Tutorial 22.   Postprocessing

Introduction: In this tutorial, the postprocessing capabilities of FLUENT are demonstrated for a 3D laminar flow involving conjugate heat transfer. The flow is over a rectangular heat-generating electronics chip which is mounted on a flat circuit board. The heat transfer involves the coupling of 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 example, you will read the case and data files (without doing the calculation) and perform a number of postprocessing exercises. In the process, you will learn how to:

- Create surfaces for the display of 3D data
- Display velocity vectors
- Display filled contours of temperature on several surfaces
- Mirror a display about a symmetry plane
- Add lights to the display at multiple locations
- Use the Scene Description and Animate panels to animate the graphics display
- Use the Sweep Surface panel to display results on successive slices of the domain
- Display pathlines
- Plot quantitative results
- Overlay and "explode" a display
- Annotate your display

Prerequisites: This tutorial assumes that you are familiar with the menu structure in FLUENT, and that you have solved Tutorial 1.

Problem Description: The problem to be considered is shown schematically in Figure 22.1. 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 W/m²-K. The circuit board conductivity is assumed to be one order of magnitude lower: 0.1 W/m²-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.
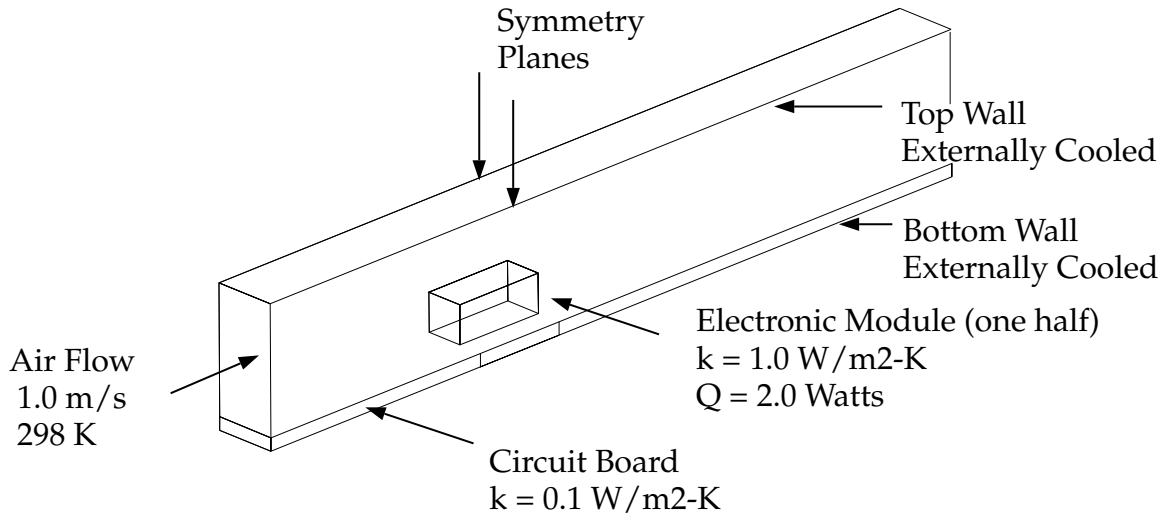
Symmetry
Planes

Top Wall
Externally Cooled

Bottom Wall
Externally Cooled

Electronic Module (one half)
k = 1.0 W/m2-K
Q = 2.0 Watts

Air Flow
1.0 m/s
298 K

Circuit Board
k = 0.1 W/m2-K

Figure 22.1: Problem Specification

Preparation

1. Copy the files chip/chip.cas and chip/chip.dat from the FLUENT documentation CD to your working directory (as described in Tutorial 1).

2. Start the 3D version of FLUENT.

Step 1: Reading the Case and Data Files

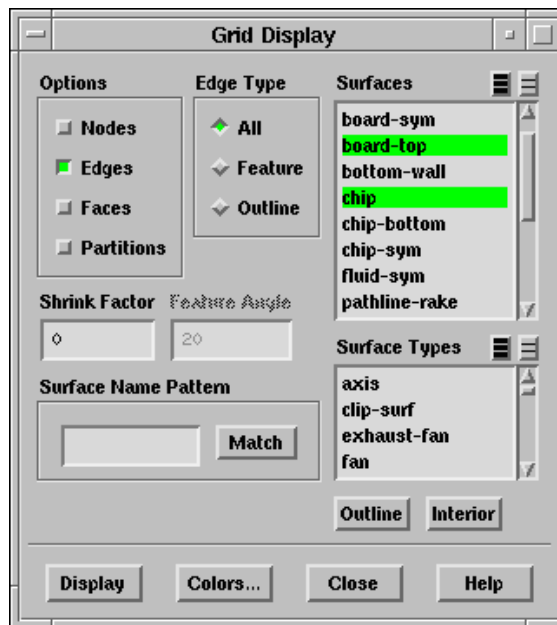1. Read in the case and data files (`chip.cas` and `chip.dat`).

   File ⟶ Read ⟶Case & Data...

   Once you select `chip.cas`*,* `chip.dat` *will be read automatically.*

Step 2: Grid Display

1. Display the grid.

   Display ⟶Grid...

(a) Under Options, select Edges.

(b) In the Surfaces list, select board-top and chip.

(c) Click Display.

Note: *You may want to scroll through the* Surfaces *list to be sure that no other surfaces are selected. You can also deselect all surfaces by clicking on the far-right button at the top of the* Surfaces *list, and then select the desired surfaces for display.*
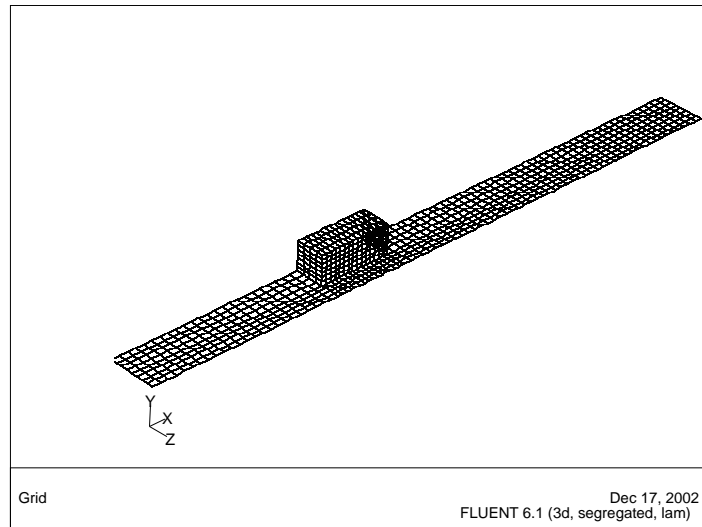
Figure 22.2: Graphics Display of the Chip and Board Surfaces

2. Use your left mouse button to rotate the view, and your middle mouse button to zoom the view until you obtain an enlarged isometric display of the circuit board in the region of the chip, as shown in Figure 22.2.

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

3. Create a filled surface display.

   (a) In the Grid Display panel under Options, deselect Edges and select Faces.

   (b) Click Display.

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

4. Add shading effects by enabling lights.

    Display ⟶Options...
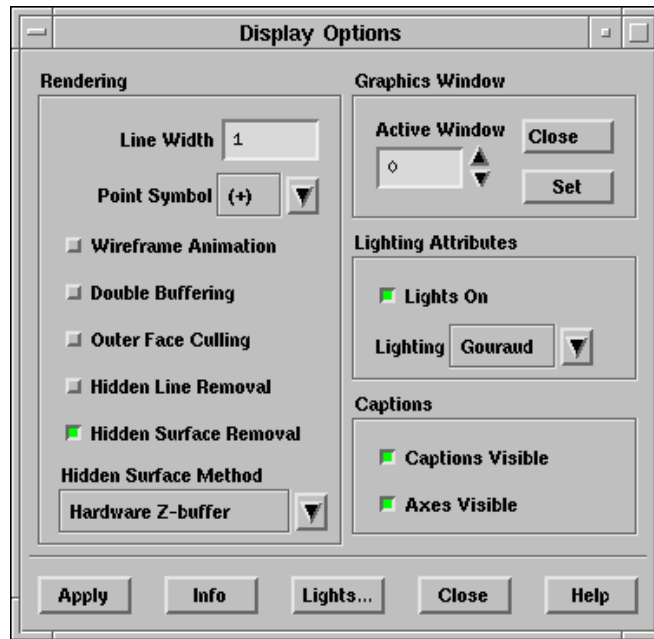
(a) Under Lighting Attributes, enable the Lights On.

(b) Click Apply.

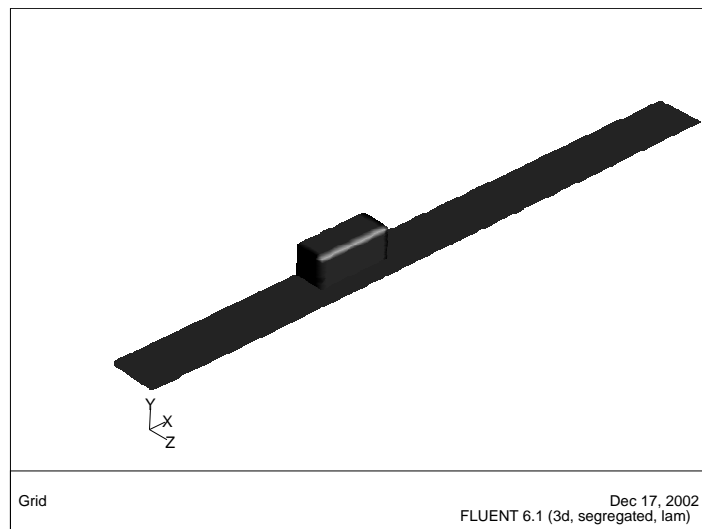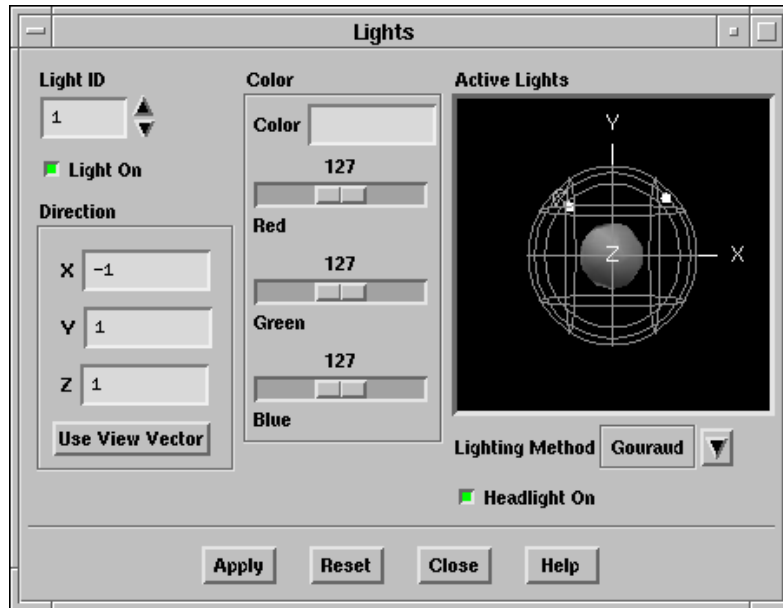*Shading will be added to the surface grid display (Figure 22.3).*

Figure 22.3: Graphics Display of the Chip and Board Surfaces with Default Lighting

(c) In the Display Options panel, click on the Lights... button.

