Tecplot®

Tecplot®

User's Manual

Version 10

Tecplot, Inc.
Bellevue, Washington
October, 2004

Contents

CHAPTER 1 *Getting Started*

Tecplot is a powerful tool for visualizing a wide range of technical data. It offers line plotting, 2- and 3-D surface plots in a variety of formats, and 3-D volumetric visualization. This chapter describes Tecplot's point-and-click interface.

1.1. Start-Up

The following sections describe how to start Tecplot on Windows or UNIX systems.

1.1.1. Windows

On Windows start Tecplot from the Start button, or from an icon on your desktop. To start Tecplot from the Start button:

1. Click Start, then select Programs.
2. Select the Tecplot 10 folder.
3. Click on Tecplot.

Following the opening banner, Tecplot appears.

1.1.2. UNIX

On UNIX systems, Tecplot is typically installed by a system administrator, who makes it available to users. Run Tecplot by typing:

```
tecplot
```

at the shell prompt. The opening banner appears, followed by the Tecplot window, as shown in Figure 1-1.

The directory in which Tecplot is installed, on any platform, is called the Tecplot home directory. You should know the absolute path of this directory and set your **`TEC100HOME`** environment variable to point to it. The Tecplot home directory includes numerous example files referred to throughout this manual; by working with these files you can quickly gain proficiency with Tecplot's features.

1.2. Interface

There are four main regions in the Tecplot window—menu bar, sidebar, workspace, and status line.

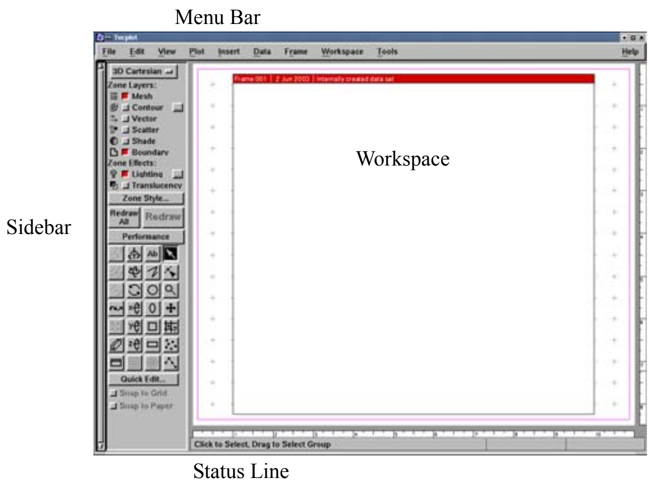


Figure 1-1. Tecplot in Motif showing the four main regions—menu bar, sidebar, workspace, status line.

1.2.1. Menu Bar

The menu bar, shown in Figure 1-2, offers rapid access to most of Tecplot's features, which are controlled primarily through dialogs.

Figure 1-2. Tecplot's menu bar.

Tecplot's features are organized into the following menus:

- **File:** Controls reading and writing of data files and plot layouts, printing and exporting of plots, recording and playing macros, setting and saving configuration preferences, and exiting Tecplot.
- **Edit:** Controls Undo functions, cutting, copying, pasting, and clearing objects, as well as pushing and popping (changing the draw order for selected items). The Edit menu also contains an option for adjusting data points.

Tecplot's Cut, Copy, and Paste options work only within Tecplot. If you are operating in Windows and want to place a graphics image of your layout into other word processing or design program, use Copy Plot to Clipboard.

- **View:** Controls the point of view of your data, including scale, view range, and 3-D rotation. You can also use the View menu to copy and paste views between frames.

  The View menu includes sizing options for convenience. Center moves the plot image so that the data points are centered within the frame. (Only the data is centered; text, geometries, and the 3-D axes are not considered.) Fit to Full Size fits the entire plot into the frame. Nice Fit to Full Size sets the axis range to begin and end on major axis increments (if axes are dependent Tecplot adjusts the vertical axis length to accommodate a major tick mark). Make Current View Nice modifies the range on a specified axis to fit the minimum and maximum of the variable assigned to that axis, then snaps the major tick marks to the ends of the axis. (If axis dependency is not independent this may affect the range on another axis.) Data Fit fits the data points to the frame.
- **Plot:** Dynamic menus feature different controls depending upon the plot type (3D Cartesian, XY Line, Polar Line, and so forth) selected from the sidebar. Plot menu options give you control over the style of your plots.
- **Insert:** Used to add text, geometries (polylines, circles, squares, ellipses, and rectangles), or image files.
- **Data:** Create, manipulate, and examine data. Types of data manipulation available in Tecplot include zone creation, interpolation, triangulation, as well as creation or alteration of variables by means of FORTRAN-like equations.
- **Frame:** Create, edit, and control frames.
- **Workspace:** Controls the attributes of your workspace, including the color map, paper grid, display options, and rulers.
- **Tools:** Run Quick Macros you may have defined, or to create simple animations of your plots. Add-ons—other than data loaders and extended curve fits—are accessed through the Tools menu.
- **Help:** Get quick help on features. By selecting About Tecplot. you can obtain specific information about your license. The Help menu also accesses information about add-ons you have loaded.

1.2.2. Sidebar

Tecplot's sidebar accesses the most frequently used controls for plotting. Many take the form tools, which control the behavior of the pointer. Additional controls determine the plot type, which zone or map layers are active, and snap modes. The controls are organized in the following functional clusters, as shown in Figure 1-3:

- **Plot Types.**
- **Zone/Map Layers.**

- **Zone Effects.**
- **Zone/Map Style.**
- **Redraw All/Redraw.**
- **Performance.**
- **Tools.**
- **Quick Edit/Object Details.**
- **Snap Modes.**

1.2.2.1. Plot Types. Plot type, combined with a frame's data set, active layers and their associated attributes, define a plot. Each plot type represents one view of the data. Plot type determines what type of plot can be drawn in the current frame. Five plot types are available:

- **3D Cartesian:** Create 3-D plots of surfaces and volumes.
- **2D Cartesian:** Create 2-D plots, such as plots of a variable located on a plane.
- **XY Line:** Create XY Line plots, such as plots of independent versus dependent variables.
- **Polar Line:** Create a Polar Line plot of radius versus angle, or vice versa. The polar axes are the radial axis (by default zero at the origin) and theta axis (by default zero for any data on the right running horizontal line).
- **S (Sketch):** Create plots without data such as drawings, flow charts, and viewgraphs.

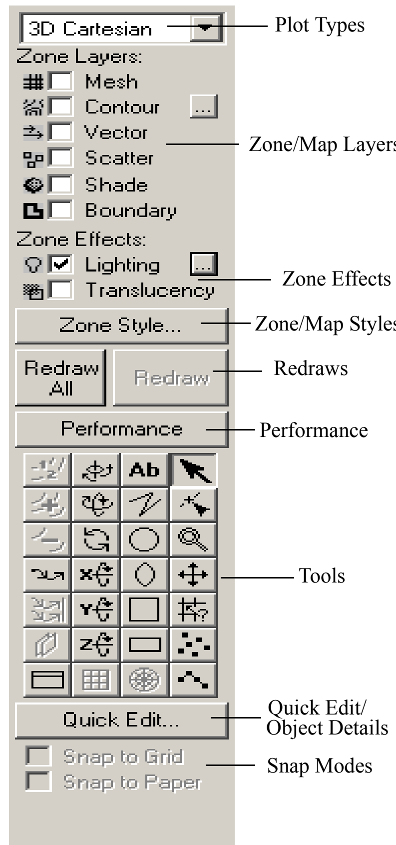


Figure 1-3. Tecplot's sidebar.

1.2.2.2. Zone Layers/Map Layers. A layer is a way of representing a frame's data set. The complete plot is the sum of all the active layers, axes, text, geometries, and other elements added to the data plotted in the layers. There are six zone layers for 2- and 3-D Cartesian, four map layers for XY Line, two for Polar Line, and none for Sketch.

The six zone layers for 2- and 3-D Cartesian plot types, shown in Figure 1-3, are:

- **Mesh:** Lines connecting the data points within each zone.
- **Contour:** Lines having a constant value, the region between these lines, or both.

- **Vector:** The direction and magnitude of vector quantities.
- **Scatter:** Symbols at the location of each data point.
- **Shade:** Used to tint each zone with a specified solid color, or to a add light-source to a 3-D surface plot. Used in conjunction with the Lighting zone effect you may set Paneled or Gouraud shading. Used in conjunction with the Translucency zone effect you may create a translucent surface for your plot.
- **Boundary:** Zone boundaries for ordered data.

The four XY Line map layers, shown in Figure 1-4, are:

Figure 1-4. Map layers for XY Line.

- **Lines:** Plots a pair of variables, X and Y, as a set of line segments or a fitted curve.
- **Symbols:** A pair of variables, X and Y, as individual data points represented by a symbol you specify.
- **Bars:** A pair of variables, X and Y, as a horizontal or vertical bar chart.
- **Error Bars:** Available in several formats.

The two map layers for Polar Line, shown in Figure 1-5, are:

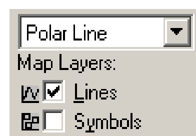


Figure 1-5. Map layers for Polar Line.

- **Lines:** A pair of variables, X and Y, as a set of line segments or a fitted curve.
- **Symbols:** A pair of variables, X and Y, as individual data points represented by a symbol you specify.

1.2.2.3. Zone Effects. For 3-D Cartesian plot types the check boxes shown in Figure 1-6 appear—Lighting; Translucency. Only shaded and flooded contour surface plot types are affected.

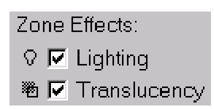


Figure 1-6. Zone Effects.

1.2.2.4. Zone Style/Mapping Style. Call up the appropriate dialog, allowing you to modify the appearance of each zone or map.

1.2.2.5. Redraw Buttons. Allow you to keep your plot up to date: Redraw All redraws all frames (Shift-Redraw All causes Tecplot to completely regenerate the workspace); Redraw redraws only the current frame.

1.2.2.6. Performance. Calls up a quick access menu of enhancements to speed plotting. Selecting Options calls up the Display Performance dialog, where you may configure Tecplot's status line, tool tips, graphics cache, image export, and performance options. For further details, see Section 30.3, “Display Performance Dialog.”

1.2.2.7. Tool Buttons. Each of the tools represented by a button is a mouse mode, specifying the behavior of the pointer anywhere in the workspace. There are 28 modes in 12 categories, as shown in Figure 1-7.

- **Contour mouse modes.**
- **Streamtrace mouse modes.**
- **Slicing mouse mode.**
- **Frame mouse mode.**
- **Zone creation mouse modes.**
- **3-D rotation mouse modes.**
- **Text mouse mode.**
- **Geometry mouse modes.**
- **Mouse pointer modes:** Selector and Adjustor.
- **View mouse modes:** Zoom and Translate/Magnify.
- **Probe mouse mode.**
- **Data extraction mouse modes.**

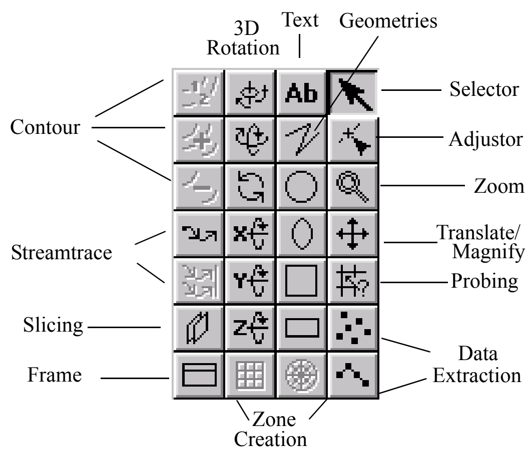


Figure 1-7. Sidebar tools and mouse modes.

1.2.2.8. Enhanced Tool Operations. Several sidebar tools offer mouse and keyboard shortcuts which can greatly speed Tecplot use, especially when working with large amounts of data. These are:

- **Contour tools:**

  +: Switch to Contour Add tool if you are using Contour Remove.

  - : Switch to Contour Remove tool if you are using Contour Add.

- **Contour Add tool:**

  Click: Place a contour line.

  Ctrl-click: Replace the nearest contour line with a new line.

  Drag: Move the new contour line.

- **Streamtrace Placement tool:**

  D: Switch to streamrods.

  R: Switch to streamribbons.

  S: Switch to surface lines.

  V: Switch to volume lines.

Alt-click/Alt-drag: Determine the XYZ-location by ignoring zones and looking only at derived volume objects (streamtraces, iso-surfaces, slices).

1-9: Change the number of streamtraces to be added when placing a rake of streamtraces.

- **Slicing tool:**

  Click: Place a start slice.

  Drag: Move the start slice.

  Alt-click/Alt-drag: Determine the XYZ-location by ignoring zones and looking only at derived volume objects (streamtraces, iso-surfaces, slices).

  Shift-click: Place the end slice.

  Shift-drag: Move the end slice.

  +: Turns on the start slice if no slices are active, or turns on the end slice if slices are already active.

  - : Turns off the end slice if the end slice is active, or conversely, turns off the start slice if the end slice is not active.

  I, J, K (ordered zones only): Switch to slicing constant I-, J-, or K-planes respectively.

  X, Y, Z: Switch to slicing constant X-, Y-, or Z-planes respectively.

  0-9: Numbers one through nine activate intermediate slices and set the number of intermediate slices to the number entered; zero turns off intermediate slices.

- **Zoom tool:**

  Click: Center a 200 percent magnification around the location of your click.

1.2.2.9. Mouse Operation Enhancements. The middle and right mouse buttons allow you to smoothly zoom and translate your data. Your middle mouse button (or Ctrl-right click) zooms smoothly, and your right mouse button translates data.

This advanced functionality is available in: all contour modes; streamline placement; slicing; 3-D rotation modes; geometry modes (except polyline); zoom; translate/magnify; probe; zone creation.

1.2.2.10. Details Button. Immediately under the sidebar tools is a single button with a context-sensitive label, referred to as the Details button. Use this button to call up the dialog most directly applicable to your current action. When the currently selected tool is either the Selector  or the Adjustor  , but no objects are selected in the workspace, the Details button is labeled Quick Edit. When either of those tools is selected and one or more objects are selected in the workspace, the label changes to Object Details. If any other tool is selected, the label changes to read Tool Details.

1.2.2.11. Quick Edit. Calls up the Quick Edit dialog, where you can make rapid changes to selected objects in the workspace.

1.2.2.12. Snap Modes. Allow you to place objects precisely by locking them to the nearest reference point, either on the axis grid or on the workspace paper.

1.2.3. Status Line

The status line, running along the bottom of the Tecplot window, gives “hover help.” When you move the pointer over a sidebar tool, a button on the Quick Edit dialog, or a menu item, a description of the control appears. Brief instructions are displayed for tools.

Configuration of the status line can be changed by selecting Display Performance from the Workspace menu, or by clicking Performance on the sidebar and selecting Options from the sub-menu.

1.2.4. Tecplot Workspace

The workspace is the portion of your screen in which you create sketches and plots. This is done within a frame, which can be manipulated much like a process window. The current state of the workspace, including the sizing and positioning of frames, the location of the data files used by each frame, and all current attributes for all frames, make up a layout. By default, the workspace displays a representation of the paper Tecplot is set up to draw on, as well as a reference grid and rulers. The active frame, in which you are currently working, is on top. All modifications are made to the current frame.

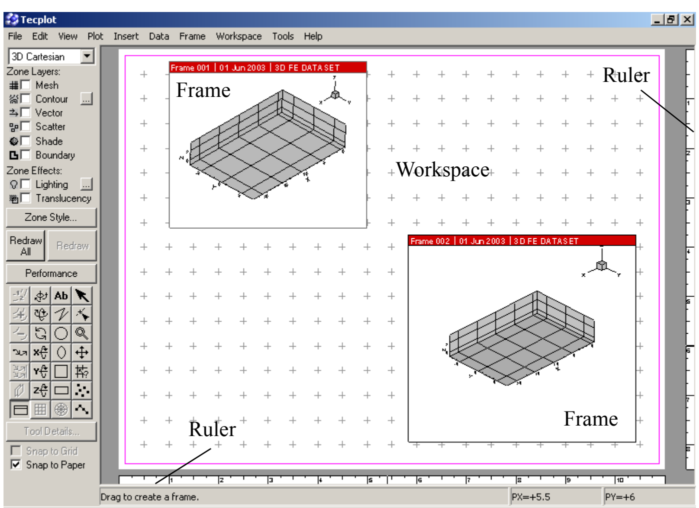


Figure 1-8. The Tecplot workspace.

1.2.5. File Dialogs

Each type of Tecplot file has at least two dialogs associated with it—one for opening files and one for saving files. These dialogs are similar, but differ when using Motif or Windows.

1.2.5.1. File Dialogs in Motif. Figure 1-9 shows the Open Layout dialog for Motif. Near the bottom of the dialog is the Selection text field. If you know the complete path of the file you want to open or save, type it into this field and click OK.

At the top of the dialog is the Filter (Name Search) text field. Use this field to specify a file name filter; Tecplot to displays all sub-directories of the current directory in the Directories scrolled list, as well as all files in the current directory ending with the extension `.lay` in the Files scrolled list. The filter determines the initial path displayed in the Selection text field. To change the default file extensions, see Section 30.1.4, "Default File Name Extensions."

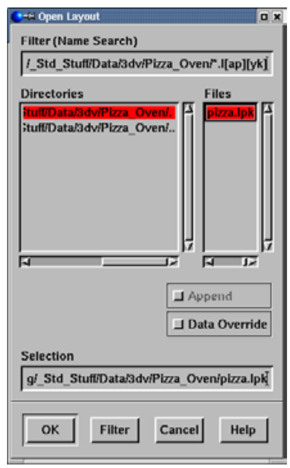


Figure 1-9. The Open Layout dialog for Motif.

1.2.5.2. File Dialogs in Windows. Figure 1-10 shows the Open Layout dialog for Windows. In the lower half of the dialog is the File name text field. If you know the complete path of the file you want to open or save, you can type it into this field, then click Open. You can also use this field to specify a file name filter. By default, the Open Layout dialog has a file name filter of `*.lay` and `*.lpk`. To change the default file extensions, see Section 30.1.4, "Default File Name Extensions."

1.2.6. Operations

The operation of Tecplot controls are familiar to anyone who has used Motif or Windows. Most actions are performed by clicking the left mouse button. (If your mouse is configured for left-hand use, click the right mouse button.)

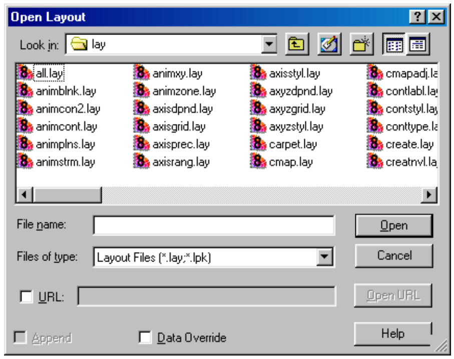


Figure 1-10. The Open Layout dialog for Windows.

Another common mouse action is dragging. Dragging is used in resizing frames, creating and modifying geometries, and to alter or adjust data. Using your keyboard in conjunction with clicking and dragging the mouse produces different actions. For example, Tecplot makes extensive use of Ctrl-click (clicking the mouse while holding down the Ctrl key) in its probing feature.

The terms click, select, and choose are sometimes used interchangeably. It is useful, however, to keep in mind that select in general means to "select an item to operate on," while choose in general means to "pick an action."

To select an object in the workspace, click on it. To select an object and call up the dialog used to modify the object, double-click on it. Double-clicking on a piece of text calls up the Text dialog, where you can edit or reformat the text. Select an object, then click on Details on the sidebar for the same effect. You can select groups of items, then act on them all at once.

After you have selected a group of items and released the mouse key, Tecplot calls up the Group Select dialog. Select the objects you want using the appropriate check boxes, as shown in Figure 1-11.

Figure 1-11. The Group Select dialog.

1.2.7. Object Positioning and Resizing

Selected objects such as frames, text, geometries, legends, and so forth, may be moved either by clicking and dragging, or by using the arrow keys on your keyboard. Arrow keys move objects in one pixel increments. For more information on moving and resizing frames, see Section 2.1.3, “Frame Positioning and Sizing.”

To scale selected objects proportionally, maintaining the vertical to horizontal aspect ratio, select the object, then press “+” on your keyboard to enlarge or “-” to reduce. Double-clicking a selected object calls up the appropriate style dialog. For example, if you double-click on a geometry, the Geometry dialog appears.

1.2.8. Quick Edit Dialog

The map and zone layer controls affecting how the individual layers are drawn can be altered using controls on the sidebar. You can also control many of these attributes using the Quick Edit dialog, shown in Figure 1-12.

To use the Quick Edit dialog, select one or more objects in the workspace, then click the appropriate button to change the attribute of the selected object(s).

Figure 1-12. The Quick Edit dialog.

1.3. Help

Tecplot features a fully integrated Help system. Quick help on menu items and sidebar controls is available from the status line or tool tips.

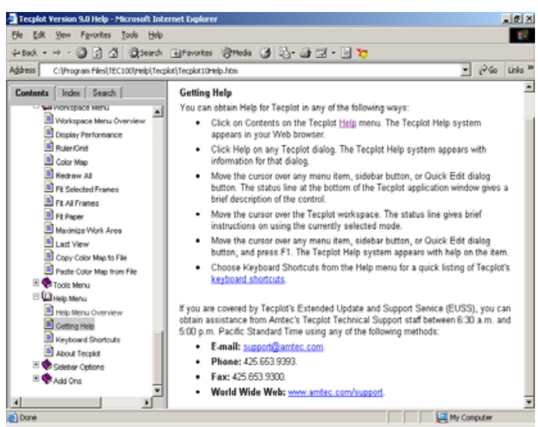


Figure 1-13. Tecplot Help in a Windows Web browser.

Detailed help is accessible by:

- Pressing the F1 key anywhere in the Tecplot window. If the pointer is over the sidebar, Quick Edit dialog, or a menu, the F1 key provides context-sensitive help on that control or menu. Otherwise, F1 calls up the Contents page of Help via your Web browser.
- Selecting Contents from the Help menu. This calls up the Contents page of the Tecplot help file via your Web browser.
- Clicking Help on any dialog.

Figure 1-13 shows Tecplot's Help as it appears in a Web browser in Windows. It supports text search, has many hypertext links, and provides detailed information on all menus and dialogs. Your answer may be in *Technical Support Notes* at *www.tecplot.com/support*. Help is also available from 6:30 A.M. to 5 P.M. Pacific Standard Time from Tecplot Technical Support at 425.653.9393. You may also send e-mail to *support@tecplot.com* with your questions.

CHAPTER 2 Frames and the Workspace

No matter which type of plot you want to create, certain operations occur repeatedly within Tecplot. Those operations concerning files are covered in Chapter 3, "Data Organization." Operations concerning software are covered here, including:

- **Working with frames**: Plots are created in a frame—a boxed area in the workspace acting as a sub-window. You control the format of each individual frame.
- **Managing your workspace**: Workspace and paper controls determine the color and orientation of your paper, as well as the ruler and grid, to precisely size and position objects.
- **Coordinate systems**: Learn when and where Tecplot uses different coordinate systems.
- **Plot view**: Zoom, translate, and fit plots within frames.
- **Cut, Copy, Paste**: Many plot elements may be cut or copied from the workspace and pasted back into other plot elements.

2.1. Frames

All plots and sketches are drawn within frames. By default, the Tecplot workspace contains one frame. You may add additional frames, up to 128. You may resize and reposition frames, modify background color, and specify border and header appearane. Tecplot acts upon only one frame—*the current frame*—at any given time, except when frames are linked.

2.1.1. Frame Creation

Create new frames interactively by drawing them in the workspace. If printing plots, draw frames within the paper displayed in the workspace. (However, this is not required.)

To create a new frame:

1. From the sidebar, select , or choose Create New Frame from the Frame menu.
2. Move the pointer into the workspace; it becomes a cross-hair. Move the cross-hair to the desired corner of a new frame, then click and drag. A rubber band box shows the outline of the frame.
3. When the box is the desired size and shape, release the mouse button.

2.1.2. Frame Deletion

Delete frames one at a time using Delete Current Frame from the Frame menu, or using Clear from the Edit menu.

To delete a group of frames:

1. Select a group of frames. The Group Select dialog appears.
2. In the Objects area of the dialog, deselect all check boxes except Frames. All the frames within the rubber band box are selected.
3. From the Edit menu, choose Clear. (In Motif, with the keyboard focused on Tecplot, type Delete.) A warning dialog appears asking if you want to delete the selected items.
4. Click OK to delete; Cancel to retain.

2.1.3. Frame Positioning and Sizing

You can size and position frames in four ways—with your mouse, with the arrow keys on your keyboard, by specifying exact coordinates using Edit Current Frame from the Frame menu, or by choosing Fit all Frames to Paper (also from the Frame menu).

2.1.3.1. Mouse Positioning and Sizing. If you click anywhere on a frame's header or border, handles appear at its corners and midpoints. Drag any of these to resize the frame. The handles on the top and bottom of the frame allow resizing vertically; the handles on the left and right of the frame allow resizing horizontally. The handles on the four corners allow simultaneous vertical and horizontal resizing. You can also obtain the resizing handles by selecting a group of frames. To scale frames proportionally, maintaining the vertical to horizontal aspect ratio, select the frames, then press "+" on your keyboard to enlarge or "-" to reduce.

2.1.3.2. Arrow Key Positioning and Sizing. After selecting frames, you may position them using the arrow keys on your keyboard. You can move frames up, down, left or right in one-pixel increments for precise location. However, you cannot resize a frame using arrow keys.

2.1.3.3. Edit Current Frame Dialog Positioning and Sizing. For precise control over frame size and location, use the Edit Current Frame dialog. You may specify the exact location for the frame's left and top sides, along with width and height. The same units are displayed on the workspace rulers as are on the dialog.

To precisely position and size your frame:

1. From the Frame menu, choose Edit Current Frame, or double-click on the header or border of the current frame. The Edit Current Frame dialog appears, as shown in Figure 2-1.
2. Enter the position of the left side of the frame in the Left Side text field and the position of the frame's top side in the Top Side text field in Paper Ruler units.
3. Using Paper Ruler units, enter the width and height of the frame in the Width and Height text fields. (Units other than Paper Ruler may be specified by typing them after the number. For example, cm for centimeters, in for inches, or pix for pixels.)

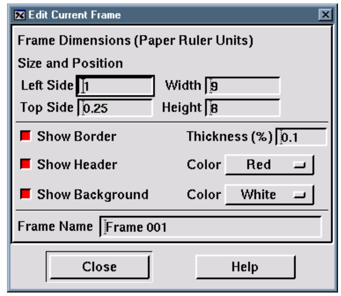


Figure 2-1. The Edit Current Frame dialog in Motif.

2.1.4. Frame Background Color Modification

Alter the frame background color for a variety of effects. To create a transparent frame, turn off the background color completely.

To turn off the background color and create a transparent frame:

1. From the Frame menu, choose Edit Current Frame.
2. On the Edit Current Frame dialog, deselect the Show Background check box. (By default, this check box is selected.)

To set a different background color:

1. From the Frame menu, choose Edit Current Frame.
2. On the Edit Current Frame dialog, verify Show Background is selected. To the right of the Show Background check box click Color to set the appropriate color.

2.1.5. Frame Border and Header Controls

Frame borders act much like a picture frame, giving a visual outline of the drawing region. However, visible borders may be undesirable. For example, if you are building a composite plot out of multiple frames, multiple borders may detract from the appearance. Tecplot allows you to turn off frame borders for any frame. You can make borders invisible (making headers invisible), or you can display the border without the header. You can also control border thickness. (Thickness is specified in frame units as a percentage of the frame height.)

The frame header contains user-configurable information which defaults to:

```
"&(FrameName) | &(date) | &(DataSetTitle)"
```

This displays information about the frame's name, date created or revised, and, if applicable, the title of the current data set. These defaults can be changed in your configuration file; see the **$!GLOBALFRAME** command in the *Tecplot Reference Manual*.

The frame header is displayed when the Show Border and Show Header check boxes are selected on the Edit Current Frame dialog. However, if you turn off the border by deselecting the Show Border check box, the header turns off as well. You can choose any of Tecplot's colors for the frame header.

On most screens, the header information is difficult to read unless you zoom into the paper (for example, by selecting Fit All Frames to Workspace from the Workspace menu).

To modify borders and headers:

1. From the Frame menu, choose Edit Current Frame.
2. On the Edit Current Frame dialog, set your header and border settings as desired.

Invisible borders frustrate positioning and resizing. To overcome this, Tecplot displays a dashed representation of invisible borders. Turn off this feature using Show Invisible Frame Borders from the Workspace menu. When Show Invisible Frame Borders is selected (the default), a small box (or check mark) appears on the Workspace menu. The default can be changed in your configuration file; see the **`$!INTERFACE ShowFrameBorder-sWhenOff`** command in the *Tecplot Reference Manual*.

2.1.6. Frame Name Modification

You may alter any frame name so it reflects the contents of the frame. One advantage of giving frames meaningful names is that they are easily distinguished in the Order Frames dialog.

To change the name of the current frame:

1. From the Frame menu, choose Edit Current Frame.
2. On the Edit Current Frame dialog, change the name as desired.

2.1.7. Frame Pushing and Popping

There are times when you want to expose—*pop*—overlapping or overlaid frames. For partially exposed frames, click on the exposed portion (in any mouse mode except Create Frame). For completely obscured frames, pop underlying frames by pushing the covering frames to the back of the plot, or by using the Frame menu's Order Frames option.

To push a frame to the back of the plot:

1. Select the appropriate frame by clicking on it. This makes it the current frame.
2. From the Frame menu, choose Push Current Frame Back.

If you have multiple overlaid frames, repeat these steps until the desired frame is on top, or pop a specific frame using Order Frames. If part of a frame is visible, pop it to the top by clicking on it.

To pop a frame by name using Order Frames:

1. From the Frame menu, choose Order Frames. The Order Frames dialog appears, as shown in Figure 2-2.

Figure 2-2. The Frame Order dialog.

2. Select the desired frame by name from the list box and click Pop, or double-click on a name.
3. To change the display order of the frame names within the list, select the List By drop-down. (List By does not affect actual frame order within the workspace, only the display of the names within the list box.)

2.2. Workspace Management

The workspace is the entire region in which you can create Tecplot frames, including, but not limited to, the region covered by the paper. You may find yourself using only the paper area in creating the screen plots. This is a natural way to work, but not essential, since the paper only limits the printing of the plots. If you are creating plots for screen use only, you may find it useful to use the entire workspace. One way to do this is to turn off display of the paper.

2.2.1. Paper Set-Up

Tecplot's representation of paper in the workspace allows you to lay out plots precisely the way you want them printed. If you place a frame on the paper and print the resulting plot, the frame appears in the exact relative location on the printed paper.

You can control the size, orientation, even the color of your paper. You can also turn off the screen representation of the paper.

2.2.1.1. Paper Size Controls. Tecplot offers the following six paper sizes:

- **Letter:** Standard U.S. letter size, 8 1/2 by 11 inches.
- **Double:** Standard U.S. ledger size, 11 by 17 inches.
- **A4:** Standard European letter size, 21 by 29.7 centimeters.
- **A3:** Standard European size, 29.7 by 42 centimeters.
- **Custom 1:** Default is 8.5 by 14 inches.
- **Custom 2:** Default is 8 by 10 inches.

To choose a paper size:

1. From the File menu, choose Paper Setup. The Paper Setup dialog appears.
2. In the Size area of the Paper Setup dialog, select the desired paper size. (In Windows, you can also set paper size using the File menu's Print option.)

All paper sizes may be customized using options in configuration or macro files. It is recommended that you only change the dimensions of the Custom 1 and Custom 2 paper sizes. To change the Custome sizes see the **`$!PAPER`** command in the *Tecplot Reference Manual*.

2.2.1.2. Paper Orientation Controls. Tecplot layouts can be landscape or portrait plots. In landscape (the default), the long axis of the paper is horizontal, while in portrait the long axis is vertical. Portrait orientation uses the width of the specified paper for the horizontal dimension, while landscape uses this for the vertical dimension. You specify the orientation as part of paper set-up.

To specify a paper orientation:

1. From the File menu, choose Paper Setup. The Paper Setup dialog appears.
2. In the Orientation area of the dialog, select the desired orientation.

2.2.1.3. Screen Paper Controls. If you are creating plots for display on your screen you can turn off the screen representation of the paper and use the full workspace.

