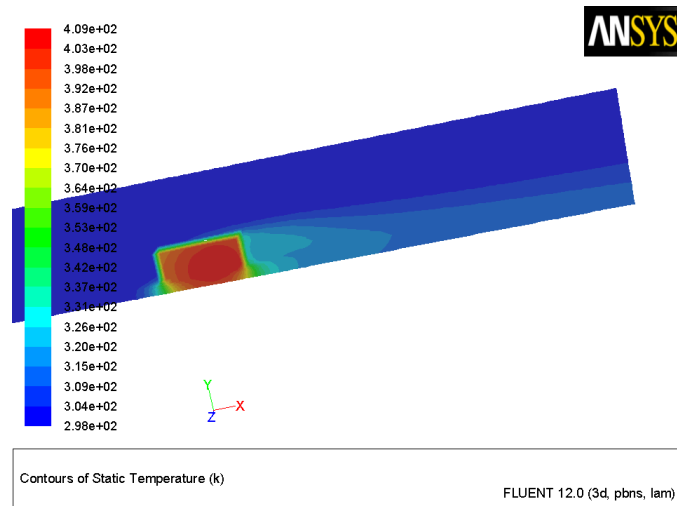


Figure 24.6: Filled Contours of Temperature on the Symmetry Surfaces

2. Display filled contours of temperature for the clipped surface (Figure 24.7).

◆ **Graphics and Animations** → **Contours** → **Set Up...**

- Deselect all surfaces from the **Surfaces** selection list and then select **fluid-sym-x-clip** and **fluid-sym-y-clip**.
- Click **Display**.
- Orient the view to obtain the display as shown in Figure 24.7.

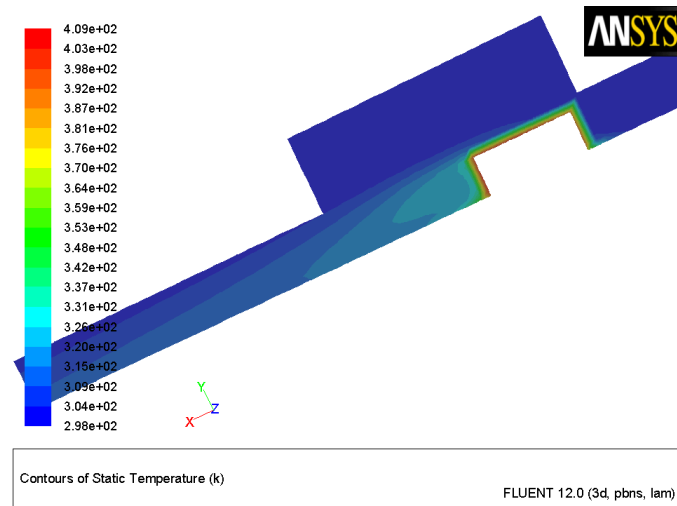


Figure 24.7: Filled Contours of Temperature on the Clipped Surface

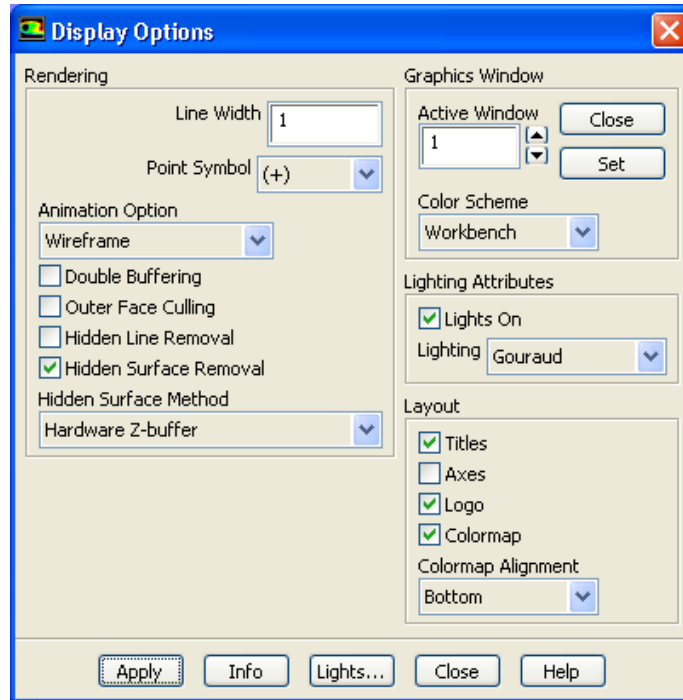
3. Display filled contours of temperature on the plane, $y=0.25\text{in}$ (Figure 24.8).

✦ **Graphics and Animations** → **Contours** → **Set Up...**

- (a) Deselect all surfaces from the **Surfaces** selection list and then select $y=0.25\text{in}$.
- (b) Click **Display** and close the **Contours** dialog box.

4. Change the location of the colormap in the graphics display window.

✦ **Graphics and Animations** → **Options...**



- (a) Disable **Axes** in the **Layout** group box.
- (b) Select **Bottom** from the **Colormap Alignment** drop-down list.
- (c) Click **Apply** and close the **Display Options** dialog box.
- (d) Zoom the display using the middle mouse button to obtain the view as shown in Figure 24.8.

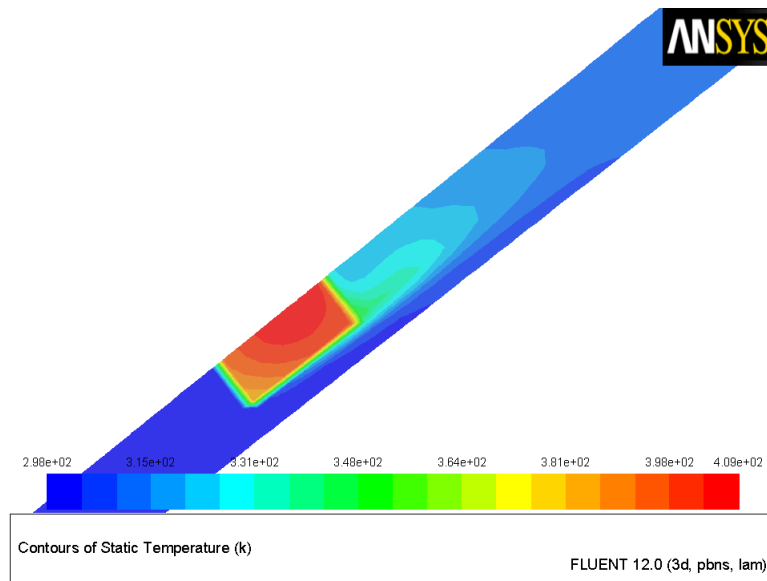
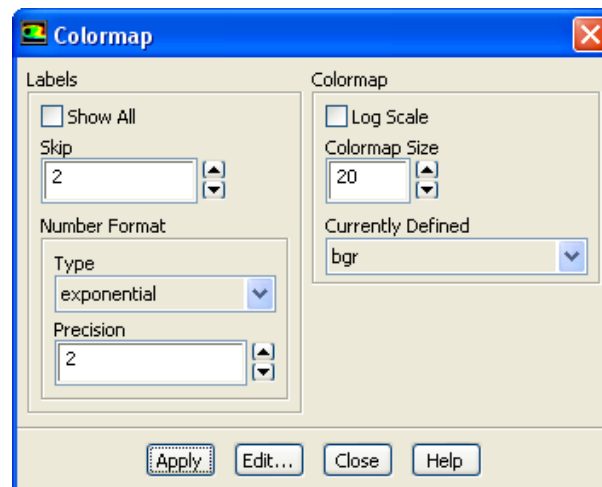


Figure 24.8: Temperature Contours on the Surface, $y = 0.25$ in.

*In Figure 24.8, the high temperatures in the wake of the module are clearly visible. You can also display other quantities such as velocity magnitude or pressure using the **Contours** dialog box.*

5. Change the display of the colormap labels.

❖ **Graphics and Animations** → Colormap...



- (a) Ensure that **Show All** is disabled.
- (b) Set **Skip** to 2.

It can also be observed that the contour labels are crowding the bottom of the screen, where the colormap is displayed. You can control the number of labels displayed on colormaps by using the skip-label function.

- (c) Click **Apply** and close the **Colormap** dialog box.

The display updates immediately with every other colormap label appearing, as shown in Figure 24.9.