This will open the Lights *panel.*

Note: *When you turn lights on, the default settings are for light 0 (indicated by the* Light ID*), corresponding to a white light at the position (1, 1, 1), as indicated by the unit vectors under* Direction*.*

(d) Add a light at (-1,1,1).

    i. Increase the Light ID to 1 and enable the Light On option.

    ii. Set X, Y, and Z to -1, 1, and 1.

    iii. Click Apply.

(e) Repeat this procedure to add a second light (Light ID=2) at (-1,1,-1).

The result is a more softly shaded display (Figure 22.4).

Note: *The* Lights *panel contains a* Headlight On *option that is turned on by default. When you create additional lights using this panel,* FLUENT *will automatically turn on a light in the direction of the flow. You can turn off the headlight by deselecting the* Headlight On *option (Figure 22.5).*

Extra: *You can use your left mouse button to rotate the ball in the* Active Lights *window in the* Lights *panel, as shown below. By doing so, you can 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.*
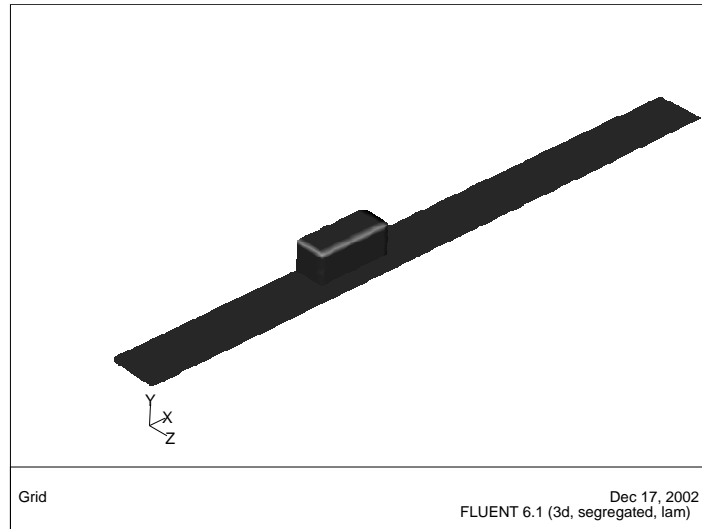
Figure 22.4: Graphics Display of the Chip and Board Surfaces with Additional Lighting
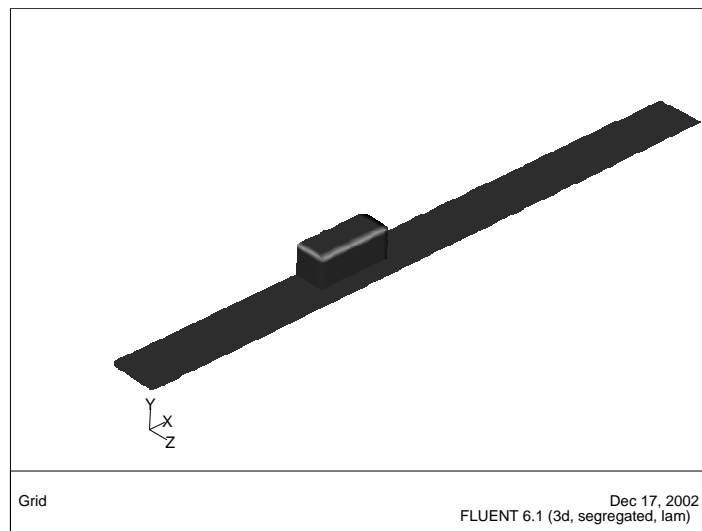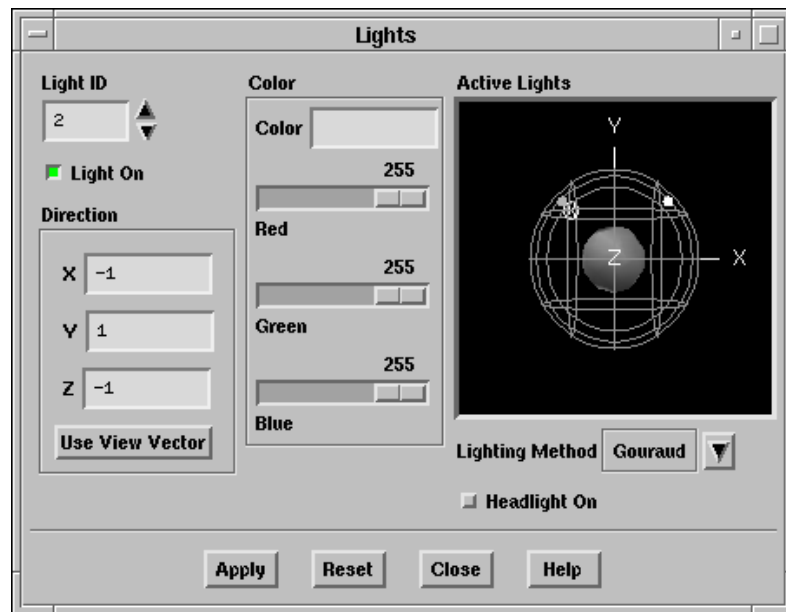

Figure 22.5: Graphics Display of the Chip and Board Surfaces with Additional Lighting: Headlight Off

You can also change the color of one or more of the lights by typing the name of a color in the Color *field or moving the* Red*,* Green*, and* Blue *sliders.*

Step 3: Isosurface Creation

To display results in a 3D model, you will need surfaces on which the data can be displayed. FLUENT *creates surfaces for all boundary zones automatically. In the case file that you have read, several of these surfaces have been renamed. Examples are* board-sym *and* board-ends, *which correspond to the side and end faces of the circuit board. In general, you may want to define additional surfaces for the purpose of viewing your results, such as a plane in Cartesian space, for example. In this exercise, you will create a horizontal plane cutting through the middle of the module. This surface will have a y value of 0.25 inches, and will be used in a later step for displaying the temperature and velocity fields.*

1. Create a surface of constant *y* coordinate.

    Surface ⟶Iso-Surface...
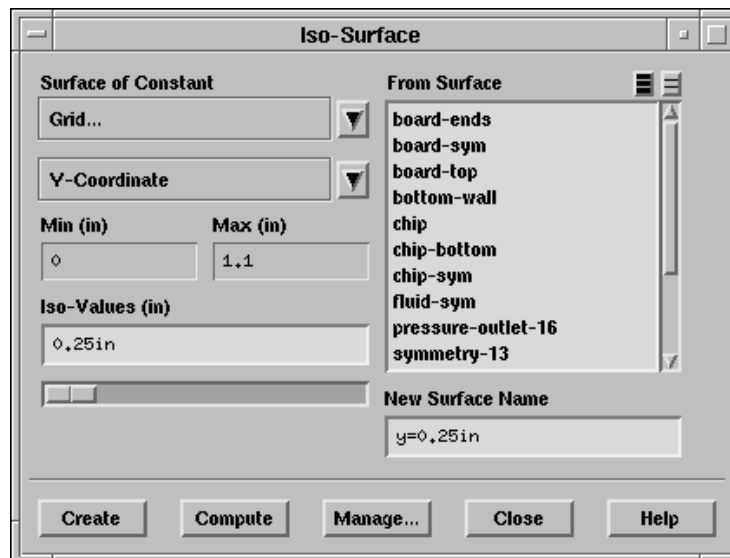
    (a) In the Surface of Constant drop-down lists, select Grid... and Y-Coordinate.

    (b) Click Compute.

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

    (c) Enter 0.25 under Iso-Values.

    (d) Enter y=0.25in under New Surface Name.

    (e) Click Create, and Close the panel.

Step 4: Contours

1. Plot filled contours of temperature on the symmetry plane (Figure 22.6).

    Display ⟶Contours...
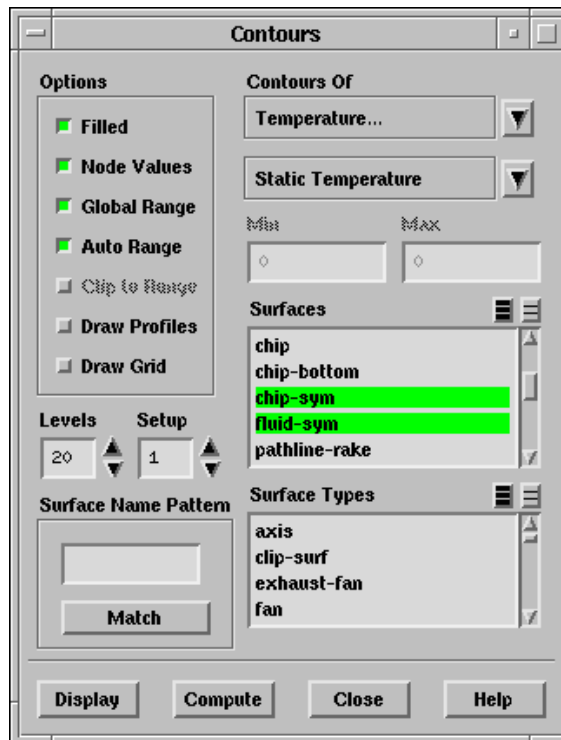
   (a) Under Options, select Filled.

   (b) Select Temperature... and Static Temperature in the Contours Of drop-down lists.

   (c) In the Surfaces list, select board-sym, chip-sym, and fluid-sym.

   (d) Click Display.

   The temperature contour will be displayed.

   (e) Rotate and zoom the display using the left and middle mouse buttons, respectively, to obtain the view shown in Figure 22.6.

   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 *panel from the* Display *pull-down menu and use the* Default *button to reset the view. Alternatively, you can revert to a previous graphics display using the keyboard shortcut* `<Ctrl>-L`.

   Hint: *You can revert to previous views in the graphics display window using the keyboard shortcut* `<Ctrl>-L`.
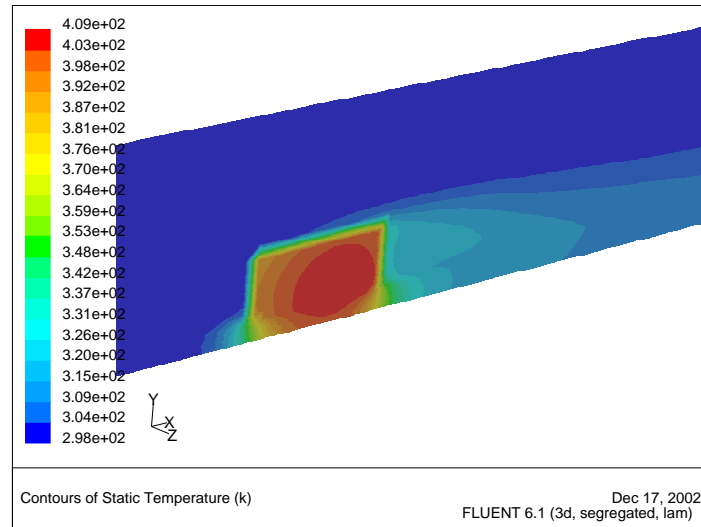
Figure 22.6: Filled Contours of Temperature on the Symmetry Surfaces

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

2. Plot filled contours of temperature on the horizontal plane at y=0.25 in (Figure 22.7).

   (a) In the Contours panel under Surfaces, deselect the symmetry planes and select y=0.25in.