To turn off the screen paper:

1. From the File menu, choose Paper Setup. The Paper Setup dialog appears.
2. Deselect Show Paper on Screen.

2.2.1.4. Paper Color Controls. You can set up your paper to show any of Tecplot's colors as a background color (the "paper fill color") on your screen, as well as use that color when printing to a color printer.

To specify paper fill color:

1. From the File menu, choose Paper Setup. The Paper Setup dialog appears.
2. Click Paper Fill Color. On the resulting dialog, choose the desired color.

When you are printing, Tecplot can flood the paper with your specified fill color. (By default, the paper fill color is ignored during printing.)

To use the paper fill color when printing:

1. From the File menu, choose Paper Setup. The Paper Setup dialog appears.
2. Select Use Paper Fill Color when Printing.

2.2.2. Grid and Ruler Set-Up

The workspace grid provides a convenient guide for placing objects on your paper. When placing text or geometric shapes, you can choose to snap the anchor points of the shapes to the grid.

Rulers provide a reference length for sizing objects. You may draw the rulers in centimeters (cm), inches (in), or points (pt), or not draw them at all.

2.2.2.1. Workspace Grid Controls. Tecplot allows you to select grid spacing from several pre-set sizes in centimeters (cm), inches (in), or points (pt). You can also specify not to show the grid. The grid is not shown if the paper is not visible onscreen, or if the Show Grid check box is not selected.

To turn off the grid:

1. From the Workspace menu, select Ruler/Grid. The Ruler/Grid dialog appears.
2. Deselect Show Grid.

To specify the grid spacing:

1. From the Workspace menu, select Ruler/Grid. The Ruler/Grid dialog appears.
2. From the Grid Spacing drop-down, choose the desired spacing.

2.2.2.2. Workspace Ruler Controls. Tecplot allows you to select the ruler markings from several pre-set sizes in centimeters (cm), inches (in), or points (pt). You can also specify whether to show the ruler.

To turn off the ruler:

1. From the Workspace menu, select Ruler/Grid. The Ruler/Grid dialog appears.
2. Deselect Show Ruler.

To specify the ruler spacing:

1. From the Workspace menu, select Ruler/Grid. The Ruler/Grid dialog appears.
2. From the Ruler Spacing drop-down, choose the desired spacing.

2.2.3. Workspace Maximization

You can create plots up to the full size of the workspace, and you can force the workspace to fill the Tecplot window, hiding the sidebar and menu bar.

To maximize your workspace:

1. From the Workspace menu, choose Maximize Workspace. The sidebar and menu bar disappear.
2. Press Esc to return to the standard view.

2.3. Coordinate Systems

Tecplot manages a number of coordinate systems. Four of these coordinate systems that are important to know about are the paper, frame, 2- and 3-D physical coordinate systems. The origins of each coordinate system and how they relate to one another is shown in Figure 2-3.

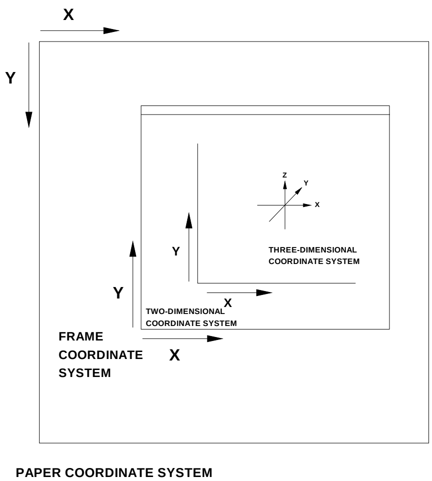


Figure 2-3. Coordinate systems in Tecplot.

Only one of the 2- or 3-D physical coordinate systems is in effect at any given time, depending on the plot type in use. The 2- and 3-D physical coordinate systems plot the X-, Y-, and or Z-coordinates of data points. Two-dimensional physical coordinates are often referred to as *grid coordinates*. The frame coordinate system is shown in Figure 2-4.

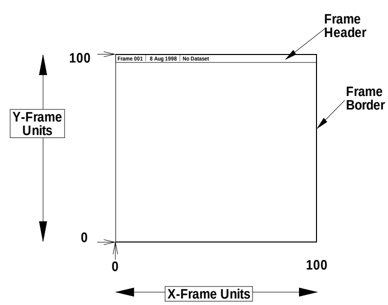


Figure 2-4. The frame coordinate system.

The vertical axis of this coordinate system always runs from zero at the bottom border of the frame to 100 at the top of the frame. Likewise, the horizontal axis runs from zero to 100 from left to right along the bottom edge of the frame. The distance of one horizontal unit is not necessarily equal to the distance of one unit in the vertical direction, since frames take on almost any aspect ratio.

Tecplot uses the height of the frame for objects scaled by frame units, such as font size. Whenever you enter a frame unit value into a dialog, or you are setting frame size and position on the paper, you may specify a different unit system as a suffix. Values are converted to frame units (or paper units when sizing or positioning a frame) for you. Valid suffixes are "in" (inches), "pt" (points), "cm" (centimeters), and "pix" (pixels). For example, if you want a piece of text exactly one inch away from the left edge of your frame, you would enter "1in" in the X-Origin text field on the Text Details dialog. Tecplot converts this to the appropriate frame unit value.

2.4. View Modification

There are two types of views inside Tecplot. The first is the view of your data inside a frame. Each frame can have a separate view of its data. This is controlled using the View menu.

The second type of view is of the frames and paper inside the workspace. This view is controlled using the Workspace menu.

Both views may be controlled by the two view mouse modes on the sidebar—Zoom and Translate. However, other actions differ depending on whether working with a frame or the workspace.

2.4.1. View of Data within a Frame

The view of your Tecplot data is the position, size, and orientation of the plot within a frame. The View menu contains controls to help you adjust the view, as well as copy the view from one frame to another. The View menu contains the following options:

- **Redraw:** Redraws the current frame.
- **Zoom:** Turns on the Zoom mode to train in on the current frame.
- **Fit to Full Size:** Resizes plots so all data points, text, and geometries are included in the frame.
- **Nice Fit to Full Size (2D Cartesian, XY Line, and Sketch plot types only):** Sets axis range to begin and end on major axis increments (if axes are dependent Tecplot adjusts the vertical axis length to accommodate a major tick mark).
- **Data Fit:** Resizes the plot so all data points are included in the frame. Text and geometries are not considered.
- **Make Current View Nice (2D Cartesian, XY Line, and Sketch plot types only):** Modifies the range on a specified axis to fit the minimum and maximum of the variable assigned to that axis, then snaps the major tick marks to the ends of the axis. (If axis dependency is not independent this may affect the range on another axis.)

- **Center:** Centers the plot within the frame.
- **Translate/Magnify:** Turns on the Translate mode and calls up the Translate/Magnify dialog, which you use to move and resize your plot with respect to the frame.
- **Last:** Restores the previous view.
- **Rotate (3D Cartesian plot type only):** Calls up the Rotate dialog for image rotation. For further information, see Section 7.6.1, “Three-Dimensional Rotation.”
- **3D View Details (3D Cartesian plot type only):** Calls up the 3D View Details dialog for setting the view position and angle of 3-D images. For further information see Section 7.6.1, “Three-Dimensional Rotation.”
- **Copy View:** Copies the current frame view to a buffer; it can then be pasted onto another frame.
- **Paste View:** Pastes a copied view onto the current frame.

Shortcuts are provided for most View menu controls. For Zoom, use  on the sidebar to select the Zoom mode. For Translate/Magnify, use  on the sidebar. For Fit to Full Size, use the keyboard shortcut Ctrl-F. For Last, use the keyboard shortcut Ctrl-L. For Paste View, use the keyboard shortcut Ctrl-A. There are no shortcuts for Data Fit, Center, or Copy View.

2.4.1.1. Mouse Zoom and Translation. The middle and right mouse buttons allow you to smoothly zoom and translate data. Your middle mouse button (Ctrl-right click) zooms smoothly, and your right mouse button translates data. (See Appendix C, “Mouse and Keyboard Operations,” for additional functionality.) Advanced functionality is available in: Contour Modes; Streamtrace Placement; Slicing; Rotation Modes; Geometry Modes (Except Polyline); Zoom; Translate/Magnify; Probe; Zone Creation.

2.4.1.2. Zoom Tool. Select the Zoom tool from the sidebar; the pointer becomes a magnifying glass. Click and drag to create a rubber band box around the region you want to magnify. The plot resizes to fit the longest dimension of the box. Zoom into a region by positioning the magnifying glass and clicking your left mouse button. It magnifies the area by 200 percent, centering on the position of the magnifying glass.

It may be useful to zoom into successively smaller regions until the area of interest includes adequate detail. For example, finite-element data file **`feexchng.plt`** shows an area of interest containing many circles. You zoom in on the circles, and then zoom in again on a single circle. Finally, you zoom in on the boundary of a circle, as illustrated in Figure 2-5.

Fit one or all frames to the workspace by using the Fit Selected Frame or Fit All Frames to Workspace on the Workspace menu. Both are alternative methods of zooming the paper. To return to the default view, choose Fit Paper to Workspace from the Workspace menu. At any stage of the zoom, you can use Ctrl-L to return to the previous (last) view, Ctrl-F to return to the full size initial view, or Ctrl-Z (Undo) to restore the previous view.

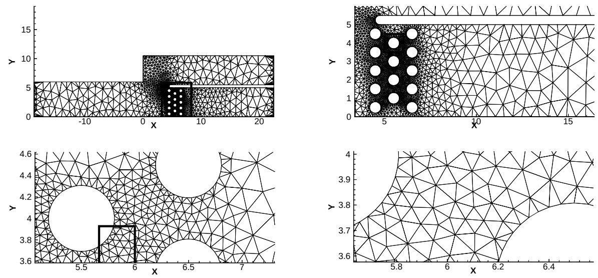


Figure 2-5. Zooming into a plot.

2.4.1.3. Data Translating and Scaling. Use the Translate/Magnify feature to translate the view of your data. Translating moves the image of your data in respect to the current frame. You can translate plots in any direction within a frame. The Translate/Magnify feature is available as both a sidebar tool and as a dialog accessed from the View menu, shown in Figure 2-6.

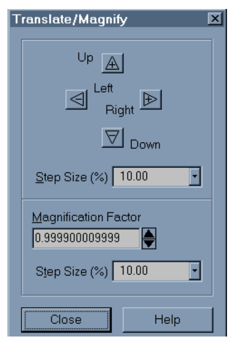


Figure 2-6. The Translate/Magnify dialog.

The following options and shortcuts are available for the Translate/Magnify dialog:

- **Up, Down, Left, Right:** Use the arrows to translate the image in the desired direction.
- **Magnification Factor:** Change magnification using the arrows, or enter a value in the text field.
- **Step Size (%):** Control the step size for each arrow using pre-set ranges from the drop-down, or by entering your own value.

To translate an image using the sidebar's Translate/Magnify tool:

1. On the sidebar, click . The pointer changes to an all-direction cursor.
2. In the workspace, drag in the direction you want to transfer the image.

Translate/Magnify tool mode offers the following keyboard options:

+: Increase scale of images.

- : Decrease scale of images.

2.4.1.4. Data Fit. You can use the Fit to Full Size option from the View menu to restore the initial view of your data after extensive zooming, scaling, or translating. Tecplot performs the Fit to Full Size operation when it first displays your data set. You can perform the operation in either of the following ways:

- Select Fit to Full Size from the View menu.
- While the window is active, press Ctrl-F.

2.4.1.5. Last View. Allows you to step backward through the resizings and repositionings of plots. Any time you change the view of a frame, either by zooming, centering, translating, or fitting the plot, the previous view is placed in a view stack. Each frame has four view stacks, one for each plot type. Each view stack stores the last sixteen views for that plot type. Move back through the view stack by choosing Last from the View menu repeatedly, or pressing Ctrl-L repeatedly.

2.4.1.6. Copy and Paste Views. When you are working with multiple frames attached to the same data set, it is often useful to make your view changes to one frame, and then propagate those changes to the other frames. You can do this using the Copy View and Paste View options under the View menu, as follows:

1. Make the changes (zooming, translating) you want to make to one frame.
2. From the View menu, choose Copy View.
3. Click in another frame sharing the same plot type.
4. From the View menu, choose Paste View (or press Ctrl-A). Note that Copy View and Paste View only affect the ranges of XY-axis and tick mark spacing. For complete duplication, use Copy Style to File and Paste Style from File from the Style menu.

2.4.2. Frame and Paper Views within the Workspace

The view of your frames and the paper within the workspace is controlled through the Workspace menu. This is called the workspace view.

The Workspace menu contains the following options:

- **Display Performance:** Calls up the Display Performance dialog where you can customize Tecplot's graphic resource management to suit your needs.

- **Ruler/Grid:** Calls up the Ruler/Grid dialog, controlling for the workspace ruler and paper grid.
- **Show Invisible Frame Borders:** Hide or display invisible frame borders for placement purposes.
- **Redraw All:** Redraws all frames and the paper.
- **Fit Selected Frames to Workspace:** Resizes the workspace view so selected frames are included in the view.
- **Fit All Frames to Workspace:** Resizes the workspace view so all frames are included in the view.
- **Fit Paper to Workspace:** Resizes the workspace view so the paper is included in the view.
- **Last Workspace View:** Restores the workspace to its previous view.
- **Maximize Workspace:** Set the workspace to its largest size. Use Esc to return to your normal view.
- **Color Map:** Calls up the Color Map dialog, where you may customize Tecplot's color map settings using five pre-defined schemes, an interpolated scheme, or a user-defined scheme.
- **Copy Color Map to File:** Allows you to save a color map file.
- **Paste Color Map from File:** Allows you to load a color map file.

2.4.2.1. Workspace Zooming. Use the  sidebar tool to zoom the workspace. The pointer changes to a magnifying glass. Shift-drag the magnifying glass to draw a box around a region to magnify. The plot resizes so the longest dimension of the box fits into the workspace.

2.4.2.2. Workspace Translating and Scaling. Use the  sidebar tool to translate and magnify paper and image simultaneously. (Magnifying the screen image does not affect the printout size.)

To translate the entire paper and image:

1. Click the  sidebar tool for Translate/Magnify mode.
2. Shift-click to operate on paper and image simultaneously. (Leave the mouse button down.)
3. Drag to move the paper.

To magnify the entire paper and image (operates on the on-screen paper and image only):

1. Click the  sidebar tool for Translate/Magnify mode.
2. Shift-click to operate on paper and image simultaneously. Remember to leave the mouse button down.
3. Press "+" to magnify paper and image, "-" to reduce.

You can move and rescale the paper simultaneously as long as you hold down the mouse button. If you release the button, "+" and "-" revert to resizing the image.

2.4.2.3. Workspace View Fit. You can use Fit Paper to Workspace from the Workspace menu to restore the initial view of the paper. Tecplot fits the paper to the workspace when it first starts.

2.4.2.4. Last Workspace View. You may restore the last workspace view with Last Workspace View from the Workspace menu. When you change the view of the workspace, using Fit Selected Frames to Workspace or Fit All Frames to Workspace for example, the previous view is placed in a view stack, which stores the last sixteen views. Cycle back through the view stack by choosing Last Workspace View repeatedly, or use Undo from the View menu.

2.5. Cut—Copy—Paste—Clear—Undo

Duplicate frames, text, and geometries with the copy and paste options of the Edit menu (or their keyboard equivalents). You can also cut objects from one location and paste them into another, or throw them away completely. To select all geometries, zones, text or streamtraces in a frame, choose the Select All option from the Edit menu.

Under Windows, the Cut, Copy, and Paste options work only within Tecplot. However, the Edit menu's Copy Plot to Clipboard, option allows you to copy Tecplot frames and paste them into other Windows applications. See Section 22.14, "Clipboard Exporting to Other Applications," for a discussion of this feature.

If you cut or clear the last Tecplot frame, Tecplot automatically creates another frame to replace it.

Tecplot can undo all plot and mapping style modifications. In addition, Tecplot allows you to undo a variety of other plot alterations. As a rule, Tecplot allows undo for reversable operations that can be restored without significant impact on the operation's performance. To undo an operation, select Undo from the Edit menu, or press Ctrl+Z in the workspace.

Specifically, the Undo option is allowed for the following conditions:

- All zone and map style changes.
- Some (though not all) frame control operations, push and pop.
- Creating new frames.
- View operations.
- Some pick operations.
- Streamtrace actions.
- The following data alterations:
  - Deleting zones and variables.
  - Renaming data set zones and zones.
  - Creation of rectangular or circular zones.
  - Duplication of zones.

- Equation processing. (Except equations containing derivatives.)

Notes: Undo is unavailable for *all* data operations once an Undo operation has been performed on an un-allowed item. In addition, once an operation is performed that is not undoable, the entire undo history for that frame is erased.

CHAPTER 3 *Data Organization*

This chapter describes Tecplot's internal handling and storage of data.

3.1. Data Hierarchy

Tecplot structures data in two levels. The highest level in Tecplot's internal data structure is a data set. Data sets consist of one or more zones. Zones—blocks of data making up a data set—are the second level in the data hierarchy. They can be created in Tecplot, or loaded from a file. A data hierarchy example is shown in Figure 3-1.

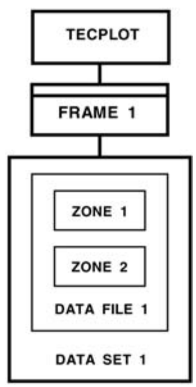


Figure 3-1. Tecplot data structure.

Starting with an empty frame, a data set is created and assigned to the active frame whenever you read one or more data files into Tecplot, or create a zone within Tecplot. Multiple frames can be attached to the same data set. How data sets and frames relate to one another is shown in Figure 3-2.

If more than one data file is read into a frame, Tecplot groups all the zones from the files into one data set. Once in Tecplot, all zones in a data set must contain the same variables defined for each data point. (This does not necessarily mean each of your data files needs to have the same number of variables in the same order. See Section 5.1, "Tecplot-Format Data File Loading," for loading dissimilar data files or parts of data files.) The number of zones in a concatenated data set is the sum of the number of zones in the data files that are read. As Figure 3-2

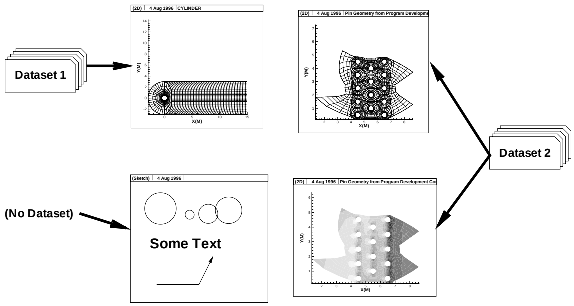


Figure 3-2. Data sets and frames.

shows, one or more frames can access data from the same data set. Frames using the same data set initially have the same header color.

A more complex example of Tecplot data structure is shown in Figure 3-3.

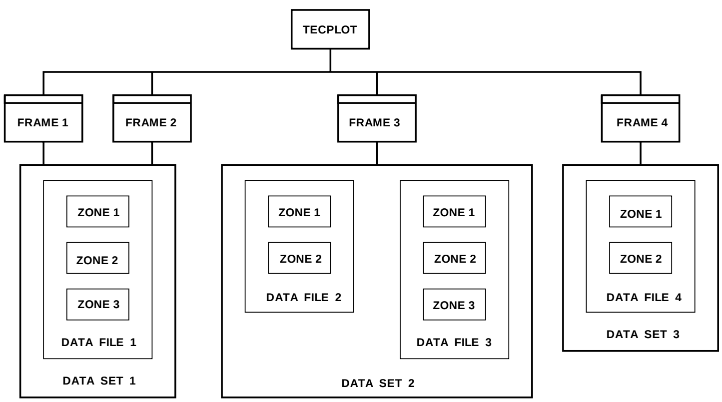


Figure 3-3. A complex data structure example.

Frames 1 and 2 both access data set 1, itself made up of a data file with three zones. Frame 3 accesses data set 2 containing two data files—one with two zones and one with three zones. Frame 4 uses data set 3, containing one data file of two zones.

3.2. Multiple Zones

Multiple zones may be used for plotting complex configurations, or sub-dividing data for convenience. You may also represent data taken at different time steps, or using measurement methods. An illustration of multiple zones is shown in the left-most plot in Figure 3-4.

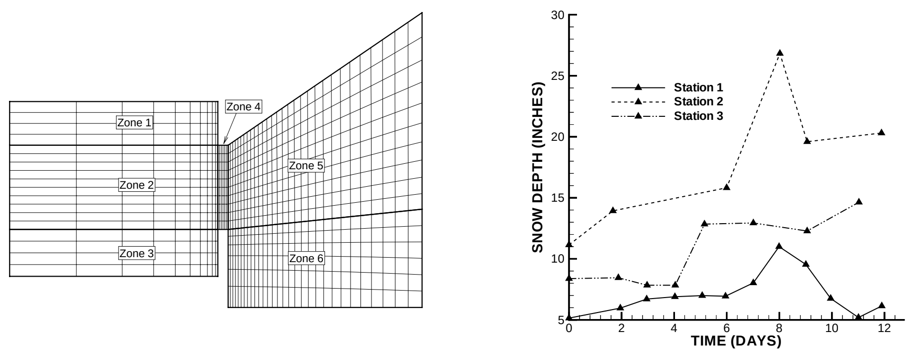


Figure 3-4. Example plots showing the use of multiple zones. A 2-D Cartesian mesh is on the right; an XY Line plot on the right.

You could use multiple zones to show the measurement of snow depth at several different stations in an XY Line plot. You take measurements once a day at each station, but there are some days on which you cannot get to all of the stations. As a result, when you are finished taking data, you have a different number of data points for each station. Since each set of data has the same number of variables per data point (time and snow depth), you can set up a Tecplot data file with the measurements from each station in a separate zone. In Tecplot, you can define a set of mappings to plot snow depth versus time for any combination of zones (in this example, stations). Figure 4-4 shows a plot of this data.

3.3. Data Structure in Zones

Tecplot accommodates two different types of data: ordered and finite-element. The following sections describe each in detail.

3.3.1. Ordered Data

Ordered data is a set of points logically stored in a one-, two-, or three-dimensional array in Tecplot. I, J, and K are used as sub-scripts to access values within the array.

The most common forms for these arrays are:

- **I-ordered:** One-dimensional array of data points where the I-dimension is greater than or equal to one and the dimension in J- and K-directions is equal to one. The I-dimension represents the total number of data points for the zone.
- **IJ-ordered:** Two-dimensional array of data points where both the I- and J-dimensions are greater than one and the K-dimension is equal to one. The number of data points is the product of the I- and J-dimensions.
- **IJK-ordered:** Three-dimensional array of data points where all three of the I-, J-, and K-dimensions are greater than one. The number of data points is the product of the I-, J-, and K-dimensions.

Other ordered data types are also valid but are not typically created in Tecplot. These may come from data sets created by other applications wishing to retain a particular data order. They are:

- **J- or K-ordered:** The same as I-ordered but the J- or K-dimension is greater than one and the remaining dimensions are equal to one.
- **JK- or IK-ordered:** Similar to IJ-ordered. In both cases two of the three dimensions are greater than one and the remaining dimension is equal to one.

In general, discussions which refer to using I-ordered data may be applied to J- or K-ordered data. All three represent a logical one-dimensional array of data. Likewise, all discussions referring to IJ-ordered data may be applied to JK- or IK-ordered data.

3.3.1.1. I-, J-, or K-Ordered Data Points. Data points for XY Line plots are usually arranged in a one-dimensional array indexed by one parameter: I for I-ordered, J for J-ordered, or K for K-ordered, with the two remaining index values equal to one. For I-ordered, the most common type, I is as follows: *I=1* at the first data point, *I=2* at the second data point, *I=3* at the third data point, and so forth to *I=IMax* for the last point. At each data point, *N* variables (**V1, V2, ..., V***N*) are defined. If you arrange the data in a table where the values of the variables (*N* values) at a data point are given in a row, and there is one row for each data point, the table would appear something like that shown in Figure 3-5. For example, if you wanted to make a simple XY-plot of pressure versus time, **V1** would be time and **V2** would be pressure.

V1	V2	V3	V4	V5	V6	. . .	V*N*	(Values at data point *I = 1*.)
V1	V2	V3	V4	V5	V6	. . .	V*N*	(Values at data point *I = 2*.)
V1	V2	V3	V4	V5	V6	. . .	V*N*	(Values at data point *I = 3*.)
V1	V2	V3	V4	V5	V6	. . .	V*N*	
V1	V2	V3	V4	V5	V6	. . .	V*N*	

Figure 3-5. Table of values for I-ordered data points (suitable for XY-plots).

`V1`	`V2`	`V3`	`V4`	`V5`	`V6`	`...`	`V`*N*	
`V1`	`V2`	`V3`	`V4`	`V5`	`V6`	`...`	`V`*N*	(Values at data point *I* = *IMax*.)

Figure 3-5. Table of values for I-ordered data points (suitable for XY-plots).

You may also input data for 2- and 3-D vector and scatter plots in I-ordered format. You could create a 3-D vector plot by setting the first six variables at each data point to the three physical coordinates (X, Y, Z) and the three velocity vector components (U, V, W). However, if you did this, you would not be able to use features like light source shading, hidden-surface removal, or streamtraces. These features depend upon a mesh structure connecting the data points (see IJ- and IJK-ordering and finite-element surface points in the next sections).

3.3.1.2. IJ-Ordered Data Points. The data points for 2- and 3-D surface field plots are usually organized in a two-parameter mesh. Each data point is addressable by a set of the two parameters (I and J) and has four neighboring data points (except at the boundaries). The points are located above, below, to the left, and to the right as shown in Figure 3-6.

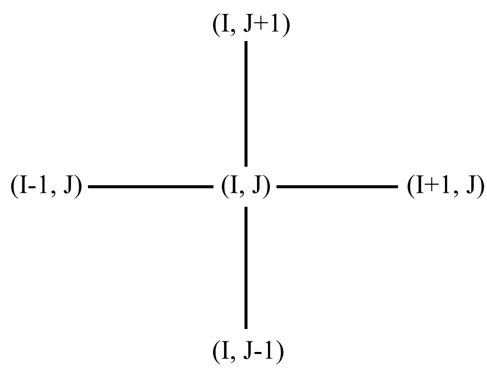


Figure 3-6. IJ-ordered data point neighbors.

At each data point define two (or three) spatial variables (X, Y, and Z) plus one or more variables like temperature, velocity components, or concentration. Plot the data points in a 2- or 3-D coordinate system where any two (or three) of the variables defined at the data points are the spatial coordinates (by default, the first two or three are used).

A family of I-lines results by connecting all of the points with the same I-index. Likewise, a family of J-lines is formed in the same way. When both the I- and J-lines are plotted in a two-dimensional coordinate system, a 2-D mesh plot results. When both the I- and J-lines are plotted in a three-dimensional coordinate system, a 3-D surface mesh plot results. Both meshes are shown in Figure 3-7.

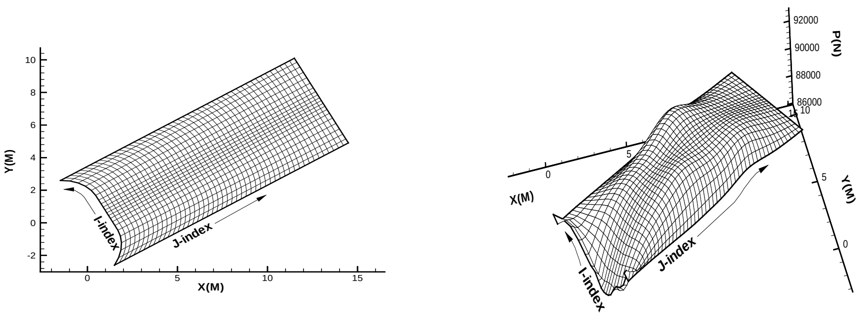


Figure 3-7. Left, a 2-D mesh of IJ-ordered data points. Right, a 3-D mesh of IJ-ordered data points.

The data points lie at the intersections of the I- and J-lines. The points along the I-lines and J-lines need not be in a straight line. The points may trace out curved, irregularly spaced, and/or nonparallel paths. They may lie in a planar 2-D or on a non-planar 3-D surface.

Data organized in IJ-order can also be used for XY Line plots. I-order is actually the same as IJ-order with J equal to one. In XY Line plots, you can specify the range (maximum and minimum) and skip interval for the I- and J-indices for plotting data points; data points outside of the specified ranges are not plotted. You can also plot the I-lines or the J-lines of an IJ-ordered zone.

3.3.1.3. IJK-Ordered Data Points. The data points for 3-D volume field plots are usually organized in a three-parameter mesh. Each point is addressable by a set of three parameters (I, J, and K) and has six neighboring data points (except at the boundaries). These neighbors are located above, below, left, right, in front of, and behind the data point as shown in Figure 3-8.

At each data point, you define three spatial variables (X, Y, Z) plus (typically) one or more variables such as pressure, vector components, and vorticity.

A mesh plot of IJK-ordered data is displayed in Figure 3-9. The directions of the I-, J-, and K-indices are shown. As you can see, the points that define the mesh can form curved, irregularly spaced, and/or nonparallel paths.

3.3.1.4. I-, J-, and K-Planes. An important concept in dealing with IJK-ordered data is that of I-, J-, and K-planes. A K-plane is the connected surface of all points with a constant K-index value. The I- and J-indices range over their entire domains; thus, a K-plane has, in effect, a two parameter ordering, much like IJ-ordering. In fact, IJ-ordered data is identical to IJK-ordered data with the K-index equal to one (*KMax=1*). Note that K-planes are not necessarily planes in the strict sense. They are called K-planes because they exist as planes in logical (IJK) space. In real (XYZ) space, the K-planes may be cones, ellipsoids, or arbitrary surfaces.

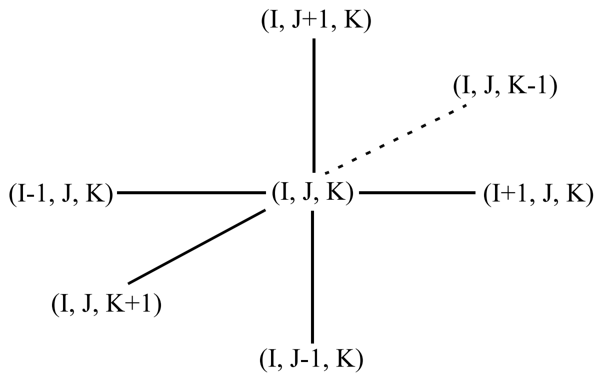


Figure 3-8. IJK-ordered data point neighbors.

An I-plane is the connected surface of all points with a constant I-index value (with *J* and *K* ranging over their entire domains), and a J-plane is the connected surface of all points with a constant J-index value (with *K* and *I* ranging over their entire domains). Examples of I-, J-, and K-planes are shown in Figure 3-9.

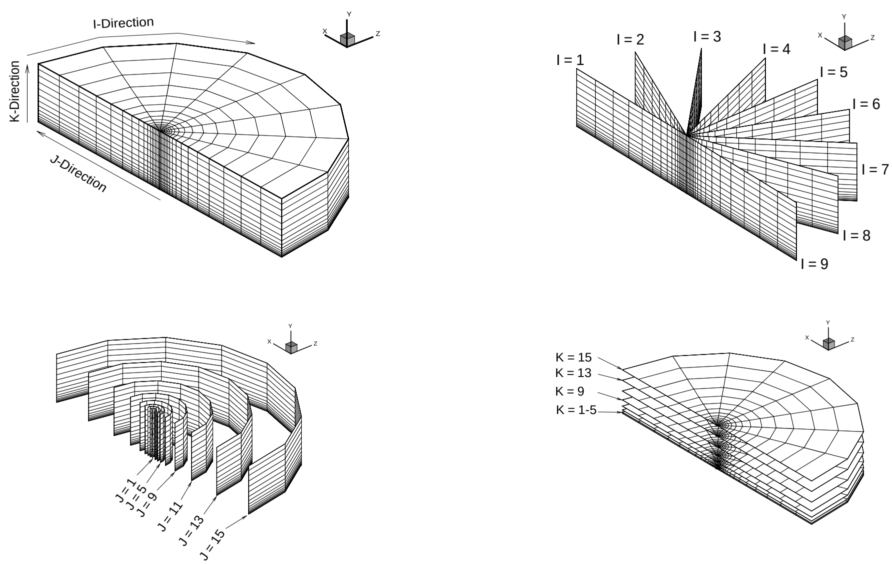


Figure 3-9. Clockwise from upper left: I-, J-, and K-directions of an IJK-ordered zone; I-planes of an IJK-ordered zone; J-planes of an IJK-ordered zone; K-planes of an IJK-ordered zone.