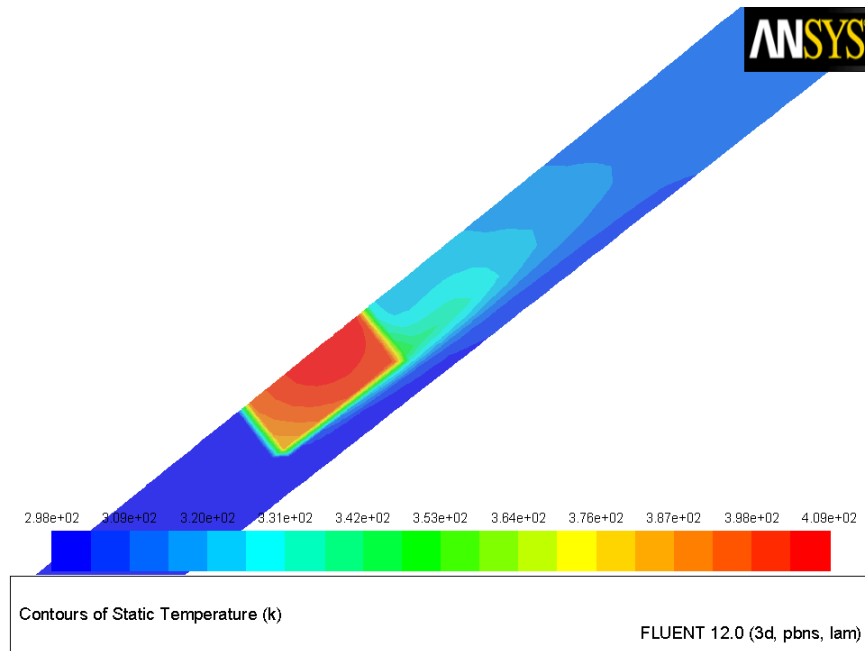


Figure 24.9: Filled Contours of Temperature on the Symmetry Surface for Skip = 2

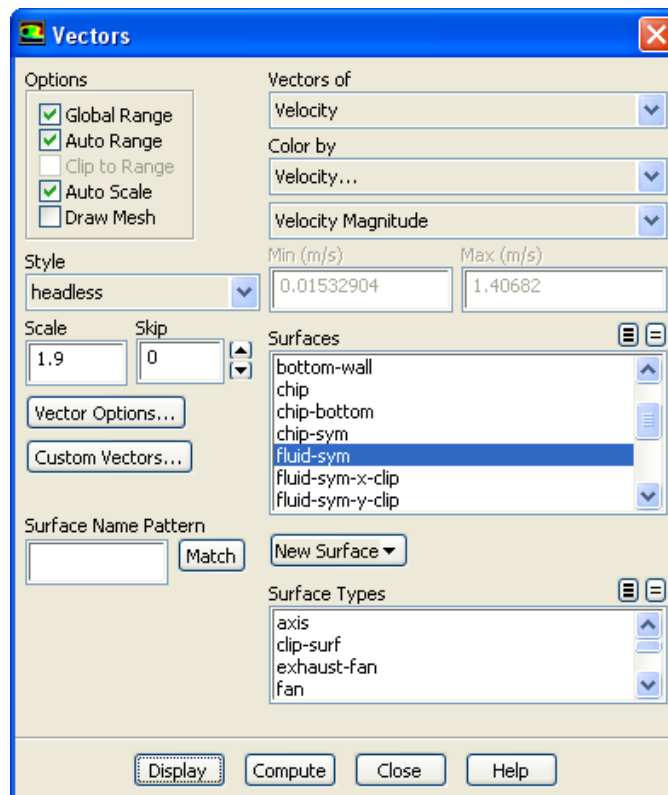
Step 6: Velocity Vectors

❖ Graphics and Animations

Velocity vectors provide an excellent visualization of the flow around the module, depicting details of the wake structure.

1. Display velocity vectors on the symmetry plane through the module centerline (Figure 24.10).

❖ Graphics and Animations →  Vectors → Set Up...



- (a) Enter 1.9 for Scale.
- (b) Deselect all surfaces from the **Surfaces** selection list and then select **fluid-sym**.
- (c) Click **Display**.

Extra: *You can display velocity vectors for the clipped surfaces.*

❖ Graphics and Animations →  Vectors → Set Up...

- (a) Deselect all surfaces from the **Surfaces** selection list and then select **fluid-sym-x-clip** and **fluid-sym-y-clip**.
- (b) Click **Display**.

2. Change the colormap layout.



- Enable Axes in the **Layout** group box.
- Select **Left** from the **Colormap Alignment** drop-down list.
- Click **Apply** and close the **Display Options** dialog box.
- Rotate and zoom the display to observe the vortex near the stagnation point and in the wake of the module (Figure 24.10).

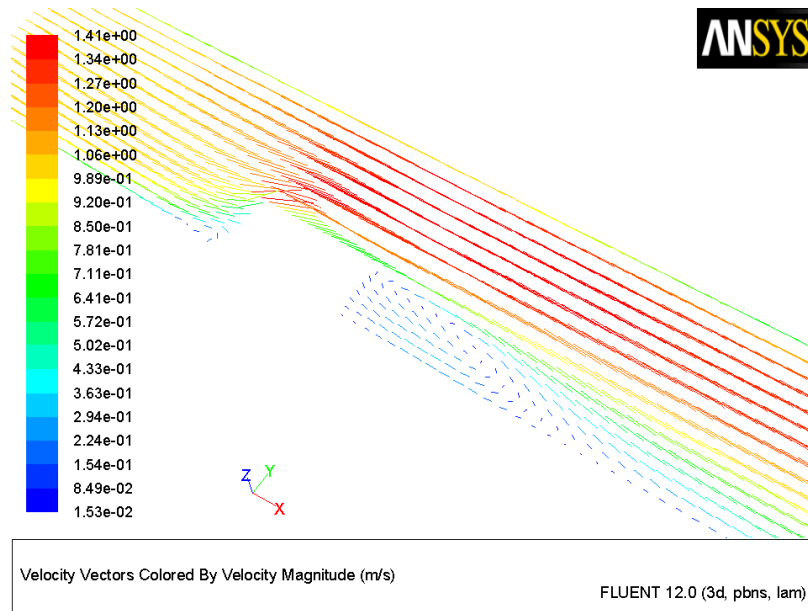


Figure 24.10: Velocity Vectors in the Module Symmetry Plane

Note: The vectors in Figure 24.10 are shown without arrowheads. You can modify the arrow style in the **Vectors** dialog box by selecting a different option from the **Style** drop-down list.

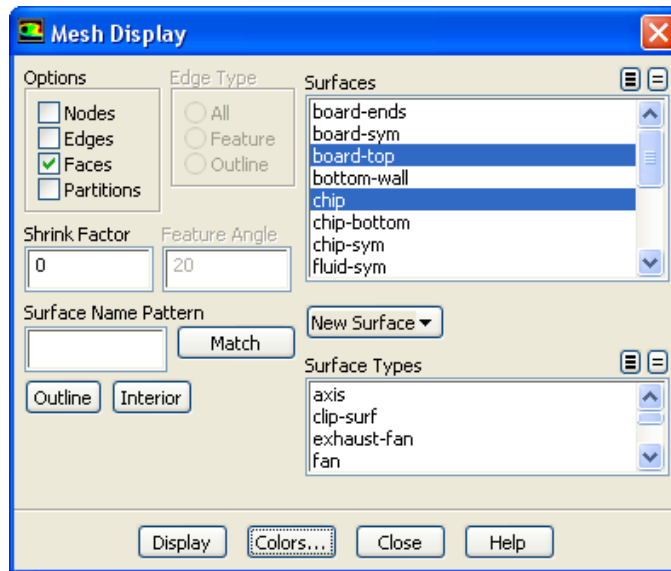
Extra: If you want to decrease the number of vectors displayed, then increase the **Skip factor** to a non-zero value.

3. Plot velocity vectors in the horizontal plane intersecting the module (Figure 24.11).

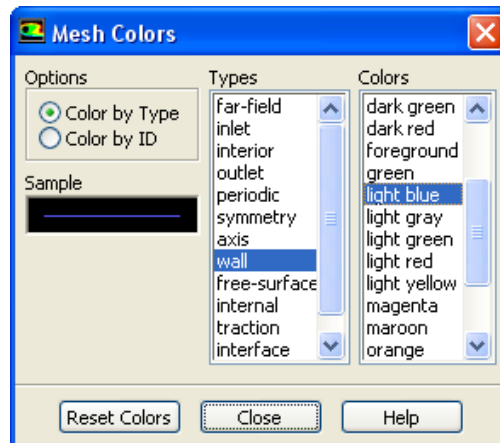


After plotting the vectors, you will enhance the view by mirroring the display about the module centerline and displaying the module surfaces.

- (a) Enable Draw Mesh in the Options group box to open the Mesh Display dialog box.



- i. Ensure that Faces is enabled in the Options group box.
- ii. Retain the selection of board-top and chip from the Surfaces selection list.
- iii. Click the Colors... button to open the Mesh Colors dialog box.



- A. Select Color by Type in the Options group box.
 - B. Select wall from the Types selection list.
 - C. Select light blue from the Colors selection list and close the Mesh Colors dialog box.
 - iv. Click Display and close the Mesh Display dialog box.
- (b) Enter 3.8 for Scale.
- (c) Deselect all surfaces by clicking the unshaded icon to the right of the Surfaces selection list.

- (d) Select $y=0.25\text{in}$ from the Surfaces selection list.
- (e) Click Display and close the Vectors dialog box.
- (f) Rotate the display with the mouse to obtain the view as shown in Figure 24.11.

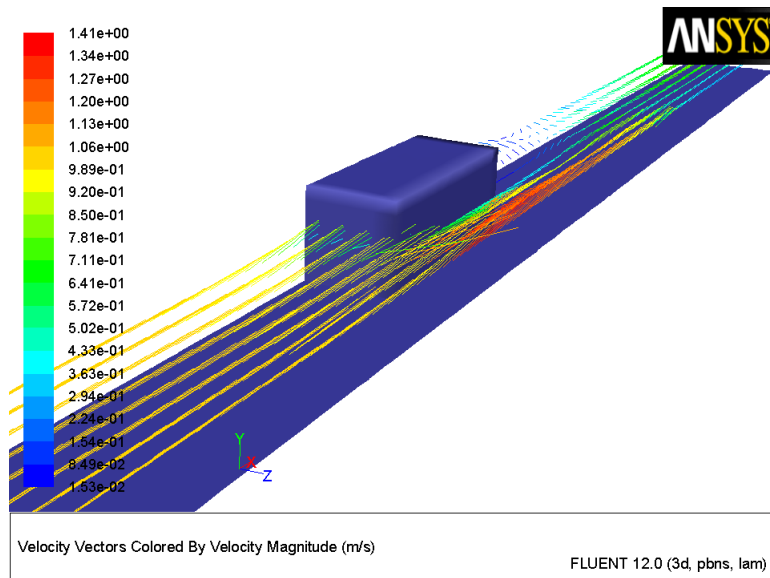
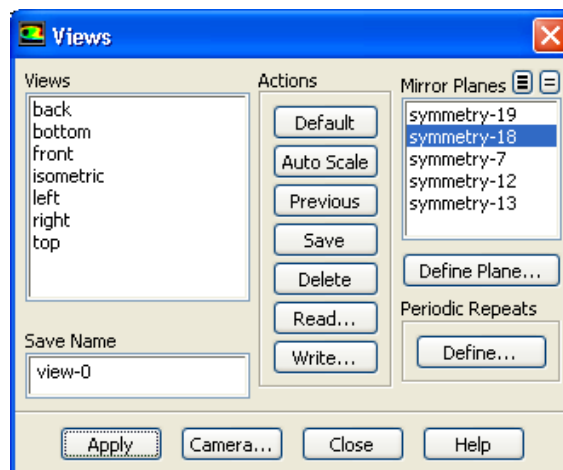


Figure 24.11: Velocity Vectors Intersecting the Surface

4. Mirror the view about the chip symmetry plane (Figure 24.12).

✦ **Graphics and Animations** → **Views...**



- (a) Select symmetry-18 from the Mirror Planes selection list.

Note: This zone is the centerline plane of the module and its selection will create a mirror of the entire display about the centerline plane.

(b) Click **Apply** and close the Views dialog box.

The display will be updated in the graphics window (Figure 24.12).