   (b) Click Display.

   (c) Zoom the display using your middle mouse button to obtain the view shown in Figure 22.7.

In the contour display (Figure 22.7), the high temperatures in the wake of the module are clearly visible. You may want to display other quantities using the Contours panel, such as velocity magnitude or pressure).
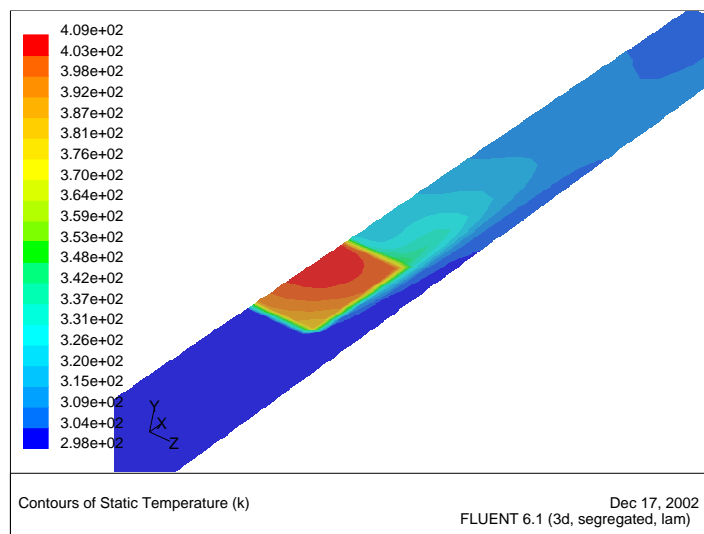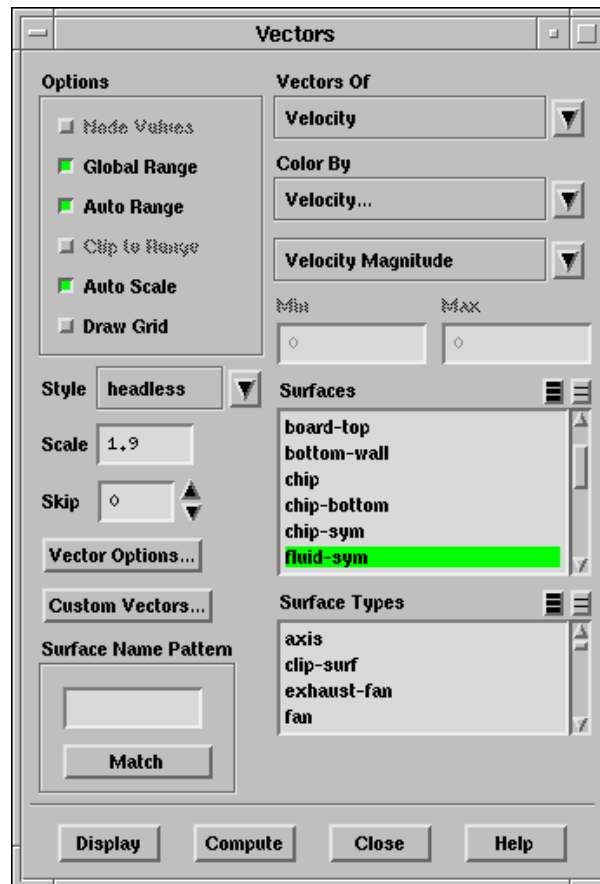
Figure 22.7: Filled Contours of Temperature on the $y = 0.25$ in. Surface

Step 5: Velocity Vectors

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 22.8).

   Display ⟶Vectors...

   (a) In the Surfaces list, select fluid-sym.

   (b) Set the Scale Factor to 1.9.

   (c) Click Display.

   (d) Rotate and zoom the display to observe the vortex near the stagnation point and in the wake of the module (Figure 22.8).

   Note: *The vectors in Figure 22.8 are shown without arrowheads. You can modify the arrow style in the* Vectors *panel by selecting a different option from the* Style *drop-down list.*
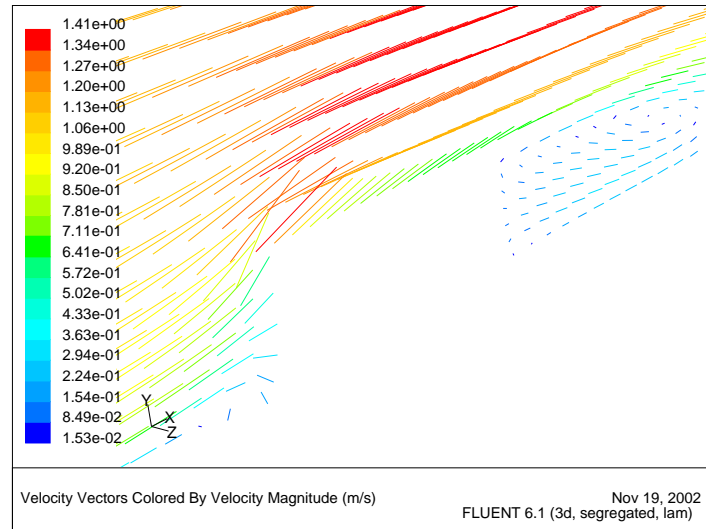
Figure 22.8: Velocity Vectors in the Module Symmetry Plane

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

2. Plot velocity vectors in the horizontal plane intersecting the module (Figure 22.10).

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

   Display ⟶Vectors...
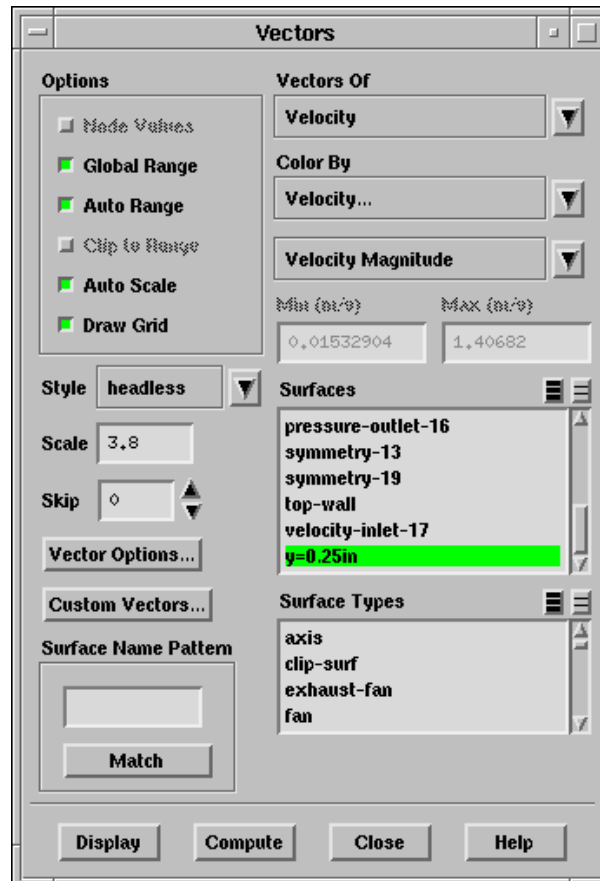
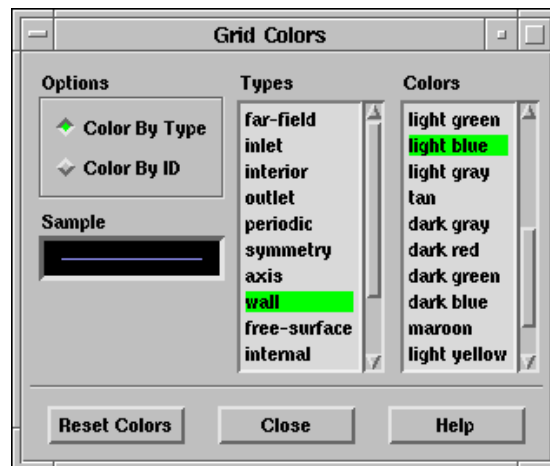   (a) Deselect all surfaces by clicking the unshaded icon to the right of Surfaces.

   (b) In the Surfaces list, select y=0.25in.

   (c) Set the Scale to 3.8.

   (d) Under Options, select Draw Grid.

       This will open the Grid Display *panel.*

(e) Under Options, check that Faces is selected.

(f) In the Surfaces list, select board-top and chip.

(g) Click Colors....

> *This will open the* Grid Colors *panel.*

(h) In the Types list, select wall.

(i) In the Colors list, select light blue, and then Close the panel.

(j) In the Grid Display panel, click Display and then Close the panel.

(k) Use your mouse to obtain the view shown in Figure 22.9.

(l) In the Vectors panel, click Display.