3.3.1.5. IJK-Ordered Data Plotting. Plotting IJK-ordered data is more complex than plotting other ordered data types such as I- or IJ-ordered. With the other data types all data will typically be plotted. IJK-ordered data offers more options as to which portions of data will be viewed, especially when creating 2- or 3-D plots. The Surfaces page of the Zone Style dialog allows you to designate which surfaces of IJK-ordered data will be plotted. You may choose to plot just outer surfaces, or you may select combinations of I-, J-, and K-planes to be plotted. For more information see Chapter 20, "Three-Dimensional Volume Data."

3.3.2. Finite-Element Data

Finite-element data, also referred to as *FE data*, is a method of structuring data as a collection of points in 2- or 3-D space with a set of instructions on connecting these points to form elements, or cells.

Finite-element data defines a set of points (nodes) and the connected elements of these points. Finite-element data can be divided into three types:

- **FE-line:** A set of line segments defining a 2- or 3-D line.
- **FE-surface:** A set of triangular or quadrilateral elements defining a 2-D field or a 3-D surface.
- **FE-volume:** A set of tetrahedral or brick elements defining a 3-D volume field.

In each of the above data-point orderings there is virtually no limit to the number of data points; the size of your data set is limited only by the amount of physical resources of your computer. You may use a different data point structure for each zone within a data set, as long as the number of variables defined at each data point is the same. Chapter 4, "ASCII Data for Tecplot," gives detailed information about how to format your data for Tecplot.

3.3.2.1. Finite-Element Line Data. Plotting connectivity lines between finite-element line data points results in a 2- or 3-D Cartesian coordinate system appearing as a line as shown in Figure 3-10. Unlike I-ordered data, a single finite-element line zone may consist of multiple disconnected sections.

The values of the variables at each data point (node) are entered in the data file similarly to I-ordered data, where the nodes are numbered with the I-index. This data is followed by another set of data defining connections between nodes. This second section is often referred to as the *connectivity list*. All elements are lines consisting of two nodes, specified in the connectivity list.

A common operation creating finite-element line zones is slicing a 3-D surface zone using Tecplot's Slice from Plane option. To do this, select Extract from Tecplot's Data menu.

3.3.2.2. Finite-Element Surface Data. In finite-element surface data, the values of the variables at each node (data point) and the finite-element connectivity lists are entered in the data file in the same manner as finite-element line data (described above). The difference is in the number of nodes per element.

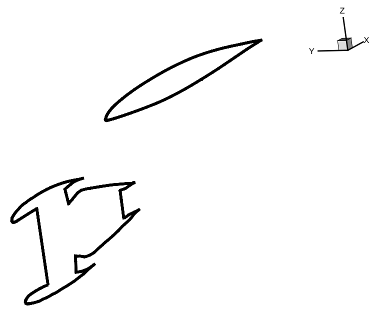


Figure 3-10. A finite-element line resulting from sliced 3-D surface data.

Plotting connecting lines between finite-element surface data points in a 2-D Cartesian coordinate system results in a mesh as shown in Figure 3-11. Plotting a finite-element surface mesh

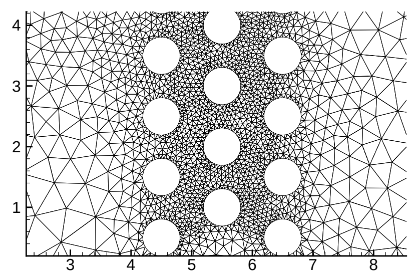


Figure 3-11. Mesh plot of finite-element surface data in two dimensions.

in a 3-D Cartesian coordinate system results in a mesh like that shown in Figure 3-12.

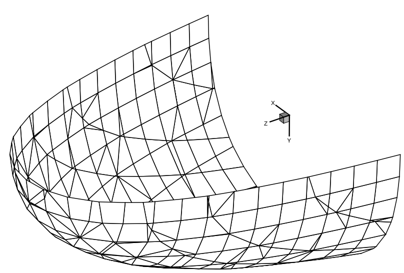


Figure 3-12. Mesh plot of finite-element surface data in three dimensions.

You can choose (by zone) to arrange your data in three point (triangle) or four point (quadrilateral) elements. The number of points per node and their arrangement are called the element type of the zone. You may repeat a node in the quadrilateral element type to create a triangle if a mixture of quadrilaterals and triangles is necessary.

3.3.2.3. Finite-Element Volume Data. Finite-element volume cells may contain four points (tetrahedron) or eight points (brick). The elements in each zone must be either all tetrahedra or all bricks. Connectivity of a tetrahedron is shown in Figure 3-13.

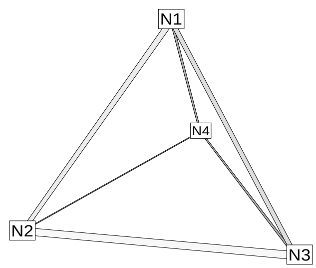


Figure 3-13. Connectivity of tetrahedron finite-element volume.

Connectivity of a brick is shown in Figure 3-14. In the brick format, points may be repeated to

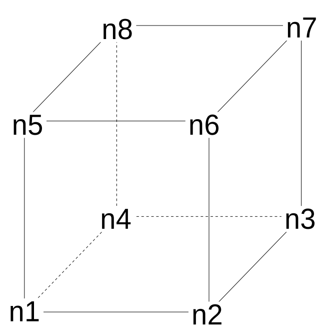


Figure 3-14. Connectivity of brick finite-element volume.

achieve 4-, 5-, 6-, or 7-point elements. For example, a node list entry of "`n1 n1 n1 n1 n5 n6 n7 n8`" results in a quadrilateral-based pyramid element. An example of a finite-element volume mesh is shown in Figure 3-15.

In finite-element volume order, the values of the variables at each node (data point) and their connectivity lists are entered in the data file in the same manner as finite-element surface data. Finite-element zones of element type brick or tetrahedron are referred to as finite-element volume zones.

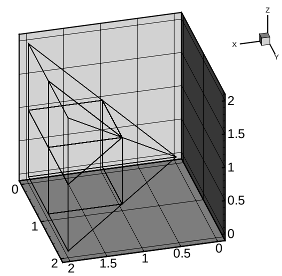


Figure 3-15. A finite-element volume mesh.

3.3.3. Variable Location (Cell-Centered or Nodal)

Our opening chapters presumes data is located at the nodes. However, it is possible for some data to be defined at the cell-centers. This is true for all zone types. You choose the location for each variable. If a variable is cell-centered, one value is specified for each element (cell).

For finite-element meshes, cell-centers are the centers (centroids) of elements. For I-ordered grids, the cell-centers are at the centers of the lines connecting points I and I+1. For IJ-ordered grids, the cell-centers are at the centroids of the quadrilaterals defined by points IJ; I+1J; IJ+1; I+1; J+1. For IJK-ordered grids, the cell-centers are at the centroids of the hexahedral-like elements defined by points IJK; I+1JK; IJ+1K; I+1J+1K; IJK+1; I+1JK+1; IJ+1K+1; I+1J+1K+1.

For many types of plots, Tecplot internally interpolates cell-centered values to the nodes.

3.4. Data Set Information

The Data Set Information dialog, accessed from the Data Set Info option on the Data menu, gives summary information about the current data set, including the data set title, zone and variable names, and the minimum and maximum values of a selected variable. You can modify the data set title, zone and variable names of any data set. The dialog and its pages (Zone/Variable Info, Data Set, Sharing, Journal, Aux Data) are shown in Figure 3-16.

The following information is provided on the Zone/Variable Info page:

- **Zone(s):** Lists all zones by number, with their titles. Select one zone to display its name in the Zone Name field, where the zone name can be modified.
- **Zone Name:** Enter a new name for a selected zone.
- **Zone Type (Ordered or FE data):** Displays the type of zone selected in the Zone(s) listing. For ordered data, it is followed by the index values for IMax, JMax and KMax (shown below). For finite-element data, it is followed by the element type, number of points, and number of elements (see below).

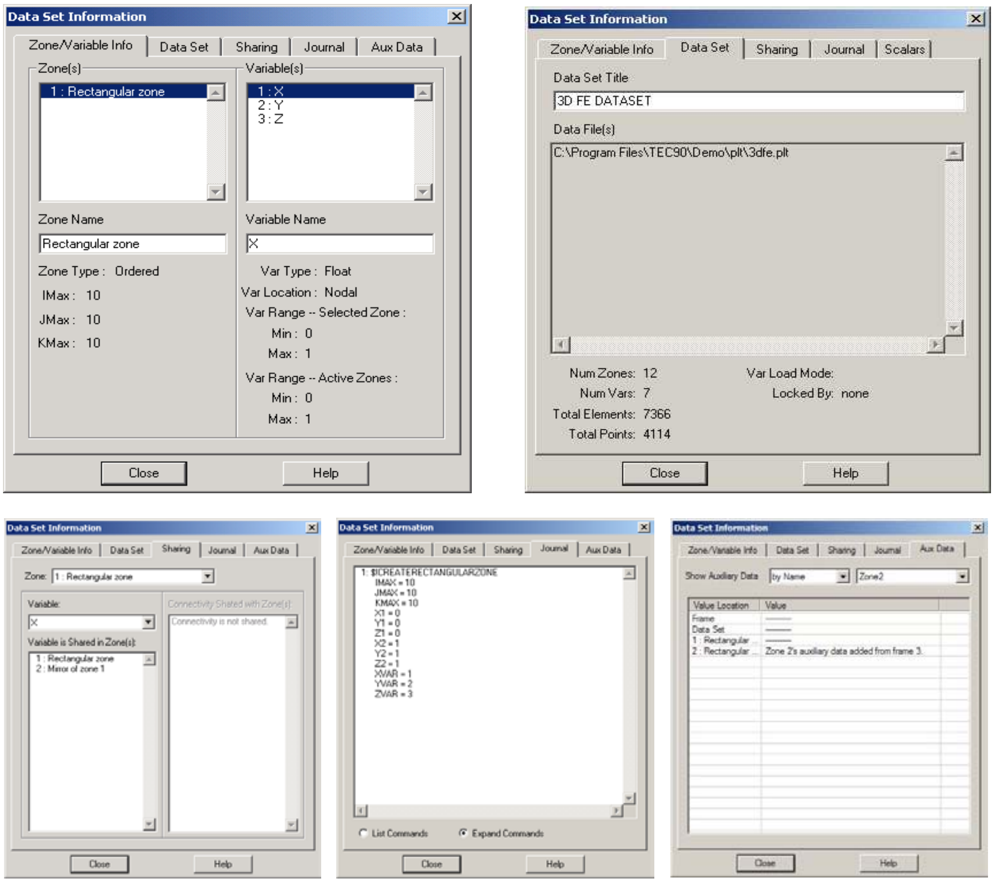


Figure 3-16. The Data Set Information dialog. Above are the Variable and Data Set pages. Below, left to right, are the Sharing, Journal and Aux Data pages.

- **IMax (ordered data):** Displays the IMax value of the zone selected in the Zone(s) listing.
- **JMax (ordered data):** Displays the JMax value of the zone selected in the Zone(s) listing.
- **KMax (ordered data):** Displays the KMax value of the zone selected in the Zone(s) listing.
- **Pts (finite-element data):** Displays the number of data points in the zone selected in the Zone(s) listing.

  - **Elem (finite-element data):** Displays the number of elements in the zone selected in the Zone(s) listing.
- **Variable(s):** Lists all variables by number, with their names. Select one variable to display its name in the Variable Name field, where the name can then be modified.
- **Variable Name:** Enter a new name for a selected variable.
- **Var Type:** Displays the type of data of the selected variable in the Variable(s) field.
- **Var Location:** Indicates if variables are located at nodes or cell-centers.
- **Var Range -- Selected Zone:** Displays the Min and Max values for the selected variable in the selected zone.
- **Var Range -- Active Zone(s):** Displays the Min and Max values for the selected variable for all active zones.

On the Data Set page are:

- **Data Set Title:** Enter a title for the current data set, or edit an existing title. The default is the result of concatenating the titles specified in each `Title` record encountered in the data files making up the data set.
- **Data File(s):** Lists the names and paths of all external data files making up the current data set.
- **Num Zones:** Number of zones in the data set.
- **Num Vars:** Number of variables in the data set.
- **Total Elements:** Total number of elements in the data set.
- **Var Load Mode:** Depending on the method used, this displays either By Position or By Name.
- **Locked By:** This field will inform you if the current data set has been locked by an add-on. Add-ons can lock a data set which in turn prevents your from deleting zones or deleting the last frame associated with the data set.

On the Sharing page are:

- **Zone:** Use the drop-down to select the appropriate zone.
- **Variable:** Use the drop-down to select the appropriate variable.
- **Variable is Shared in Zone(s):** This list box displays and allows you to select individual shared variables.
- **Connectivity Shared with Zone(s):** This list box displays and allows you to select specific connectivity lists.

On the Journal page are:

- **Journaled data list box:** Lists currently journaled data.
- **List Commands:** Briefly summarizes actions in Tecplot as they apply to the data set.

- **Expand Commands:** Displays the commands above in detail, including such things as the zone number, variable, and value.

On the Aux Data page are:

- **Show Auxiliary Data:** Use the drop-down to display auxiliary data for zones, data sets, frames or names. If Zone or Name is selected, a second drop-down is available for further selection.
- **Data Name/Value:** Displays the names and values of any auxiliary data.

CHAPTER 4 ASCII Data for Tecplot

This chapter discusses how to format data so data files may be loaded directly into Tecplot. Data files read by Tecplot may be binary or ASCII. The following sections describe the format of ASCII data files. Reading an ASCII data file into Tecplot can be much slower than reading a binary data file, as binary data files take up less disk space.

Tecplot or Preplot converts ASCII data files to binary. See Section 5.1, "Tecplot-Format Data File Loading," for converting with Tecplot, or Section 4.5, "ASCII Data File Conversion to Binary," for converting with Preplot. Documentation on the binary format is included as comments in the Preplot source code. Finally, if your data is generated in FORTRAN or C, you may be able to generate binary data files directly using the utilities described in Chapter 11, "Writing Binary Data for Loading into Tecplot," of the *Tecplot Reference Manual*.

You can also load data generated by, or tabulated in, other software packages. In addition, data loaders using Tecplot's Add-on Developer's Kit (ADK) are available from Amtec. These convert data from a number of software packages into a Tecplot-readable format. (This is described in Chapter 6, "Data Loaders: Tecplot's Import Feature.") Using ADK, you can write loaders of your own.

4.1. ASCII Data File Records

An ASCII data file begins with an optional file header defining a title for the data file and or the names of the variables. The header is followed by optional zone records containing the plot data. Zone records may contain ordered or finite-element data. You may also include text, geometry, and custom-label records that create text, geometries, and/or custom labels on plots. Each data file may have up to 32,700 zone records, ten custom label records, and any number of text and geometry records. These records may be in any order.

The first line in a zone, text, geometry, custom label, or data set auxiliary data record begins with the keyword **ZONE**, **TEXT**, **GEOMETRY**, **CUSTOMLABELS**, **DATASETAUXDATA,** or **VARAUXDATA.** The maximum length of a line in a data file is 4,000 characters (unless you edit and recompile the Preplot source code). Any line may be continued onto one or more following lines (except for text enclosed in double quotes ["]). Double quotes must be used to enclose character strings with embedded blank spaces or other special characters. A backslash (****) may be used to remove the significance of (or escape) the next character (that is, **\"** produces a single double-quote). Any line beginning with an octothorp (**#**) is treated as a comment and ignored.

The following simple example of a Tecplot ASCII data file has one small zone and a single line of text:

```
TITLE="Simple Data File"
VARIABLES="X" "Y"
ZONE I=4 F=POINT
1 1
2 1
2 2
1 2
TEXT X=10 Y=90 T="Simple Text"
```

The format of the ASCII data file is summarized in Section 4.1.8, “Summary of Data File Records.”

4.1.1. File Header

In the file header of your data file, you may specify an optional title that is displayed in the headers of Tecplot frames. The title line begins with **TITLE=**, followed by the title text enclosed in double-quotes. You may also assign a name to each of the variables by including a line that begins with **VARIABLES=**, followed by each variable’s name enclosed in double quotes. The quoted variable names should be separated by spaces or commas. Tecplot calculates the number of variables (*N*) from the list of variable names. If you do not specify the variable names (and your first zone has **POINT** data packing), Tecplot sets the number of variables equal to the number of numeric values in the first line of zone data for the first zone, and names the variables **V1**, **V2**, **V3**, and so forth.

Initially, Tecplot uses the first two variables in data files as the X- and Y-coordinates, and the third variable for the Z-coordinate of 3-D plots. You may, however, order the variables in the data file any way you want, since you can interactively reassign the variables to the X-, Y-, and or Z-axes using Tecplot dialogs.

Dataset and variable auxiliary data is added to the datafile using the DATASETAUXDATA and VARAUXDATA records. Auxiliary data are name/value pairs that a user can specify and then use in Tecplot with dynamic text, equations, macros, or add-ons. Multiple auxiliary data can be added at the dataset level as follows:

```
DATASETAUXDATA SampleNumber=”5”
DATASETAUXDATA AOA=”5.7”
```

Variable auxiliary data is added to Tecplot on a per variable basis. Like dataset auxiliary data multiple items can be added for each variable:

```
VARAUXDATA 1 MyData=”Hello”
VARAUXDATA 1 MoreData=”World”
VARAUXDATA 2 MyData=”More information”
VARAUXDATA 2 MoreData=”hi mom”
VARAUXDATA 2 MyExtraData=”Some extra data”
```

The variable number with which the auxiliary data is associated immediately follows the VARAUXDATA record. Also note that the data associated with a particular auxiliary data name are unique for each variable. Therefore the same named item can be added to each variable if desired. Conversely a particular auxiliary data item can be added to only one variable.

If the file header occurs in a place other than at the top of the data file, a warning is printed and the header is ignored. This allows you to concatenate two or more ASCII data files before using Tecplot (provided each data file has the same number of variables per data point).

4.1.2. Zone Records

A zone record consists of a control line that begins with the keyword **ZONE** followed by a set of numerical data called the zone data. The format of the zone control line is shown in Section 4.1.8, “Summary of Data File Records.”

4.1.2.1. The ZONETYPE Parameter. The zone data are of the type specified by the **ZONETYPE** parameter in the control line. There are two basic types of zones: ordered and finite-element. Ordered zones have the formats **ZONETYPE=ORDERED**. Finite-element zones have the specific **ZONETYPE** of **FELINESEG**, **FETRIANGLE**, **FEQUADRILATERAL**, **FETETRAHEDRON**, or **FEBRICK**. **ORDERED** is presumed if the **ZONETYPE** parameter is omitted. See Section 4.2, “Ordered Data,” for more information on ordered zones, and Section 4.3, “Finite-Element Data,” for details on finite-element data.

4.1.2.2. The DATAPACKING Parameter. The zone data packing is specified by the **DATAPACKING** parameter in the control line. There are two data packing options: **POINT** and **BLOCK**.

In **POINT** format, the values for all variables are given for the first point, then the second point, and so on. In **BLOCK** format, all of the values for the first variable are given in a block, then all of the values for the second variable, then all of the values for the third, and so forth. Zones with cell-centered data must use **BLOCK** data packing. More detail on this is given below.

4.1.2.3. POINT Format Example. If you have only one zone of data in **POINT** data packing format, and it is one-dimensional (that is, *JMax=1*, *KMax =1*), you may omit the zone control line. If you want Tecplot to determine the number of variables, you may create a data file with only the zone data, such as the following:

```
12.5 23 45 1.
14.3 24 46 2.
12.2 24 50 3.
13.3 26 51 4.
13.5 27 55 5.
```

Tecplot calculates the number of data points (*IMax*) in the zone by assuming that each row represents a data point and each column represents a variable, and creates an I-ordered zone. This type of structure is good for XY-plots and scatter plots. If there are multiple zones, two- or three-dimensional zones, finite-element zones, or **BLOCK**-format zone data, you must include a zone control line at the beginning of each zone record.

4.1.2.4. Data Types. Each variable in each zone in the data file may have its own data type. Tecplot supports the following six data types:

- **SINGLE** (four-byte floating point values).
- **DOUBLE** (eight-byte floating point values).
- **LONGINT** (four-byte integer values).
- **SHORTINT** (two-byte integer values).
- **BYTE** (one-byte integer values, from 0 to 255).
- **BIT.**

The data type determines the amount of storage Tecplot assigns to each variable. Therefore, the lowest level data type should be used whenever possible. For example, imaging data, which usually consists of numerical values ranging from zero to 255, should be given a data type of **BYTE**. By default, Tecplot treats numeric data as data type **SINGLE**. If any variable in the zone uses the **BIT** data type, the zone format must be **BLOCK** or **FEBLOCK**; you cannot use **POINT** or **FEPOINT** format.

4.1.2.5. Variable Location. Each variable in each zone in a data file may be located at the nodes or the cell-centers. Each variable is specified as **NODAL** or **CELLCENTERED** in the **VARLOCATION** parameter array, located in the control line. The format is:

VARLOCATION=([*set-of-vars*]=var-location,[*set-of-vars*]=var-location, ...)

where *set-of-vars* is the set of the variables and var-location is either **NODAL** or **CELLCENTERED**. Variables omitted from the list are assumed to be **NODAL**. For example:

```
VARLOCATION=([3-7,10]=CELLCENTERED, [11-12]=CELLCENTERED)
```

specifies that variables 3 through 7, 10, 11 and 12 are cell-centered and all other variables are, by default, nodal for this zone.

All cell-centered variables must list one value for each element. With nodal variables, one value must be listed for each node. Zones with cell-centered variables must be in **BLOCK** data packing format.

4.1.2.6. Data Lists. Numerical values in zone data must be separated by one or more spaces, commas, tabs, new lines, or carriage returns. Blank lines are ignored. Integer (**101325**), floating point (**101325.0**), and exponential (**1.01325E+05**) numbers are accepted. To repeat a particular number in the data, precede it with a repetition number as follows: "*Rep**Num*," where *Rep* is the repetition factor and *Num* is some numeric value to be repeated. For example, you may represent 37 values of 120.5 followed by 100 values of 0.0 as follows:

```
37*120.5, 100*0.0
```

4.1.2.7. Zone Auxiliary Data. Auxiliary data strings associated with the current zone are specified with **AUXDATA** parameter in the control line. This auxiliary data may be used in dynamic text, equations, macros, or add-ons. Auxiliary data is provided as named strings:

```
AUXDATA TIME="October 13, 2002, 8 A.M."
```

There may be multiple **AUXDATA** parameters in the control line for a zone, but names must be unique.

4.1.2.8. Variable Sharing between Zones. Frequently, some variables are exactly the same for a set of zones. For example, a series of zones may contain measurement or simulation data at the same XYZ-locations, but different times. In this case, Tecplot's memory usage may be dramatically reduced by sharing the coordinate variables between the zones. The zones that variables are shared from are specified in the **VARSHARELIST** in the control line of the current zone. The format is:

VARSHARELIST=([*set-of-vars***]=***zzz***, [***set-of-vars***]=***zzz***)**

where *set-of-vars* is the set of variables that are shared and *zzz* is the zone they are shared from. If *zzz* is omitted, the variables are shared from the previous zone. For example:

```
VARSHARELIST=([4-6,11]=3, [20-23]=1, [13,15])
```

specifies that variables 4, 5, 6 and 11 are shared from zone 3, variables 20, 21, 22, and 23 are shared from zone 1, and variables 13 and 15 are shared from the previous zone. For variable sharing, ordered zones may only share with ordered zones having the same dimensions. Finite-element zones may share with any zone having the same number of nodes, for nodal variables, or the same number of cells, for cell-centered data.

The connectivity list (finite-element only) and face-neighbors may be shared between zones using the **CONNECTIVITYSHAREZONE** parameter in the control line of the current zone. The format is:

CONNECTIVITYSHAREZONE=*nnn*

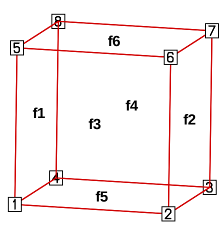
where *nnn* is the number of the zone that the connectivity is shared from. To use connectivity sharing, the zone must have the same number of points and elements, and be the same zone type.

4.1.2.9. Face Neighbors. The implicit connections between elements in a zone may be overridden, or connections between cells in adjacent zones established, using the **FACENEIGHBORMODE** parameter and **FACENEIGHBORCONNECTIONS** list in the control line of the zone. **FACENEIGHBORMODE** has four options: **LOCALONETOONE**, **LOCALONETOMANY**, **GLOBALONETOONE**, and **GLOBALONETOMANY**. **LOCALONETOONE** is the default.

The nature of the **FACENEIGHBORCONNECTIONS** list depends upon the **FACENEIGHBORMODE**, described in the table below. To connect the cells along one edge to cells on another edge of the same zone, use **LOCAL**. To connect cells of one zone to cells of another zone or zones, use **GLOBAL**. If the points of the cells are exactly aligned with the neighboring cell points, use **ONETOONE**. If even one cell face is neighbor to two other cell faces, use **ONETO-MANY**.

Mode	Number of Values	Data
LOCALONETOONE	**3**	**cz, fz, nc**
LOCALONETOMANY	**nz+4**	**cz, fz, oz, nz, nc1, nc2, ..., ncn**
GLOBALONETOONE	**4**	**cz, fz, zr, cr**
GLOBALONETOMANY	**2*nz+4**	**cz, fz, oz, nz, zr1, cr1, zr2, cr2, ..., zrn, crn**

In this table, **cz** is the cell number in the current zone, **fz** is the number of the cell face in the current zone, **nc**, is the cell number of the neighbor cell in the current zone, **oz** is face obscuration flag (zero for face partially obscured, one for face entirely obscured), **nz** is the number of neighboring cells for the **ONETOMANY** options, **ncn** is the number of the *nth* local zone neighboring cell in the list, **zr** is the remote zone number, **cr** is the cell number of the neighboring cell in the remote zone, **zrn** is the zone number of the nth neighboring cell in the **GLOBALONETOMANY** list, and **crn** is the cell number in the remote zone of the *nth* neighboring cell in the **GLOBALONETOMANY** list. The **cz**, **fz** combinations must be unique; multiple entries are not allowed. The face numbers for cells in the various zone types are defined in Figure 4-1.



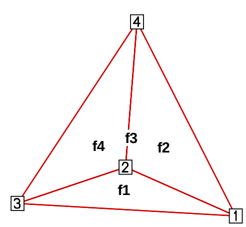


Figure 4-1. Examples of brick (left) and tetrahedron (right) face neighbors.

A connection must be specified for two matching cell faces to be effective. For example, for data with a FACENEIGHBORMODE of GLOBALONETOONE, if cell 6, face 2 in zone 9 should be connected to cell 1, face 4 in zone 10, the connections for zone 9 must include the line:

6 2 10 1 (cell# face#, connecting zone#, connecting cell#)

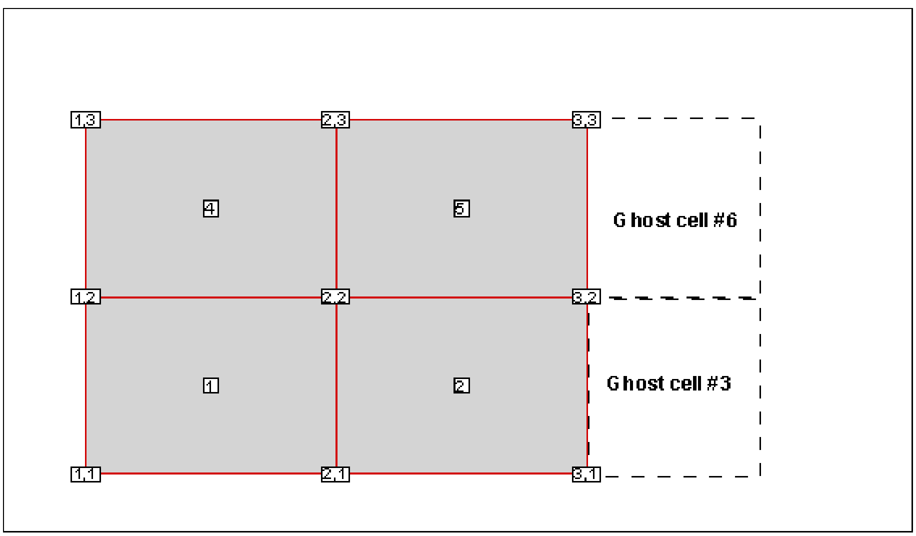
And the connections for zone 10 must include this line:

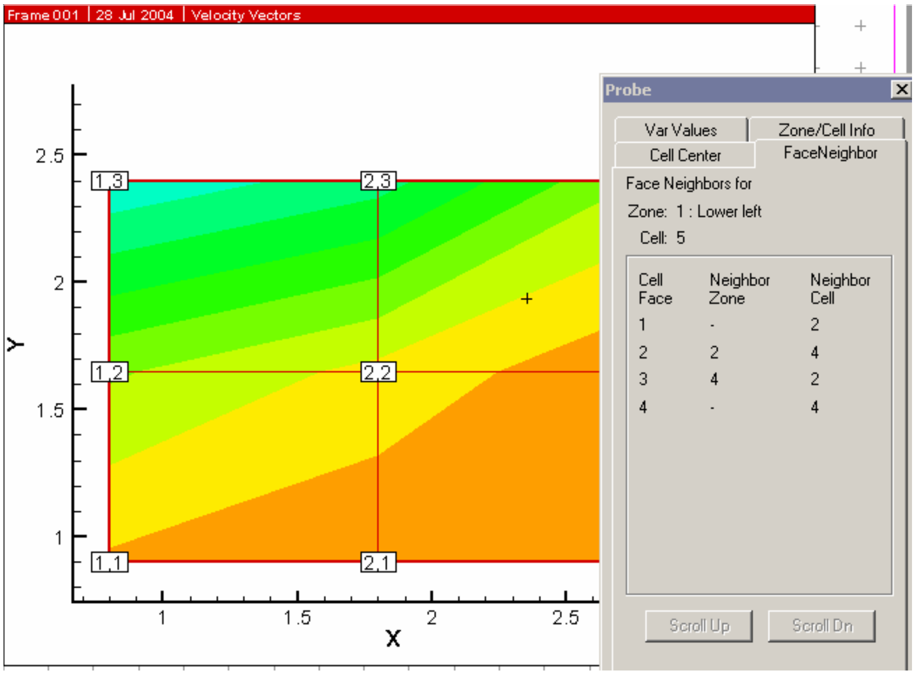
1 4 9 6

Global face neighbors are useful for telling Tecplot about the connections between zones. This could be used, for example, to smooth out the crease in Gouraud surface shading at zone boundaries. For cell-centered data, they can make contours and streamtraces more continuous at zone boundaries.

Cell numbering

For ordered (IJ or IJK) zones, cell numbers are defined by the index value of the first node, where Index = I + (J-1)*|MAXI| + (K-1)*|MAXI|*|MAXJ|. Because the number of nodes in each direction is one greater than the number of cells in that direction, there is no cell to correspond with the last point in each row. In the example below, there is no cell numbered "3", yet the first cell in the second row is numbered "4". As you define face neighbors, it may help you to think of a "ghost cell" at the end of each row (where I = MaxI) and at the end of each column in 3D (where J = MaxJ). If you probe any cell, the Face Neighbor tab of the Probe dialog will show the correct cell number.





4.1.2.10. Zone Types and Control Lines. As stated above, there are two distinct types of zones: ordered zones and finite-element zones. Ordered zones are I-, IJ-, and IJK-ordered zones (**ZONETYPE=ORDERED**). Finite-element zones are FE-line, -surface and -volume zones (**ZONETYPE** of **FELINESEG**, **FETRIANGLE**, **FEQUADRILATERAL**, **FETETRAHEDRON**, and **FEBRICK**). The control lines for these zone types differ in the parameters needed. Both zone types can use the **C** (*color*), **T** (*zonetitle*), **DATAPACKING**, **VARLOCATION**, **AUXDATA**, and **DT** (*datatype*) parameters.

The **T** parameter specifies a title for the zone. This may be any text string up to 64 characters in length. If you supply a longer text string, it is automatically truncated to the first 64 characters. The titles of zones appear in the Zone Style and other dialogs, and, optionally, in the XY- plot legend. (You can use keywords in the zone titles to identify sets of zones to enable/disable or to change zone attributes.) The **C** parameter sets an initial color for the zone. This may be overridden interactively, or by use of a stylesheet. The **DT** (*type1, type2, type3, ...*) parameter specifies the data types for the variables in a zone.

For ordered zones, you may specify the **I** (*IMax*), **J** (*JMax*), and **K** (*KMax*) parameters, which store the number of data points in the I, J, and K directions. **J** and **K** both default to 1. **I** must be specified if **J** is used; **I** and **J** must be specified if **K** is used. If all are omitted, Tecplot assumes an I-ordered zone and calculates *IMax* for you.