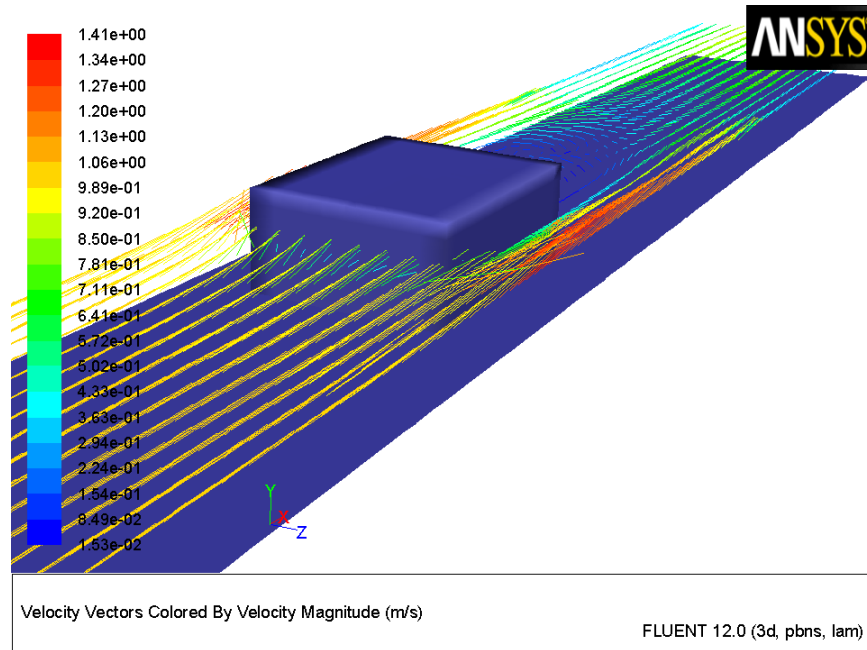


Figure 24.12: Velocity Vectors After Mirroring

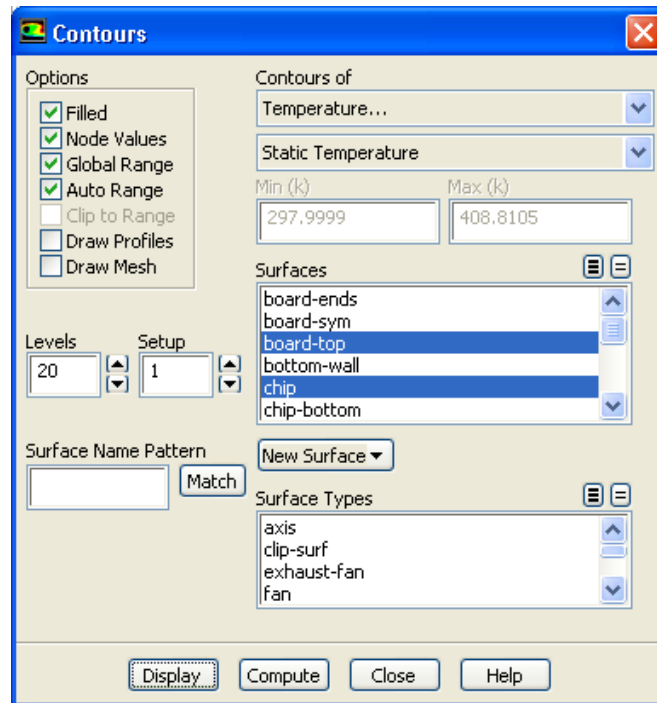
Step 7: Animation

Graphics and Animations

Using ANSYS FLUENT, you can animate the solution and also a scene. For information on animating the solution, see Tutorial 13, Step 10. In this tutorial, you will animate a scene between two static views of the graphics display.

You will display the surface temperature distribution on the module and the circuit board by selecting the corresponding boundaries. You will also create the key frames and view the transition between the key frames, dynamically, using the animation feature.

1. Display filled contours of surface temperature on the board-top and chip surfaces. (Figure 24.13).



- (a) Ensure that **Filled** is enabled in the **Options** group box.
- (b) Retain the default selection of **Temperature...** and **Static Temperature** from the **Contours of** drop-down lists.
- (c) Deselect all surfaces by clicking the unshaded icon to the right of **Surfaces**.
- (d) Select **board-top** and **chip** from the **Surfaces** selection list.
- (e) Click **Display** and close the **Contours** dialog box.
- (f) Zoom the display as needed to obtain the view shown in Figure 24.13.

Figure 24.13 shows the high temperatures on the downstream portions of the module and relatively localized heating of the circuit board around the module.

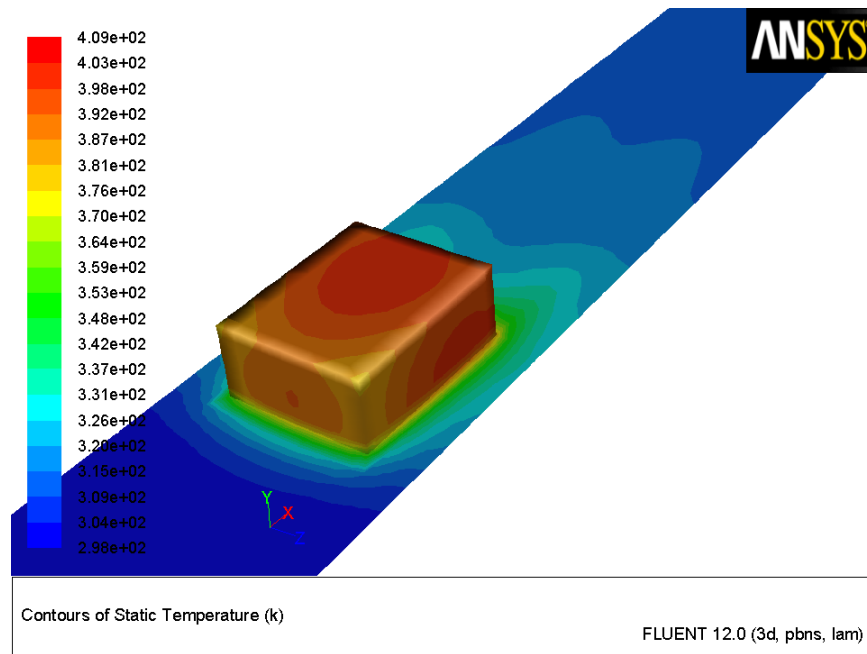
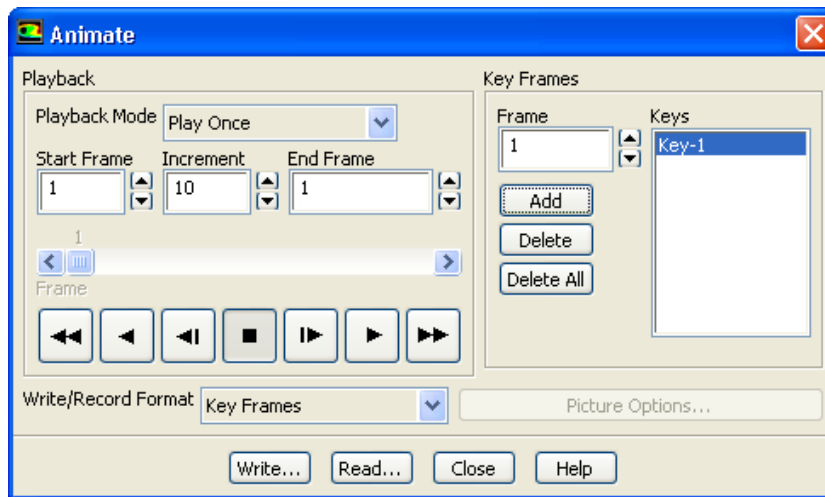


Figure 24.13: Filled Temperature Contours on the Chip and Board Top Surfaces

2. Create the key frames by changing the point of view.

◆ **Graphics and Animations** → **Scene Animation** → Set Up...



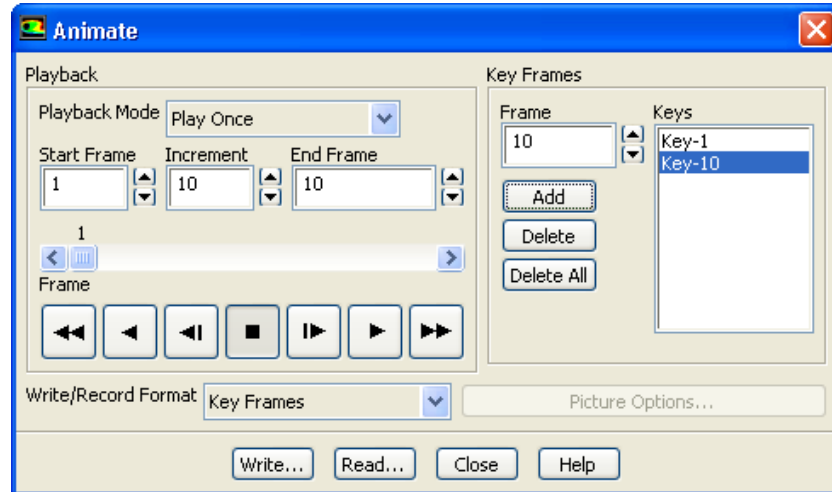
You will use the current display (Figure 24.13) as the starting view for the animation (Frame = 1).

- (a) Click Add in the Key Frames group box.

This will store the current display as Key-1.

- (b) Zoom the view to focus on the module region.
- (c) Enter 10 for Frame in the Key Frames group box.
- (d) Click Add.

This will store the new display as Key-10.



The zoomed view will be the tenth key frame of the animation, with intermediate displays (2 through 9) to be filled in during the animation.

- (e) Rotate the view and zoom out the display so that the downstream side of the module is in the foreground (Figure 24.14).