(m) Rotate the display with your mouse to obtain the view shown in Figure 22.10.
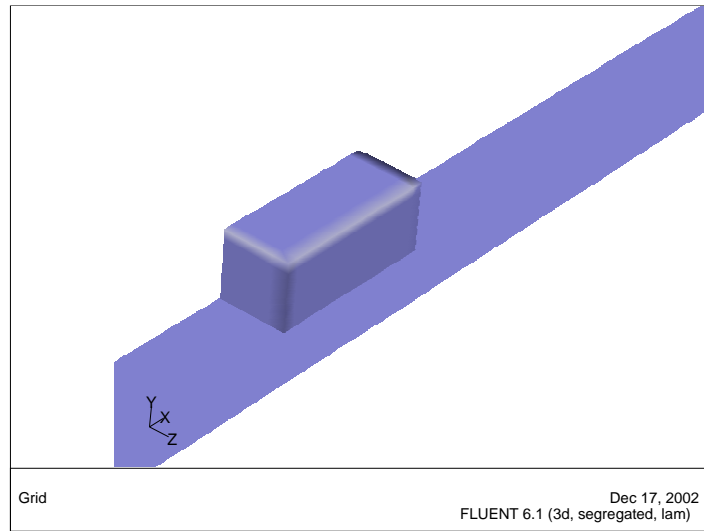
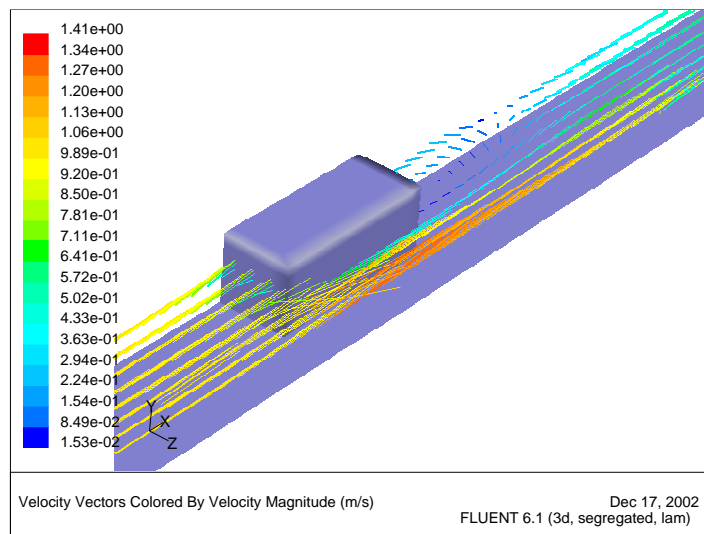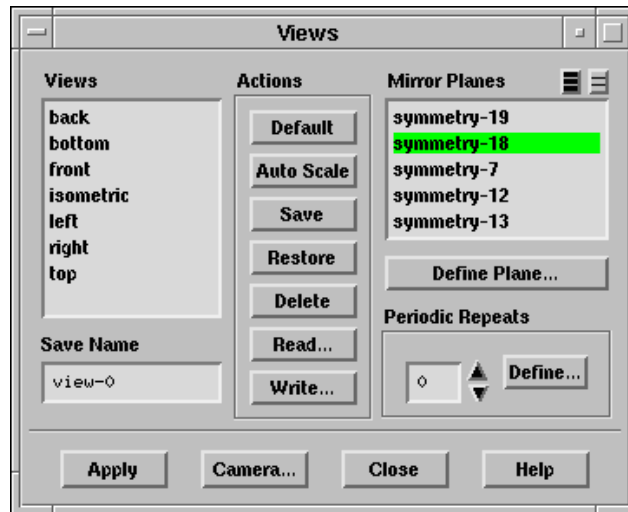Figure 22.9: Filled Surface Display for the Chip and Board Top

Figure 22.10: Velocity Vectors and Chip and Board Top Surfaces

3. Mirror the view about the chip symmetry plane (Figure 22.11).

   Display ⟶Views...

(a) In the Mirror Planes list, select symmetry-18.

   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.

   The display will be updated in your graphics window (Figure 22.11).

Extra: *You may want to experiment with different views and/or scale factors for the velocities to examine different regions (upstream and downstream of the chip, for example).*
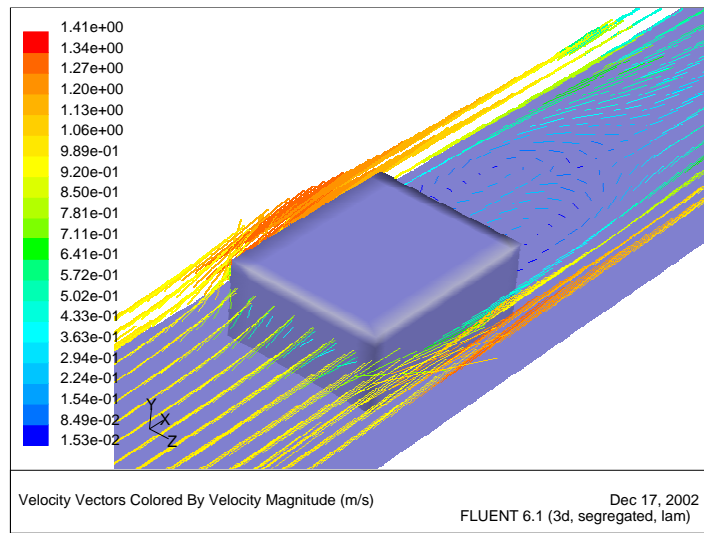
Figure 22.11: Velocity Vectors and Chip and Board Top Surfaces after Mirroring

Step 6: Animation

The surface temperature distribution on the module and on the circuit board can be displayed by selecting these boundaries for display of temperature contours. You can then view the display dynamically, using the animation feature. While effective animation is best conducted on "high-end" graphics workstations, you can follow the procedures below on any workstation. If your 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 turning on the Double Buffering *option in the* Display Options *panel.*

1. Display filled contours of surface temperature on the board-top and chip, excluding the symmetry surfaces (Figure 22.12).

   Display ⟶Contours...
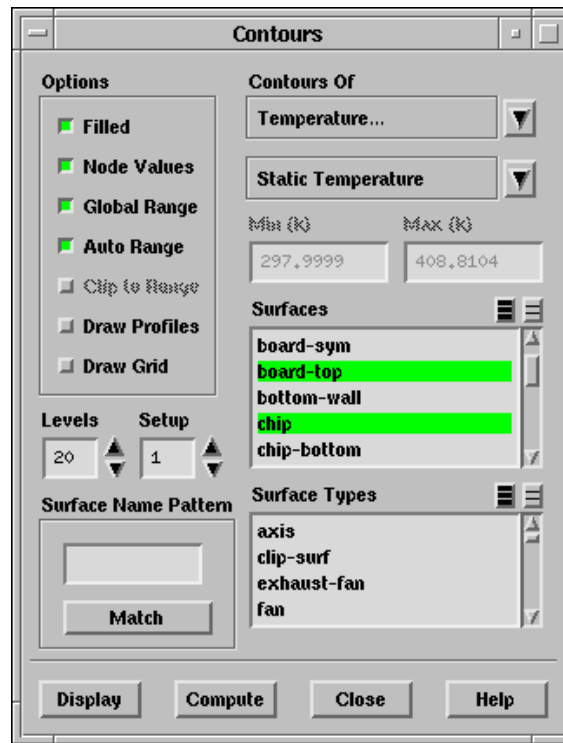
   (a) Under Options, select Filled.
   (b) Select Temperature... and Static Temperature in the Contours Of drop-down lists.
   (c) Deselect all surfaces by clicking the unshaded icon to the right of Surfaces.
   (d) In the Surfaces list, select board-top and chip.

(e) Click Display.

(f) Zoom the display as needed to obtain the view shown in Figure 22.12.
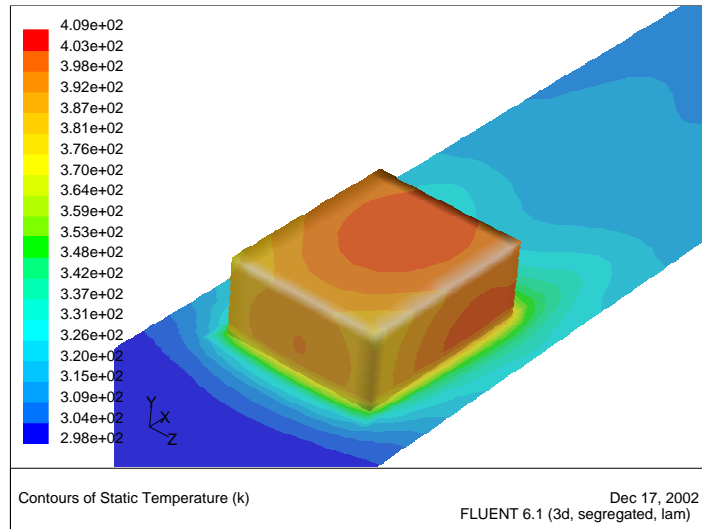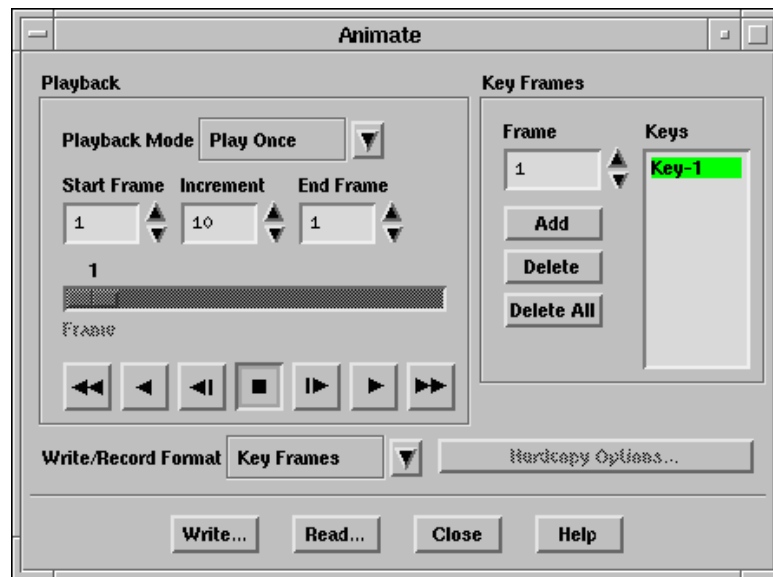

Figure 22.12: Filled Temperature Contours on the Chip and Board Top Surfaces

The temperature display (Figure 22.12) shows the high temperatures on the down-stream portions of the module and the relatively localized heating of the circuit board around the module.

2. Animate the surface temperature display by changing the point of view.

Display —→Scene Animation...

*You will use the current display (Figure 22.12) as the starting view for the animation (*Frame = 1*).*

(a) Under Key Frames, click Add.

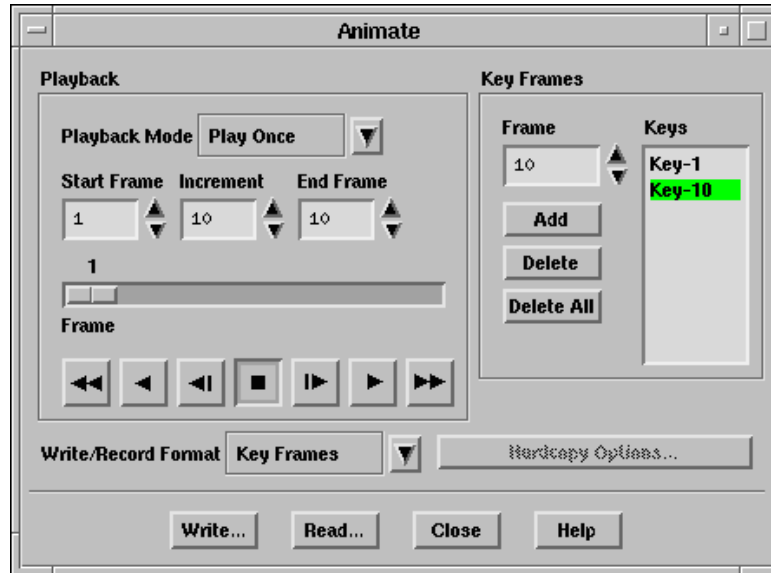*This will store the current display as* Key-1.

(b) Zoom the view to focus on the module region.

(c) Under Key Frames, change the Frame number to 10.

(d) Click Add.

This will store the new display as Key-10.

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

(e) Rotate the view and un-zoom the display so that the downstream side of the module is in the foreground, as shown in Figure 22.13.

(f) Change the Frame number to 20.

(g) Click Add.

This will store the new display as Key-20.

3. To animate the view, click on the "play" arrow button (second from the right in the row of playback buttons) in the Playback section of the Animate panel.