Note: **`I`** and **`J`** are not equivalent to either the number of variables or the number of data points. The number of data points is equal to the product of **`I`**, **`J`**, and **`K`**.

For finite-element zones, described in Section 4.3, "Finite-Element Data," you must specify the **`N`** (*numnodes*) and may optionally include **`E`** (*numelements*), and or the **`NV`** (*nodevalue*) parameter. If the **`E`** parameter is not specified, Tecplot calculates it from the number of node sets in the connectivity list following the node data. The **`NV`** (*nodevalue*) parameter specifies the number of the variables representing the "Node" value in finite-element data. The **`NV`** parameter is used infrequently, mostly when the order in which nodes are listed in the data file do not match the node numbering desired in the plot.

Section 4.2, "Ordered Data," provides examples of zone data in various formats, as well as sample pieces of FORTRAN code that you can use as templates to print out your own data. Our sample code is intended only as a general example—the zone data that it produces contains only one value per line. You may want to modify the code to suit your own needs.

4.1.3. Text Record

Text records are used to import text directly from a data file. Text can also be imported into Tecplot using a macro file. Text may be titles, labels, or other information. You may create data files containing only text records and read them into Tecplot just as you would read any other data file. You may delete and edit text originating from data files just like text created interactively.

The text record consists of a single control line. The control line starts with the keyword **`TEXT`** and has one or more options:

- The text string is defined in the required **`T`** (*text*) parameter.
- The color is controlled by the **`C`** (*color*) parameter.
- Use the **`CS`** (*coordinatesys*) parameter to specify the text coordinate system, either **`FRAME`** or **`GRID`**. If you specify the frame coordinate system (the default), the values of the **`X`** (*xorigin*) and **`Y`** (*yorigin*) parameters are in frame units; if you specify grid coordinates, **`X`** and **`Y`** are in grid units (that is, units of the physical coordinate system). **`X`** and **`Y`** locate the anchor point of the text string. For Polar Line plots, you may specify **`THETA`** and **`R`** instead of **`X`** and **`Y`**.
- Use the **`AN`** (*textanchor*) parameter to specify the position of the anchor point relative to the text. There are nine possible anchor positions, as shown in Figure 4-2.
- Use the **`HU`** (*heightunits*) parameter to assign units for character heights. If the **`CS`** parameter is **`FRAME`**, you can set **`HU`** to either **`FRAME`** or **`POINT`**. If the **`CS`** parameter is **`GRID`**, you can set **`HU`** to either **`GRID`** or **`FRAME`**.
- Use the **`H`** parameter to specify the height; it is measured in the units defined by the **`HU`** parameter.
- To include multiple lines of text in a single text record, include **`\\n`** in the text string to indicate a new line.

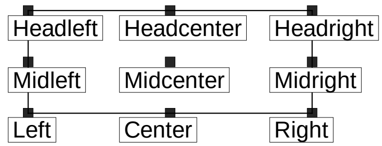


Figure 4-2. Text anchor positions—values for the **AN** parameter.

- You can assign the line spacing for multi-line text using the **LS** (*linespacing*) parameter. The default value, 1, gives single-spacing. Use 1.5 for line-and-a-half spacing, 2 for double-spacing, and so on.

Optionally, you may draw a box around the text string using the **BX** (*boxtype*) parameter. The parameters **BXO** (*boxoutlinecolor*), **BXM** (*boxmargin*), and **LT** (*linethickness*) are used if the *boxtype* is **HOLLOW** or **FILLED**. The parameter **BXF** (*boxfillcolor*) is used only if the *boxtype* is **FILLED**. The default *boxtype*, **NOBOX**, ignores all other *box* parameters.

The **S** (*scope*) parameter specifies the text scope. **GLOBAL** scope is the same as selecting the check box Show in “Like” Frames in the Text Options dialog. See Section 16.1.8, “Text Scope,” for details.

You may also use the **ZN** (*zone*) parameter to attach text to a specific zone or XY mapping. For further information, see Section 16.1.6.4, "Attaching Text to Zones or X-Y Mapping."

4.1.3.1. Text Record Examples. You may attach a macro command to the text with the **MFC** parameter. See Section 16.6, “Text and Geometry Links to Macros.”

Some simple examples of text records are shown below. The first text record specifies only the origin and the text. The next text record specifies the origin, color, font, and the text. The last text record specifies the origin, height, box attributes, and text. Note that the control line for the text can span multiple file lines if necessary (as in the last text record below).

```
TEXT X=50, Y=50, T="Example Text"

TEXT X=10, Y=10, F=TIMES-BOLD, C=BLUE, T="Blue Text"

TEXT X=25, Y=90, CS=FRAME, HU=POINT, H=14,
    BX=FILLED, BXF=YELLOW, BXO=BLACK, LS=1.5,
    T="Box Text \\n Multi-lined text"
```

4.1.4. Geometry Record

Geometry records are used to import geometries from a data file. Geometries are line drawings that may be boundaries, arrows, or even representations of physical structures. You may create data files containing only geometry and text records and read them into Tecplot. You may delete and edit geometries originating from data files just like the geometries that you create interactively.

The geometry record control line begins with the keyword **GEOMETRY**. Use the **CS** (*coordinatesys*) parameter to specify the geometry coordinate system, either **FRAME** or **GRID**. If you specify the frame coordinate system (the default), the values of the **X** (*xorigin*) and **Y** (*yorigin*) parameters are in frame units; if you specify grid coordinates, **X** and **Y** are in grid units (that is, units of the physical coordinate system). For Polar Line plots, you may specify **THETA** and **R** for **X** and **Y**. **X** and **Y** (or **THETA** and **R**) locate the anchor point, or origin, of the geometry, which is the center of a circle or ellipse, the lower left corner of a square or rectangle, and the anchor point of a polyline. The anchor point specifies the offset of all the points: if *X=1*, *Y=1*, and the first point is (1, 2), and the second point is (2, 4), then Tecplot draws at (2, 3) (1+1, 2+1) then (3, 5) (2+1, 4+1). In other words, the points for any geometry are always relative to the specified anchor point. The **Z** (*zorigin*) is specified only for **LINE3D** geometries, and, since **LINE3D** geometries are always in grid mode, **Z** is always in units of the Z-axis.

Geometry types are selected with the **T** (*geomtype*) parameter. The available geometry types are listed below:

- **SQUARE**: A square with lower left corner at **X**, **Y**.
- **RECTANGLE**: A rectangle with lower left corner at **X**, **Y**.
- **CIRCLE**: A circle centered at **X**, **Y**.
- **ELLIPSE**: An ellipse centered at **X**, **Y**.
- **LINE**: A set of 2-D polylines (referred to as multi-polylines) anchored at **X**, **Y**.
- **LINE3D**: A set of 3-D polylines (referred to as multi-polylines) anchored at **X**, **Y**, **Z**.

The color of the geometry is controlled by the **C** (*color*) parameter. Any geometry type except **LINE3D** may be filled with a color by using the **FC** (*fillcolor*) parameter. With both **C** (*color*) and **FC** (*fillcolor*) on the control line, the geometry is outlined in one color and filled with another. Each polyline of a **LINE** geometry is filled individually (by connecting the last point of the polyline with the first). Not specifying the **FC** (*fillcolor*) parameter results in a hollow, or outlined, geometry drawn in the color of the **C** (*color*) parameter.

You can control how geometries are drawn using the **L** (*linetype*), **LT** (*linethickness*), and **PL** (*patternlength*) parameters. You can set **L** to any of Tecplot's line patterns (**SOLID**, **DASHED**, **DOTTED**, **DASHDOT**, **LONGDASH**, **DASHDOTDOT**). You can set **LT** and **PL** to any value, using frame units.

The control line of the geometry is followed by geometry data. For **SQUARE**, the geometry data consists of just one number: the side length of the square.

For **RECTANGLE**, the geometry data consists of two numbers: the first is the width (horizontal axis dimension), and the second is the height (vertical axis dimension).

For **CIRCLE**, the geometry data is one number: the radius. For **ELLIPSE**, the geometry data consists of two numbers: the first is the horizontal axis length and the second is the vertical axis length. For both circles and ellipses, you can use the **EP** (*numellipsepts*) parameter to specify the number of points used to draw circles and ellipses. All computer-generated curves are simply collections of very short line segments; the **EP** parameter allows you to control how many line segments Tecplot uses to approximate circles and ellipses. The default is 72.

For **LINE** and **LINE3D** geometries, the geometry data is controlled by the **F** (*format*) parameter. These geometries may be specified in either **POINT** or **BLOCK** format. By default, **POINT** format is assumed. Each geometry is specified by the total number of polylines, up to a maximum of 50. Each polyline is defined by a number of points and a series of XY- or XYZ-coordinate points between which the line segments are drawn. In **POINT** format, the XY- or XYZ-coordinates are given together for each point. In **BLOCK** format, all the X-values are listed, then all the Y-values, and (for **LINE3D** geometries) all the Z-values. All coordinates are relative to the **X**, **Y**, and **Z** specified on the control line. You can specify points in either single or double precision by setting the **DT** (*datatype*) parameter to either **SINGLE** or **DOUBLE**.

For **LINE** geometries, you can specify arrowheads using the **AAT** (*arrowheadattach*), **AST** (*arrowheadstyle*), **ASZ** (*arrowheadsize*), and **AAN** (*arrowheadangle*) parameters. See Section 4.1.8, "Summary of Data File Records," for details. These parameters provide the same functionality available when you create a line geometry interactively.

The **S** (*scope*) parameter specifies the geometry's scope. **GLOBAL** scope is the same as selecting the check box Show in Like Frames in the Geometry dialog. See Section 16.2.2.8, "Geometry Scope," for details.

You may also use the **ZN** (*zone*) parameter to attach geometry to a specific zone or XY-mapping.

You may attach a macro command to the text with the **MFC** parameter. See Section 16.6, "Text and Geometry Links to Macros."

LINE3D geometries must be created in a data file. They may not be created interactively. **LINE3D** geometries are always in grid mode. To view **LINE3D** geometries in Tecplot, your plot type must be in 3D Cartesian, which requires at least one zone. Thus, a data file with only **LINE3D** geometries is useful only as a supplement to other data files.

4.1.4.1. Geometry Record Examples. The following geometry record defines a rectangle of **40** width and **30** height:

```
GEOMETRY  T=RECTANGLE
40 30
```

The following geometry record defines an origin and a red circle of **20** radius, with an origin of (**75**, **75**) that is filled with blue:

```
GEOMETRY X=75, Y=75, T=CIRCLE, C=RED, FC=BLUE,CS=FRAME
 20
```

The following geometry record defines an origin and two polylines, drawn using the Custom 3 color. The first polyline is composed of three points, the second of two points.

```
GEOMETRY X=50, Y=50, T=LINE, C=CUST3
 2
```

```
3
0 1
0 0
2 0
2
0 0
1 2
```

In **BLOCK** format, the same geometry appears as:

```
GEOMETRY X=50, Y=50, T=LINE, C=CUST3, F=BLOCK, CS=FRAME
 2
 3
 0 0 2
 1 0 0
 2
 0 1
 0 2
```

The next geometry record defines a purple ellipse with a horizontal axis length of `20` and a vertical axis length of `10`, with an origin of (`10, 70`), that is filled with yellow.

```
GEOMETRY X=10, Y=70, T=ELLIPSE, C=PURPLE, FC=YELLOW
 20 10
```

The final geometry record is a 3-D polyline with four points that is composed of one polyline using the default origin of (`0, 0, 0`):

```
GEOMETRY T=LINE3D
 1
 4
 0 0 0
 1 2 2
 3 2 3
 4 1 2
```

In **BLOCK** format, this geometry record can be written as follows:

```
GEOMETRY T=LINE3D, F=BLOCK
 1
 4
 0 1 3 4
 0 2 2 1
 0 2 3 2
```

4.1.5. A More Extensive Example of a Geometry Record

In the **TextGeom** file shown below, there are four text records (showing a circle, ellipse, rectangle, and line). A plot of the file is shown in Figure 4-3.

```
TEXT X=20, Y=85, F=HELV-BOLD, C=BLUE, H=7.5,
      T="Example Text"
TEXT X=20, Y=75, F=TIMES-BOLD, H=5, T="Subtitle"
TEXT X=80, Y=25, F=TIMES-ITALIC-BOLD, H=4, C=RED,
      BX=FILLED, BXF=YELLOW, BXM=50, BXO=CYAN,
      T="Filled Box"
```

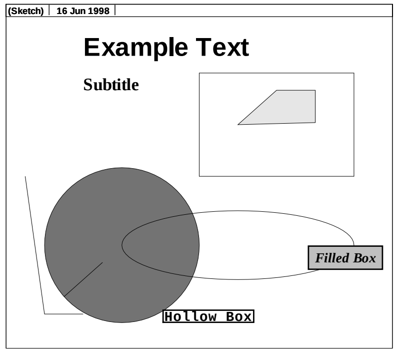


Figure 4-3. Text and geometries created from the sample in Section 4.1.5, "A More Extensive Example of a Geometry Record."

```
TEXT X=41, Y=8, H=4, F=COURIER-BOLD,
      C=CUST3, BX=HOLLOW, BXO=CUST4, T="Hollow Box"
GEOMETRY X=50, Y=50, T=RECTANGLE, FC=WHITE, C=BLUE
   40 30
GEOMETRY X=30, Y=30, T=CIRCLE, FC=BLUE, C=GREEN
   20
GEOMETRY X=70, Y=65, T=LINE, FC=PURPLE, C=BLACK
   1
   4
   -10 0
   0 10
   010 10
   10 0.6
GEOMETRY T=LINE, C=CUST1
   2
   3
   5 50
   10 10
   20 10
   2
   15 15
   25 25
GEOMETRY X=60, Y=30, T=ELLIPSE, C=CUST8
   30 10
```

4.1.6. Custom Label Record

The custom label record is an optional record to define sets of text strings for use in custom labeling the values of an axis, contour legend or value labels, or variable-value node labels. The custom label record begins with the keyword **`CUSTOMLABELS`**, followed by one or more text strings. The text strings *must* be enclosed within double quotes ("`"`") if they contain any commas, spaces or other special characters, or if they might be confused with valid data file keywords. Enclosing the strings in double quotes is always recommended.

The first custom label string corresponds to a value of one on the axis, the next to a value of two, the next to a value of three, and so forth. Custom labels may appear one to a line, or there may be more than one on a line, separated by a comma or space. Multiple custom label records can be present in a data file. If this is the case, you choose which set to assign to a given axis, contour legend, or variable-value node labels. Custom labels are discussed in more detail in Section 17.5.2.3, “Custom Labels.”

A simple example of a custom-label record is shown below. **MON** corresponds to a value of **1**, **TUE** corresponds to **2**, **WED** to **3**, **THU** to **4**, and **FRI** to **5**. Since custom labels have a wrap-around effect, **MON** also corresponds to the values **6**, **11**, and so forth.

```
CUSTOMLABELS "MON", "TUE", "WED", "THU", "FRI"
```

4.1.7. Data Set Auxiliary Data Record

There is frequently auxiliary data (or Metadata) that helps describe the data set. For example, experimental data my have information about the facility and time at which the data was taken, and other parameter that describe the experiment. Likewise, simulation results have auxiliary data (such as reference quantities for non-dimensional data) needed to fully analyze and present the results. This data may be concerning the data set as a whole or it can vary from zone to zone. The ASCII file format for specifying auxiliary data associated with the data set are described here. The format for zonal auxiliary data is described in Section 4.1.2.7.

The data set auxiliary data control line is as follows:

```
DATASETAUXDATA name-string = “value string”
```

where name-string is a unique character string with no spaces. There may be multiple **DATASETAUXDATA** records but name-string must be unique for each one.

Auxiliary data may be used in text, macros, equations (if it is numeric), and accessed from add-ons. It may also be viewed directly in the AuxData page of the Data Set Information dialog.

4.1.7.1. Data Set Auxiliary Data Examples. The following auxiliary data contain flow field information that might be found in output from a computational fluid-dynamics simulation.

```
DATASETAUXDATA MachNo = "1.2"
DATASETAUXDATA Alpha = "5"
DATASETAUXDATA RefTemperature = "250"
DATASETAUXDATA RefPressure = "101325"
DATASETAUXDATA Configuration = "A2 No. 3"
DATASETAUXDATA Date = "August 5, 2003"
DATASETAUXDATA Region = "NE Quadrant of Sector 47"
```

You may then use the numerical values in equations to modify the variables like this:

```
{P} = {P_non_dim} * AuxDataSet:RefPressure
```

`Configuration` and **`Date`** may then be included in text on the plot. This makes it easier to automate your plotting tasks using layout files and/or macros.

4.1.8. Summary of Data File Records

The following table summarizes the records and parameters allowable in Tecplot data files.

Data File Section	**Records**	**Parameter Descriptions**
File Header	**`TITLE = "filetitle"`**	Title for data file.
	`VARIABLES =`	
	`"`*vname1*`"`	Name of first variable.
	`"vname2" ...`	Name of second variable.
	`DATASETAUXDATA`name=”Value String”	name/value pair
	`VARAUXDATA` varnum name=”Value String”	variable #’s name/value pair

Data File Section	Records	Parameter Descriptions
Ordered Zone Record	**`ZONE`**	
	`T="zonetitle"`	Title for zone.
	`I=IMax`	Number of points in I-direction.
	`J=JMax`	Number of points in J-direction.
	`K=KMax`	Number of points in K-direction.
	`C=color`	One of the following: **`BLACK`**, **`RED`**, **`GREEN`**, **`BLUE`**, **`CYAN`**, **`YELLOW`**, **`PURPLE`**, **`WHITE`**, **`CUST1`**, **`...`**, **`CUST8.`**
	`ZONETYPE=ordered`	
	`DATAPACKING=datapacking format`	Either **`POINT`** or **`BLOCK.`**
	`VARLOCATION=([set-of-vars]=var-location,[set-of-vars]=var-location,...)`	Lists specifying variable locations. The list, *nnn*, contains variable numbers and *var-location* is either **`CELL-CENTERED`** or **`NODAL`**.
	`VARSHARELIST=([set-of-vars]=zzz,[set-of-vars]=zzz,...)`	Lists of variables to share from specified previous zones. For each pair specified, the variables listed in *set-of-vars* are shared from zone *zzz*.
	`DT=(datatypelist)`	List specifying data type for each variable, from among the following: **`SINGLE`**, **`DOUBLE`**, **`LONGINT`**, **`SHORTINT`**, **`BYTE`**, **`BIT.`**
	`AUXDATA namestring="value string"`	Saves the name/value pairs of strings associated with the zone. May have multiple **`AUXDATA pairs defined for each zone.`**
	`FACENEIGHBORMODE=faceneighbor-mode`	One of the following: **`LOCALONETOONE`**, **`LOCALONETOMANY`**, **`GLOBALONETOONE`**, **`GLOBALONETOMANY.`**
	`FACENEIGHBORCONNECTIONS=face-neighborconnectlist`	**`Choice of FACE-NEIGHBORMODE will change what is specified in`**

Data File Section	**Records**	**Parameter Descriptions**
Finite-element Zone Record	**ZONE** **T="zonetitle"** **N=numnodes** **E=numelements** **C=color** **ZONETYPE=zonetype** **DATAPACKING=datapacking format** **VARLOCATION=([set-of-vars]=var-location,[set-of-vars]=var-location,...)** **VARSHARELIST=([set-of-vars]=zzz,[set-of-vars]=zzz,...)** **AUXDATA namestring="value string"** **FACENEIGHBORMODE=faceneighbor-mode** **FACENEIGHBORCONNECTIONS=face-neighborconnectlist** **NV=nodevariable** **DT=(datatypelist)** **CONNECTIVITYSHAREZONE=nnn**	Title for zone. Number of nodes. Number of elements. **For finite-element zones the zonetype options are FELINESEG, FETRIANGLE, FEQUADRILATERAL, FETETRAHEDRON, and FEBRICK.** Either **POINT** or **BLOCK.** Lists specifying variable locations. The list, *nnn*, contains variable numbers and *var-location* is either **CELLCENTERED** or **NODAL**. Lists of variables to share from specified previous zones. For each pair specified, the variables listed in *set-of-vars* are shared from zone *zzz*. Saves the name/value pairs of strings associated with the zone. May have multiple **AUXDATA pairs defined for each zone.** One of the following: **LOCALONETOONE, LOCALONETOMANY, GLOBALONETOONE, GLOBALONETOMANY. Choice of FACENEIGHBORMODE will change what is specified in FACENEIGHBORCONNECTIONS.** *Faceneighborconnectlist*

Data File Section	Records	Parameter Descriptions
Text Record	**TEXT**	
	X=xorigin	*X* origin of object in *coordinatesys* units.
	Y=yorigin	*Y* origin of object in *coordinatesys* units.
	Z=zorigin	*Z* origin of object in *coordinatesys* units.
	Theta=thetaorigin	*Theta* origin of object in *coordinatesys* units.
	R=rorigin	*R* origin of object in *coordinatesys* units.
	F=font	One of the following: **HELV**, **HELV-BOLD**, **TIMES**, **TIMES-ITALIC**, **TIMES-BOLD**, **TIMES-ITALIC-BOLD**, **COURIER**, **COURIER-BOLD**, **GREEK**, **MATH**, **USER-DEF**.
	THETA=originangle	**THETA** coordinate of origin for Polar Line plots (only if **X** and **Y** are not specified).
	R=originradius	**R** coordinate of origin for Polar Line plots (only if **X** and **Y** are not specified).
	CS=coordinatesys	One of **FRAME**, **GRID** or **GRID3D**.
	HU=heightunits	One of **FRAME**, **POINT** or **GRID** (**GRID** not allowed for **FRAME** coord sys).
	AN=textanchor	One of the following: **LEFT**, **CENTER**, **RIGHT**, **MIDLEFT**, **MIDCENTER**, **MIDRIGHT**, **HEADLEFT**, **HEADCENTER**, **HEADRIGHT**.
	C=color	See description above.
	A=angle	Angle in degrees, counterclockwise from horizontal.
	H=height	Character height in *heightunits*.
	LS=linespacing	Line spacing for multiple-line text.
	S=scope	Either **LOCAL** or **GLOBAL**.
	T="text"	Alphanumeric text string.
	BX=boxtype	One of **NOBOX**, **HOLLOW**, or **FILLED**.
	BXM=boxmargin	Margin around text as fraction of text height.
	BXF=boxfillcolor	Fill color for box: use *color*
	BXO=boxcolor	
	LT=boxlinethickness	
	ZN=zone	
	CLIPPING=clipping	
	MFC="*macrofunctioncommand***"**	

Data File Section	Records	Parameter Descriptions

Geometry Record	`GEOMETRY`
	`X=xorigin`
	`Y=yorigin`
	`Z=zorigin`
	`THETA=originangle`
	`R=originradius`
	`CS=coordinatesys`
	`C=color`
	`L=linetype`
	`PL=patternlength`
	`LT=linethickness`
	`T=geomtype`
	`EP=numellipsepts`
	`AST=arrowheadstyle`
	`AAT=arrowheadattach`
	`ASZ=arrowheadsize`
	`AAN=arrowheadangle`
	`DT=datatype`
	`S=scope`
	`F=geomformat`
	`FC=geomfillcolor`
	`ZN=zone`
	`DRAWORDER=draworder`
	`CLIPPING=clipping`
	`MFC="`*`macrofunctioncommand`*`"`
		X-origin of object in *coordinatesys* units.
		Y-origin of object in *coordinatesys* units.
		For `GRID3D` geoms only, the *Z*-origin of object in *coordinatesys* units.
		For `GRID CS` only, `THETA` coordinate of origin for Polar Line plots (only if `X` and `Y` are not specified).
		For `GRID CS` only, `R` coordinate of origin for Polar Line plots (only if `X` and `Y` are not specified).
		One of `FRAME`, `GRID`, or `GRID3D`.
		One of the following: `SOLID`, `DASHED`, `DASHDOT`, `DOTTED`, `LONGDASH`, `DASHDOTDOT`.
		Pattern length for specified line type.
		Line thickness for geometry outline.
		One of the following: `LINE`, `SQUARE`, `RECTANGLE`, `CIRCLE`, `ELLIPSE` (Image geoms cannot be in data files).
		Number of points to use to approximate circles and ellipses.
		One of `PLAIN`, `HOLLOW`, or `FILLED`.
		One of the following: `NONE`, `BEGINNING`, `END`, `BOTH`.
		Size of arrowhead in Frame units.
		Angle of arrowhead in degrees.
		Either `SINGLE` or `DOUBLE` (applies to 2- and 3-D polylines only).
		Either `LOCAL` or `GLOBAL`.
		Either `POINT` or `BLOCK`.
		Fill color for geometry; use *color* options
		Zone (or XY-mapping) num-

Data File Section	Records	Parameter Descriptions
Custom Labels Record	**CUSTOMLABELS** **"label1"** **"label2" ...**	 String for value of one when using custom labels. String for value of two when using custom labels.
Data Set Auxiliary Data Record	**DATASETAUXDATA namestring="value string"**	Saves name/value pairs of strings associated with the data set. May have multiple **DATASETAUXDATA** pairs defined for the data set.

4.2. Ordered Data

For ordered data, the numerical values in the zone data must be in either **POINT** or **BLOCK** format, specified by the **DATAPACKING** parameter.

4.2.1. I-Ordered Data

I-ordered data has only one index, the I-index. This type of data is typically used for XY-plots, scatter plots, and irregular (random) data for triangulation or for interpolation into an IJ-or IJK-ordered zone within Tecplot.

In I-ordered data, the I-index varies from one to *IMax*. The total number of data points is *IMax*. For zones with only nodal variables, the total number of values in the zone data is *IMax*N* (where *N* is the number of variables). For a mixture of nodal and cell-centered variables, the number of values in the zone data is *IMax*Nn+(IMax-1)*Nc*, where *Nn* is the number of nodal variables and *Nc* is the number of cell-centered variables. For data in **POINT** format, *IMax* is calculated by Tecplot from the zone data if it is not explicitly set by the zone control line (using the **I-**parameter).

4.2.1.1. I-Ordered Data in POINT Format Example. A simple example of I-ordered data in **POINT** format is listed below. There are two variables (**X**, **Y**) and five data points. In this example, each row of data corresponds to a data point and each column to a variable. This data set is plotted in Figure 4-4; each data point is labeled with its I-index.

```
VARIABLES = "X","Y"
ZONE I=5, DATAPACKING=POINT
2    4
3    9
5   25
6   36
7   49
```

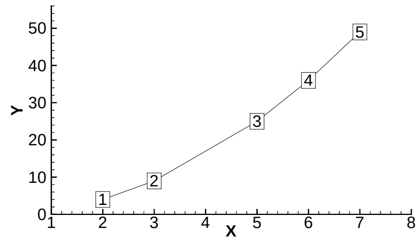


Figure 4-4. An I-ordered data set.

For this data try omitting the **VARIABLES** and **ZONE** lines, leaving two columns of information. In this case, Tecplot would count the columns to determine the number of variables, count rows to determine I-dimension, and label the variables V1 and V2.

4.2.1.2. FORTRAN Code Example to Generate I-Ordered Data in POINT Format.

The following sample FORTRAN code shows how to create I-ordered data in **POINT** format:

```
 INTEGER VAR
   .
   .
   .
 WRITE (*,*) ´ZONE DATAPACKING=POINT, I=´, IMAX
 DO 1 I=1,IMAX
     DO 1 VAR=1,NUMVAR
1        WRITE (*,*) ARRAY(VAR,I)
```

4.2.1.3. I-Ordered Data in BLOCK Format Example. The same data as in Section 4.2.1.1. is shown below in **BLOCK** format. In this example, each column of zone data corresponds to a data point; each row to a variable.

```
VARIABLES = "X", "Y"
ZONE I=5, DATAPACKING=BLOCK
2 3 5 6 7
4 9 25 36 49
```

In **BLOCK** format all *IMax* values of each variable are listed, one variable at a time.

4.2.1.4. FORTRAN Code to Generate I-Ordered Data in BLOCK Format Example.

The following sample FORTRAN code shows how to create I-ordered data in **BLOCK** format:

```
 INTEGER VAR
   .
   .
   .
```

```
 WRITE (*,*) ´ZONE DATAPACKING=BLOCK, I=´, IMAX
 DO 1 VAR=1,NUMVAR
     DO 1 I=1,IMAX
1        WRITE (*,*) ARRAY(VAR,I)
```

4.2.1.5. Multi-Zone XY Line Plot Example. The two tables below show the values of pressure and temperature measured at four locations on some object at two different times. The four locations are different for each time measurement.

Time = 0.0 seconds:

Position	Temperature	Pressure
71.30	563.7	101362.5
86.70	556.7	101349.6
103.1	540.8	101345.4
124.4	449.2	101345.2

Figure 4-5.

Time = 0.1 seconds:

Position	Temperature	Pressure
71.31	564.9	101362.1
84.42	553.1	101348.9
103.1	540.5	101344.0
124.8	458.5	101342.2

Figure 4-5.

For this case, we want to set up two zones in the data file, one for each time value. Each zone has three variables (**`Position`**, **`Temperature`**, and **`Pressure`**) and four data points (one for each location). This means that *IMax=4* for each zone. We include a text record (discussed in Section 4.1.3., "Text Record") to add a title to the plot. A data file in **`POINT`** format is given below. The plot shown in Figure 4-6 can be produced from this file.

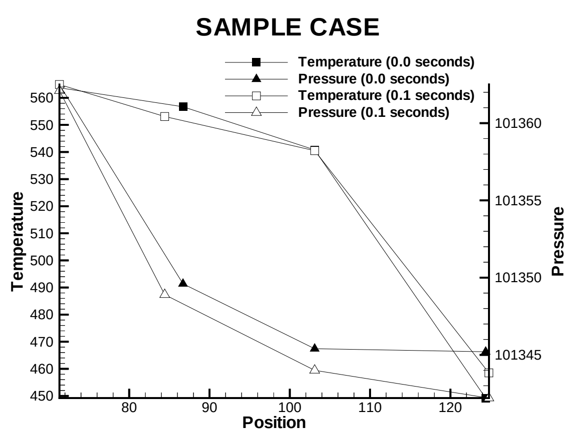


Figure 4-6. A multi-zone XY Line plot.

```
TITLE = "Example: Multi-Zone XY Line Plot"
VARIABLES = "Position", "Temperature", "Pressure"
ZONE T="0.0 seconds", I=4
```

```
71.30 563.7 101362.5
86.70 556.7 101349.6
103.1 540.8 101345.4
124.4 449.2 101345.2
ZONE T="0.1 seconds", I=4
71.31 564.9 101362.1
84.42 553.1 101348.9
103.1 540.5 101344.0
124.8 458.5 101342.2
TEXT CS=FRAME, HU=POINT, X=16, Y=90, H=28, T="SAMPLE CASE"
```

A data file in **BLOCK** format is shown below. All of the values for the first variable (**Position**) at each data point are listed first, then all of the values for the second variable (**Temperature**) at each data point, and so forth.

```
TITLE = "Example: Multi-Zone XY Line Plot"
VARIABLES = "Position", "Temperature", "Pressure"
ZONE DATAPACKING=BLOCK, T="0.0 seconds", I=4
71.30 86.70 103.1 124.4
563.7 556.7 540.8 449.2
101362.5 101349.6 101345.4 101345.2
ZONE DATAPACKING=BLOCK, T="0.1 seconds", I=4
71.31 84.42 103.1 124.8
564.9 553.1 540.5 458.5
101362.1 101348.9 101344.0 101342.2
TEXT CS=FRAME, HU=POINT, X=16, Y=90, H=28, T="SAMPLE CASE"
```

A more compact data file for this example is in the point format shown below. Tecplot determines the number of variables from the number of values in the first line of data under the first zone. The variables and zones are assigned default names.

```
ZONE
71.30 563.7 101362.5
86.70 556.7 101349.6 103.1 540.8 101345.4 124.4 449.2 101345.2
ZONE
71.31 564.9 101362.1 84.42 553.1 101348.9 103.1 540.5 101344.0
124.8 458.5 101342.2
TEXT CS=FRAME, HU=POINT, X=16, Y=90, H=28, T="SAMPLE CASE"
```

4.2.1.6. Multi-Zone XY Line Plot with Variable Sharing Example. If the data from the section above was taken at the same position for both times, variable sharing could reduce memory usage and file size. That file appears as:

```
TITLE = "Example: Multi-Zone XY Line Plot with Variable Sharing"
VARIABLES = "Position", "Temperature", "Pressure"
ZONE T="0.0 seconds", I=4
71.30 563.7 101362.5
86.70 556.7 101349.6
103.1 540.8 101345.4
124.4 449.2 101345.2
ZONE T="0.1 seconds", I=4, VARSHARELIST=([1]=1)
564.9 101362.1
553.1 101348.9
```

```
540.5 101344.0
458.5 101342.2
TEXT CS=FRAME, HU=POINT, X=16, Y=90, H=28, T="SAMPLE VARIABLE SHARING CASE"
```

4.2.2. IJ-Ordered Data

IJ-ordered data has two indices: I and J. IJ-ordered data is typically used for 2- and 3-D surface mesh, contour, vector, and shade plots, but it can also be used to plot families of lines in XY-plots. See Chapter 7, "Line Plots," for more information. In IJ-ordered data, the I-index varies from 1 to *IMax*, and the J-index varies from one to *JMax*. The total number of data points (nodes) is *IMax**JMax*. For zones with only nodal variables, the total number of numerical values in the zone data is *IMax*JMax*N* (where *N* is the number of variables). For a mixture of nodal and cell-centered variables, the number of values in the zone data is *IMax*JMax*Nn+(IMax-1)*(JMax-1)*Nc*, where *Nn* is the number of nodal variables and *Nc* is the number of cell-centered variables. Both *IMax* and *JMax* must be specified in the zone control line (with the **`I`** and **`J`** parameters). The I- and J-indices should not be confused with the X- and Y-coordinates—on occasions the two may coincide, but this is not the typical case.