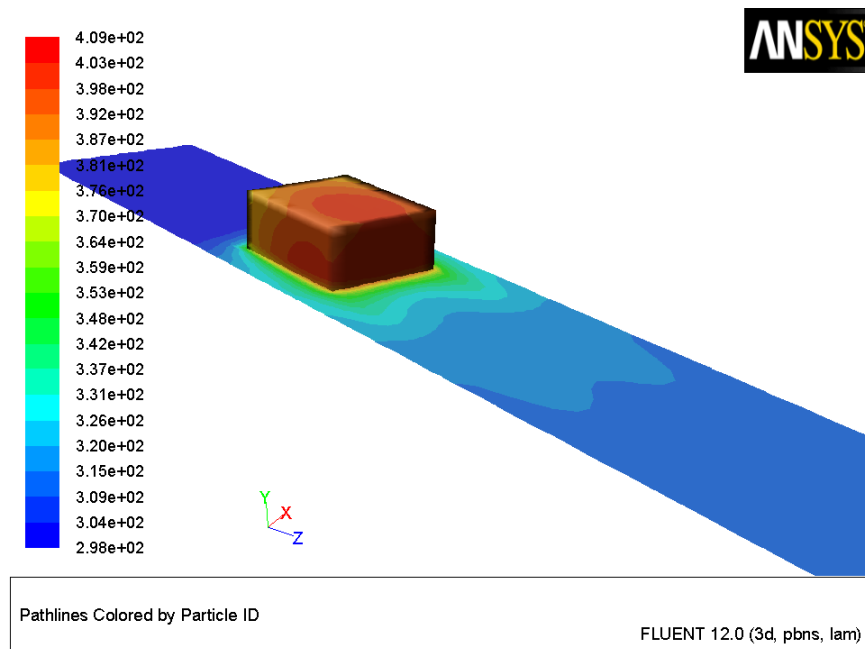


Figure 24.14: Filled Temperature Contours on the Chip and Board Top Surfaces

- (f) Enter 20 for **Frame**.
- (g) Click **Add**.

This will store the new display as Key-20.

3. View the scene animation by clicking on the “play” button () in the **Playback** group box.

*While effective animation is best conducted on “high-end” graphics workstations, you can view scene animations on any workstation. If the graphics display speed is slow, the animation playback will take some time and will appear choppy, with the redrawing very obvious. On fast graphics workstations, the animation will appear smooth and continuous and will provide an excellent visualization of the display from a variety of spatial orientations. On many machines, you can improve the smoothness of the animation by enabling the **Double Buffering** option in the **Display Options** dialog box.*

Note: *You can also make use of animation tools of ANSYS FLUENT for transient cases as demonstrated in Tutorial 4.*

Extra: *You can change the **Playback** mode if you want to “auto repeat” or “auto reverse” the animation. When you are in either of these **Playback** modes, you can click on the “stop” button () to stop the continuous animation.*

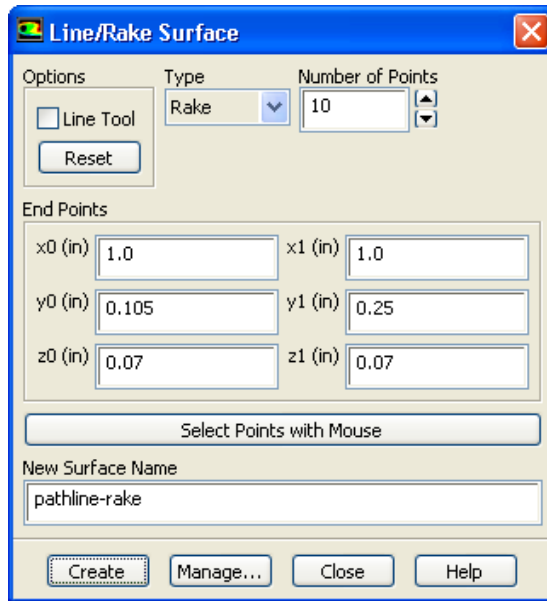
4. Close the **Animate** dialog box.

Step 8: Pathlines

Pathlines are the lines traveled by neutrally buoyant particles in equilibrium with the fluid motion. Pathlines are an excellent tool for visualization of complex three-dimensional flows. In this example, you will use pathlines to examine the flow around and in the wake of the module.

1. Create a rake from which the pathlines will emanate.

Surface → Line/Rake...



- (a) Select **Rake** from the **Type** drop-down list.

*A rake surface consists of a specified number of points equally spaced between two specified endpoints. A line surface (the other option in the **Type** drop-down list) is a line that includes the specified endpoints and extends through the domain; data points on a line surface will not be equally spaced.*

- (b) Retain the default value of 10 for **Number of Points**.

This will generate 10 pathlines.

- (c) Enter a starting coordinate of (1.0, 0.105, 0.07) and an ending coordinate of (1.0, 0.25, 0.07) in the **End Points** group box.

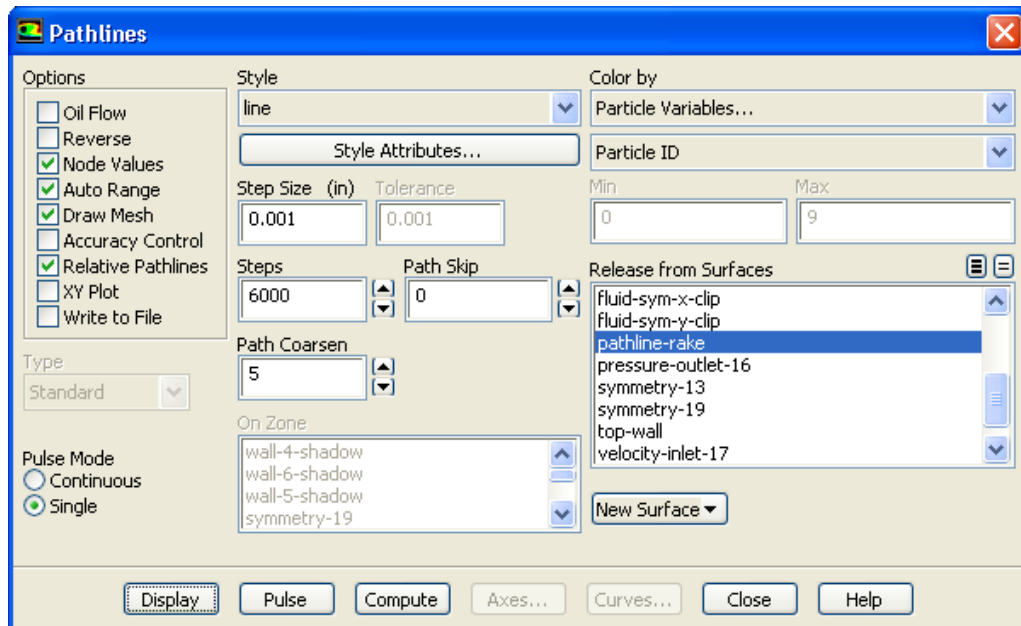
This will define a vertical line in front of the module, about halfway between the centerline and edge.

- (d) Enter **pathline-rake** for **New Surface Name**.

You will refer to the rake by this name when you plot the pathlines.

- (e) Click **Create** and close the **Line/Rake Surface** dialog box.

2. Draw the pathlines (Figure 24.15).



- (a) Enable Draw Mesh in the Options group box to open the Mesh Display dialog box.
 - i. Ensure that Faces is enabled in the Options group box.
 - ii. Retain the selection of board-top and chip from the Surfaces selection list.
These surfaces should already be selected from the earlier exercise where the mesh was displayed with velocity vectors, Step 6: Velocity Vectors.
 - iii. Close the Mesh Display dialog box.
- (b) Enter 0.001 inch for Step Size.
- (c) Enter 6000 for Steps.

Note: A simple rule of thumb to follow when you are setting these two parameters is that if you want the particles to advance through a domain of length L , the Step Size times the number of Steps should be approximately equal to L .

- (d) Set Path Coarsen to 5.

Coarsening the pathline simplifies the plot and reduces the plotting time. The coarsening factor specified for Path Coarsen indicates the interval at which the points are plotted for a given pathline in any cell.

- (e) Select pathline-rake from the Release from Surfaces selection list.

- (f) Click Display.

The pathlines will be drawn on the surface.

- (g) Rotate the display so that the flow field is in front and the wake of the chip is visible as shown in Figure 24.15.

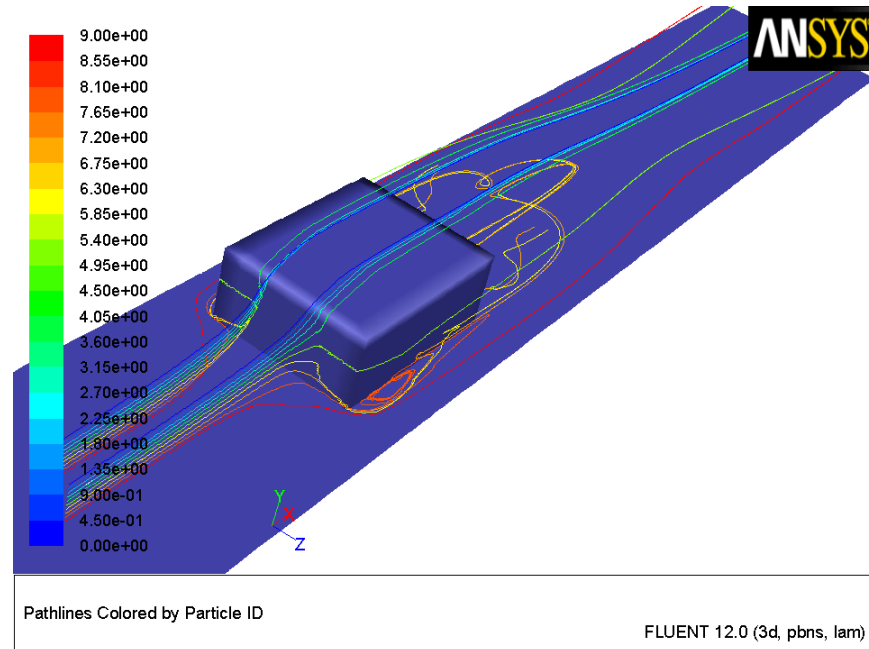


Figure 24.15: Pathlines Display

3. Write the pathlines to a file.

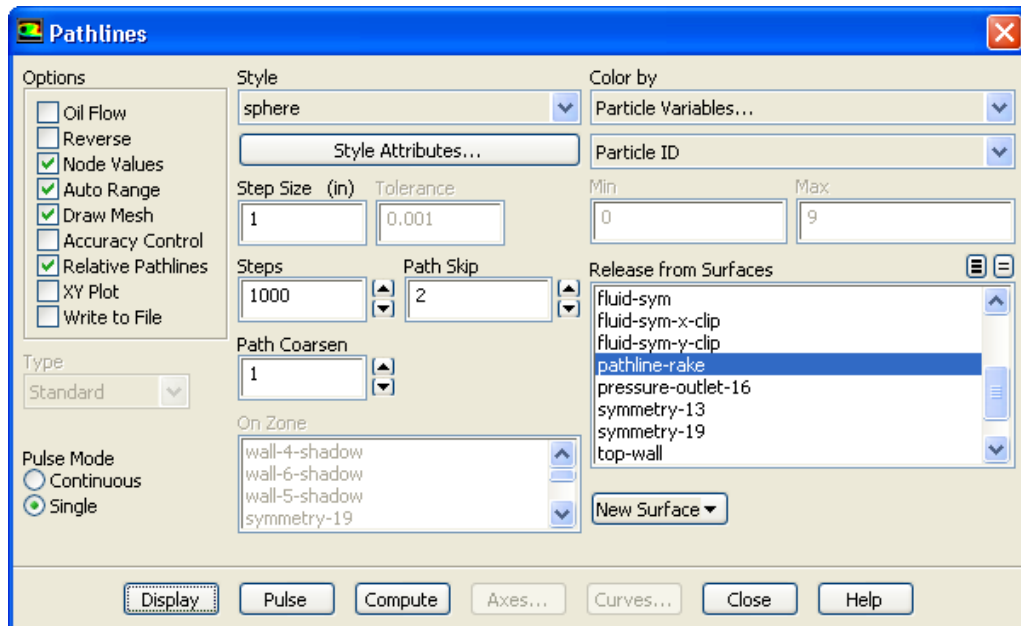
- (a) Enable Write to File in the Options group box.
 (b) Click the Write... button to open the Select File dialog box.