Note: *You can also make use of* FLUENT*'s animation tools 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 (square) to stop the continuous animation.*
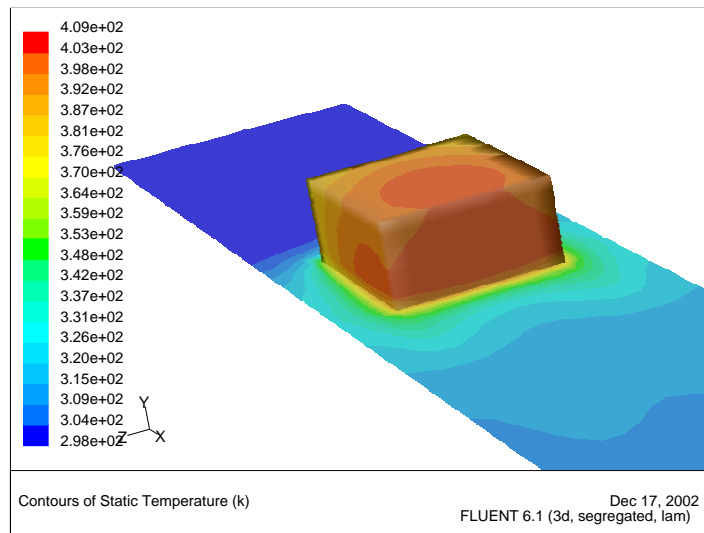
Figure 22.13: Filled Temperature Contours on the Chip and Board Top Surfaces

Step 7: Displaying 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...
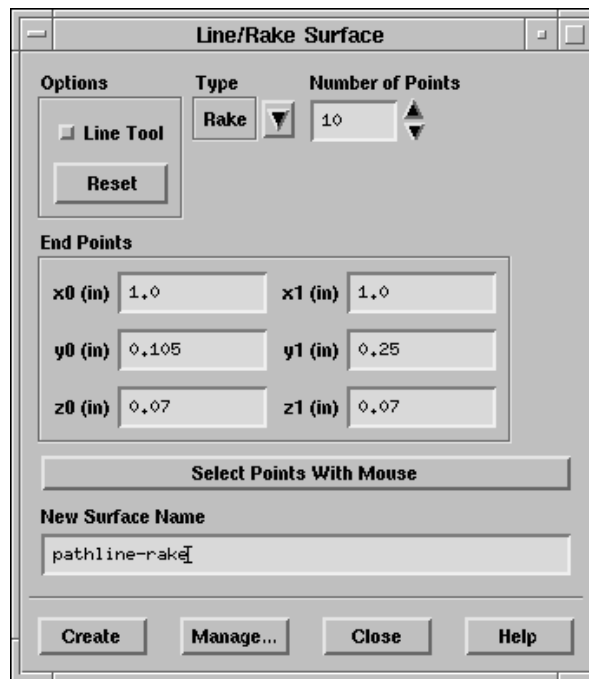
(a) In the Type drop-down list, select Rake.

   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 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) Keep the default of 10 for the Number of Points along the rake.

   This will generate 10 pathlines.

(c) Under End Points, enter the coordinates of the line, using a starting coordinate of (1.0, 0.105, 0.07) and an ending coordinate of (1.0, 0.25, 0.07), as shown in the panel above.

   This will define a vertical line in front of the module, about halfway between the centerline and edge.
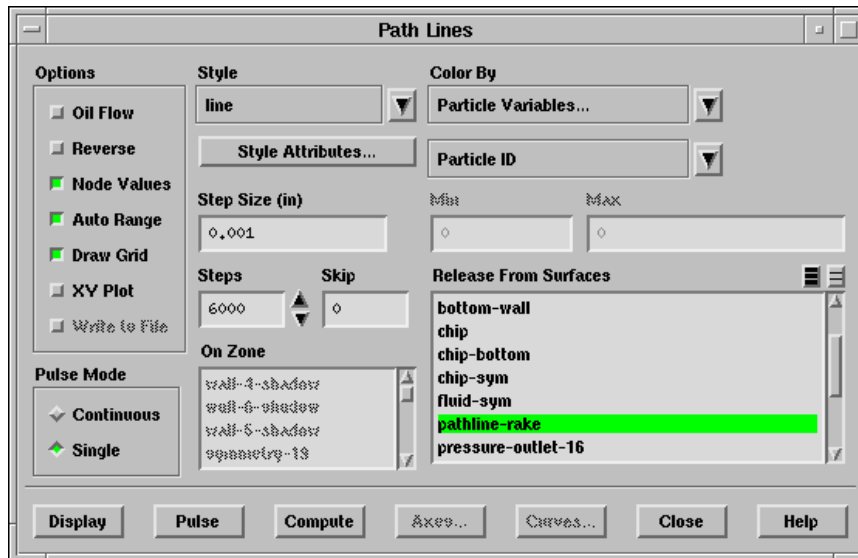
(d) Enter `pathline-rake` for the New Surface Name.

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

(e) Click Create, and Close the panel.

2. Draw the pathlines (Figure 22.14).

Display $\longrightarrow$ Path Lines...

(a) In the Release From Surfaces list, select pathline-rake.

(b) Set the Step Size to 0.001 inch and the number of Steps to 6000.

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.*

(c) Under Options, select Draw Grid.

This will open the Grid Display panel.

(d) In the Surfaces list, select board-top and chip.

These surfaces should already be selected from the earlier exercise where the grid was displayed with velocity vectors, Step 5: Velocity Vectors.

(e) Under Options, check that Faces is selected, and then Close the panel.

(f) In the Path Lines panel, click Display.

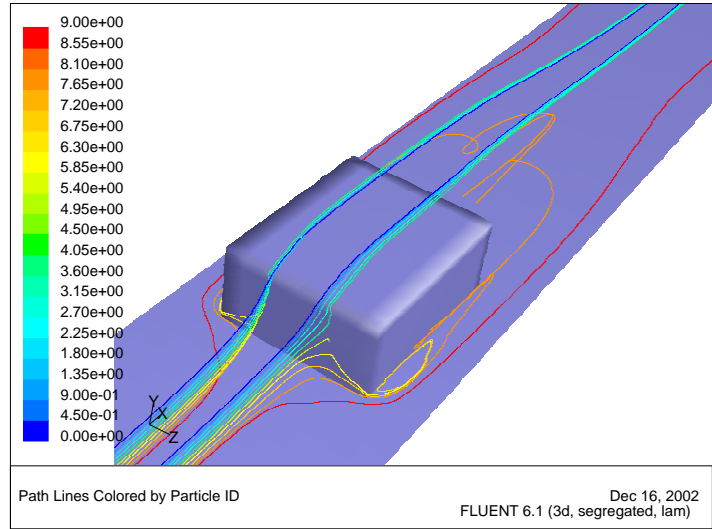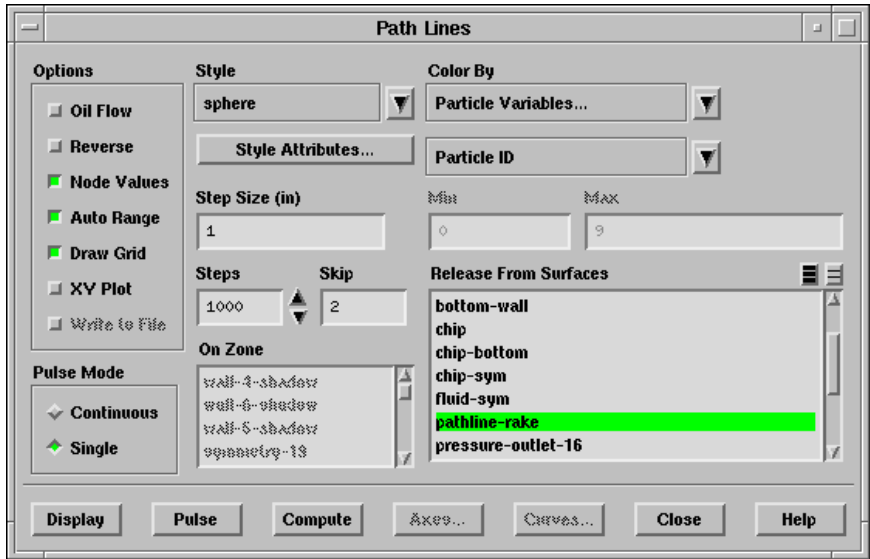*The pathlines will be drawn on the surface.*

Figure 22.14: Pathlines Shown on a Display of the Chip and Board Surfaces.

(g) Rotate the display so that the flow field in front and in the wake of the chip is visible, as shown in Figure 22.14.

3. Display pathlines as spheres.

Display ⟶Path Lines...

(a) In the Release From Surfaces list, keep the selection of pathline-rake.

(b) Select sphere from the Style drop-down list.

(c) Click the Style Attributes button.

This displays the Path Style Attributes *panel.*

i. In the Path Style Attributes panel, set the Diameter to 0.0005.

ii. Click OK to dismiss the Path Style Attributes panel with the new diameter setting.

(d) Set the Step Size to 1 inch and the number of Steps to 1000.

(e) Set Skip to 2.

(f) In the Path Lines panel, click Display.

The spherical pathlines will be drawn along the surface.

(g) Rotate the display so that the flow field in front and in the wake of the chip is visible, as shown in Figure 22.15.

Note: *Path lines can be colored by the surface they are released from using the* Color By *drop-down lists. Select* Particle Variables... *and* Surface ID *and then select* Display *in the* Path Lines *panel. An example of coloring path lines based on the surface can be seen in Figure 22.16.*
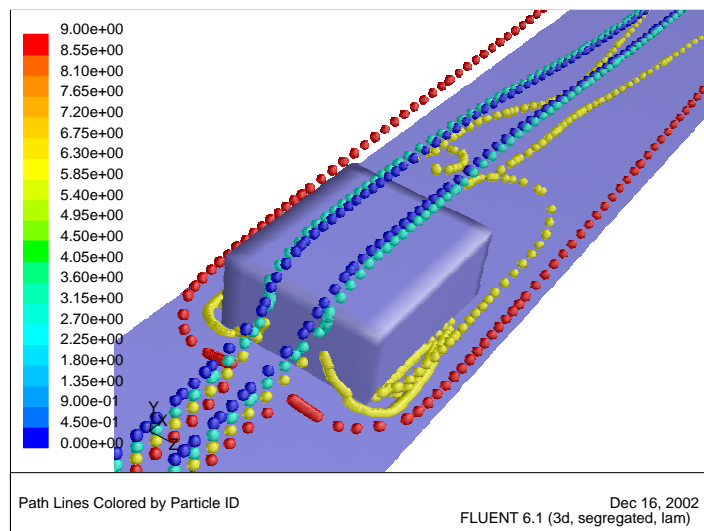
Figure 22.15: Sphere Pathlines Shown on a Display of the Chip and Board Surfaces.
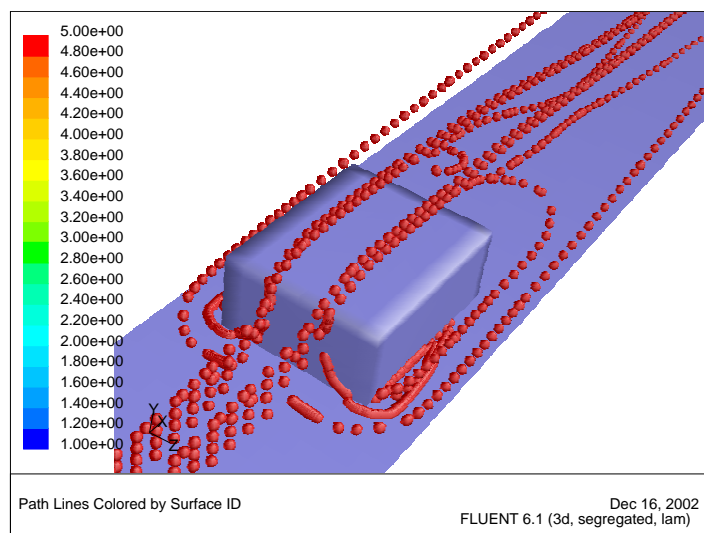
Figure 22.16: Sphere Pathlines Colored by Surface ID

Step 8: Overlaying Velocity Vectors on the Pathline Display

The overlay capability, provided in the Scene Description *panel, 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.