The I-index varies the fastest. That is, when you write programs to print IJ-ordered data, the I-index is the inner loop and the J-index is the outer loop. Note the similarity between I-ordered data and IJ-ordered data with *JMax=1*.

4.2.2.1. IJ-Ordered Data in POINT Format Example. An example of IJ-ordered data in **`POINT`** format is listed below. There are four variables (**`X`**, **`Y`**, **`Temperature`**, **`Pressure`**)

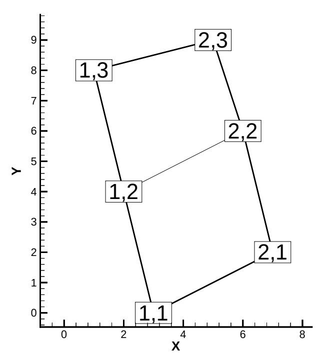


Figure 4-7. An IJ-ordered data set.

and six data points. In this example, each row of data corresponds to a data point; each column to a variable. The first two lines are for *J=1*, the next two for *J=2*, the last two for *J=3*. The first, third, and fifth lines are for *I=1*; the second, fourth, and sixth lines are for *I=2*. This data is plotted in Figure 4-7; each data point is labeled with its IJ-index.

```
VARIABLES = "X", "Y", "Temperature", "Pressure"
ZONE I=2, J=3, DATAPACKING=POINT
3 0 0 50
7 2 0 43
2 4 1 42
6 6 0 37
1 8 1 30
5 9 1 21
```

4.2.2.2. FORTRAN Code to Generate IJ-Ordered Data in POINT Format Example.

The following sample FORTRAN code shows how to create IJ-ordered data in **POINT** format:

```
WRITE (*,*) ´VARIABLES = "X", "Y", "Temperature", "Pressure"’
WRITE (*,*) ´ZONE I=’, IMAX, ’, J=’, JMAX, ’, ’DATAPACKING=POINT’
 DO 1 J=1,JMAX
    DO 1 I=1, IMAX
1            WRITE (*,*) X(I,J), Y(I,J), T(I,J), P(I,J)
```

4.2.2.3. IJ-Ordered Data Set in BLOCK Format Example. The same data set as in Section 4.2.2.1. is shown in **BLOCK** format below. In this example, each column of data corresponds to a data point; each row to a variable.

```
VARIABLES = "X", "Y", "Temperature", "Pressure"
ZONE I=2, J=3, DATAPACKING=BLOCK
3 7 2 6 1 5
0 2 4 6 8 9
0 0 1 0 1 1
50 43 42 37 30 21
```

In **BLOCK** format, all *IMax*JMax* values of each variable are listed, one variable at a time. Within each variable block, all the values of a variable at each data point are listed.

4.2.2.4. FORTRAN Code to Generate IJ-Ordered Data in BLOCK Format Example.

The following sample FORTRAN code shows how to create IJ-ordered data in **BLOCK** format:

```
 INTEGER VAR
   .
   .
   .
 WRITE (*,*) ´ZONE DATAPACKING=BLOCK, I=´, IMAX,  ´, J=´, JMAX
 DO 1 VAR=1,NUMVAR
     DO 1 J=1,JMAX
         DO 1 I=1,IMAX
1            WRITE (*,*) ARRAY(VAR,I,J)
```

4.2.2.5. IJ-Ordered Data with Cell-Centered Data. An example of IJ-ordered data with cell-centered variables might include four variables (**X**, **Y**, **Temperature**, **Pressure**), nine data points, and four cells where **Temperature** and **Pressure** are cell-centered.

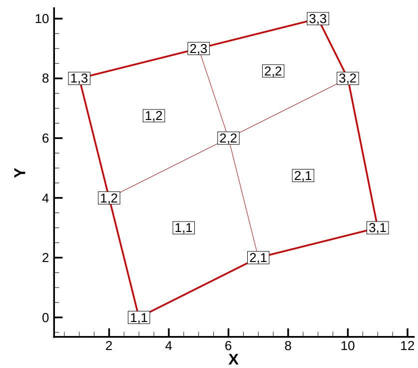


Figure 4-8. An IJ-ordered data set with cell-centered data.

```
VARIABLES = "X", "Y", "Temperature", "Pressure"
ZONE I=2, J=3, DATAPACKING=BLOCK, VARLOCATION=(3=CELLCENTERED,
4=CELLCENTERED)
3 7 11 2 6 10 1 5 9
0 2 3 4 6 8 8 9 10
0 2 1 3
45 60 35 70
```

The nodal variables of **X** and **Y** are specified at all nine nodes, but cell-centered variables and have values specified for the four cells **[(IMax-1)*(JMax-1)]**. Zones with cell-centered data must have **DATAPACKING=BLOCK**.

4.2.3. IJK-Ordered Data

IJK-ordered data has three indices: I, J, and K. This type of data is typically used for 3-D volume plots, although planes of the data can be used for 2- and 3-D surface plots. See Chapter 20, “Three-Dimensional Volume Data,” for more information.

In IJK-ordered data, the I-index varies from 1 to *IMax*, the J-index varies from one to *JMax*, and the K-index varies from one to *KMax*. The total number of data points (nodes) is *IMax*JMax*KMax*. For zones with only nodal variables the total number of values in the zone data is *IMax*JMax*KMax*N*, where *N* is the number of variables. For a mixture of nodal and cell-centered variables, the number of values in the zone data is *IMax*JMax*KMax*Nn+(IMax-1)*(JMax-1)*(KMax-1)*Nc*, where *Nn* is the number of nodal variables and *Nc* is the number of cell-centered variables. The three indices, *IMax*, *JMax*, and *KMax*, must be specified in the zone control line using the **I**-, **J**-, and **K**-parameters.

The I-index varies the fastest; the J-index the next fastest; the K-index the slowest. If you write a program to print IJK-ordered data, the I-index is the inner loop, the K-index is the outer loop, and the J-index is the loop in between. Note the similarity between IJ-ordered data and IJK-ordered data with *KMax=1*.

4.2.3.1. IJK-Ordered Data in POINT Format Example. An example of IJK-ordered data in **POINT** format is listed below. There are four variables (**X**, **Y**, **Z**, **Temperature**) and twelve data points. Each row of data corresponds to a data point; each column to a variable. This data is plotted in Figure 4-9; each data point is labeled with its IJK-index.

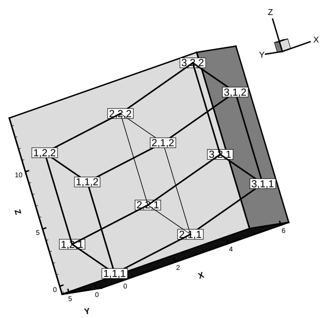


Figure 4-9. An IJK-ordered data set.

```
VARIABLES = "X" "Y" "Z" "Temp"
ZONE I=3, J=2, K=2,DATAPACKING=POINT

 0 0 0 0
 3 0 1 5
 6 0 3 10
 0 6 3 10
 3 6 4 41
 6 6 6 72
 0 0 8 0
 3 0 9 29
 6 0 11 66
 0 6 11 66
 3 6 12 130
 6 6 14 169
```

4.2.3.2. BLOCK Format of the Same Data. The same data set as Section 4.2.3.1., this time in **BLOCK** format, is shown below. For this example, each column of data corresponds to a data point; each row to a variable.

```
VARIABLES = "X" "Y" "Z" "Temp"
ZONE I=3, J=2, K=2, DATAPACKING=BLOCK
 0 3 6 0 3 6 0 3 6 0 3 6
 0 0 0 6 6 6 0 0 0 6 6 6
 0 1 3 3 4 6 8 9 11 11 12 14
 0 5 10 10 41 72 0 29 66 66 130 169
```

4.2.3.3. FORTRAN Code to Generate an IJK-Ordered Zone in POINT Format Example. The following sample FORTRAN code shows how to create an IJK-ordered zone in **POINT** format:

```
    WRITE (*,*) 'VARIABLES = "X", "Y", "Z", "Temp"'
    WRITE (*,*) 'ZONE I=',IMAX,' J=',JMAX,' K=',KMAX,' DATAPACKING=POINT'
   DO 1 K=1,KMAX
      DO 1 J=1,JMAX
         DO 1 I=1,IMAX
   1         WRITE (*,*) X(I,J,K), Y(I,J,K), Z(I,J,K), Temp(I,J,K)
```

In **BLOCK** format, all *IMax*JMax*KMax* values of each variable are listed, one variable at a time. Within each variable block, all the values of the variable at each data point are listed.

4.2.3.4. FORTRAN Code to Generate IJK-Ordered Data in BLOCK Format Example. The following sample FORTRAN code shows how to create an IJK-ordered zone in **BLOCK** format:

```
    INTEGER VAR
      .
      .
      .
      .
    WRITE (*,*) ´ZONE DATAPACKING=BLOCK, I=´, IMAX, ´, J=´, JMAX, ´, K=´, KMAX
    DO 1 VAR=1,NUMVAR
         DO 1 K=1,KMAX
            DO 1 J=1,JMAX
                DO 1 I=1,IMAX
   1                WRITE (*,*) ARRAY(VAR,I,J,K)
```

4.2.4. One Variable Data Files

For ordered data, it is possible to read in a data file that has only one variable. Tecplot then creates the other required variables. That is, if your data is I-ordered, a variable containing the I-index values is created, numbered **V1**, and called **I**. For IJ-ordered data, two variables, numbered **V1** and **V2** and called **I** and **J**, are created to contain the I- and J-index values. For IJK-ordered data, three variables **I**, **J**, and **K** are created and numbered **V1**, **V2**, and **V3**. The variable in the data file is numbered with the next available variable number, that is, **V2** for I-ordered data, **V3** for IJ-ordered data, and **V4** for IJK-ordered data. The created variables are the default X-, Y-, and Z-variables. The data type for the created variables is determined according to the following table:

Maximum of IMax, JMax, and KMax	Data Type
< 256	**BYTE**
<32,766	**SHORTINT**
>=32,766	**SINGLE**

For example, if you have an ASCII file with 256 by 384 numbers representing intensities of a rasterized image, you could make a data file similar to the following:

```
VARIABLES = "TEMPERATURE"
 ZONE I=256, J=384
```
List all 98,304 values of temperature here.

Read the data file into Tecplot. Two new variables of type **SHORTINT** are created and used as the default X- and Y-coordinates. These variables are the I- and J-index values; they are named **I** and **J**. You can now create any type of 2-D plot with the data.

If you have finite-element data, Tecplot will not create any new variables for you. If you need to add variables to finite-element data, you can do so using the Data menu.

4.3. Finite-Element Data

For finite-element data, the zone types, specified in the **ZONETYPE** parameter, may be **FELINESEG**, **FETRIANGLE**, **FEQUADRILATERAL**, **FETETRAHEDRON**, or **FEBRICK**. For any of these **DATAPACKING** may be **POINT** or **BLOCK**.

The number of nodes (data points) is given by the **N=***numnodes* parameter, and the number of elements is given by the **E=***numelements* parameter (this is also the total length of the connectivity list).

Zone data is divided into two logical sections. It has no markers, but you may place blank lines between the sections to distinguish them. The first section, the node (and sometimes element) data, lists the values of the variables at the data points (or nodes) or cell-centers (elements) as if they were I-ordered (one-dimensional) zone data. The second section, the connectivity list, defines how the nodes are connected to form elements. There must be *numelements* lines in the second section; each line defines one element. The number of nodes per line in the connectivity list depends on the element type specified in the zone control line (**ZONETYPE** parameter). For example, **ZONTYPE=FETRIANGLE** has three numbers per line in the connectivity list. If nodes 5, 7, and 8 are connected, one line reads: **5 7 8**.

In the descriptions below, **N***E* is the *E*th node at a vertex of an element. The subscripts of **N***E* refer to the element number. For example, $\mathbf{N2}_3$ represents the second node of the third element.

For the triangle element type, each line of the connectivity list contains three node numbers that define a triangular element:

$\mathbf{N1}_M, \mathbf{N2}_M, \mathbf{N3}_M$

For the quadrilateral element type, each line of the connectivity list contains four node numbers that define a quadrilateral element:

$\mathbf{N1}_M, \mathbf{N2}_M, \mathbf{N3}_M, \mathbf{N4}_M$

If you need to mix quadrilateral and triangle elements, either create two zones or use the quadrilateral element type with node numbers (**$N4_M$**=**$N3_M$**) repeated to form triangles.

Zones created from the quadrilateral and triangle element types are called FE-surface zones.

For the tetrahedron element type, each line of the second section of the zone data contains four node numbers that define a tetrahedral element:

$N1_M$ $N2_M$ $N3_M$ $N4_M$

For the brick element type, each line of the second section contains eight node numbers that define a "brick-like" element:

$N1_M$ $N2_M$ $N3_M$ $N4_M$ $N5_M$ $N6_M$ $N7_M$ $N8_M$

Tecplot divides the eight nodes into two groups of four; nodes **$N1_M$**, **$N2_M$**, **$N3_M$**, and **$N4_M$** make up the first group, and **$N5_M$**, **$N6_M$**, **$N7_M$**, and **$N8_M$** make up the second group. Each node is connected to two nodes within its group and the node in the corresponding position in the other group. For example, **$N1_M$** is connected to **$N2_M$** and **$N4_M$** in its own group, and to **$N5_M$** in the second group. To create elements with fewer than eight nodes, repeat nodes as necessary, keeping in mind the basic brick connectivity just described. Figure 4-10 shows the basic brick connectivity. For example, to create a tetrahedron, you can set **$N3_M$**=**$N4_M$** and **$N5_M$**=**$N6_M$** =**$N7_M$**=**$N8_M$**. To create a quadrilateral-based pyramid, you can set **$N5_M$**=**$N6_M$**=**$N7_M$**=**$N8_M$**. If you need a mixture of bricks and tetrahedra, either use two zones or use the brick element type with node numbers repeated so that tetrahedra result.

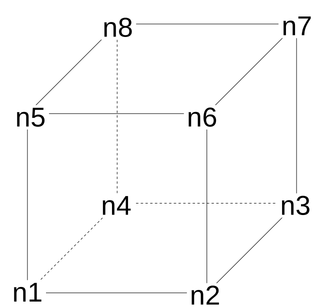


Figure 4-10. Basic brick connectivity.

Zones created from the brick and tetrahedron element types are called finite-element volume zones.

If **CONNECTIVITYSHAREZONE=***nnn* is in the zone control line, the connectivity list is shared from zone *nnn*. In this case, no connectivity list is given, just the node (and possibly element) data. If *nnn* is greater than or equal to the current zone number, Tecplot generates an error message.

4.3.1. Triangle Data in POINT Format Example

An example of triangle element type finite-element data with **POINT** datapacking is listed below. There are two variables (**X**, **Y**) and five data points. In this example, each row of the data section corresponds to a node and each column to a variable. Each row of the connectivity list corresponds to a triangular element and each column specifies a node number. This data set is plotted in Figure 4-11. Each data point is labeled with its node number.

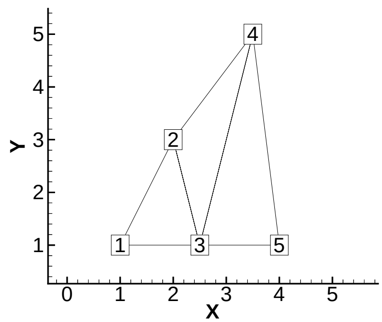


Figure 4-11. A finite-element triangle data set.

```
VARIABLES = "X", "Y"
ZONE N=5, E=3, DATAPACKING=POINT, ZONETYPE=FETRIANGLE
1.0 1.0
2.0 3.0
2.5 1.0
3.5 5.0
4.0 1.0

1 2 3
3 2 4
3 5 4
```

4.3.1.1. BLOCK Format of the Same Data. The same data in **BLOCK** format is shown below. In this example, each column of the data section corresponds to a node and each row to a variable. As above, each row of the connectivity list corresponds to a triangular element and each column specifies a node number.

```
VARIABLES = "X", "Y"
 ZONE N=5, E=3, DATAPACKING=BLOCK, ZONETYPE=FETRIANGLE
 1.0 2.0 2.5 3.5 4.0
 1.0 3.0 1.0 5.0 1.0
 1 2 3
 3 2 4
 3 5 4
```

4.3.2. FORTRAN Code Generating Triangle Data in POINT Format Example

The following sample FORTRAN code shows how to create triangle element type finite-element data in **POINT** format:

```
      INTEGER VAR
      .
      .
      .
      WRITE (*,*) ´ZONE DATAPACKING=POINT, ZONETYPE=FETRIANGLE,N=´,
     &    NNODES,´,E=´,NELEM
      DO 1 N=1,NNODES
        DO 1 VAR=1,NUMVAR
 1          WRITE(*,*) VARRAY(VAR,N)

      DO 2 M=1,NELEM
        DO 2 L=1,3
 2         WRITE (*,*) NDCNCT(M,L)
```

4.3.3. FORTRAN Code Generating Triangle Data in BLOCK Format Example

This FORTRAN code creates triangle element type finite-element data in **BLOCK** format:

```
      INTEGER VAR
      .
      .
      .
      WRITE (*,*) ´ZONE DATAPACKING=BLOCK, ZONETYPE=FETRIANGLE,N=´,NNODES,
     &´,E=´,NELEM
       DO 1 VAR=1,NUMVAR
         DO 1 N=1,NNODES
 1          WRITE(*,*) VARRAY(VAR,N)
       DO 2 M=1,NELEM
         DO 2 L=1,3
 2          WRITE (*,*) NDCNCT(M,L)
```

4.3.4. Finite-Element Zone Node Variable Parameters Example

The node variable parameter allows setting of the connectivity to match the value of the selected node variable. In the example below, the files appear to be identical in Tecplot, although the connectivity list has changed to reflect the values of the node variable node order. Notice that the index value of the nodes is not changed by the node variable value.

The original data set:

```
TITLE     = "Data with original node ordering"
VARIABLES = "X"
"Y"
ZONE T="Triangulation"
 N=6, E=5,DATPACKING=POINT, ZONETYPE=FETRIANGLE
DT=(SINGLE SINGLE )
 2.00E+000 3.00E+000
 2.20E+000 3.10E+000
 3.10E+000 4.20E+000
```

```
 2.80E+000 3.50E+000
 2.40E+000 2.10E+000
 4.30E+000 3.20E+000
 1 2 5
 6 4 3
 5 4 6
 2 3 4
 5 2 4
```

The data set with the nodes re-ordered for connectivity:

```
TITLE     = "Data with modified node ordering"
VARIABLES = "X"
"Y" "Node-Order"
ZONE T="Triangulation"
 N=6, NV = 3, E=5,DATAPACKING=POINT, ZONETYPE=FETRIANGLE
DT=(SINGLE SINGLE )
 2.00E+000 3.00E+000 5
 2.20E+000 3.10E+000 4
 3.10E+000 4.20E+000 1
 2.80E+000 3.50E+000 2
 2.40E+000 2.10E+000 6
 4.30E+000 3.20E+000 3
 5 4 6
 3 2 1
 6 2 3
 4 1 2
 6 4 2
```

4.4. Variable and Connectivity List Sharing

The **VARSHARELIST** in the **ZONE** record allows you to share variables from specified previous zones. The **CONNECTIVITYSHAREZONE** parameter in the **ZONE** record allows you to share the connectivity list from a specified previous zone. The following is an example to illustrate these features.

The table below shows Cartesian coordinates X and Y of six locations, and the pressure measured there at three different times (*P1*, *P2*, *P3*). The XY locations have been arranged into finite-elements.

X	Y	P_1	P_2	P_3
-1.0	0.0	100	110	120
0.0	0.0	125	135	145
1.0	0.0	150	160	180
-0.5	0.8	150	165	175

Figure 4-12.

X	Y	P_1	P_2	P_3
0.5	0.8	175	185	195
0.0	1.6	200	200	200

Figure 4-12.

For this case, we want to set up three zones in the data file, one for each time measurement. Each zone has three variables: X, Y, and P. The zones are of the triangle element type, meaning that three nodes must be used to define each element. One way to set up this data file would be to list the complete set of values for X, Y, and P for each zone. Since the XY-coordinates are exactly the same for all three zones, a more compact data file can be made by using the **VARSHARELIST**. In the data file given below, the second and third zones have variable sharing lists that share the values of the X- and Y-variables and the connectivity list from the first zone. As a result, the only values listed for the second and third zones are the pressure variable values. Note that the data could easily have been organized in a single zone with five variables. Since blank lines are ignored in the data file, you can embed them to improve readability. A plot of the data is shown in Figure 4-13.

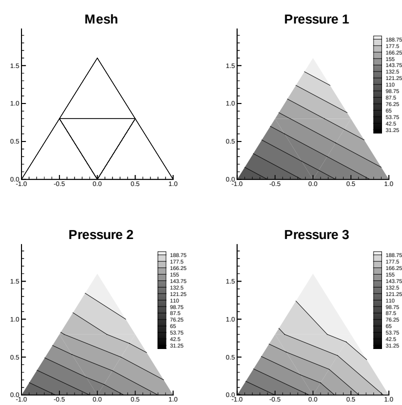


Figure 4-13. A plot of finite-element zones.

```
TITLE = "Example: Variable and Connectivity List Sharing"
VARIABLES = "X", "Y", "P"
ZONE T="P_1", DATAPACKING=POINT, N=6, E=4, ZONETYPE=FETRIANGLE
-1.0  0.0  100
0.0  0.0  125
1.0  0.0  150
-0.5  0.8  150
0.5  0.8  175
0.0  1.6  200
```

```
1 2 4
2 5 4
3 5 2
5 6 4
ZONE T="P_2", DATAPACKING=POINT, N=6, E=4, ZONETYPE=FETRIANGLE, VARSHARELIST
= ([1, 2]=1), CONNECTIVITYSHAREZONE = 1
110 135 160 165 185 200

ZONE T="P_3", DATAPACKING=POINT, N=6, E=4, ZONETYPE=FETRIANGLE, VARSHARELIST
= ([1, 2]=1), CONNECTIVITYSHAREZONE = 1
120 145 180 175 195 200
```

4.5. ASCII Data File Conversion to Binary

Although Tecplot can read and write ASCII or binary data files, binary data files are more compact and are read into Tecplot much more quickly. Your Tecplot distribution includes Preplot, which converts ASCII to binary data files. You can also use Preplot to debug ASCII data files Tecplot cannot read.

4.5.1. Preplot Options

To use Preplot, type the following command from the UNIX shell prompt, from a DOS prompt, or using the Run command in Windows:

`preplot` *infile [outfile] [options]*

where *infile* is the name of the ASCII data file, *outfile* is an optional name for the binary data file created by Preplot, and *options* is a set of options from either the standard set of Preplot options or from a special set of options for reading PLOT3D format files. If *outfile* is not specified, the binary data file has the same base name as the *infile* with a **`.plt`** extension. You may use a minus sign ("**`-`**") in place of either the *infile* or *outfile* to specify standard input or standard output, respectively.

Any or all of **`-iset`**, **`-jset`**, and **`-kset`** can be set for each zone, but only one of each per zone.

For more Preplot command lines, see Appendix B.3, "Preplot."

4.5.2. Preplot Examples

If you have an ASCII file named **`dset.dat`**, you can create a binary data file called **`dset.plt`** with the following Preplot command:

```
preplot dset.dat dset.plt
```

By default, Preplot looks for files with the **`.dat`** extension, and creates binary files with the **`.plt`** extension. Thus, either of the following commands is equivalent to the above command:

```
preplot dset
preplot dset.dat
```

Preplot checks the input ASCII data file for errors such as illegal format, numbers too small or too large, the wrong number of values in a data block, and illegal finite-element node numbers. If Preplot finds an error, it issues a message displaying the line and column where the error was first noticed. This is only an indication of where the error was *detected*; the actual error may be in the preceding columns or lines.

If Preplot encounters an error, you may want to set the debug option to get more information about the events leading up to the error:

```
preplot dset.dat -d
```

You can set the flag to **-d2**, or **-d3**, or **-d4**, and so forth, to obtain even more detailed information.

In the following Preplot command line, the number of points that are written to the binary data file **dset.plt** is less than the number of points in the input file **dset.dat**:

```
preplot dset.dat -iset 3,6,34,2 -jset 3,1,21,1 -iset 4,4,44,5
```

For zone 3, Preplot outputs data points with I-index starting at 6 and ending at 34, skipping every other one, and J-index starting at one and ending at 21. For zone 4, Preplot outputs data points with the I-index starting at four, ending at 44, and skipping by five.

In the following Preplot command line, every other point in the I-, J-, and K-directions is written to the binary data file:

```
preplot dset.dat -iset ,,,2 -jset ,,,2 -kset ,,,2
```

The *zone*, *start*, and *end* parameters are not specified, so all zones are used, starting with index 1, and ending with the maximum index. The overall effect is to reduce the number of a data points by a factor of about eight.

4.5.3. Preplot Conversion of the PLOT3D Format

PLOT3D is a graphics plotting package developed at NASA. Some numerical simulation packages and other programs can create graphics in PLOT3D format. There are two paths by which you can get files in PLOT3D format into Tecplot. This section describes the Preplot path; you can also use the PLOT3D Loader described in Section 6.8, “The PLOT3D Data Loader.” The PLOT3D Loader has more advanced capabilities than Preplot.

Preplot can read files in the PLOT3D format and convert them to Tecplot binary data files through the use of special switches. You do not need to know about these switches unless you have data in PLOT3D format.

PLOT3D files typically come in pairs consisting of a grid file (with extension **.g**) and a solution file (with extension **.q**). Sometimes only the grid file is available. The grid itself may be either a single grid, or a multigrid, and the data may be 1D, 2D, 3D-planar, or 3D-whole (equivalent to Tecplot’s 3-D volume data). The PLOT3D files may be binary or ASCII. The PLOT3D-specific switches to Preplot allow you to read PLOT3D files with virtually any combination of these options.

The *ilist*, *jlist*, and *klist* are comma-separated lists of items of the form:

```
start[:end][:skip]]
```

where *start* is the number of the starting I-, J-, or K-plane, *end* is the number of ending I-, J-, or K-plane, and *skip* is the skip factor between planes. If *end* is omitted, it defaults to the starting plane (so if just *start* is specified, only that one plane is included). The *skip* defaults to one (every plane) if omitted; a value of two includes every other plane, a value of three include every third plane, and so on.

You must specify one of the flags **-1d**, **-2d**, **-3dp**, or **-3dw**. You may also specify only one of **-ip**, **-jp**, or **-kp** and only one of **-b** or **-f**.

If the input PLOT3D file is 3-D whole (**-3dw**) and none of the plane-extraction switches **-ip**, **-jp**, or **-kp** is specified, the PLOT3D file is converted directly to an IJK-ordered zone (or multiple zones if the file is multi-grid).

For example, in the following command line, Preplot reads from the PLOT3D files **aero.g** and **aero.q**. The input is binary and 3-D whole. The J-planes 2, 3, 4, 45, 46, and 47 are processed and made into six IJ-ordered zones, in a binary data file named **aero.plt**:

```
preplot aero -plot3d -b -3dw -jp 2,3,4,45,46,47
```

In the following command line, the plane-extraction switches are omitted, so Preplot creates a single IJK-ordered zone:

```
preplot aero -plot3d -b -3dw
```

The following command line reads an ASCII file **airplane.g** for which there is no corresponding **.q** file; the data is 3-D whole:

```
preplot airplane -plot3d -gridonly -3dw
```

The following command line reads a multi-grid, 3-D planar, binary-FORTRAN pair of PLOT3D files, **multgrid.g** and **multgrid.q**:

```
preplot multgrid -plot3d -m -f -3dp
```

CHAPTER 5 Working with Tecplot Files

Tecplot reads and writes a number of different files. In general, these operations are performed using the File menu. With the File menu, you can instruct Tecplot to read or write data files, layout files, and macro files.

This chapter presumes your data files are in a form readable by Tecplot. Chapter 4, “ASCII Data for Tecplot,” discusses data files structured for direct reading by Tecplot. Chapter 6, “Data Loaders: Tecplot’s Import Feature,” discusses importing a variety of other formats. Some files, such as stylesheets and equation files, are read and written from other menus and dialogs. Other files can be loaded from the Tecplot command line. For more information, please see Appendix A, “Command Line Options.”

5.1. Tecplot-Format Data File Loading

This section describes the process for loading Tecplot-format data files, as well as reading data files formatted for other software.

Tecplot cares about the format of a data file and the context in which the file is read, but is not concerned about file names (including extensions). Tecplot uses the standard extensions **`.dat`** for ASCII and **`.plt`** for binary.

There are four ways to work with Tecplot-format data files:

- Generate a Tecplot-format ASCII data file. Read the file into Tecplot and work without conversion. If the data set is altered, save it as an ASCII data file. This method works for smaller data sets where the convenience of an ASCII file outweighs any inefficiencies.
- Generate a Tecplot-format ASCII data file. Read it into Tecplot, then save it as a binary data file, then work with the binary file. Once you have saved a binary version, you can delete the ASCII version. This works well for large data sets where ASCII inefficiencies are noticeable.
- Generate a Tecplot-format ASCII data file, then convert it to a binary file with Preplot. (Preplot, a utility program included with Tecplot, converts ASCII and PLOT3D to binary Tecplot-format data files. Preplot also converts PLOT3D files to Tecplot-format binary data files.) Once the binary file is created, delete the ASCII version to save space. This works well for identifying problems with data files, since Preplot’s error messages include precise

details. This method also works well in batch processing, or if the ASCII data files are generated on another machine. (See Section 4.5, “ASCII Data File Conversion to Binary,” for a description of Preplot.)

- Generate a Tecplot-format binary data file. Read the binary data file into Tecplot and work without conversion. This works well if you use routines provided by Amtec to write Tecplot-format binary files from C or FORTRAN programs. See Chapter 11, “Writing Binary Data for Loading into Tecplot,” in the *Tecplot Reference Manual*, for complete details.

5.1.1. Data File Loading

Tecplot allows you to read multiple data files to create a data set. However, you usually start with a single data file. Tecplot uses separate procedures for loading single versus multiple data files.

To load a single data file:

1. From the File menu, choose Load Data File(s). The Load Data File dialog appears, as shown in Figure 5-1.

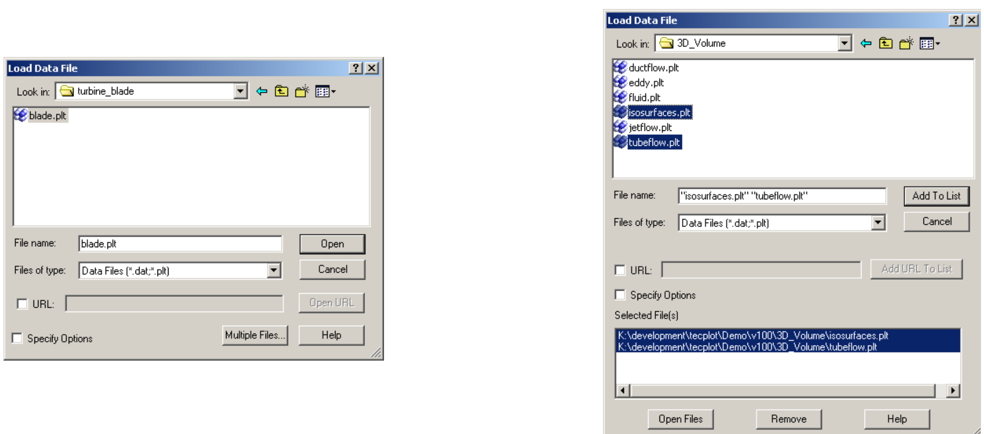


Figure 5-1. Load Data File (left) and the Load Multiple Data Files dialogs (right).