The Display button will change to a Write... button when you enable the Write to File option.

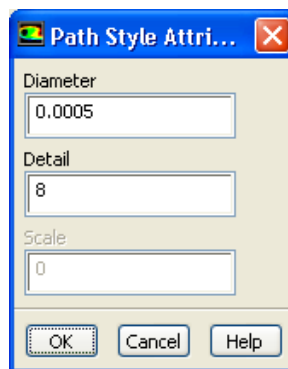
- i. Enter chip-pathline for Fieldview File.
- ii. Click OK to close the Select File dialog box.

ANSYS FLUENT will save the file in Fieldview format .fvp extension.

4. Display pathlines as spheres.



- (a) Disable Write to File in the Options group box.
- (b) Select **sphere** from the Style drop-down list.
- (c) Click the **Attributes...** button to open the Path Style Attributes dialog box.



- i. Enter 0.0005 for Diameter.
- ii. Click OK to close the Path Style Attributes dialog box.
- (d) Enter 1 inch for Step Size and 1000 for Steps respectively.
- (e) Set Path Skip to 2 and Path Coarsen to 1.
- (f) Retain the selection of **pathline-rake** in the Release from Surfaces selection list.
- (g) Click **Display**.

The spherical pathlines will be drawn along the surface.

- (h) Rotate the display so that the flow field is in front and the wake of the chip is visible as shown in Figure 24.16.

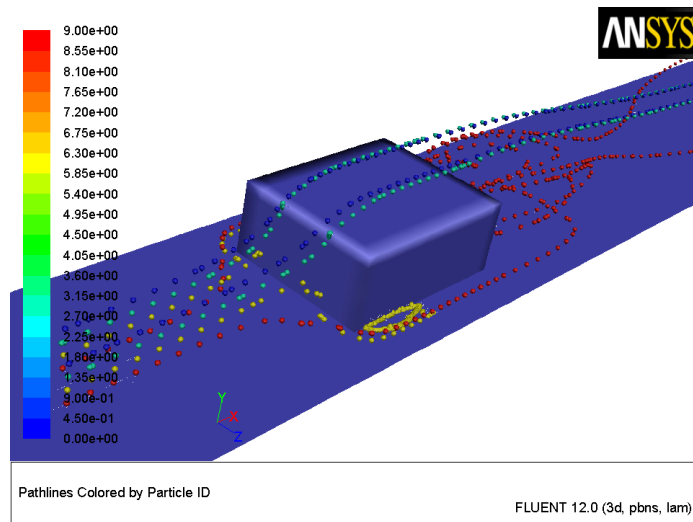


Figure 24.16: Sphere Pathlines Display

- (i) Select Surface ID from the lower Color by drop-down list.
 (j) Click Display and close the Pathlines dialog box (Figure 24.17).
This will color the pathlines by the surface they are released from.

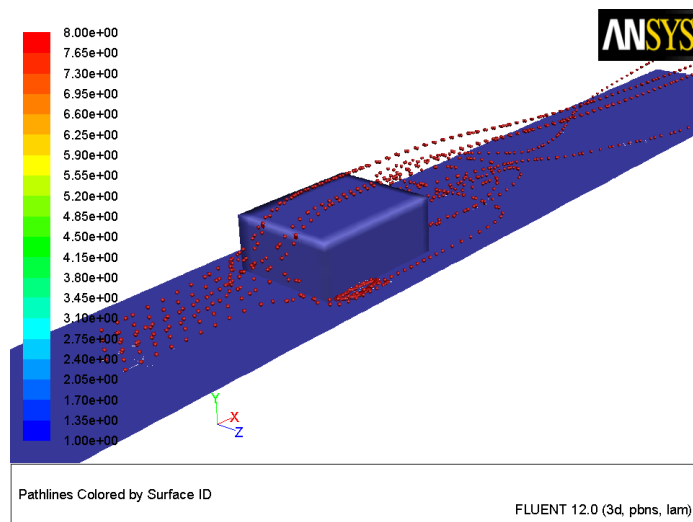


Figure 24.17: Sphere Pathlines Colored by Surface ID

Note: You can also create solution animations for pathlines using Animation Sequence dialog box.

◆ **Calculation Activities** (Solution Animations) → **Create/Edit...**

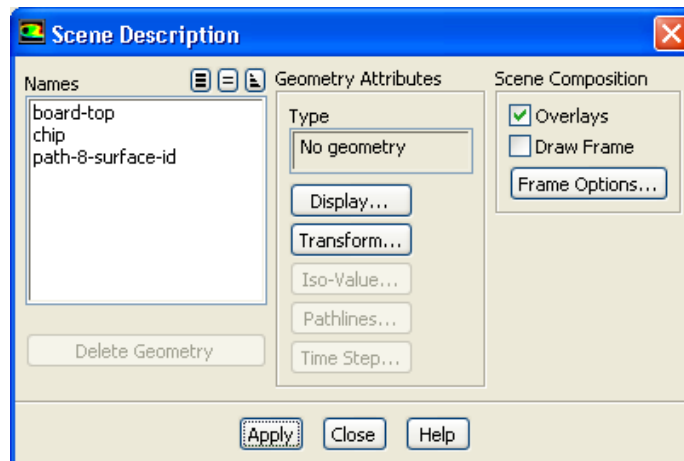
Step 9: Overlaying Velocity Vectors on the Pathline Display

◆ **Graphics and Animations**

The overlay capability, provided in the Scene Description dialog box, allows you to display multiple results on a single plot. You can exercise this capability by adding a velocity vector display to the pathlines just plotted.

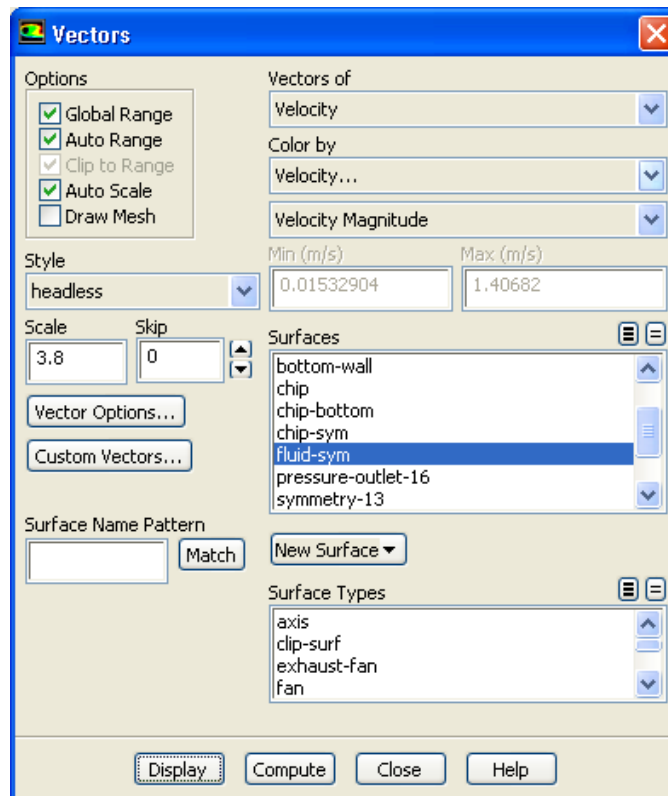
1. Enable the overlays feature.

◆ **Graphics and Animations** → **Scene...**



- (a) Enable Overlays in the Scene Composition group box.
- (b) Click **Apply** and close the Scene Description dialog box.

2. Add a plot of vectors on the chip centerline plane.



- Disable Draw Mesh in the Options group box.
- Retain the value of 3.8 for Scale.
- Deselect all surfaces by clicking the unshaded icon to the right of Surfaces.
- Select fluid-sym from the Surfaces selection list.

Because the mesh surfaces are already displayed and overlaying is active, there is no need to redisplay the mesh surfaces.

- Click Display and close the Vectors dialog box.
- Use the mouse to obtain the view that is shown in Figure 24.18.

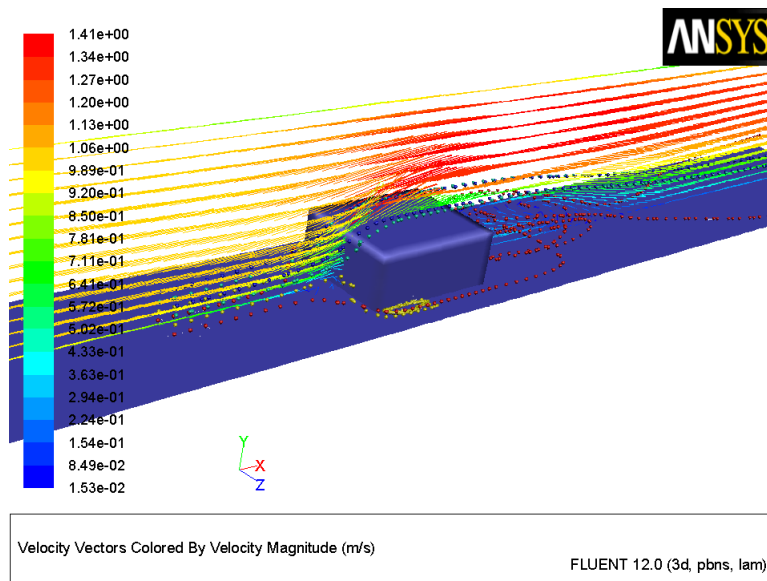


Figure 24.18: Overlay of Velocity Vectors and Pathlines Display

Note: The final display (Figure 24.18) does not require mirroring about the symmetry plane because the vectors obscure the mirrored image. You may disable the mirroring option in the Views dialog box at any stage during this exercise.