   Display ⟶Scene...
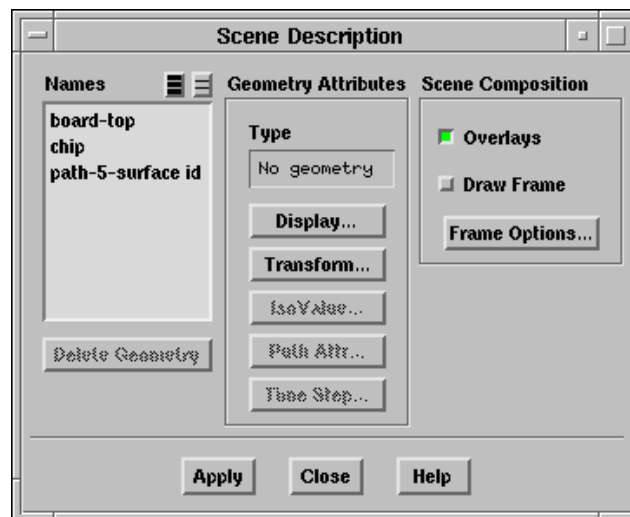
   (a) Under Scene Composition, select Overlays.

   (b) Click Apply.

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

   $\boxed{\text{Display}}$ $\longrightarrow$ Vectors...
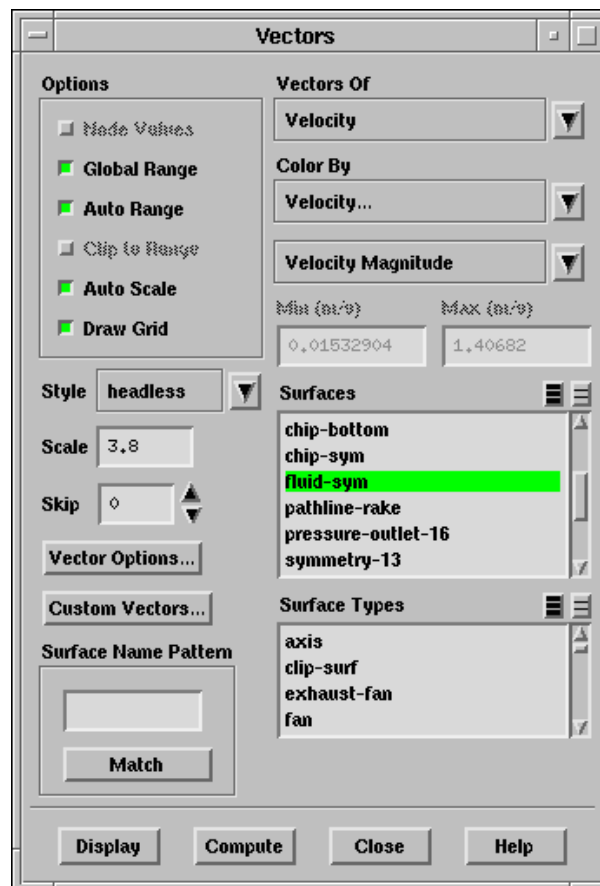
   (a) Under **Options**, deselect **Draw Grid**.

   (b) Deselect all surfaces by clicking the unshaded icon to the right of **Surfaces**.

   (c) In the **Surfaces** list, select **fluid-sym**.

   (d) Set the **Scale** to `3.8`.

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

   (e) Click **Display**.

   (f) Use your mouse to obtain the view that is shown in Figure 22.17.

Note: *The final display (Figure 22.17) does not require mirroring about the symmetry plane because the vectors obscure the mirrored image. You may turn off the mirroring option in the* Views *panel at any stage during this exercise.*
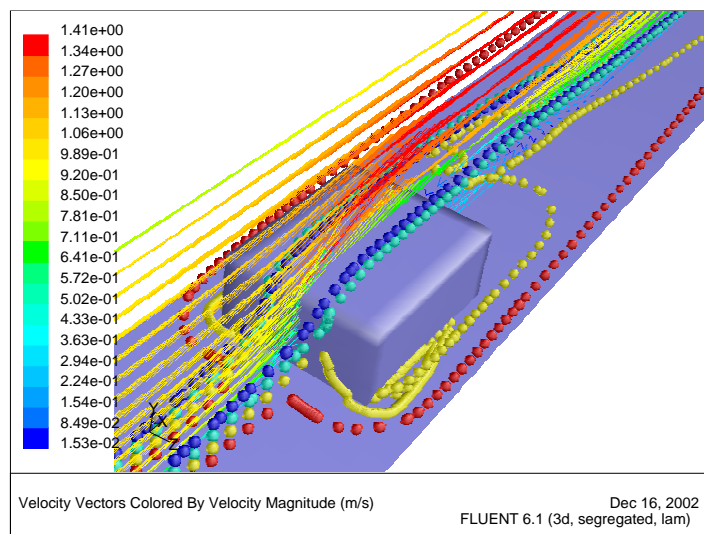
Figure 22.17: Overlay of Velocity Vectors and Pathlines Display

Step 9: Exploded Views

The Scene Description *panel 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. Below, 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.

   Display ⟶Scene...
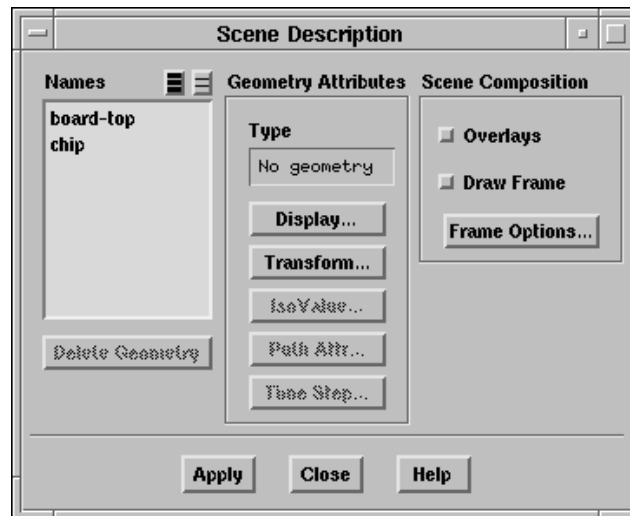
   (a) In the Names list, select the velocity vectors and pathlines.

   (b) Click Delete Geometry.

   (c) Click Apply.

   The Scene Description *panel should then contain only the two grid surfaces (*board-top *and* chip*).*

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

   Surface ⟶Iso-Surface...
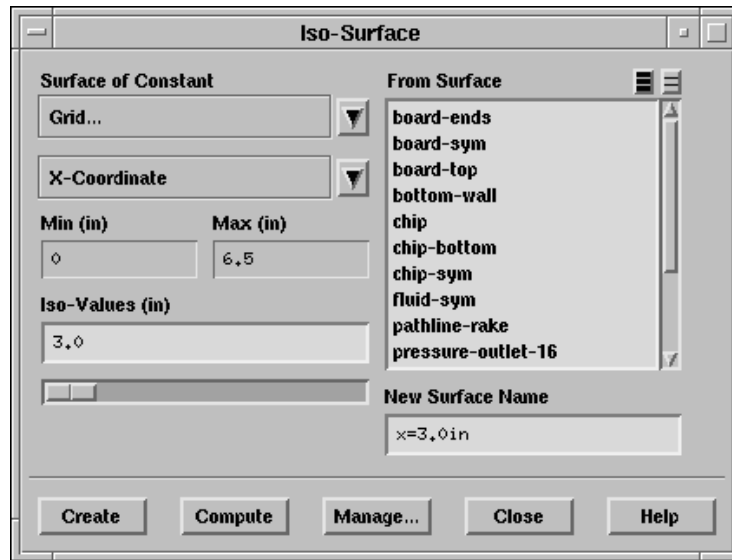
Hint: *If you forget how to create an isosurface, see Step 3: Isosurface Creation.*

3. Add the display of filled temperature contours on the x=3.0in surface.

   Display ⟶Contours...

(a) Under Options, deselect Draw Grid.

(b) Deselect all surfaces by clicking on the unshaded icon to the right of Surfaces.

(c) In the **Surfaces** list, select x=3.0in.

(d) Click **Display**, and **Close** the panel.

The filled temperature contours will be displayed on the x=3.0 in. surface.