2. Navigate to the appropriate file, or enter the file name in the File Name field and click Open.
3. The Select Initial Plot dialog appears. Use the Initial Plot Type drop-down to set the plot type (2- or 3-D, XY, Polar, Sketch, or Automatic, where Tecplot chooses the most appropriate type.) Use Show First Zone Only if you have many zones and intend to view a small fraction of them at any given time. This reads all zones, but only performs initial calculations and drawings for the first zone. The Select Initial Plot dialog is shown in Figure 5-2.

To load multiple data files:

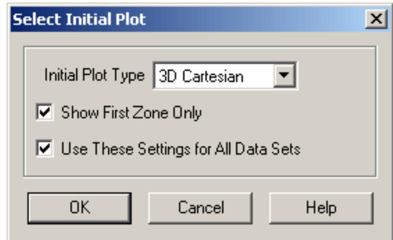


Figure 5-2. The Select Initial Plot Type dialog.

1. From the File menu, choose Load Data File(s). The Load Data File dialog appears.
2. Click Multiple Files. The Load Data File dialog for multiple files appears, also shown in Figure 5-1.
3. Navigate to the appropriate file, or enter the file name in the File Name field. Open now reads Add To List. Click to add the file, repeating the process for additional files. Specify a URL by selecting the URL check box. Your URL must start with ftp:// or http://. For example, ftp://ftp.microsoft.com/myplot.plt. To add a disk file name you must deselect the URL check box. You may add any combination of URLs and disk file names to the Selected File(s) list.

   or

   Double-click on the name of the file in the Look in drop-down. The file is added to the Selected File(s) scrolled list.
4. To remove files from the Selected File(s) scrolled list, select the files, then click Remove.
5. When the Selected File(s) scrolled list contains only those files you want to read in, click Open Files to read in the files.

The order of the list of files to be read in is important. Tecplot numbers the zones based on the ordering in this list.

5.1.1.1. Data File Loading Options. Tecplot allows you to specifically control what is loaded from your data files. This gives you the ability to load only certain zones or variables, or even part of a zone. You can also choose to load the variables in your data files by name or by position. Use the Load Data File Options dialog to specify these and other options.

To select options when loading one or more data files:

1. Select files for loading as described in Section 5.1.1, “Data File Loading,” without clicking Open.
2. Select Specify Options.
3. Click Open. The Load Data File Options dialog appears, as shown in Figure 5-3.
4. Select the desired options (described in detail below).

5. Click OK to read the selected data.
6. On the Select Initial Plot dialog, make the appropriate choices, then click OK.

The Load Data File Options dialog has three pages—General, Zones, Vars (Variables)—discussed separately in Sections 5.1.1.2. through 5.1.1.6.

5.1.1.2. Data File Loading General Options. On the General page (Figure 5-3), you have the option to load a subset of record types or load only portions of the data.

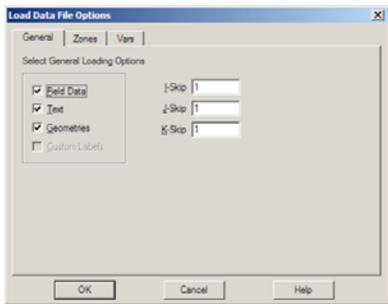


Figure 5-3. The General page of the Load Data File Options dialog.

To load specific record types from the data file, select the desired record types by choosing the appropriate check boxes:

Field Data	Load zone records (the actual data). If not selected, the Zones and Vars pages of the dialog are inactive.
Text	Load text records.
Geometries	Load geometry records.
Custom Labels	Load custom label records.

The check boxes are available only if those records exist in the data files. By default, all of the records in the data files are selected.

If you want to load a portion of the data points, specify skip factors for the I-, J-, and K-dimensions in the corresponding text fields. Each skip factor *n* tells Tecplot to read in every *n*th point in the specified direction. By default, all the skip factors are set to one, so every data point is loaded.

5.1.1.3. Data File Loading Zone Options. On the Zones page, shown in Figure 5-4, allows selection of specific zones to load from data files and, if appropriate, whether to collapse the zone list.To specify which zones to load, select them in the Select Zones to Load list box. Zones are listed in the order (that is, all zones from the first data file, then all zones from the second, and so forth). By default, all zones are selected to be loaded.

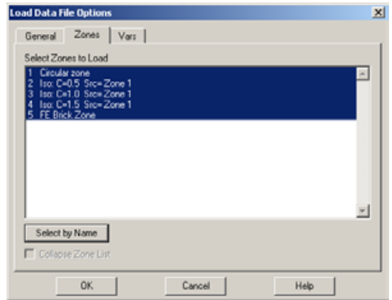


Figure 5-4. The Zones page of the Load Data File Options dialog.

If you have selected to only load specific zones and want them renumbered upon loading, select Collapse Zone List. (If you are loading variables by position, the check box reads Collapse Zone and Variable Lists.) See Section 5.1.1.7, “Zone and Variable List Collapsing,” for more information.

5.1.1.4. Data File Loading Variable Options. The Variables page loads variables by name or position. The default is to load variables by name. When loading variables by name, Tecplot creates variables based on the variable names in the data files. When loading variables by position, Tecplot creates variables based on their order in the data files. Tecplot creates variable order based on their order in the first data file loaded. This is true when loading by position or by name.

For example, consider two data files: **`file1.plt`** and **`file2.plt`**. File **`file1.plt`** has variables **`X`**, **`Y`**, and **`P`**, in that order. File **`file2.plt`** has variables **`P`**, **`X`**, and **`Y`**, in that order. If these two files are loaded by position, the first variable in Tecplot is created from **`X`** in **`file1.plt`** and **`P`** in **`file2.plt`**. The second variable is from **`Y`** in **`file1.plt`** and **`X`** in **`file2.plt`**. The third variable is from **`P`** in **`file1.plt`** and **`Y`** in **`file2.plt`**.

Data File	**`file1.plt`**	**`file2.plt`**
Variable 1	**`X`**	**`P`**
Variable 2	**`Y`**	**`X`**
Variable 3	**`P`**	**`Y`**

If these two files are loaded by name, the first variable is created from **`X`** in **`file1.plt`** and **`X`** in **`file2.plt`**. The second is from **`Y`** in **`file1.plt`** and **`Y`** in **`file2.plt`**. The third variable is from **`P`** in **`file1.plt`** and **`P`** in **`file2.plt`**.

5.1.1.5. Variable Loading by Name. Using the Load Variables by Name area of the Load Data File Options dialog’s Vars page, you may select specific variable names to load from the data files. This is shown in Figure 5-5.

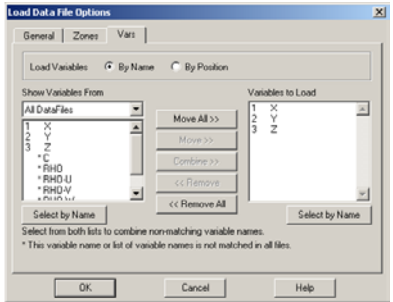


Figure 5-5. The Load Variables By Name area of the Load Data File Options dialog.

When loading variables by name, variables are associated by name then loaded into Tecplot. Variable names can be combined; two variables with different names in different files can be loaded into a single Tecplot variable. When a variable name is missing the variable is set to all zeros for all zones loaded from that file.

The Show Variables From list box displays variable names from the data files to load. Filter the list with the drop-down above the list. Choosing All Data Files shows variable names from all data files in order (that is, all of the variable names from the first data file are listed, then all variable names from the second file, and so forth). Identical variable names from more than one file appears only once in the list. An asterisk (*) next to a variable name indicates the variable name does not exist in all the files. A number next to a variable name indicates the variable to be loaded into. For example, a "2" next to a variable name means that variable name is loaded into the second Tecplot variable.

The Variables to Load list box displays variables to be loaded into Tecplot. By default, it shows only variable names existing in all of the data files selected. If no matching variable names exist, the list is empty. An asterisk (*) next to a variable name indicates the name does not exist for all files. If you load a file with an asterisk, the file's zones are set to zero for that variable. Duplicate variable names are not allowed.

The following options allow you to manipulate the lists:

- 

Take all of the variable names in Show Variables From and add them to Variables to Load.
- 

Take the selected variable name in Show Variables From and add it to Variables to Load.
- 

Take the variable name selected in Show Variables From and combine it with the variable name selected in Variables to Load.

- 

Remove the selected variable name from Variables to Load list.
- 

Remove all variable names from Variables to Load list.

You have limited options in changing Variables to Load when appending data files to the current data set or replacing the current data set while retaining the plot style. The list is partially determined by the current data set. You can add names or combine new names, but you cannot remove any variable names.

When appending data files to the current data set by adding names to Variables to Load, adding a new name which already exists in the current data set, but which was not loaded initially, causes Tecplot to reload the affected data files to include the new name. First change the data set using the Data menu's Alter option.

5.1.1.6. Variable Loading by Position. The Load Variables By Position area of the Load Data File Options dialog's Vars page allows you to select specific variables to load from data files, and to collapse the zone and variable lists when possible. The Loading Variables by Position section of the dialog is shown in Figure 5-6.

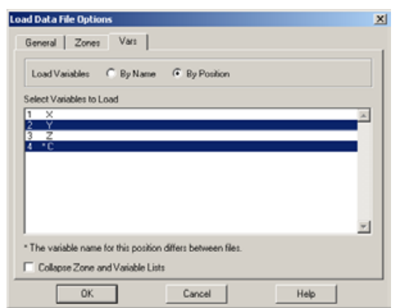


Figure 5-6. The Load Variables by Position section of the Load Data File Options dialog.

To specify variables to load from data files, select them in Select Variables to Load. This is a multiple selection list box where you can click-and-drag, Ctrl-click, or Shift-click to choose variable. The variable names listed come from the first data file. If variable names in the other files do not match those in the first, an asterisk (*) appears next to the name. The number of variables listed is limited to the minimum number in all of the files. By default, all of the variables are selected to be loaded.

If you have selected to only load specific variables and want them renumbered, select the "Collapse Zone and Variable Lists" check box. See Section 5.1.1.7, "Zone and Variable List Collapsing," for more information.

When appending files to the current data set or replacing the current data set while retaining the plot style, you cannot select the variables to load. These are determined by the variables currently in Tecplot. When appending files to the current data set, the new files must have at least as many variables as are currently in Tecplot.

5.1.1.7. Zone and Variable List Collapsing. When loading files you have the option of reading only selected zones (and variables when loading by position). You may either preserve existing zone and variable numbering, or “collapse” the data read so zones and variables are renumbered according to their positions in Tecplot.

For example, zones 2 and 5 of a five zone data file are loaded. If the zones and variables are not collapsed (the default), Tecplot reads them in as zones 2 and 5. Writing this data set to an ASCII file, it has five zones; zones 1, 3, and 4 have no data (“Zombie” zones). Selecting the collapse option, Tecplot reads them in as zones 1 and 2. Writing this data set to a file yields only two zones.

In most cases collapsing zones and variables is unnecessary. All dialogs showing zones or variables list the zones read in, though they may not be numbered sequentially. Do not collapse zones and variables when:

- You have a large data set and read a portion of the data to reduce the amount of memory used in processing. You then create a stylesheet to use at a later time with a different sub-set of the data.
- You have many zones and variables and you are familiar with certain ranges of them. (For example, you may know that zones 150-200 represent a known portion of the data.) If you partially read the data and do not collapse it, these zones continue to be designated with their familiar numbers.

5.2. Data File Writing

You can write out the data set in the current frame as either an ASCII or binary data file. Tecplot asks you to choose which part of the data to write, as well as to specify the format for the saved file.

To write the data set in the current frame to a file:

1. From the File menu, choose Write Data File. The Write Data File Options dialog appears, as shown in Figure 5-7.
2. (Optional) Select the zones and or variables you want to write to the saved data file.
3. (Optional) Specify which record types you want to write to the saved data file by selecting the appropriate check boxes:
   - **Text:** Select this check box to save any text attached to the current frame.
   - **Geometries:** Save any geometries attached to the current frame.
   - **Custom Labels:** Save any custom labels attached to the current data set.

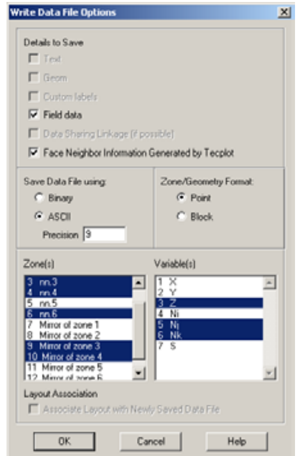


Figure 5-7. The Write Data File Options dialog.

- **Field Data:** Save the zone data.
- **Data Sharing Linkage (If Possible):** Save the variable and connectivity sharing between zones existing in the data set, reducing its size and loading time.
- **Face Neighbor Information Generated by Tecplot:** Automatically save the face neighbor information generated by Tecplot for finite-element zones. This increases the data set size and loading time, but speeds performance after loading.

By default, all record types that are present in the current data set are selected.

4. (Optional) Choose whether to save the file as ASCII or Binary.
5. (Optional) For ASCII, choose to write the file in **`POINT`** format or **`BLOCK`** format (**`BLOCK`** is required if any variables are cell-centered). (**`POINT`** format organizes the data by data points, **`BLOCK`** format organizes it by variables. See Chapter 4, “ASCII Data for Tecplot,” for a complete description of both formats.)
6. (Optional) You can also specify the precision of your Float and Double variables. These variable types are written in exponent format and the precision determines the number of digits included past the decimal point (a precision of three would result in numbers of the form **`4.657E+02`**).
7. (Optional) Associate Layout with Newly Saved Data File: Tecplot associates this data file with the layout’s style. If not selected, Tecplot asks you for a file name when writing out the file.
8. Click OK; the Write Data File dialog appears, as shown in Figure 5-8.
9. Specify a file name, then click OK.

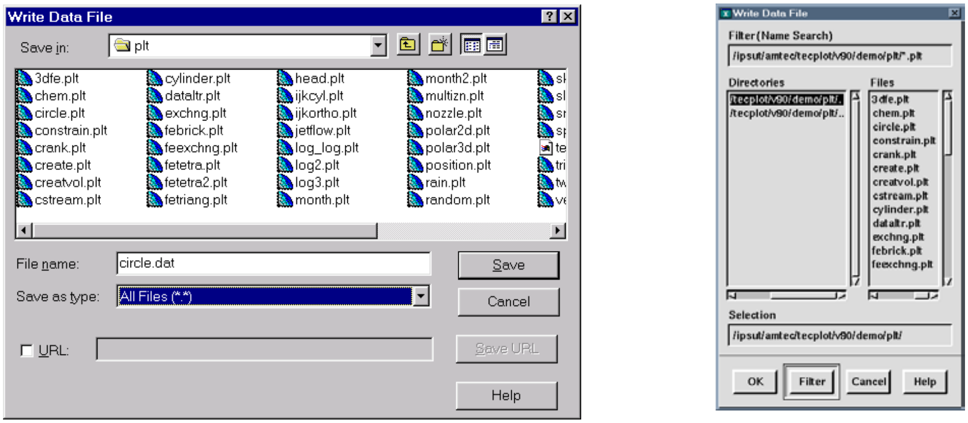


Figure 5-8. The Write Data File dialogs in Windows (left) and Motif (right).

5.3. Layout Files, Layout Package Files, Stylesheets

Tecplot has three different types of files for storing plot information:

- **Stylesheets:** Stylesheets store information about a single frame, but do not include any information about the data used by the frame.
- **Layout files:** Layout files store information about all the frames in the workspace, including identification of the data used by each frame.
- **Layout Package files:** Layout package files are an extension to layout files where data and an optional preview image are included.

This section describes the three types of files and gives suggestions on when to use each one.

5.3.1. Stylesheets

Layout and layout package files for the most part are the preferred medium for saving the style of your plot. They are quick and easy to load and save, and they save your complete picture. Unlike layout and layout package files, which contain the complete style of all the frames in your workspace, stylesheets contain only the style of a single frame in Tecplot. Despite this limitation, there are situations where stylesheets are useful, including:

- When pre-processing must be done to a data set prior to attaching a style. You may need to load a data set and run some equations or do interpolation or zone extraction before assigning a style. The style may reference objects or variables that do not exist in the original data and it is necessary to assign the style after they are created. Note that Tecplot's data journaling capabilities eliminate this situation in many cases.

- When switching styles on large data sets. You may want to load a large data set and generate two full page plots. Each plot has a different style. By using a stylesheet for the second plot you avoid having to reload the data set.
- When copying the style of one frame to another frame in the same layout.
- When saving just part of a frame's style, such as just the contour levels.

A stylesheet is a special type of macro file that contains commands used to define the style of a single frame in Tecplot. The style includes attributes such as the type of plot (a 2-D contour plot in 2-D or an XY Line plot), the colors used, the current view of the data, how axes are displayed, or enhancements such as text or geometries. Some items comprising the style of a frame are shown in Figure 5-9.

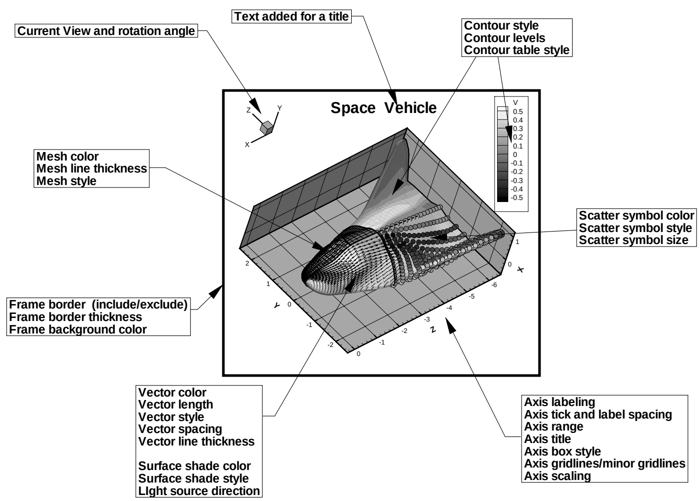


Figure 5-9. Some of the items considered part of the frame style.

If the frame contains any image geometries (see Section 16.3. "Images" on page 294), Tecplot will save references to the image files in the stylesheet. If the images came from a layout package file, Tecplot will save references to this file in the stylesheet.

Stylesheets are copied (created, by extracting from a frame and writing to a file) or pasted (applied to a frame) by using the Copy Frame Style to File or Paste Frame Style from File options of the Frame menu. A stylesheet does not contain any information about the frame's data (if required), or where that data set comes from.

To select part or all of a frame style and create a stylesheet:

1. On the Frame menu, select Copy Frame Style to File; the Copy Style to File dialog appears.

2. Select a path and file name.
3. By default, Tecplot saves the names of the image files used in the stylesheet with their relative file paths. If you want to save your stylesheet using absolute file paths, turn off the Use Relative Path check box.
4. (Optional) Click Options. The Copy Style Options dialog appears. On the Copy Style Options dialog, select the aspects of the frame style that you want to save, then click Close.
5. On the Copy Style to File dialog, click OK.

To apply all or part of a stylesheet to a data set:

1. In the Frame menu, click Paste Frame Style from File. The Paste Style from File dialog appears.
2. Specify the path and name of the file containing the style to be copied.
3. (Optional) Click Options. The Paste Style Options dialog appears. Specify style items to paste, then click Close.
4. When you have finished click OK.

5.3.2. Layout Files

A plot often consists of multiple frames or even multiple data sets. To capture all the information on the plot, use a special type of Tecplot macro file called a layout file (usually with an extension of `.lay`), with a complete description of your plot. Layout files include the names of the data files used to create the plot, the frame layout and data set attachments, axis and plot attributes, the current color map, and so forth.

Figure 5-10 shows a layout with four frames. The frame in the upper left-hand corner is attached to data set 1. The two frames on the right are both attached to data set 2. The frame in the lower left is not attached to a data set.

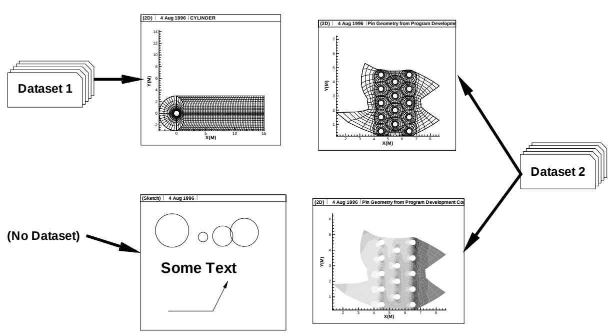


Figure 5-10. Layout of four frames using two data sets.

If a frame defined in a layout file requires an attached data set, the data files necessary to build the data set are referenced in the layout file. These data files can be referenced using absolute paths or relative paths. Use relative paths if you intend to move both the layout file and the data files to some other location on disk, or some other platform, at a later date. When using relative paths under Windows, the data files must be on the same drive as the layout file.

A layout file may also contain the data journal; a set of macro commands which alter the data or create new data. The data journal commands replicate the data modifications made to the original data (in files) during prior Tecplot sessions. Not all data operations are supported by the data journal. For more information, see Chapter 22, “Data Operations.”

Aside from the commands needed to build the individual style of each frame, layout files also include macro commands to set the following:

- Page layout information including the size and orientation of the paper.
- Some print setup information, including how colors are to be mapped to monochrome gray scales for monochrome output.
- Color spectrum information, including what basic global color map is installed and what adjustments have been made.

For complete details on the structure of Tecplot macro files and the commands available in layout files, see Chapter 27, “Macro Commands,” and the *Tecplot Reference Manual*.

To include the field data with a layout, use a layout package file. For more information, see Section 5.3.3, “Layout Package Files.”

5.3.2.1. Layout File Saving. You save layout files using the Save Layout or Save Layout As options under the File menu.

To save your layout, do the following:

1. From the File menu, select Save Layout As. The Save Layout dialog appears.
2. Choose “Linked Data (*.lay)” using the File of type drop-down, as shown in Figure 5-11.
3. By default, Tecplot saves the name of the data files used in the layout with their relative file paths. If you want to save your layout using absolute file paths, turn off the Use Relative Path check box.
4. Specify a file name for the layout file. In Windows, you may specify a URL by selecting the URL check box. When selected, you may enter a full URL as the file name, as shown in Figure 5-12.

   The URL must start with either ftp:// or http://. For example, ftp://ftp.microsoft.com/myplot.plt. The two file types are mutually exclusive. You cannot browse disk files when the URL check box is selected, and you cannot enter a URL if the URL check box is not selected. To open (or save) the URL, click Open URL (or Save URL).
5. If you have changed a data set since you last read it in or wrote it out and only data operations supported by the data journal have been performed, Tecplot asks if you want to use

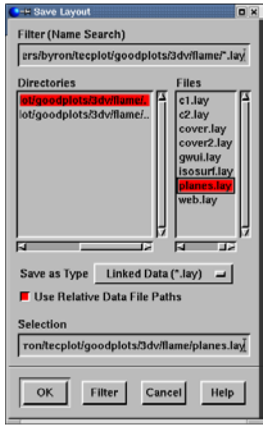


Figure 5-11. The Save Layout dialog in Motif, showing the Linked Data (*.lay) option.

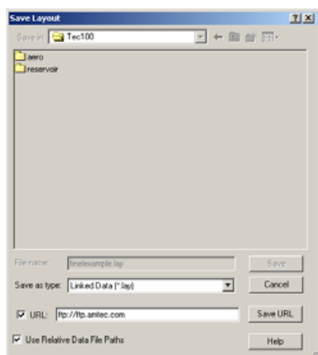


Figure 5-12. The Save Layout dialog in Windows, showing the URL check box.

the original data set along with journaled data instructions. This is the default; it continues to reference the original data files, minimizing disk storage requirements and changes to the original data are reflected in later Tecplot sessions.

If you choose to save the current data to new files and reference the new files in the layout, or if you have modified the data in ways not supported by the data journal, Tecplot prompts you for a file name under which to save the changed data. If your layout has multiple data sets, Tecplot prompts you for a file name for each modified data set.

Once you save a layout, the layout file name appears in the Tecplot header. To save your layout to this file again, choose Save Layout from the File menu.

5.3.2.2. Layout File Opening. You open layout files using the Open Layout option under the File menu.

To open your layout file:

1. From the File menu, select Open Layout. The Open Layout dialog appears.
2. Specify a file name, then click OK. If you have an unsaved layout, Tecplot asks you if you want to save it before opening a new layout file.
3. (Optional) To combine the layout file with the current layout in Tecplot, click on the Append check box prior to clicking OK.

5.3.2.3. Layout File Opening with Different Data Files. When you open your layout files in Tecplot, you have the option of overriding the data files that are referenced in the layout file. This does not change the layout file, however, you can save a layout file with the new data files.

This section describes the process for overriding a data file from the Open Layout dialog. A second method is described in Appendix A, Section A.5, “Overriding the Data Sets in Layouts by Using "+" on the Command Line.”

To open a layout file with different data files than those specified in the layout file:

1. From the File menu, choose Open Layout. The Open Layout dialog appears.
2. Select Data Override.
3. Specify the layout file name, and click Open.
4. The Override Layout Data dialog appears, as shown in Figure 5-13.

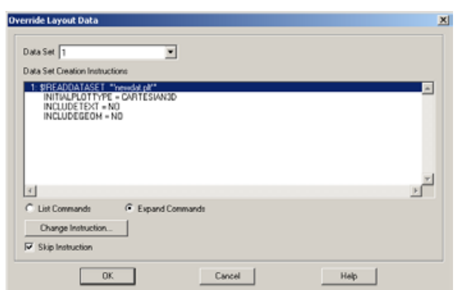


Figure 5-13. The Override Layout Data dialog.

5. One line is listed for each data set in the layout file. Each line contains the data reader name (**`TECPLOT`** for Tecplot-format data files). If the data set is being loaded by the Tecplot reader, this line shows the number of files making up the data set, and a partial list of file names. If a data loader add-on is used, instructions used by the loader are listed here. (This could be a list of file names identical to the Tecplot loader’s list.)

6. To change the data files or instructions making up a data set, double-click on the appropriate line, or select the line and click Change.
7. This process includes one or more dialogs allowing you to change the list of file names or instructions. Tecplot-format data files include a dialog to select new files. If the data loader does not have the capability to override the instructions, an error message appears.
8. Repeat steps 6 and 7 until the files you want are listed, then click OK.

5.3.3. Layout Package Files

Layout package files allow you to transmit raw data, along with style information, so colleagues can view the results in Tecplot. With layout package files the view can be changed, different plot types tested, and so forth.

Layout package files are very useful if you are making large documents containing many images, or other situations when you need to catalog your images. You can extract contents using the command line utility **`lpkview`**. This utility allows you to look at thumbnail sketches of each image in a layout packing file without having to load each separately into Tecplot. For more information see Section 5.3.3.3, “Layout Package Utility.”

Layout package files have the same properties as standard layout files. (For more information about layout files, see Section 5.3.2, “Layout Files.”) Layout package files also contain all data associated with frames in the layout, and an optional preview image of the Tecplot workspace. An extension of **`.lpk`** is used.

5.3.3.1. Layout Package File Opening. Open a layout package file using the Open Layout option of the File menu. You may open both standard layout files and layout package files from this dialog.

The Data Override check box is only applicable when the file being opened is a standard layout file. If you select Data Override, and the file you open is a layout package file, a warning dialog appears; Tecplot loads the layout package with the original data.

5.3.3.2. Layout Package File Saving. Save a layout package file using the Save Layout or Save Layout As options of the File menu.

To save your layout package file:

1. From the File menu, select Save Layout As. The Save Layout dialog appears.
2. Choose “Packaged Data (*.lpk)” from the Files of type drop-down, as shown in Figure 5-14.
3. Choose whether or not you want a preview image included with your layout package file.
4. Specify a file name for the layout package file.

Once you save a layout package file, the layout package file name appears in the Tecplot header. To save your layout package to this file again, choose Save Layout from the File menu.

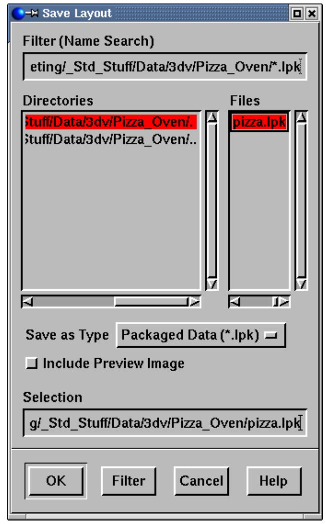


Figure 5-14. The Save Layout dialog in Motif, showing the Packaged Data (*.lpk) and Include Preview Image options.

5.3.3.3. Layout Package Utility. As a convenience a command line utility, **`lpkview`**, is also provided to catalog and unpack layout packages. In its simplest form, only a layout package file is given to the utility; all default options are assumed. Using default options cause the utility to unpack the preview image (if present), the layout, and all associated data files into the directory in which the utility was run. For example:

```
lpkview myplot.lpk
```

might unpack the following files in the current directory:

```
myplot.png
myplot.lay
welldata.plt
grid_1.plt
grid_2.plt
```

Tecplot determines the names for unpacked files when the package is created. Tecplot eliminates name conflicts within the package by appending unique numbers to non-unique names. However, no attempt is made by **`lpkview`** to ensure that names are unique with other files located in the directory where the items are unpacked.

If only the layout and associated data files are desired then run the command as follows:

```
lpkview -l -d myplot.lpk
```

Using the **-t** options will output the names of the items within the layout package file without unpacking them. For example:

```
lpkview -t wingperf.lpk
```

might output the following catalog to standard output:

```
wingperf.png
wingperf.lay
run_alpha12beta5.plt
```

For a complete list of lpkview command lines, see Appendix B.2, “LPKView.”

5.4. Plot Publishing for the Web

Publish allows saving plots directly to an HTML file, from which you may read and write data and layout files to ftp:// and http:// sites. A Tecplot HTML file could include a reference to a layout package file of your analysis, enabling other Tecplot users browsing your files to review your results directly. The Publish Options dialog and a Tecplot HTML file are shown in Figure 5-15.

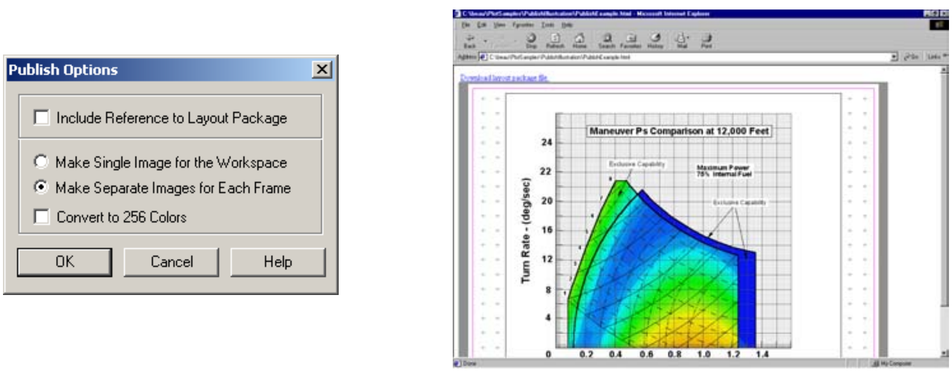


Figure 5-15. The Publish Options dialog creates layout package files for the Web.

Publish creates an HTML file referencing the plot images in your Tecplot workspace. Publish also creates layout package files with a link from the HTML file to the layout package file.

To create a Publish file:

1. Choose Publish from the File menu.
2. Decide whether you would like to reference a layout package file. Selecting this option creates a layout package file, along with a reference to that file in the resulting Publish file.
3. Select what images are to be referenced in your Publish file.

Selecting the Make Single Image for the Workspace check box creates a single image file of your entire Tecplot workspace. A single reference is added to your Publish file for this image file.

Selecting the Make Separate Images for Each Frame check box creates an image for each frame in your Tecplot workspace. A separate reference for each frame is added to your Publish file.

4. Click OK. You will be presented with a standard file input/output dialog. The name you enter for your file will be used to create the HTML file, create the associated image files, and optionally, the layout package file.