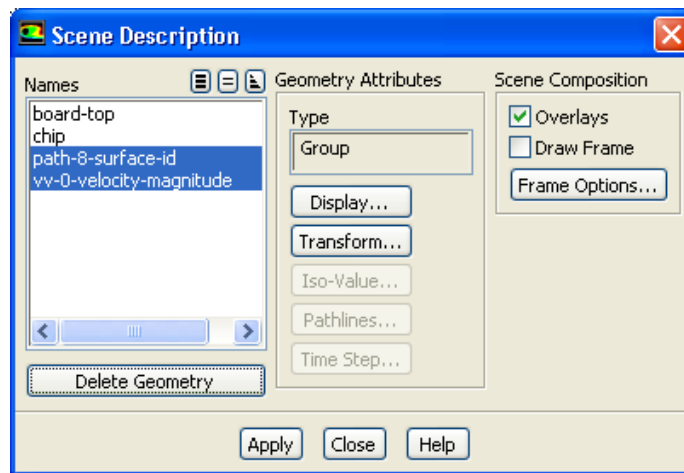
Step 10: Exploded Views

Graphics and Animations

The Scene Description dialog box stores each display that you request and allows you to manipulate the displayed items individually. This capability can be used to generate “exploded” views, in which results are translated or rotated out of the physical domain for enhanced display. As shown in the Scene Description dialog box, you can experiment with this capability by displaying “side-by-side” velocity vectors and temperature contours on a streamwise plane in the module wake.

1. Delete the velocity vectors and pathlines from the current display.

✦ **Graphics and Animations** → **Scene...**

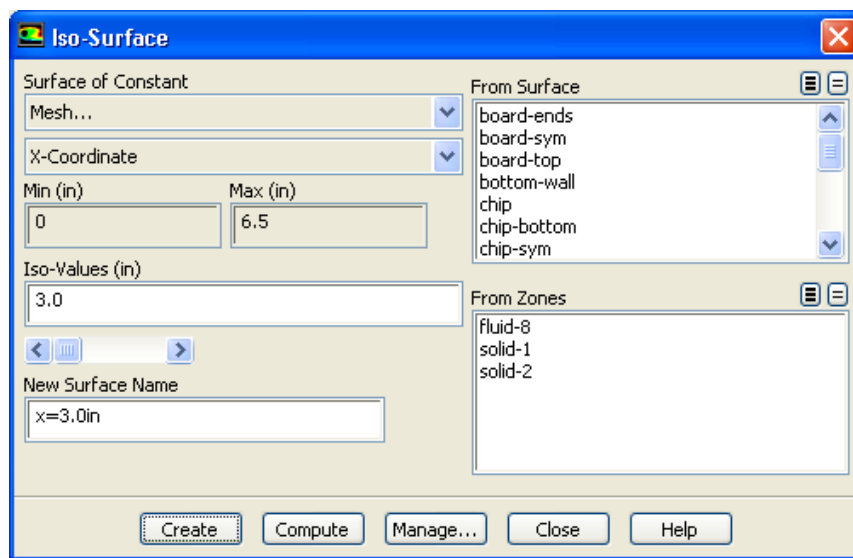


- (a) Select the pathlines (path-8-surface-id) and velocity vectors (vv-0-velocity-magnitude) from the Names selection list.
- (b) Click Delete Geometry.
- (c) Click Apply and close the Scene Description dialog box.

The Scene Description dialog box should then contain only the two mesh surfaces (board-top and chip).

2. Create a plotting surface at $x=3$ inches (named $x=3.0\text{in}$), just downstream of the trailing edge of the module.

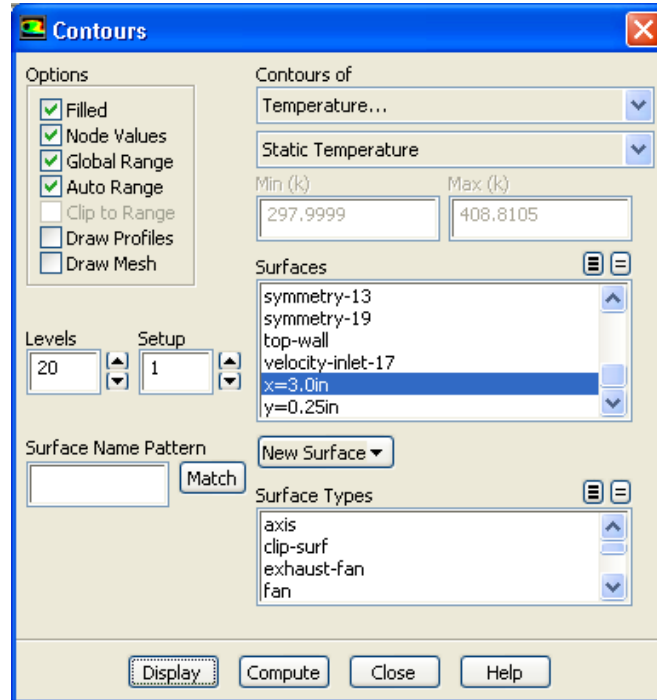
Surface → **Iso-Surface...**



Hint: For details on creating an isosurface, see Step 4: Creating Isosurfaces.

3. Add the display of filled temperature contours on the $x=3.0\text{in}$ surface.

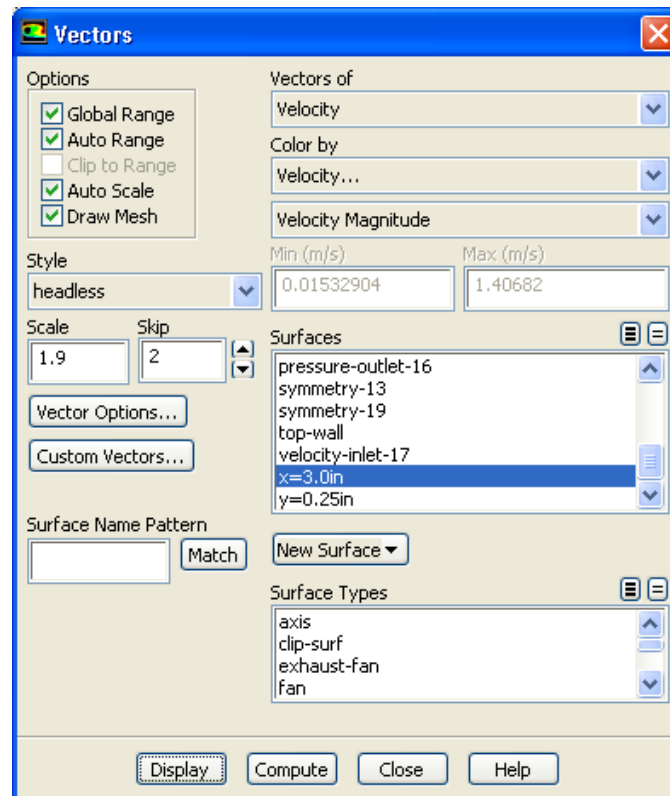
◆ Graphics and Animations →  Contours → Set Up...



- (a) Ensure that **Draw Mesh** is disabled in the **Options** group box.
- (b) Deselect all surfaces by clicking on the unshaded icon to the right of **Surfaces**.
- (c) Select $x=3.0\text{in}$ from the **Surfaces** selection list.
- (d) Click **Display** and close the **Contours** dialog box.

The filled temperature contours will be displayed on the $x=3.0\text{ in.}$ surface.

4. Add the velocity vectors on the $x=3.0$ in plotting surface.



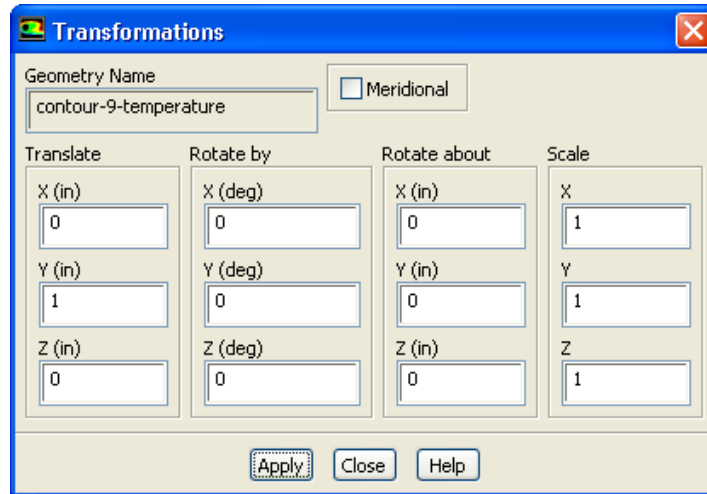
- (a) Enable the **Draw Mesh** option in the **Options** group box to open the **Mesh Display** dialog box.
 - i. Retain the default settings.
 - ii. Close the **Mesh Display** dialog box.
- (b) Enter 1.9 for **Scale**.
- (c) Set **Skip** to 2.
- (d) Deselect all surfaces by clicking on the unshaded icon to the right of **Surfaces**.
- (e) Select $x=3.0$ in from the **Surfaces** selection list.
- (f) Click **Display** and close the **Vectors** dialog box.

The display will show the vectors superimposed on the contours of temperature at $x=3.0$ in.

5. Create the exploded view by translating the contour display, placing it above the vectors (Figure 24.19).

◆ **Graphics and Animations** → **Scene...**

- (a) Select **contour-9-temperature** from the **Names** selection list.
- (b) Click the **Transform...** button to open the **Transformations** dialog box.



- i. Enter 1 inch for Y in the Translate group box.
- ii. Click **Apply** and close the **Transformations** dialog box.

The exploded view allows you to see the contours and vectors as distinct displays in the final scene (Figure 24.19).

- (c) Deselect **Overlays**.
- (d) Click **Apply** and close the **Scene Description** dialog box.