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

$\boxed{\text{Display}}$ ⟶Vectors...

```
┌─────────────────────────────────────────────────────┐
│  ─                    Vectors                  ▫  □  │
│ ┌───────────────────────────────────────────────────┐
│  Options              Vectors Of                     │
│  ┌──────────────────┐ ┌──────────────────────────┐  │
│  │ ▢ Node Values    │  Velocity                 ▼  │
│  │ ▣ Global Range   │ Color By                     │
│  │ ▣ Auto Range     │  Velocity...              ▼  │
│  │ ▢ Clip to Range  │                              │
│  │ ▣ Auto Scale     │  Velocity Magnitude       ▼  │
│  │ ▢ Draw Grid      │  Min (in/s)      Max (in/s)  │
│  └──────────────────┘  0.01532904     1.40682      │
│                                                      │
│  Style  headless  ▼   Surfaces          ☰ ☰        │
│                        symmetry-13                   │
│  Scale  1.9            symmetry-19                   │
│                        top-wall                      │
│  Skip   2   ▲▼         velocity-inlet-17             │
│                        x=3.0in                       │
│  Vector Options...     y=0.25in                      │
│                                                      │
│  Custom Vectors...    Surface Types     ☰ ☰        │
│  Surface Name Pattern  axis                          │
│                        clip-surf                     │
│  ┌──────────────┐      exhaust-fan                   │
│  └──────────────┘      fan                           │
│       Match                                          │
└───────────────────────────────────────────────────┘
│   Display    Compute     Close      Help            │
└─────────────────────────────────────────────────────┘
```
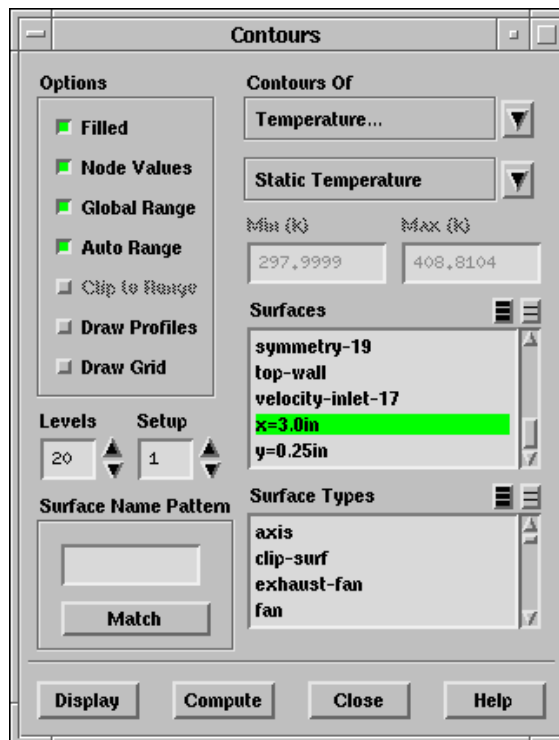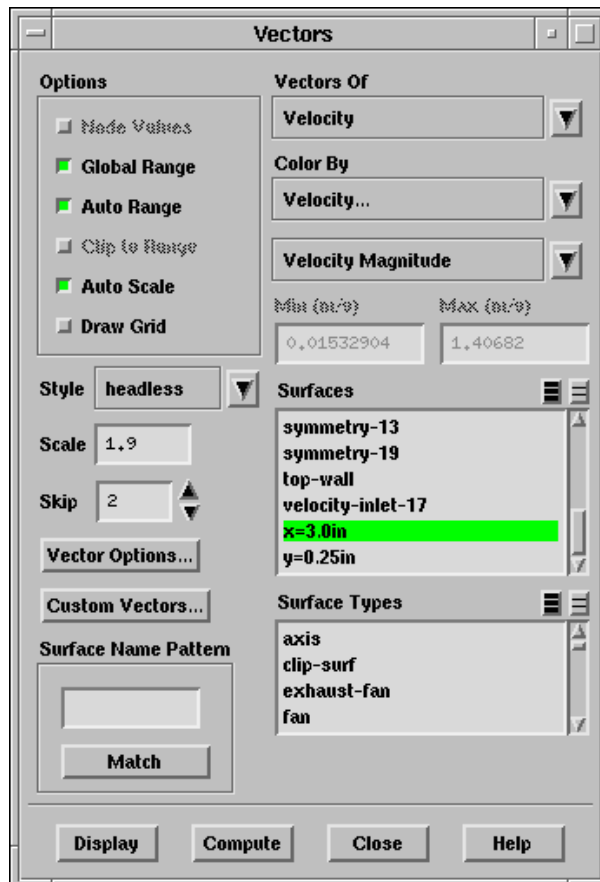
(a) Under **Options**, deselect **Draw Grid**.

(b) Deselect all surfaces by clicking on the unshaded icon to the right of **Surfaces**.

(c) In the **Surfaces** list, select x=3.0in.

(d) Increase the **Skip** to 2.

(e) Change the **Scale** to 1.9.

(f) Click **Display**.

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

5. Create the exploded view (Figure 22.18) by translating the contour display, placing it above the vectors.

   Display ⟶ Scene...

   (a) In the Scene Description panel, select contour-6-temperature in the Names list.

   (b) Click Transform....

   This will open the Transformations *panel.*
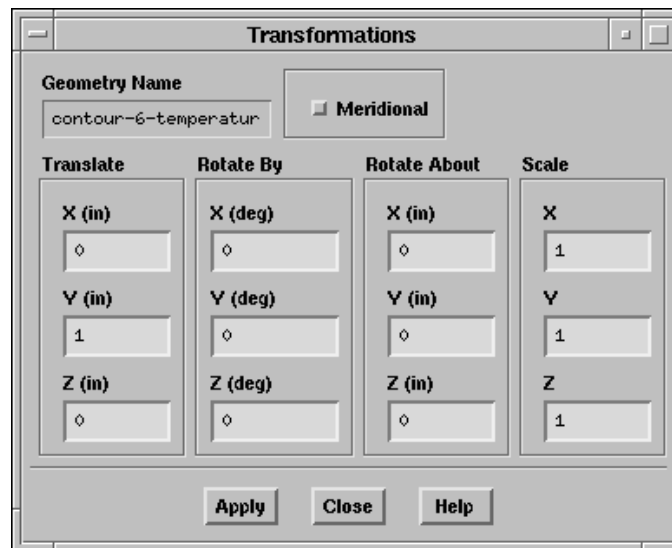
   (c) Under Translate, enter 1 inch for Y.

   (d) Click Apply, and Close the Transformations panel.

   The exploded view allows you to see the contours and vectors as distinct displays in the final scene (Figure 22.18).

6. Turn off the Overlays.

   (a) In the Scene Description panel, deselect the Overlays option.
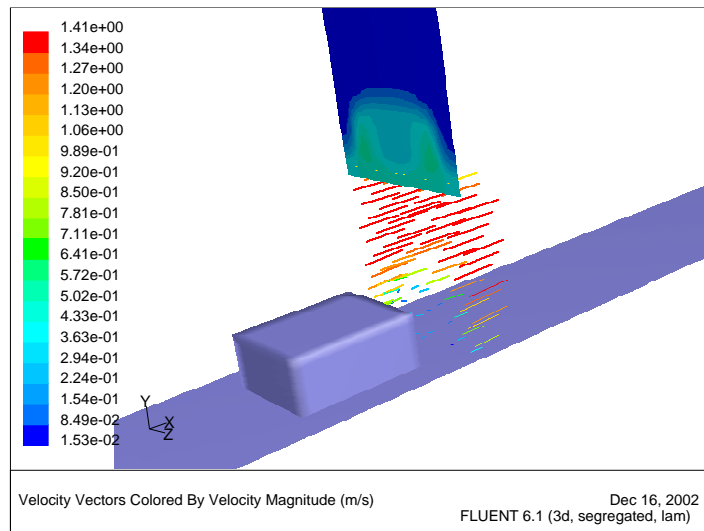
   (b) Click Apply, and Close the panel.

Figure 22.18: Exploded Scene Display of Temperature and Velocity

Step 10: Animating the Display of Results in Successive Streamwise Planes

Often, 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 *panels, here you will use the* Sweep Surface *panel 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 of constant x coordinate.*

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

   Display $\longrightarrow$Scene...

   (a) In the Scene Description panel, select contour-6-temperature and vv-6-velocity-magnitude in the Names list.

   (b) Click Delete Geometry.

   (c) Click Apply, and Close the panel.

   *The panel and display window will be updated to contain only the grid surfaces.*

2. Use your mouse to un-zoom 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.

   Display $\longrightarrow$Sweep Surface...
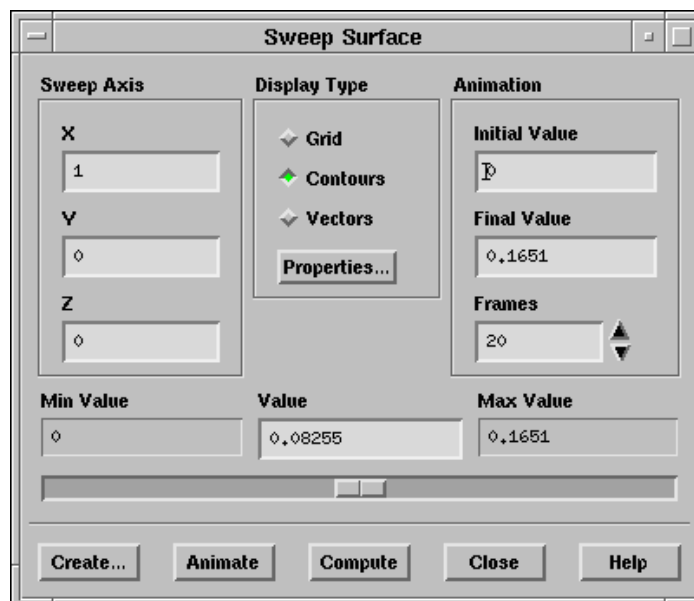
(a) Keep the default Sweep Axis (the x axis).

(b) Under Animation, set the Initial Value to 0 m and the Final Value to 0.1651 m.

> **!** *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) Set the number of Frames to 20.

(d) Select Contours under Display Type.

This will open the Contours panel.
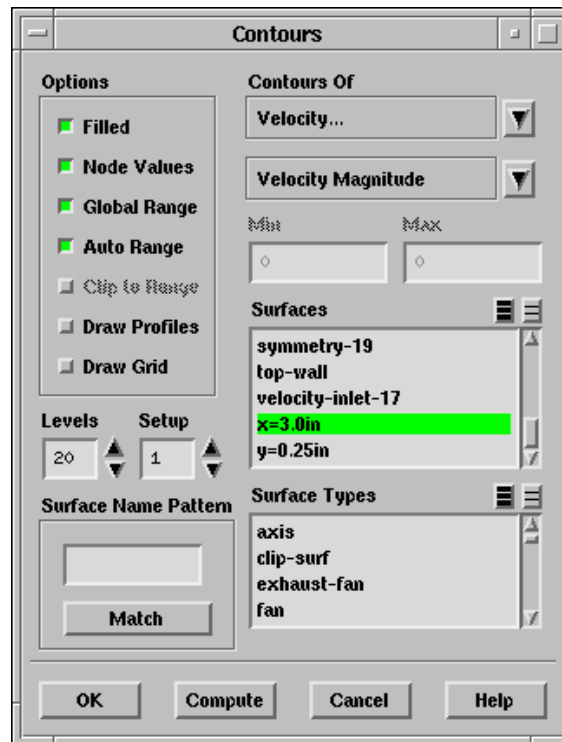
i. In the Contours panel, select Velocity... and Velocity Magnitude in the Contours Of drop-down lists.

ii. In the Contours panel, click OK.

(e) Click on Animate in the Sweep Surface panel.

You will see the velocity contour plot displayed at 20 successive streamwise planes. FLUENT automatically interpolates 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 FLUENT's animation tools for transient cases as demonstrated in Tutorial 4.*

Step 11: XY Plots

XY plotting can be used to display quantitative results of your CFD simulations. Here, you will complete your 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...
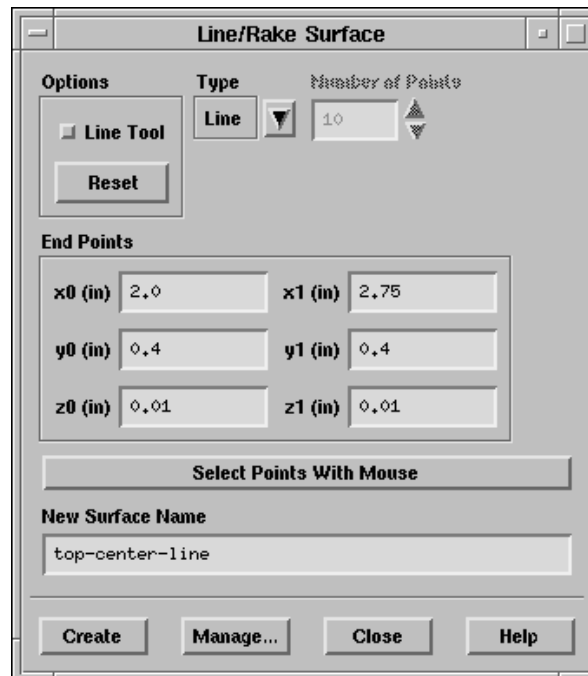
   (a) In the Type drop-down list, select Line.