5.5. Other Tecplot Files

In addition to the basic ASCII and binary data files, Tecplot uses a number of other files:

- **Equation:** Equation files are used to store equations for data manipulation. They are discussed in their entirety in Chapter 24, “Data Operations.”
- **Macro:** Macro files are used to record and play back Tecplot operations and to set up Tecplot for animation or batch mode. See Chapter 27, “Macro Commands,” for more details, and the *Tecplot Reference Manual* for an annotated list of all macro commands.
- **Color map:** A color map file is a Tecplot macro file that saves and restores RGB color values used for contour flooding and multi-coloring. See Chapter 10, “Contour Plots,” for details on creating and modifying Tecplot color map files.
- **Print:** Print files are files created with Tecplot (or Windows) printer drivers. See Chapter 21, “Printing,” for more details.
- **Export:** Export files are graphics files created by Tecplot for import into graphics editing or word processing programs. All the print file formats are available for export, as are several types of bitmaps, Windows Metafile (WMF) Format, and Encapsulated PostScript (EPS) format. For more information on creating bitmap files for export, see Chapter 22, “Exporting.”
- **Curve-coefficient:** These files contain the coefficients for the equations used to draw curves in XY-plots. These are output files only; they cannot be read back into Tecplot. See Chapter 7, “Line Plots,” for more details.

CHAPTER 6 *Data Loaders: Tecplot's Import Feature*

Tecplot allows you to load data in a number of formats with loaders that Amtec has produced using the Add-on Developer's Kit. The Import option on the File menu accesses a scrolled list of data loaders.

This chapter tells you how to load data in the following formats: Computational Fluid Dynamics General Notation System (CGNS); Digital Elevation Map (DEM); Digital eXchange Format (DXF); Excel (Windows only); Fluent Version 5 and 6 (**`.cas`** and **`.dat`**); Gridgen; Hierarchical Data Format (HDF); Image; PLOT3D; Polygon (PLY); text spreadsheet.

Access data loaders using the File menu's Import option. The Import dialog appears, as shown in Figure 6-1.

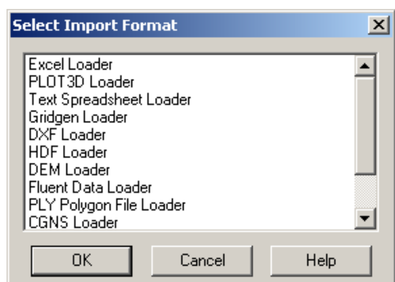


Figure 6-1. The Import dialog, accessed via the File menu.

New data loaders are posted at our Web site, www.tecplot.com, as they become available. You can also build your own data loaders using the Add-on Developer's Kit.

6.1. The CGNS Loader

The CGNS Loader reads files created with CGNSLib Version 2.3 or earlier into Tecplot. Choose either all or specific bases, zones, and solutions for loading into Tecplot zones. You can also select field variables individually. Define index ranges to load specific sub-zone blocks or planes for structured-grid zones.

Only CGNS bases and zones with valid grids can be read by the CGNS Loader. For unstructured grids Version 2.1 of the CGNS Loader supports BAR_2, TRI_3, QUAD_4, TETRA_4, PYRA_5, PENTA_6, HEXA_8, MIXED element types and their combinations on every section. However, the CGNS Loader does not support higher-order element types.

Only vertex and cell-center field variable locations are supported. Cell-centered data is averaged to the nodes when the file is read. For cell-centered structured grids arithmetic averaging is used. Rind data is used in the averaging if it is available. For cell-centered unstructured grids either a Laplacian averaging or arithmetic averaging can be selected to average the cell data to the surrounding nodes.

The dialog for the CGNS Loader is shown in Figure 6-2.

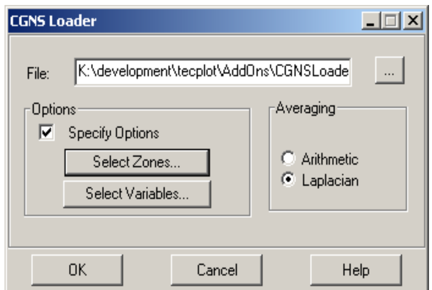


Figure 6-2. The dialog for the CGNS Loader.

The following options are available:

- **File:** Enter the name of the file to load, including the complete path, or click "**...**" to browse for the file.
- **Specify Options:** Active when a valid file is entered or selected. If not selected, every base, zone, solution, and variable is loaded into Tecplot when you click OK. This option allows you to control the data loaded from your CGNS file, including loading only particular zones, field variables, or partial zones.
- **Select Zones:** Launches the Load CGNS Options: Zones dialog, which allows you to select specific zones and partial zones to load.
- **Select Variables:** Launches the Load CGNS Options: Variables dialog, which allows you to select specific field variables to load. Grid variables are always loaded automatically.
- **Averaging:** When the field variables are stored at cell centers, either a Laplacian averaging or arithmetic averaging may be used to average the cell data to the nodes they surround. This can result in a bias at the boundary nodes. Arithmetic averaging is automatically used for ordered/structured zones. When available, Rind data is used in the averaging.

- **CGNS Section Mapping:** CGNS files sometimes have multiple node-maps for each finite-element zone. These are called sections, and a zone may contain sections with different cell-types and cell dimensions. For example, one section may be TETRA_4 (tetrahedral element - 3D cell dimension) for the volume zone and another section may be TRI_3 (triangular element - 2D cell dimension) for various surfaces surrounding the volume zone. If you select *one Tecplot zone per CGNS zone/solution* (default), it will combine all sections with the zone cell-dimension into one Tecplot zone. If you select *Load each section as separate Tecplot zone*, it will creat a separate Tecplot zone for each section regardless of cell dimension.

6.1.1. CGNS Loader Options: Zones Dialog

Tecplot zones are not always equivalent to CGNS zones. This dialog specifies zones to load from CGNS data files; it is shown in Figure 6-3.

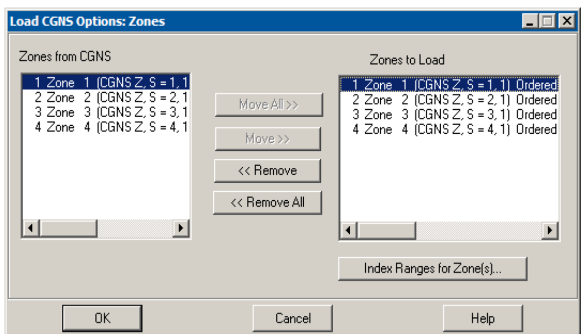


Figure 6-3. CGNS Loader: Zones dialog.

For example, each solution for a CGNS zone is considered a unique Tecplot zone. The CGNS base, zone, and solution hierarchy orders the zones. The integer preceding the word Zone is the Tecplot zone number assigned to that zone. The integer following Zone represents the order the zone was found in the CGNS file. The zone description includes the CGNS hierarchy information. “CGNS B, Z, S =” followed by three integers represents the CGNS order for the base, zone, and solution, respectively. “CGNS Z, S =” and two integers are displayed if a single base is found. The description also indicates whether the zone is ordered (structured) or finite-element (unstructured). I-, J-, and K-dimensions are provided for ordered zones; the number of nodes and elements are provided for finite-element zones.

By default all zones are selected for reading and displayed in Zones to Load. Delete the list by clicking Remove All. You can also remove zones by selecting them, then clicking Remove. This does not affect the order of the zones in Zones from CGNS. However, zones not shown Zones to Load are not assigned a Tecplot zone number. Similarly, all zones can be added to Zones to Load by clicking Move All, or added selectively using Move. Zones in Zones to Load are listed in the order they were added to the list, regardless of their order in Zones from

CGNS. This order is the basis for their Tecplot zone number and how they are identified in Tecplot.

To load a partial zone or sub-zone, first highlight the zone of interest in Zones to Load, then click Index Ranges for Zone(s). The Load CGNS: Index Ranges dialog appears.

6.1.1.1. CGNS Loader Options: Index Ranges Dialog. This dialog specifies a sub-set of the selected ordered/structured zone(s) to be loaded. Define a block, plane, or line of points for extraction on loading.

This is accomplished by inputting the ranges for the I-, J-, and K-indices. Each index requires Start, End, and Skip values. The Skip value reduces the number of points loaded by skipping points. A Skip value of three loads every third point after the Start point. (Start and End points are always loaded.) If multiple zones are selected prior to calling up the CGNS Loader: Index Ranges dialog, the default values for End are *Mx*, indicating the maximum value for each zone. You can enter any value for End; it is only be used in the appropriate zones. If the value is greater than the maximum index for a zone, End is replaced by the maximum index.

For multi-dimensional zones the number of points that can specified to load for the I- and J-directions must be greater than one. If the inputs for Start, End, and Skip result in a single point in either direction, an error message appears; change the inputs or cancel the dialog. The CGNS Loader: Index Ranges dialog is shown in Figure 6-4

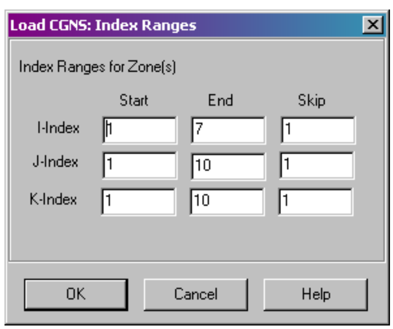


Figure 6-4. CGNS Loader: Index Ranges dialog.

6.1.2. CGNS Loader Options: Variables Dialog

The CGNS Loader: Variables dialog includes the Variables from CGNS and Variables to Load list boxes. It is shown in Figure 6-5.

The Variables from CGNS list includes all field variables that were found in the CGNS data file regardless of the zone(s) to which they belong, since CGNS files may contain zones that have different field variables. The Variables to Load list contains the field variables that have been selected to load into Tecplot. Initially, both lists are the same. A Tecplot variable number

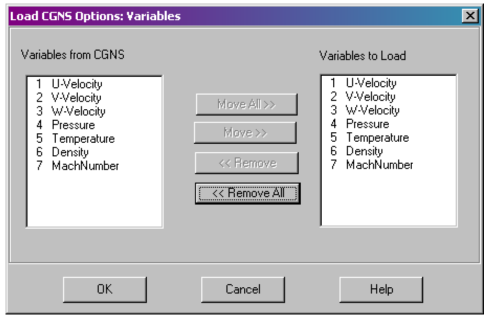


Figure 6-5. CGNS Loader: Variables dialog.

is assigned to each CGNS field variable that appears in the Variables to Load list. Since Tecplot requires every zone to have the same number of variables, each zone that is loaded into Tecplot will include every variable in the Variables to Load list. This will occur regardless of whether the zone included that field variable in the CGNS file. The variables that were not originally in the zone will be initialized to zero before they are loaded into Tecplot. Since this can unnecessarily increase the size of the data set in Tecplot, you are cautioned to carefully examine the Variables from CGNS list to guarantee that the field variables selected for the Variables to Load list will be consistent with the zones to be loaded.

Remove All lets you delete all variables from the Variables to Load list. Move All lets you include all variables from the Variables from CGNS list. You can also selectively remove variables from the Variables to Load list by highlighting the variable(s) with a click, click-and-drag, Ctrl-click, or Shift-click, and then clicking Remove. The field variables that do not appear in the Variables to Load list will not have a Tecplot variable number assigned to them. Similarly, variables can be added to the Variables to Load by selectively highlighting the variable(s) to be added in the Variables from CGNS list and then clicking Move. Note that variables in the Variables to Load list will be listed in the order in which they were added to the list regardless of their order in the Variables from CGNS list. This order is the basis for their Tecplot variable number, which is how they will be ordered in Tecplot.

6.2. The DEM Loader

The DEM Loader add-on can load Digital Elevation Map files that have the same file format as the U.S. Geological Survey's standard DEM format. These files are generally used by cartographers and geologists to map terrain. The DEM Loader will not accept Spatial Data Transfer Standard (SDTS) formatted data.

DEM files are available on the Web for a number of states within the U.S. For more information, refer to the following references:

- **General:** edcwww.cr.usgs.gov/doc/edchome/ndcdb/ndcdb.html.

- **User’s guide:** edcwww.cr.usgs.gov/glis/hyper/guide/l_dgr_dem.

The DEM Loader first launches a multi-file selection dialog. After choosing one or more DEM files to load, you are presented with a simple dialog where you can set the I- and J-skipping. Since DEM files are quite large, you will likely want to set both of these to be 10 or more.

Figure 6-6 shows the DEM Loader dialog.

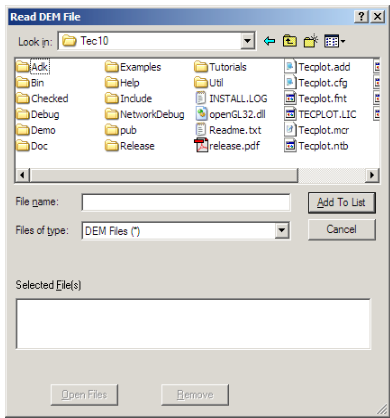


Figure 6-6. The Digital Elevation Map (DEM) Loader dialog.

6.3. The DXF Loader

The DXF Loader add-on can import AutoCAD DXF (drawing interchange) files. When importing a file, Tecplot creates an appropriate geometry for each of the following entity types:

- Text.
- Lines.
- Arc.
- Circle.
- Point.
- Solid.
- 3-D faces.

Note: When importing a DXF file, no zones are created. Instead, the geometries representing each entity type are simply added to the frame. Be aware that a typical DXF file can contain several thousand geometries, and these are all included when you save a layout file.

6.3.1. The Load DXF File Dialog

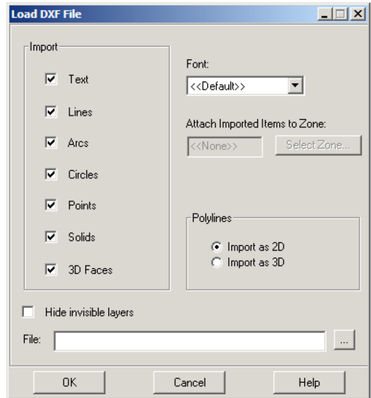


Figure 6-7. The Load DXF File dialog.

The Load DXF File dialog (Figure 6-7) has a variety of features, most of which are self-explanatory.

You can select any of the following:

- **Import:** Select any or all geometries to import -- Text, Lines, Arcs, Circles, Points, Solids, 3D Faces.
- **Font:** Select the font to use for text.
- **Attach Imported Items to Zone:** Specify a zone to which all imported geometries will be attached. Clicking the Select Zone button produces a menu of zone options.
- **Polylines/Import as 2D:** All lines and polylines are stored with three coordinates in DXF files. If you select this option, the loader will add 2-D line geometries for all lines and polylines in the DXF file (the third coordinate will be ignored).
- **Polylines/Import as 3D:** If you select this option, the loader will add 3-D line geometries for all lines and polylines in the DXF file. To view a 3-D DXF file, create or load a 3-D zone, import your DXF file, then choose Fit to Full Size from the View menu.
- **Hide Invisible Layers:** If this option is checked, objects in layers which are “off” in the DXF file will be imported with the background color.

6.3.2. DXF Loader Limitations

The DXF Loader does not create any field data. Loading a DXF file only adds geometries to your existing frame.

Since most geometries in Tecplot are 2-D, best results will be obtained by loading “flat” DXF files, such as maps.

Binary AutoCAD (**`*.dwg`**) are not supported in this release.

6.4. The Excel Loader

The Excel Loader add-on can read numeric data from Microsoft Excel version 5.0 or higher **`.xls`** files and import the data into Tecplot. It is shown in Figure 6-8. The Excel Loader is available only for Windows platforms.

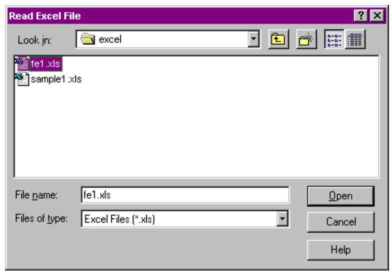


Figure 6-8. The initial Excel Loader dialog.

This Import function is only useful for basic formats, requiring your Excel file to contain only values, no functions. Amtec recommends use of the Excel macro from the **`Util/Excel`** folder as an easier method to open Excel data with Tecplot.

The Excel loader is a point-and-click operation if your spreadsheet is arranged in either of two ways, which we refer to as table format and carpet format. Other formats can also be loaded and imaged, but they require a little more work, and in some cases more sophistication, on your part.

Once you have chosen an Excel file to load into Tecplot, the Excel Loader leads you through a series of dialogs that let you specify a variety of attributes such as the format of the data in the Excel spreadsheet, the variables to read into Tecplot, and zone information.

6.4.1. Spreadsheet Data Formats

The Excel Loader will automatically identify blocks of data in table or carpet format, that is, blocks that satisfy the conditions of these formats. The characteristics of these formats are described in Section 6.4.1.1, "Table Format," and Section 6.4.1.2, "Carpet Format." If the loader has identified any carpet or table format blocks, you may select them from a list.

The loader will list blocks of data in standard Excel notation. For example, a block found on worksheet *sheet1*, cells A1-D8, is listed as follows: **`(sheet1 ! A1:D8)`**. It is important to verify that the matrix specified is actually the matrix of data you wish to load.

If you select a user-defined format (or if the loader did not identify any carpet or table blocks), then you will be prompted to enter the names and number of variables, and one or more zones and associated properties. For each zone you will also have to enter the location of the field data in the spreadsheet.

6.4.1.1. Table Format. Table format is especially applicable to spreadsheets containing data that will be plotted in line plots—generally, data that represent an independent and one or more dependent variables. Many spreadsheets containing data to be plotted in 2D or 3D Cartesian plots will also satisfy the conditions of table format. A table formatted data set has the following characteristics:

- The data set is arranged in one or more adjacent columns.
- Each column is the same length and contains numeric data.
- At the top of each column is a variable name (that is, a cell containing the text label of the variable).
- The spreadsheet data set is imported as a single I-ordered zone in **`POINT`** format with *N* variables, where *N* is the number of columns in the table.
- There must be no blank cells within the block of data. An empty cell will prevent the loader from recognizing the block. You can satisfy this condition by filling blank cells with 0.0.
- The block of data must be surrounded by empty cells, text-filled cells, or table boundaries. The loader will not recognize a block of data as being in table format if any cell adjacent to the block is filled with a number.

Figure 6-9 shows an Excel block in table format.

	A	B	C
1	Month	Seattle Rainfall	
2	1	4.3	
3	2	4.5	
4	3	4	
5	4	4.2	
6	5	3.5	
7	6	2.1	
8	7	2	
9	8	1.5	
10	9	2.1	
11	10	2.5	
12	11	3.3	
13	12	3.5	
14			

Figure 6-9. A block of data in table format. Note that for the block to be recognized as such by Tecplot, it must be bounded by spreadsheet edges, text, or empty cells.

6.4.1.2. Carpet Format. A spreadsheet to be plotted in a 2D or 3D Cartesian plot is likely to be in carpet format. The carpet formatted data set, shown in Figure 7-10, has the following characteristics:

- The spreadsheet data set is imported as an IJ-ordered zone. In Figure 6-10, the spreadsheet is imported as *I=4* and *J=4*. The three variables are X, Y and V. In the spreadsheet cell 2B is index **`1`**, **`1`**, cell 3B is index **`2`**, **`1`**. See section 4.2.2, "IJ-Ordered Data."
- The top row in the block contains the values of the X-variable, the first column of the block contains the values of the Y-variable, and the V-values are the interior data. This format is useful if your data set was generated from a function *f*, such that *f*(X, Y) = V. This may be a simple arithmetic function of X and Y, or may represent measurements of some variable at points on a grid.
- The block is a rectangular arrangement of numeric data in the spreadsheet, with a blank cell in the upper left hand corner.
- There must be no blank cells within the block of data. An empty cell will prevent the loader from recognizing the block. You can satisfy this condition by filling blank cells with **`0.0`**.
- The block of data must be surrounded by empty cells, text-filled cells, or table boundaries. The loader will not recognize a block of data as being in carpet format if any cell adjacent to the block is filled with a number.

	A	B	C	D	E	F
1		1	2	3	4	
2	1	1	2	3	4	
3	2	2	4	6	8	
4	3	3	6	9	12	
5	4	4	8	12	16	
6						

Figure 6-10. The carpet table shows values as a simple arithmetic function of X and Y.

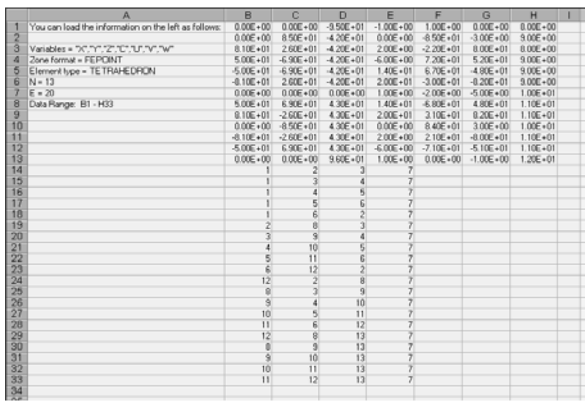
6.4.1.3. Other Formats. The Other format option gives you a great deal of flexibility in loading data into Tecplot, but also requires you to give the loader more information about the block of data you are loading. A series of dialogs leads you through the process of describing your data, similar to the way you would specify this information in a Tecplot ASCII file. Some of the most relevant attributes of your data set and its format are described in Section 4.1.2.4, "Data Types," Section 4.2, "Ordered Data," and Section 4.3, "Finite-Element Data."

- **Default format:** The Excel Loader offers a semiautomatic option that requires only that you specify the upper left and lower right corners of your data block. Once you've specified those corners, it handles the data in the same way that Tecplot handles an unformatted block in an ASCII file. That is, it assumes one zone of I-ordered data in **`POINT`** format.

- **Custom format:** Using the Custom format option, you can specify characteristics of your data set. Custom format has the following features:
  - It allows you to work with spreadsheets containing blank cells or text cells.
  - For XY-, IJ- and IJK-ordered data, you'll tell the loader the boundaries of the block to load, and how many data points there are within that block (*IMax*, *JMax*, *KMax*).
  - For finite-element data, the number of data points is implied by the number of nodes and number of elements.
  - Allows you to load blocks of cells that you delimit interactively.
  - It is the only option for loading finite-element, IJK-ordered, or zone data from Excel. If a user wants to read in data from an Excel spreadsheet into more than one Tecplot zone the custom format must be used. The default assumes that all data read should be put in a single I-ordered zone.

6.4.2. FEPOINT Excel File in User-Defined Format Example

The Excel spreadsheet in **`Tec90/examples/loaders/xls/fe1.xls`** (Figure 6-11) contains data in finite-element **`POINT`** format (refer to Section 4.3, "Finite-Element Data," for a discussion of FE-point format). The procedure for loading the data into Tecplot is as follows:



	A	B	C	D	E	F	G	H	I
1	You can load the information on the left as follows:	0.00E+00	0.00E+00	-9.50E+01	-1.00E+00	1.00E+00	0.00E+00	0.00E+00	
2		0.00E+00	8.50E+01	-4.20E+01	0.00E+00	-8.50E+01	-3.00E+00	9.00E+00	
3	Variables = "X","Y","Z","C","U","V","W"	8.10E+01	2.60E+01	-4.20E+01	2.00E+00	-2.20E+01	8.00E+01	8.00E+00	
4	Zone format = FEPOINT	5.00E+01	-6.90E+01	-4.20E+01	-6.00E+00	7.20E+01	5.20E+01	9.00E+00	
5	Element type = TETRAHEDRON	-5.00E+01	-6.90E+01	-4.20E+01	1.40E+01	6.70E+01	-4.80E+01	9.00E+00	
6	N = 13	-8.10E+01	2.60E+01	-4.20E+01	2.00E+01	-3.00E+01	-8.20E+01	9.00E+00	
7	E = 20	0.00E+00	0.00E+00	0.00E+00	1.00E+00	-2.00E+00	-5.00E+00	1.00E+01	
8	Data Range: B1 - H33	5.00E+01	6.90E+01	4.30E+01	1.40E+01	-6.80E+01	4.80E+01	1.10E+01	
9		8.10E+01	-2.60E+01	4.30E+01	2.00E+01	3.10E+01	8.20E+01	1.10E+01	
10		0.00E+00	-8.50E+01	4.30E+01	0.00E+00	8.40E+01	3.00E+00	1.00E+01	
11		-8.10E+01	-2.60E+01	4.30E+01	2.00E+00	2.10E+01	-8.00E+01	1.10E+01	
12		-5.00E+01	6.90E+01	4.30E+01	-6.00E+00	-7.10E+01	-5.10E+01	1.10E+01	
13		0.00E+00	0.00E+00	9.60E+01	1.00E+00	0.00E+00	-1.00E+00	1.20E+01	
14		1	2	3	7				
15		1	3	4	7				
16		1	4	5	7				
17		1	5	6	7				
18		1	6	2	7				
19		2	8	3	7				
20		3	9	4	7				
21		4	10	5	7				
22		5	11	6	7				
23		6	12	2	7				
24		12	2	8	7				
25		8	3	9	7				
26		9	4	10	7				
27		10	5	11	7				
28		11	6	12	7				
29		12	8	13	7				
30		8	9	13	7				
31		9	10	13	7				
32		10	11	13	7				
33		11	12	13	7				
34									

Figure 6-11. Excel file **`fe1.xls`**, used in the example in Section 6.4.2.

1. Select the Import option from the File menu.
2. Choose Excel.
3. In the Read Excel File, specify a path and a file, and click OK.
4. In the Import Excel File—Step 1 dialog, you are restricted to Other format, because **`fe1.xls`** does not satisfy the conditions of table or carpet format. Click Next>.
5. In the Step 2 of 4 dialog, add seven variables of type Double, and a title if you wish. Click Next>.

6. In the Step 3 of 4 dialog, click Add.
7. From the Add menu select the Edit Zone option and specify that:
   - You choose the block of data that extends from B1 to H33.
   - The format of the data file is **`FEPOINT`**.
   - The data set contains 13 nodes.
   - Those nodes are connected into 20 elements.
   - The element type is **`TETRAHEDRON`**.
8. Click OK. The Step 3 of 4 dialog now displays the zone you have described, with a + button that you can press to display your parameters.
9. Click Next>.
10. The Import dialog displays some of your choices. Confirm them and click Finish.
11. The initial plot is 2D Cartesian, which you can convert to 3D Cartesian for a full view of the finite-element volume (Figure 6-12).

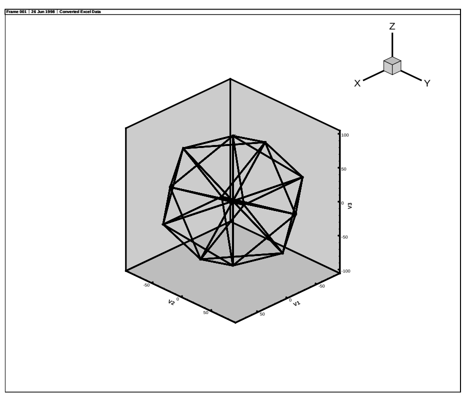


Figure 6-12. Excel spreadsheet **`fe1.xls`** 3D Cartesian plot.

6.4.3. Excel Loader Restrictions

Recall that a block of data is a rectangular group of numbers in the spreadsheet. The loader places the following restrictions on blocks:

- Carpet and table format, which the loader detects and loads automatically, are fairly narrowly defined. All other formats must be loaded on the user-defined pathway.
- Numeric cells within each block should contain only numbers or numeric characters such as **+**, **-**, and so forth. For example, a cell which contains *X=34* is interpreted by the loader as text, since it begins with text.

- Cells containing formulas (therefore displaying calculated values) will be skipped by the loader. You can convert the formulas to values within Excel.
- The spreadsheet file must have been written by Excel Version 5.0 or higher. Earlier versions of Excel are not supported.

6.5. The Fluent Loader

The Fluent Data Loader allows you to read Fluent Version 5 and 6 case (**`.cas`**) and data (**`.dat`**) files into Tecplot. To load files from earlier versions of Fluent, you must first import them into Fluent 5 or 6, then save them as Fluent 5 or 6 files.

Fluent stores solution data at cell centers (face centers for boundary zones). By default, the Fluent data loader loads the data cell-centered as well, but it has options to average the data to the nodes if this should be necessary (for use by other add-ons, for example). Two averaging options are available. Arithmetic averaging is faster, but calculates values at hanging nodes (nodes in the center of a cell face or edge) only from those cells of which the node is a corner. This can lead to discontinuous contours due in part to this one-sided averaging. The slower Laplacian averaging option takes additional neighboring cells into account, and gives smoother contours when hanging nodes are present.

The Fluent Data Loader dialog is shown in Figure 6-13.

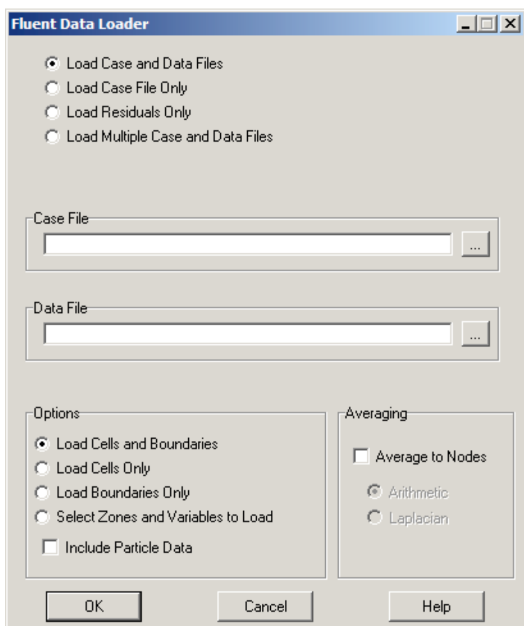


Figure 6-13. T

The following options are available:

- **Load Case and Data Files:** Loads both a case and a data file. The grid comes from the case file, and the solution comes from the data file.
- **Load Case File Only:** Loads the grid from a case file.
- **Load Residuals Only:** Loads the residual data (convergence history) from a data file. The residuals are not scaled or normalized.
- **Load Multiple Case and Data Files:** Displays the File List form in the dialog. You can load matched pairs of case and data files, or one cae file and any number of data files that match it (that is, that have the same zones).

For all load options above except Load Multiple Case and Data Files, the following controls are available:

- **Case File:** Type the name of the case file you wish to load, or click Select, then select the name of the file from the resulting dialog.
- **Data File:** The data (.dat) file contains the solution and the residual (convergence history) data. Type the name of the data file, or click Select, then select the name of the file from the resulting dialog.

For the Load Multiple Case and Data Files load option, the following controls are available:

- **Add Files:** Choose case and data files to load from a file selection dialog. Selected files are appended to the file list.
- **Remove:** Remove files you have selected in the file list.
- **Remove All:** Remove all files in the file list.
- **Flow Solution is Unsteady:** Indicates that the set of case and data files represents an unsteady solution. The loader adds a TIME auxiliary data item to each loaded zone. Tecplot does not use this data, but other add-ons may.
- **Time Interval:** The time interval between successive data files in the file list. The TIME auxiliary data item will be incremented by this amount (starting at zero) for each successive data file loaded.

For all load options except Load Residuals, some or all of the following controls are available:

- **Load Cells and Boundaries:** Loads the cell (solution) and boundary zones from the case file. Each fluid or solid cell zone and each boundary zone will be displayed as a separate zone in Tecplot.
- **Load Cells Only:** Loads only the cell (solution) zones. Each zone will be displayed as a separate zone in Tecplot.
- **Load Boundaries Only:** Loads only the boundary zones. Each zone will be displayed as a separate zone in Tecplot.

- **Select Zones and Variables to Load:** Select in a separate dialog which zones and variables to load. The option requires the loader to pre-scan all files, which can be time-consuming.
- **Include Particle Data:** Some Fluent simulations include the effects of discrete particles, such as sand grains or water droplets. Select this option to load this particle data along with the flow solution. All particles from a particular injection will be displayed in a single Tecplot zone (one zone per injection). If you have chosen to select which zones and variables to load, this option is disabled, but the particle zones and variables will be displayed in the selection lists, allowing you to load them with the flow solution.
- **Average to Nodes:** By default, non-grid variables are stored at cell centers, consistent with Fluent. Selecting this option directs the loader to average Fluent's cell-centered data to the grid nodes. This can speed up subsequent operations in Tecplot, especially slicing. Choose the averaging method below.
- **Arithmetic:** A simple, fast arithmetic averaging will be performed.
- **Laplacian:** A more accurate, much slower averaging will be performed that accounts for hanging nodes and cell sizes.

If you chose the Select Zones and Variables to Load option, the Fluent Loader Options dialog, as shown in Figure 6-14, will be displayed when you click OK. This dialog allows you to select only those zones and variables you wish to load from the files.