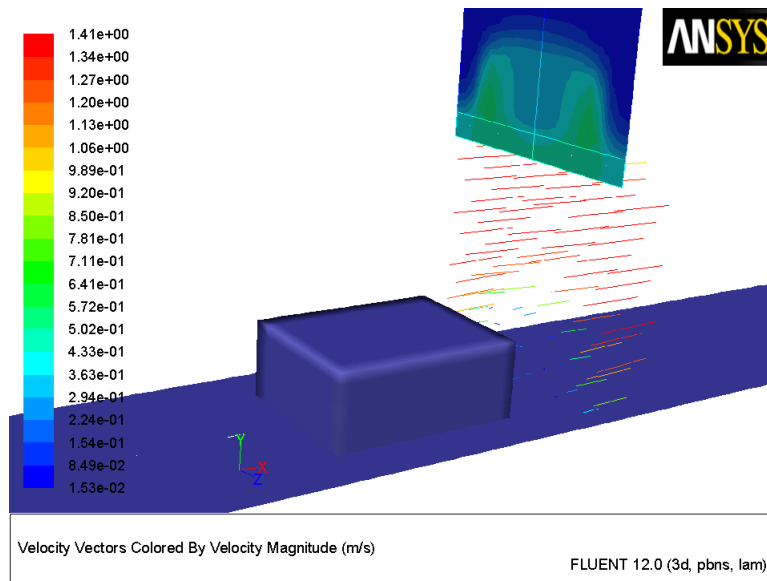


Figure 24.19: Exploded Scene Display of Temperature and Velocity

Step 11: Animating the Display of Results in Successive Streamwise Planes

◆ Graphics and Animations

*You may want to march through the flow domain, displaying a particular variable on successive slices of the domain. While this task could be accomplished manually, plotting each plane in turn, or using the **Scene Description** and **Animate** dialog boxes, here you will use the **Sweep Surface** dialog box to facilitate the process. To illustrate the display of results on successive slices of the domain, you will plot contours of velocity magnitude on planes along the X axis.*

1. Delete the vectors and temperature contours from the display.

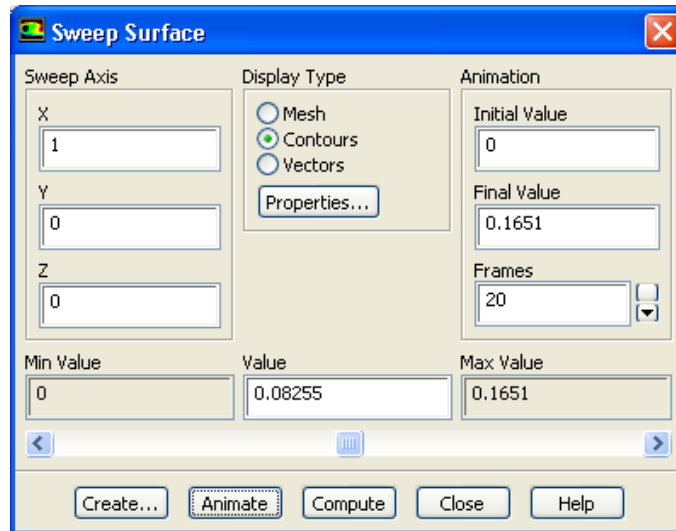
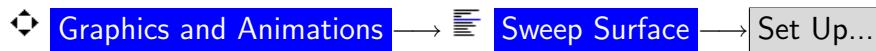
◆ **Graphics and Animations** → **Scene...**

- (a) Select **contour-9-temperature** and **vv-9-velocity-magnitude** from the **Names** selection list.
- (b) Click **Delete Geometry**.
- (c) Click **Apply** and close the **Scene Description** dialog box.

The dialog box and display window will be updated to contain only the mesh surfaces.

2. Use the mouse to zoom out the view in the graphics window so that the entire board surface is visible.

3. Generate contours of velocity magnitude and sweep them through the domain along the X axis.



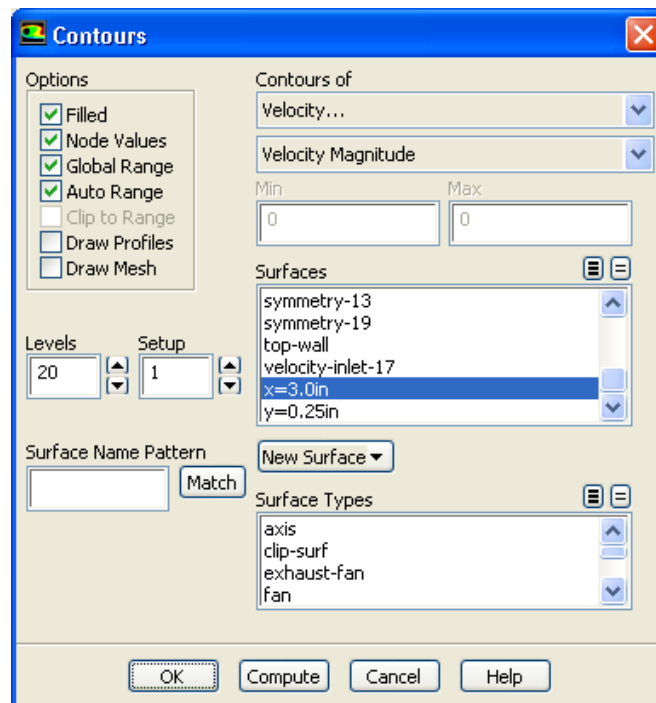
- (a) Retain the default settings in the **Sweep Axis** group box.
- (b) Retain the default value of 0 m for **Initial Value** and 0.1651 m for **Final Value** in the **Animation** group box.



The units for the initial and final values are in meters, regardless of the length units being used in the model. Here, the initial and final values are set to the **Min Value** and **Max Value**, to generate an animation through the entire domain.

- (c) Enter 20 for **Frames**.

(d) Select **Contours** in the **Display Type** list to open the **Contours** dialog box.



- i. Select **Velocity...** and **Velocity Magnitude** from the **Contours of** drop-down lists.
- ii. Click **OK** to close the **Contours** dialog box.

(e) Click **Animate** and close the **Sweep Surface** dialog box.

You will see the velocity contour plot displayed at 20 successive streamwise planes. ANSYS FLUENT will automatically interpolate the contoured data on the streamwise planes between the specified end points. Especially on high-end graphics workstations, this can be an effective way to study how a flow variable changes throughout the domain.

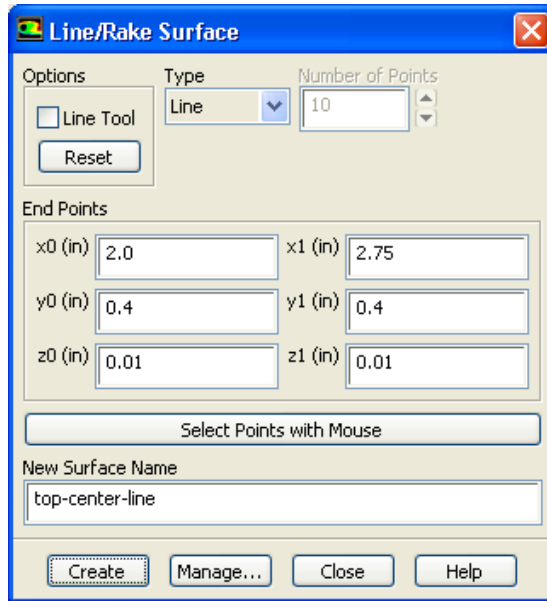
Note: *You can also make use of animation tools of ANSYS FLUENT for transient cases as demonstrated in Tutorial 4.*

Step 12: XY Plots

XY plotting can be used to display quantitative results of your CFD simulations. Here, you will complete the review of the module cooling simulation by plotting the temperature distribution along the top centerline of the module.

1. Define the line along which to plot results.

Surface → Line/Rake...

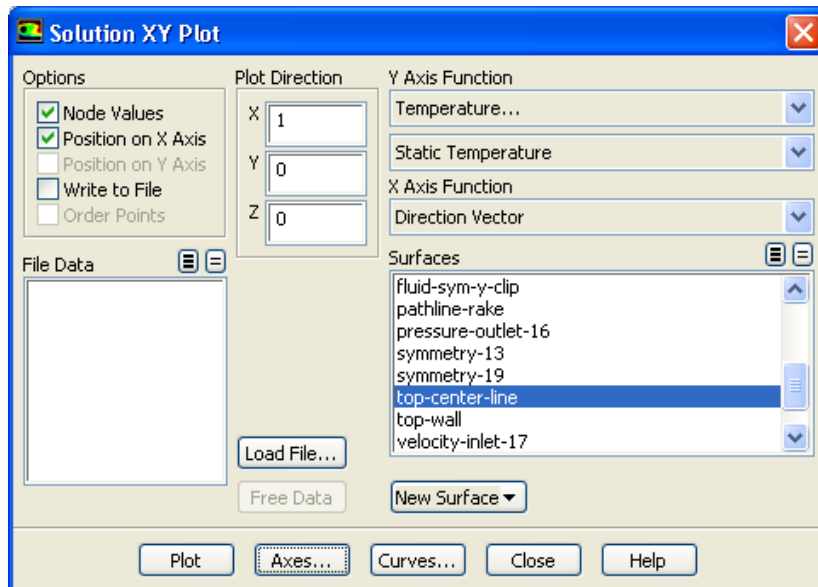


- (a) Select **Line** from the **Type** drop-down list.
- (b) Enter the coordinates of the line using a starting coordinate of (2.0, 0.4, 0.01) and an ending coordinate of (2.75, 0.4, 0.01).

These coordinates define the top centerline of the module.

- (c) Enter **top-center-line** for **New Surface Name**.
- (d) Click **Create** and close the **Line/Rake Surface** dialog box.