   (b) Under End Points, 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), as shown in the panel above.

   *These coordinates define the top centerline of the module.*

   (c) Enter top-center-line as the New Surface Name.

   (d) Click Create.

2. Plot the temperature distribution along the top centerline of the module (Figure 22.19).

    Plot ⟶XY Plot...
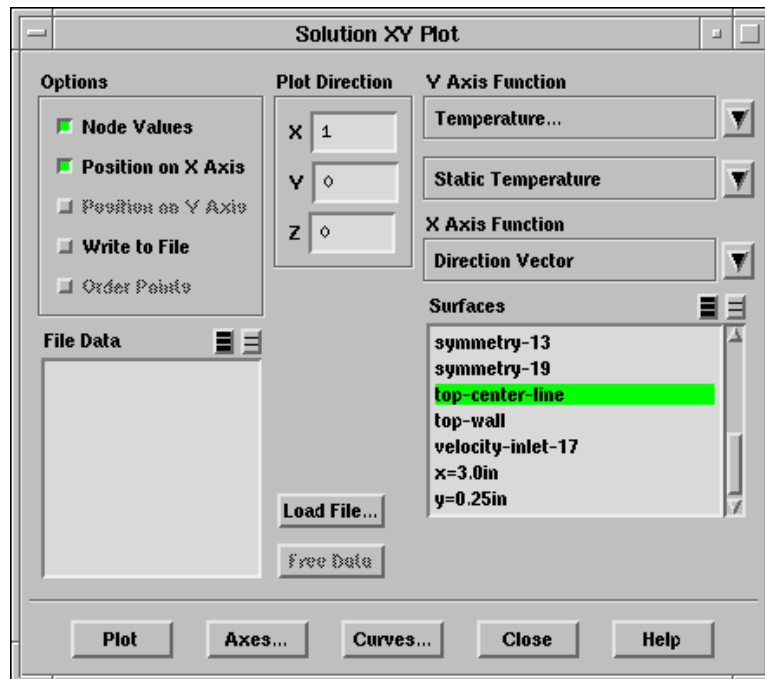
(a) Select Temperature... and Static Temperature in the Y Axis Function drop-down lists.

(b) In the Surfaces list, select top-center-line.

(c) Keep the default Plot Direction of X.

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

(d) Click Axes... to modify the axis range.

    This will open the Axes - Solution XY Plot *panel .*
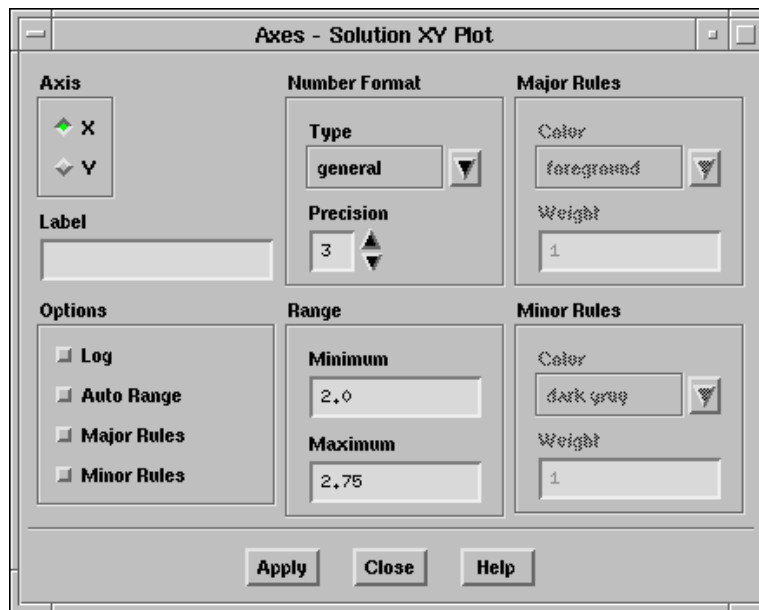
(e) Under Axis, select X.

(f) Under Options, deselect Auto Range.

(g) Set the Range using a Minimum of 2.0 and a Maximum of 2.75.

(h) Click Apply, and Close the panel.

(i) In the Solution XY Plot panel, click Plot.

   The temperature distribution (Figure 22.19) shows the temperature increase across the module surface as the thermal boundary layer develops in the cooling air flow.
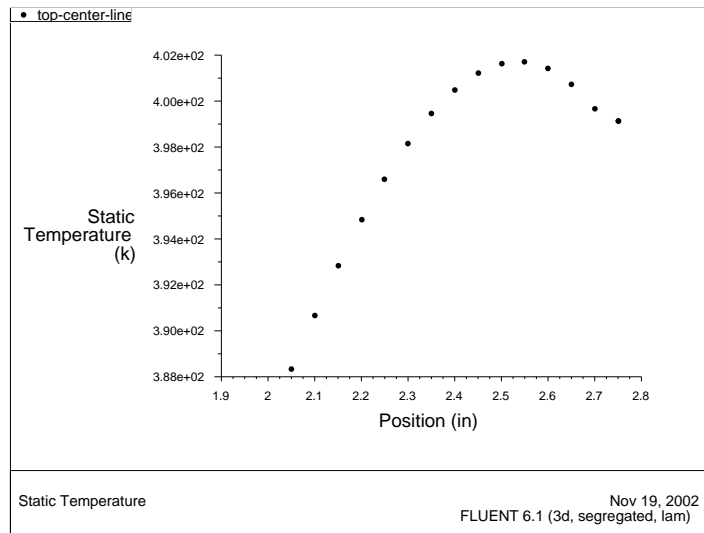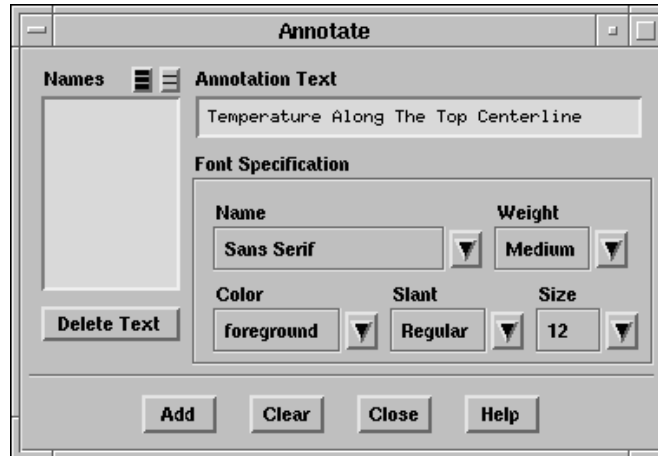
Figure 22.19: Temperature Along the Top Centerline of the Module

Step 12: Annotation

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

$\boxed{\text{Display}}\longrightarrow$Annotate...

1. In the Annotation Text field, enter the text describing your plot (e.g., `Temperature Along the Top Centerline`).

2. Click Add.

   A Working *dialog box will appear telling you to select the desired location of the text using the mouse-probe button, which is, by default, the right button.*

3. Click your 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 22.20).

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

   Note: *Depending on the size of your 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.*
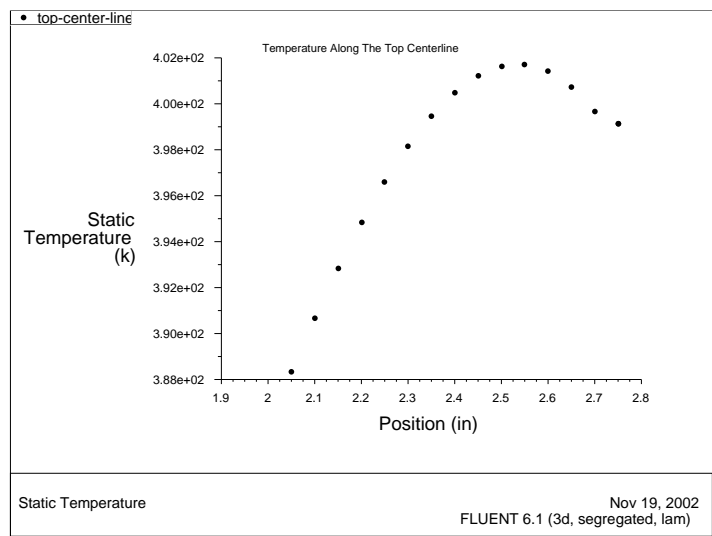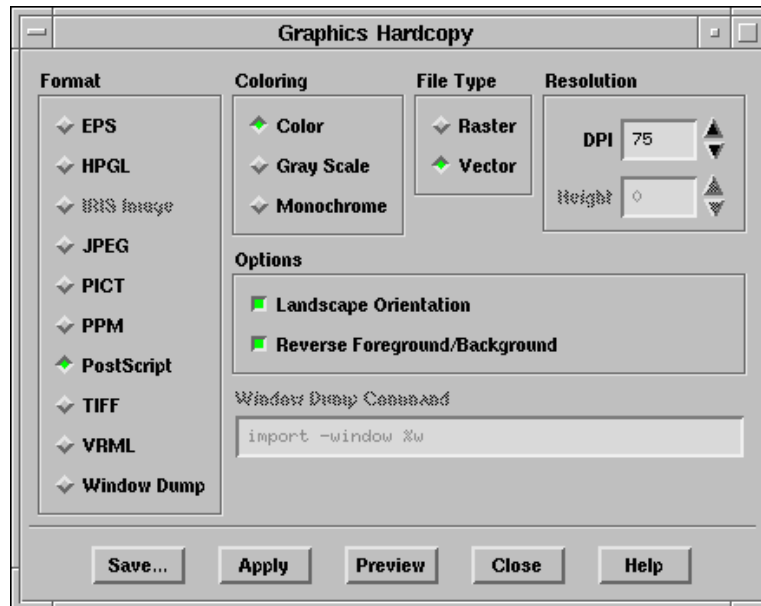
Figure 22.20: Temperature Along the Top Centerline of the Module

Step 13: Saving Hardcopy Files

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

File ⟶ Hardcopy...

1. Under Format, select PostScript.

2. Under Coloring, select Color.

3. Click Save....

   This will open the Select File *dialog box.*

4. In the Select File dialog box, enter a name for the hardcopy file.

Summary: This tutorial has demonstrated the use of many of the extensive postprocessing features available in FLUENT. For more information on these and related features, see Chapter 27 and Chapter 28 in the User's Guide.