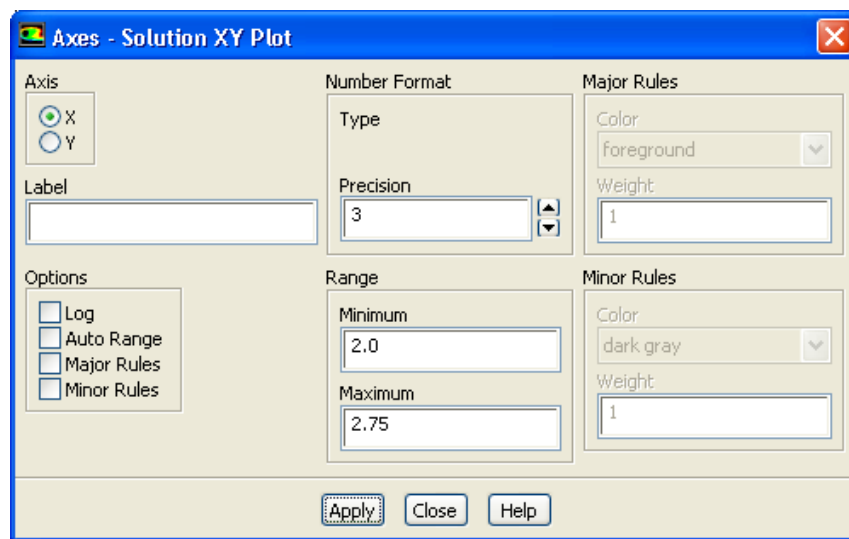
2. Plot the temperature distribution along the top centerline of the module (Figure 24.20).



- Retain the default Plot Direction of X.
- Select **Temperature...** and **Static Temperature** from the Y Axis Function drop-down lists.
- Select **top-center-line** from the Surfaces selection list.

This will plot temperature vs the x coordinate along the selected line (top-center-line).

- Click the **Axes...** button to open the Axes - Solution XY Plot dialog box.



- i. Retain the selection of X in the Axis list.
 - ii. Disable Auto Range in the Options group box.
 - iii. Enter 2.0 for Minimum and 2.75 for Maximum in the Range group box.
 - iv. Click Apply and close the Axes - Solution XY Plot dialog box.
- (e) Click Plot and close the Solution XY Plot dialog box.

The temperature distribution (Figure 24.20) shows the temperature increase across the module surface as the thermal boundary layer develops in the cooling air flow.

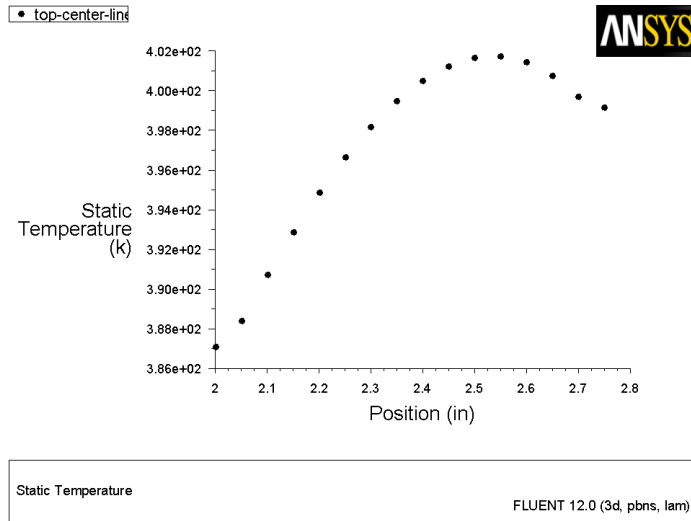
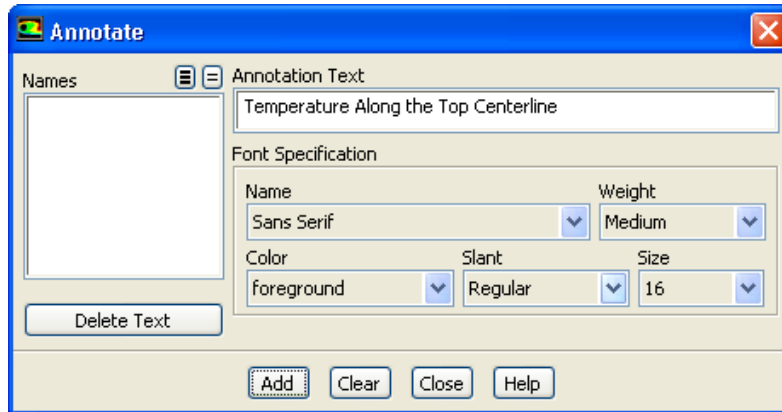


Figure 24.20: Temperature Along the Top Centerline of the Module

Step 13: Annotation



You can annotate the display with the text of your choice.



1. Enter the text describing the plot (e.g., **Temperature Along the Top Centerline**), in the **Annotation Text** field.
2. Click **Add**.

A Working dialog box will appear telling you to select the desired location of the text using the mouse-probe button.

3. Click the right mouse button in the graphics display window where you want the text to appear, and you will see the text displayed at the desired location (Figure 24.21).

Extra: *If you want to move the text to a new location on the screen, click **Delete Text**, and click **Add** once again, defining a new position with the mouse.*

Note: *Depending on the size of the graphics window and the hardcopy file format you choose, the font size of the annotation text you see on the screen may be different from the font size in a hardcopy file of that graphics window. The annotation text font size is absolute, while the rest of the items in the graphics window are scaled to the proportions of the hardcopy.*

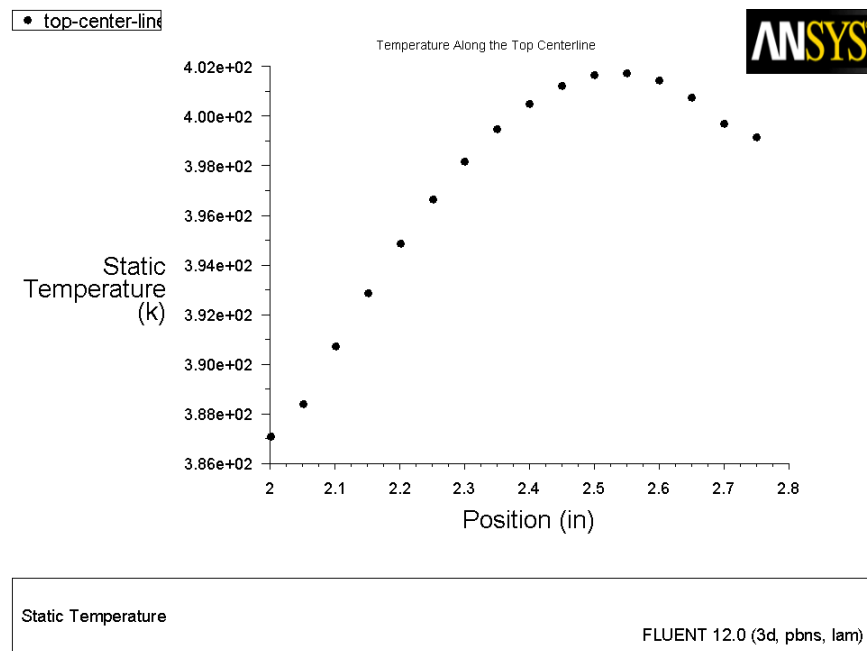
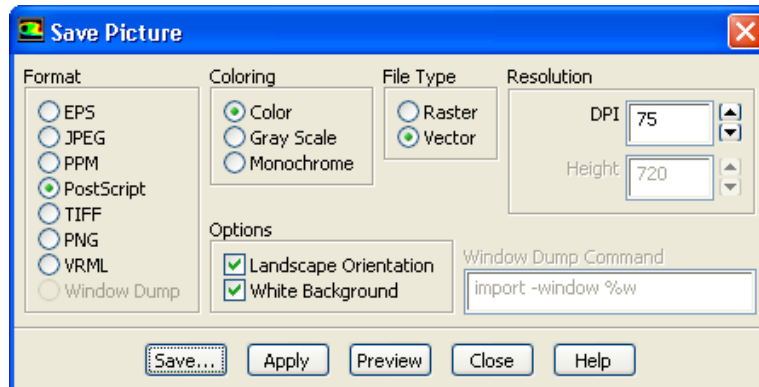


Figure 24.21: Temperature Along the Top Centerline of the Module

Step 14: Saving Hardcopy Files

File → Save Picture...

You can save hardcopy files of the graphics display in many different formats, including PostScript, encapsulated PostScript, TIFF, PNG, PPM, JPEG, VRML and window dumps. Here, the procedure for saving a color PostScript file is shown.

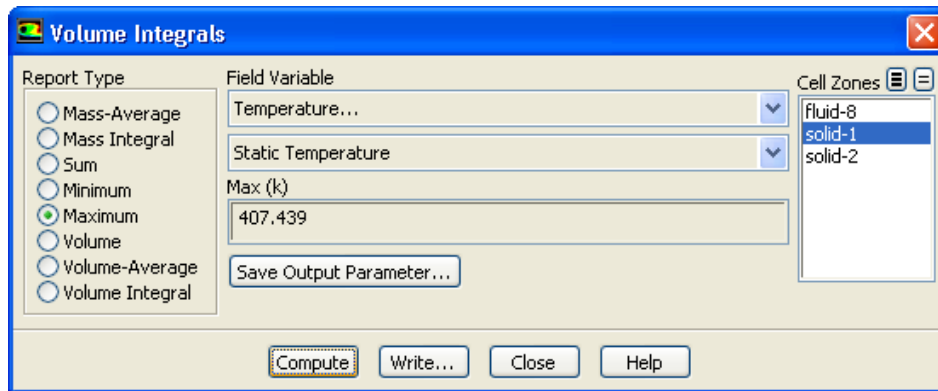


1. Select PostScript in the Format list.
2. Select Color in the Coloring list.
3. Click the Save... button to open the Select File dialog box.
 - (a) Enter a name for Hardcopy File.
 - (b) Click OK to close the Select File dialog box.
4. Close the Save Picture dialog box.

Step 15: Volume Integral Reports



Reports of Volume Integral can be used to determine the Volume of a particular fluid region (i.e. fluid zone), the sum of quantities or the maximum and minimum values of particular variables. Here we will use the Volume Integral reports to determine the maximum and minimum temperature in the chip, board, and the airflow.



1. Select Maximum in the Report Type group box.
2. Select Temperature... and Static Temperature from the Field Variable drop-down lists.
3. Select solid-1 from the Cell Zones selection list.
4. Click Compute to calculate the maximum temperature.
The maximum temperature in the solid-1 cell zone (the chip) will be displayed.
5. Select Minimum in the Report Type group box and click Compute.
The minimum temperature will be displayed in the dialog box.
6. Repeat the operations to determine the maximum and minimum temperatures in the solid-2 and fluid-8 cell zones, corresponding to the board and fluid volume, respectively.

Summary

This tutorial demonstrated the use of many of the extensive postprocessing features available in ANSYS FLUENT.

For more information on these and related features, see Chapter 30 or Chapter 29 in the separate [User's Guide](#).