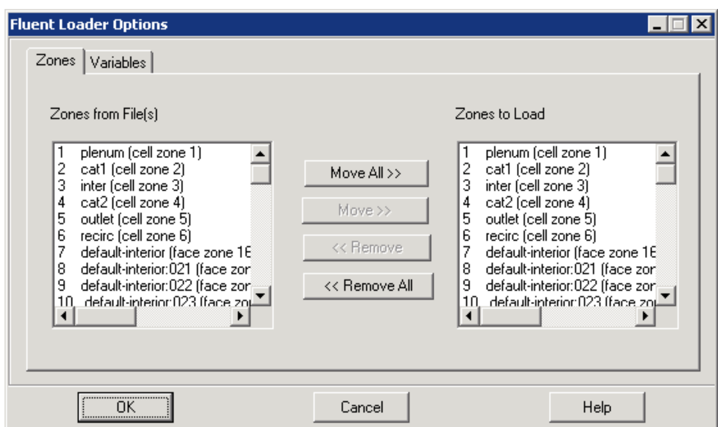


Figure 6-14. The Fluent Loader Options dialog.

This dialog has a Zones page and a Variables page. The left-hand list of each page shows, respectively, all zones and variables contained in the files you selected. The right-hand list of each page shows the zones and variables that will be loaded when you click OK. Both pages have the following controls:

- **Move All:** Adds all zones or variables to the list of those to be loaded.

- **Move:** Adds the zones or variables selected in the left-hand list to the list of those to be loaded.
- **Remove:** Removes the zones or variables selected in the right-hand list so they will not be loaded.
- **Remove All:** Empties the list of items to be loaded. You must subsequently add at least one item to load.

6.6. The Gridgen Loader

The Gridgen Loader add-on accepts output from Pointwise, Inc.'s Gridgen Version 11. (Amtec has not tested previous versions.) The Gridgen Loader can import the following types of Gridgen files into Tecplot:

- Database Network (**`*.net`**)—one IJ-ordered zone is created for each network in the file.
- Volume Grid (**`*.dat`**)—one IJK-ordered zone is created for each block of data in the file.

More information on Gridgen Volume Grid and Database Network files can be found in the *Gridgen User's Manual*.

The files can be in any of the following formats, which are automatically detected:

- ASCII.
- Binary formatted.
- Binary unformatted.
- Single or double precision.

The data set is given a default title of "Imported Gridgen Data," which you may change by selecting the Data Set Info option from the Data menu.

Variables names default to "**`X`**," "**`Y`**," and "**`Z`**." These can be changed within Tecplot after the data set is loaded.

The Gridgen Loader leads you through several screens, each of which allows you to specify one or more attributes of the input files (Figure 6-15).

6.6.1. Gridgen Data Loading with Tecplot

The Gridgen Loader dialog asks for the following information:

- File Type: Select the type of file you wish to import.
- **I-Skip, J-Skip, K-Skip:** Select the I-, J-, and K-Skip values. A value of 1 will read every data point, 2 will read every other data point, and so on.

After you have selected the file type and skip values, click OK and you will be prompted for one or more files to load. Select one or more files and click OK to load the files.

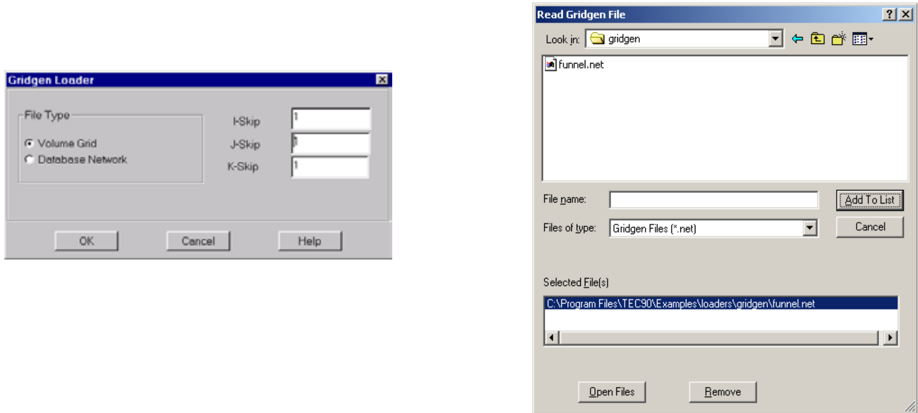


Figure 6-15. The Gridgen Loader dialogs.

6.7. The HDF Loader

The Tecplot HDF Loader add-on can load 1-D, 2-D, and 3-D Scientific Data Sets (SDS) from HDF files.

The HDF Loader dialog is shown in Figure 6-16.

Figure 6-16. The HDF Loader dialog.

A data set from an HDF file is imported as follows:

1. The file is scanned and a list of all SDS in the file is created.
2. You select one or more SDS to import. Each SDS that you select must have the same dimension.
3. A rectangular I-, IJ-, or IJK-ordered zone (for 1-, 2-, or 3-D data, respectively) is created for each SDS that you select to load.
4. The data is imported.

The HDF Loader dialog asks the following information:

- **Scientific Data Sets to load:** Select one or more SDS's to load. Each SDS that you select must have the same rank (dimension).
- **I-Skip:** Select the I-Skip value. A skip value of 1 loads every data point, a skip value of two loads every second data point, and so on.
- **J-Skip:** Select the J-Skip value.
- **K-Skip:** Select the K-Skip value.
- **Select File:** Select an HDF file.
- **Attributes:** Displays attributes of each SDS found, such as number type, rank, label, and so on.

Note: The HDF Loader uses the public-domain HDF API code library from the National Center for Supercomputing Applications (NCSA), University of Illinois, Urbana-Champaign.

6.7.1. HDF Loader Limitations

The HDF Loader can import only Scientific Data Sets from HDF files, and these are imported in a manner similar to NCSA's own HDF viewer. The way in which the data file is interpreted cannot be altered in this release of the loader. However, it is possible to write a Tecplot add-on (using the NCSA code library) which loads HDF data in a manner more suited to your particular use of the HDF format. See the *ADK User's Manual* for more information on writing add-ons.

6.8. The PLOT3D Data Loader

The PLOT3D Loader add-on can import data files formatted for the PLOT3D program developed by Pieter Buning at the NASA Ames Research Center. Some extensions such as unstructured data that are now available in FAST, the successor to PLOT3D, are also supported.

6.8.1. File Combinations

The File Selection page of the PLOT3D Loader dialog is shown in Figure 6-17.

You may specify to load just the grid file, both the grid and solution files, or just solution files.

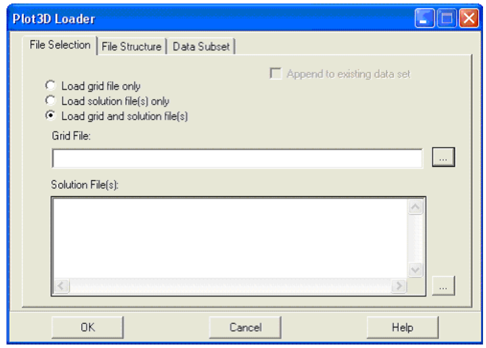


Figure 6-17. The File Selection page of the PLOT3D Loader dialog.

The following table describes all six scenarios and what the PLOT3D loader does:

Load	Not Appending	Appending
Grid Only	Existing data set is deleted and zones (one per grid are loaded).	New zones are added (one per grid). Solution vars in new zones are zeroed out.
Grid and Solution	Existing data set is deleted and zones (one for each grid in each solution file) are loaded. Each set of zones loaded shares spatial vars with the first set of grids loaded.	Same as Not Appending except original data set is preserved. Existing data set must have at least as many variables as the number needed by the incoming data.
Solution Only	A data set must already be present. The existing data set is reduced to contain the same number of zones as there are grids in each incoming solution file. Solution vars in the first solution file replace the solution vars in the original zones. Subsequent solution files create new sets of zones with spatial variables shared with the first set of zones.	Same as Not Appending except original data set is preserved. Existing dataset must have at least as many variables as the number contained in incoming solution file. Spatial vars are shared with last *n* original zones where *n* is the number of grids in each incoming solution file.

6.8.2. PLOT3D File Structure

The File Structure page of the PLOT3D Loader dialog allows you to choose to have the PLOT3D Loader auto detect the file structure, or override and manually describe the structure. The dialog is shown in Figure 6-18.

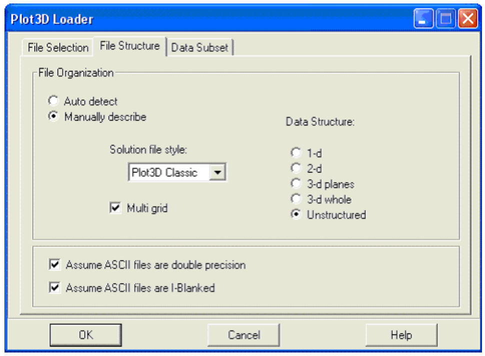


Figure 6-18. The File Structure page of the PLOT3D Loader dialog.

The PLOT3D Loader can auto detect most PLOT3D file variants. ASCII files are the most difficult to auto detect as there are a few combinations that have the exact same signature. Pure binary files also have some combinations that have the same signature. If all else fails use the manual settings to load in the files.

6.8.2.1. Unstructured Data Files. You must manually specify the data structure for all unstructured solution files if loaded without an accompanying grid file. Auto detect will think solution files are 3DW if loaded by themselves.

The following ASCII file conditions require special attention:

Condition	Notes
Double Precision	You must tell the loader if the incoming file is single or double precision.
I-Blanking	You must tell the loader if the incoming file contains I-blanking.
3-D Planar	There are some cases where these files can appear exactly the same if they are 3-D Whole. The PLOT3D loader always favors 3-D Whole. If you need to load 3-D Planar in 3-D Planar ASCII files you must specify the data structure manually.

6.8.2.2. Pure Binary Files. The following pure binary files (binary files without record markers) require special attention:

Condition	Notes
3-D Planar	There are some cases where these files can appear exactly the same if they are 3-D Whole. The PLOT3D loader always favors 3-D Whole. If you need to load in 3-D Planar pure binary files you must specify the data structure manually.

6.8.3. PLOT3D Data Subsets

The Data Subset page of the PLOT3D Loader (see Figure 6-19) allows you to read subsets of

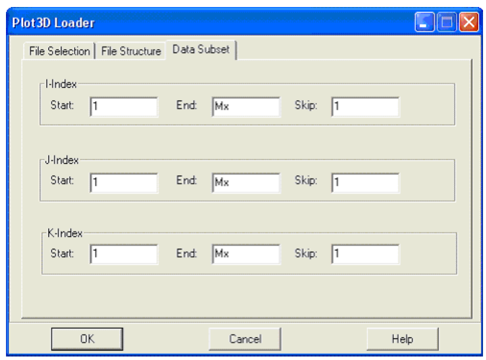


Figure 6-19. The Data Subset page of the PLOT3D Loader dialog.

ordered zones within the files. Specify the desired beginning and ending index values to read and a skip value for each index direction. A skip of one results in every value in the specified index range being read. A skip of 2 reads every second value, and so on.

6.8.4. Macro Language

The macro language syntax for the PLOT3D Loader shipped with Tecplot version 10 has changed from that of previous versions. Layouts created with previous versions can still be read, but will be saved with the newer syntax.

6.8.4.1. New Instruction Syntax. The new loader uses the Standard syntax so layouts can be saved and automatically use relative paths for file names. The following table lists the standard syntax name-value pairs used by the PLOT3D Loader:

Keyword	Value(s)	Default	Notes
STANDARDSYNTAX	1.0	None/ Required	Must be the first instruction.
FILELIST_SOLUTION FILES	"n" *"file-1" "file-2"* .... *"file-n"*	Empty	
FILENAME_GRIDFILE	*"filename"*	Empty	
IINDEXRANGE	*"indexrange"*	"1, ,1"	Start, End, Skip
JINDEXRANGE	*"indexrange"*	"1, ,1"	Start, End, Skip
KINDEXRANGE	*"indexrange"*	"1, ,1"	Start, End, Skip
APPEND	"Yes" or "No"	"No"	
ASCIIISDOUBLE	"Yes" or "No"	"No"	
ASCIIHASBLANK	"Yes" or "No"	"No"	
AUTODETECT	"Yes" or "No"	"Yes"	
DATASTRUCTURE	*"datastructure"*	---	Required if **AUTODETECT** is "No," otherwise ignored.
ISMULTIGRID	"Yes" or "No"	---	Required if **AUTODETECT** is "No," otherwise ignored.
STYLE	*"plot3dstyle"*	---	Required if **AUTODETECT** is "No," otherwise ignored.

Where

datastructure is **"1D", "2D", "3DP", "3DW",** or **"UNSTRUCTURED"**

plot3dstyle is **"PLOT3DCLASSIC", "PLOT3DFUNCTION",** or **"OVERFLOW"**

6.8.5. PLOT3D Auxiliary Data

The following auxiliary data is created by the PLOT3D Loader:

Auxiliary Name	Assigned To
`Common.ReferenceMachNumber`	Data Set and Individual Zones (1)
`Common.AngleOfAttack`	Data Set and Individual Zones (1)
`Common.ReynoldsNumber`	Data Set and Individual Zones (1)
`Common.Time`	Individual Zones
`Common.DensityVar`	Data Set
`Common.UVar`	Data Set
`Common.VVar`	Data Set
`Common.WVar`	Data Set
`Common.StagnationEnergyVar`	Data Set
`Common.GammaVar`	Data Set
`Common.TurbulentKineticEnergyVar`	Data Set
`Common.TurbulentDissipationRateVar`	Data Set
`Common.VectorVarsAreVelocity`	Data Set
`Common.SpeedOfSound`	Data Set
`G (2)`	Individual Zones
`B (2)`	Individual Zones
`T (2)`	Individual Zones
`I (2)`	Individual Zones
`H (2)`	Individual Zones
`H1 (2)`	Individual Zones
`H2 (2)`	Individual Zones

Notes: (1) Auxiliary data assigned to both zones and the data set assign the value from the last zone processed to the data set; (2) Overflow specific constants.

6.8.6. PLOT3D Loader Limitations

The **-ip**, **-jp**, **-kp** options in older PLOT3D Loader are not supported in the initial release. Tecplot handles I-, J- and K-planes well, so loading 3-D planar files as a single zone is typically sufficient.

6.9. The Text Spreadsheet Loader

The Text Spreadsheet Loader add-on is both an example of how to write a loader add-on and a utility which lets you import simple data from ASCII files. The complete source code for the Text Spreadsheet Loader is included in the ADK Examples directory.

6.9.1. Data File Format

The Text Spreadsheet Loader can read ASCII files of the following format (blank lines are ignored):

```
Variable 1, Variable 2, ..., Variable N
datapoint1,datapoint2, ..., datapoint N
.
.
.
datapoint1,datapoint2, ..., datapointN
```

Here is an example of a valid ASCII spreadsheet file:

```
Month, Rainfall
1, 15.0
2, 21.0
3, 21.0
4, 32.0
5, 10.3
6, 5.1
7, 2.3
8, 0.2
9, 1.4
10, 8.3
11, 12.2
12, 15.4
```

6.9.2. Text Spreadsheet Loader Limitations

All of the variable names must be on the first line.

CHAPTER 7 Field Plots

A field plot is any 2D Cartesian or 3D Cartesian plot. Such plots combine one or more of the following zone layers:

- Mesh.
- Contour.
- Vector.
- Scatter.
- Shade.
- Boundary.

By default, 2- and 3-D field plots consist of the Mesh and Boundary zone layers. For example, when you read in the data set **`cylinder.plt`** (included with your Tecplot distribution), you automatically see the plot shown in Figure 7-1. The **`cylinder.plt`** data set contains three zones. By default all zones are plotted; Tecplot assigns basic colors cyclically to the individual

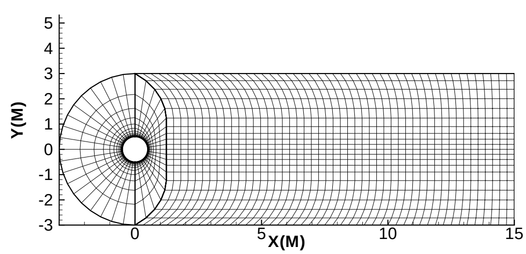


Figure 7-1. A 2-D mesh plot.

zones. However, you can assign any of Tecplot's basic colors to each zone. By default, the X- and Y-axes are dependent; if you change the range of one, the other changes to preserve the XY-aspect ratio. You can change virtually all attributes of the plot using the Axis, Field, and Style menus. This chapter discusses the basic plot attributes that are common to all field plots.

7.1. Two-Dimensional Field Plots

A 2-D field plot typically uses an IJ-ordered or finite-element surface data set. (You can view I-ordered data in a 2-D field plot, but XY-plots are typically more informative. Similarly, you can view IJK-ordered and FE-volume data with 2-D field plots, but 3-D views are usually better.) An IJ-ordered data file has the basic structure shown below:

```
TITLE = "Example: Multi-Zone 2-D Plot"
VARIABLES = "X", "Y", "Press", "Temp", "Vel"
ZONE T="BIG ZONE", I=3, J=3, DATAPACKING=POINT
1.0 2.0 100.0 50.0 1.0
1.0 3.0 95.0 50.0 1.00
1.0 4.0 90.0 50.0 0.90
2.0 2.0 91.0 40.0 0.90
2.0 3.0 85.0 40.0 0.90
2.0 4.0 80.0 40.0 0.80
3.0 2.0 89.0 35.0 0.85
3.0 3.0 83.0 35.0 0.80
3.0 4.0 79.0 35.0 0.80
ZONE T="SMALL ZONE", I=3, J=2, DATAPACKING=POINT
3.0 2.0 89.0 35.0 0.85
3.5 2.0 80.0 35.0 0.85
4.0 2.0 78.0 35.0 0.80
3.0 3.0 83.0 35.0 0.80
3.5 3.0 80.0 35.0 0.85
4.0 3.0 77.0 33.0 0.78
```

This data file has two zones and five variables, and is included with Tecplot as the file **examples/dat/multzn2d.dat**. The first zone has nine data points arranged in a three-by-three grid. The second zone has six data points in a three-by-two mesh. Reading this data file yields the mesh plot shown in Figure 7-2.

A 2-D finite-element data file is shown below:

```
TITLE = "Example: 2D Finite-Element Data"
VARIABLES = "X", "Y", "P", "T"
ZONE N=8, E=4, DATAPACKING=POINT, ZONETYPE=FEQUADRILATERAL
0.0 1.0 75.0 1.6
1.0 1.0 100.0 1.5
3.0 1.0 300.0 2.0
0.0 0.0 50.0 1.0
1.0 0.0 100.0 1.4
3.0 0.0 200.0 2.2
4.0 0.0 400.0 3.0
2.0 2.0 280.0 1.9
1 2 5 4
2 3 6 5
```

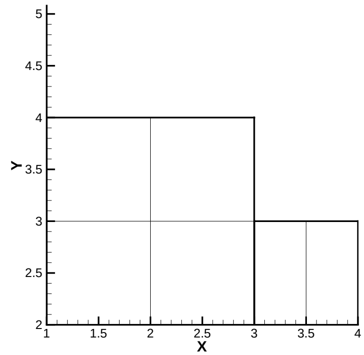


Figure 7-2. A 2-D field plot.

```
6 7 3 3
3 2 8 8
```

The above finite-element data file has eight nodes and four elements, with four variables, and is included in your Tecplot distribution as **`examples/dat/2dfed.dat`**. It yields the simple mesh plot shown in Figure 7-3.

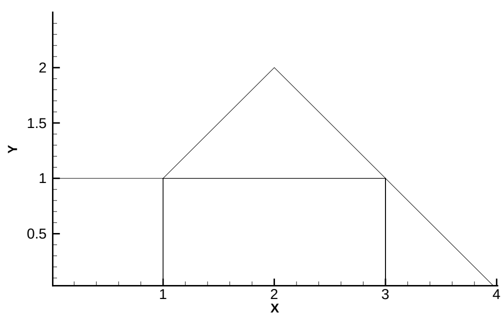


Figure 7-3. A 2-D mesh plot of a finite-element data set.

7.2. Three-Dimensional Field Plots

Creating a 3-D field plot from a 2-D plot is usually as simple as selecting 3D Cartesian from the plot type dropdown on the sidebar. For example, suppose you read in the file **`spc-ship.plt`** from the **`demo/3d`** directory, an IJ-ordered data set which by default is displayed in 2-D. Selecting 3D Cartesian yields the 3-D surface mesh plot shown at the left in Figure 7-4. The spaceship appears on its side by default; you can change this either by rotating the plot around the X-axis or by interchanging the axis assignments of the X- and Y-variables.

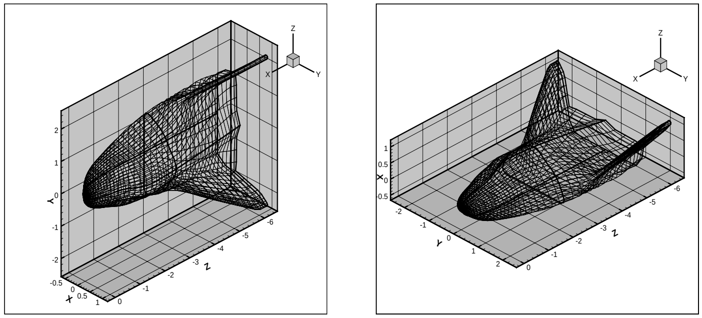


Figure 7-4. A 3-D field plot (left). The same plot after interchanging X and Y (right).

To change the axis assignments:

1. From the Plot menu, choose Assign XYZ. The Select Variables dialog appears.
2. For the Y-axis, choose variable Y, and for the Z-axis, choose variable X.
3. Redraw the plot to obtain the view displayed at the right of Figure 7-4.

For 3-D volume data sets, which include both IJK-ordered and FE-volume data sets, 3D Cartesian is the default plot type. That is, when you read in such data sets, Tecplot automatically displays them in 3-D.

IJK-ordered data sets have the general form shown below:

```
TITLE = "Example: Simple 3-D Volume Data"
 VARIABLES = "X", "Y", "Z", "Density"
 ZONE I=3, J=4, K=3, DATAPACKING=POINT
 1.0 2.0 1.1 2.21
 2.0 2.1 1.2 5.05
 3.0 2.2 1.1 7.16
 1.0 3.0 1.2 3.66
...
```

The complete ASCII data file is included with Tecplot as **`simp3dpt.dat`** (POINT format), and in block format as **`simp3dbk.dat`**. When you read either of these files into Tecplot, you immediately get the plot shown in Figure 7-5.

Finite-element volume data sets, like FE-surface data sets, consist of two separate lists—the value list and the connectivity list. A portion of a finite-element volume data file is shown below:

```
TITLE = "Example: FE-Volume Brick Data"
 VARIABLES = "X", "Y", "Z", "Temperature"
 ZONE N=14, E=5, DATAPACKING=POINT, ZONETYPE=FEBRICK
 0.0 0.0 0.0 9.5
```

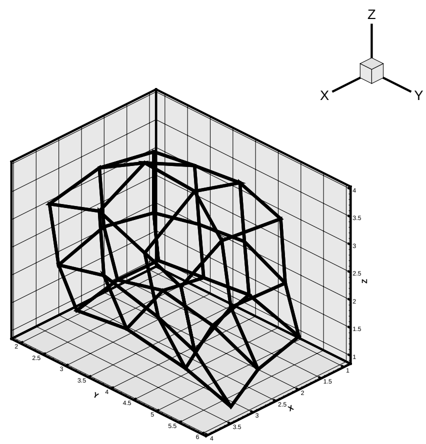


Figure 7-5. Plot of a 3-D volume.

```
 1.0 1.0 0.0 14.5
 1.0 0.0 0.0 15.0
 1.0 1.0 1.0 16.0
...
 1 1 1 1 2 4 5 3
 2 4 5 3 7 10 11 8
 4 4 5 5 10 13 14 11
 4 4 4 4 9 12 13 10
 2 2 4 4 7 6 9 10
```

The full data file, consisting of a single FE-Brick zone, is included with Tecplot as **`febrfep.dat`** (POINT format), and in BLOCK format as **`febrfeb.dat`**. When you read either of these files into Tecplot, you obtain the plot shown in Figure 7-6.

7.3. Field Plot Modification

Once you have read in your data, you can modify your field plot attributes using one or more pages of the Zone Style dialog (Mesh, Contour, and so forth) or the Quick Edit dialog. No matter which zone layer you are currently modifying, you can control the following attributes from its corresponding page in the Zone Style dialog:

- Which zones are active.
- Whether the zone layer is visible for each active zone.
- The color. (For the Scatter zone layer, you may specify two colors, an outline color and a fill color.)

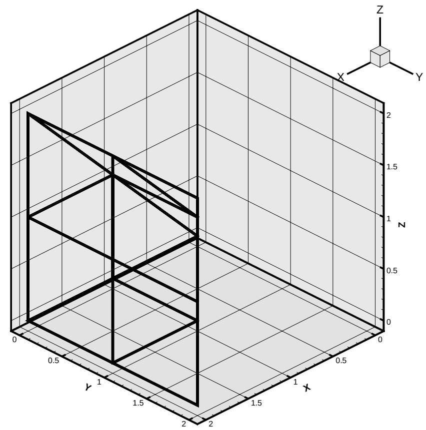


Figure 7-6. A 3-D field plot of a finite-element volume data set.

A few other attributes are common to some, but not all, zone layers:

- The plot type. (For the Scatter zone layer, this is the symbol shape.)
- The line pattern.
- The pattern length.
- The line thickness.

7.3.1. Zone Style Dialog

The following pages are available in the Zone Style dialog:

- Mesh.
- Contour.
- Vector.
- Scatter.
- Shade.
- Boundary.
- Points.
- Surfaces.
- Volume.
- Effects Attributes.

The last two are only available for 3D Cartesian plots. These pages allow you to specify plot attributes for each zone layer, zone by zone. On each page, the zone information is in the form of a scrolled list. To modify an attribute, the general procedure is the following:

1. Call up the Zone Style dialog via the Plot menu or the Zone Style button on the sidebar. The Mesh page of the dialog is shown in Figure 7-7.
2. Select one or more zones, listed in the left hand column of the scrolled list.
3. Click on a column header, which is usually a drop-down showing all the options for that attribute.
4. Choose the desired option to change the attribute for all selected zones.
5. You can quickly move between the different Zone Style dialog pages by clicking the page tabs at the top of the dialog.

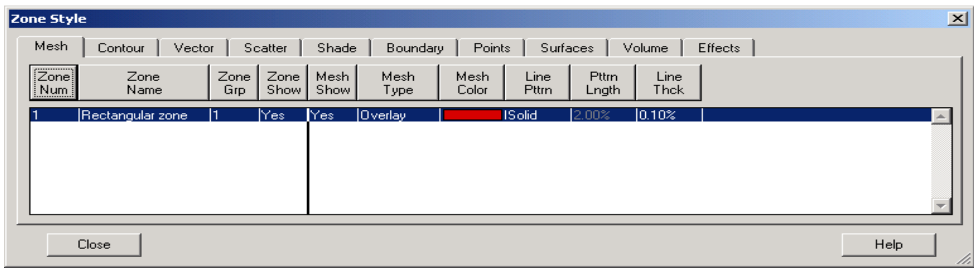


Figure 7-7. The Mesh page of the Zone Style dialog.

7.3.2. Zone Display

By default, all zones are active, meaning capable of being displayed. If a zone is not active, it will not be plotted. At least one zone must be active at all times; if you attempt to deactivate all zones, Tecplot activates the first zone of the group which you tried to deactivate. For example, suppose zones 2 and 3 are active, and you try to turn off both of them. Tecplot automatically turns zone 2 back on. You can activate and deactivate zones from any page of the Zone Style dialog.

To activate or deactivate a zone or zones:

1. From any page of the Zone Style dialog, select the zone or zones you want to activate or deactivate.
2. Click Zone Show. The choices Activate and Deactivate appear.
3. Click Activate to activate the zones, Deactivate to deactivate the zones, or Show Selected Only to activate the selected zones and deactivate all others.

7.3.3. Zone Layer Display

Whether a given zone layer is displayed for a given zone depends on three things:

- Whether the zone layer is active (controlled via the zone layer buttons on the sidebar).
- Whether the zone is active (controlled in the Zone Style dialog).

- Whether the zone is enabled to show the layer (controlled in the relevant zone layer's page of the Zone Style dialog).

When you read in a data set, all three conditions are true for all zones and for the Mesh and Boundary zone layers, and the Mesh and Boundary zone layers are shown for all zones. However, you can deactivate certain zones, and for those zones nothing is plotted. You can also disable individual zone layers for any zone. This is useful if you are creating a complex plot with different plot types for different zones. For example, you might have one zone plotted with contour flooding and another with multi-color mesh lines. In this case, you would selectively turn off the Mesh zone layer in the contour flooded zone and turn off the Contour zone layer in the mesh zone. Both the Mesh and the Contour zone layers would be active globally.

To enable or disable a field layer for a zone or zones:

1. Choose the appropriate page of the Zone Style dialog.
2. Select the zone or zones for which you want to enable or disable the zone layer.
3. Click on the column header for showing the zone layer (Mesh Show, Contour Show, and so forth). The choices Yes and No appear.
4. Click Yes to enable the zone layer for the selected zones, or No to disable the zone layer for the selected zones.

If you click Yes to enable plotting when the corresponding zone layer is disabled, Tecplot displays a dialog asking if you want to turn on the corresponding zone layer. Click Yes to turn on the zone layer, No to leave it turned off. In either case, the zone has that zone layer enabled, and that zone layer will be displayed at the first Redraw after turning on the corresponding zone layer.

You can also enable or disable each zone layer using the Quick Edit dialog for zones you select interactively in the workspace. Figure 7-8 shows the region of the Quick Edit dialog you use to enable or disable mesh display and also to control the mesh plot type.

Figure 7-8. Mesh display and plot type region of the Quick Edit dialog.

To enable or disable a field layer for a zone from the Quick Edit dialog:

1. In the workspace, use the Selector tool to select the zone or zones for which you want to enable or disable a plotting layer.
2. Call up the Quick Edit dialog by selecting Quick Edit... from the Edit menu.
3. Click Y in the appropriate display area to enable the zone layer for the selected zones; click N to disable the zone layer for the selected zones.

7.3.4. Color Choice

For each zone layer, you can pick a color independently for each zone in the data set. The color chosen for each zone layer is independent of the colors chosen for the other layers. Thus, the mesh color is independent of the colors that you choose for contour lines, vectors, scatter symbols, solid shading, or boundaries. You can choose from any of Tecplot's basic colors, or (for mesh, contour lines, scatter symbols, and vectors) choose a Multi-color option or RGB coloring. When you select a Multi-color option, the zone layer is colored as a function of the contour variable for the selected contour group (C1, C2, C3 or C4). If no contour variable is currently active, the Contour Details dialog appears with the default contour variable highlighted. You can either select a new contour variable, or click Close to accept the default. A multi-colored plot varies in color like a flooded contour plot—the color of each line segment (between adjacent data points) is determined from the average value of the contour variable at the two data points (together with other options such as the number of contour levels and the current color map).

If you choose RGB coloring, you will be prompted for three variables to use for the red, green, and blue color components. Choose the desired variables and click OK. RGB coloring is covered in more detail in CHAPTER 10 "Contour Plots" on page 203.

Specifying a color is essentially the same for all zone layers; to be specific, the procedure for modifying mesh plots is given below. The procedure for changing the color for other zone layers is similar.

To choose a mesh color for a zone or zones:

1. From the Zone Style dialog, select the zone or zones for which you want to specify a color.
2. Click Mesh Color. A drop-down appears containing Tecplot's basic colors and the Multi-color option.
3. Click on the desired color option.

You can also choose the mesh color from the Quick Edit dialog for zones chosen interactively in the workspace. The color region consists of a row of options labeled Fill, Line, and Text, respectively, followed by a button that displays the Select Color dialog and a button labeled X, which is not used for mesh plots. When you choose a color via the Quick Edit dialog, the color changes for all visible zone layers for the selected zones.

To choose a mesh color for a zone or zones from the Quick Edit dialog:

1. In the workspace, use the Selector tool to select the zone or zones for which you want to assign a new mesh color.
2. Call up the Quick Edit dialog by choosing Quick Edit... from the Edit menu.
3. Click Line.
4. Click Color...
5. Click on the desired color option.

7.3.5. Line Pattern

For mesh plots, contour line plots, and vector plots, you can pick a line pattern independently for each zone in the data set. The line pattern chosen for one layer is independent of the line patterns of the other plot layers. You can choose from any of Tecplot's six line patterns:

- Solid.
- Dashed.
- Dash Dot.
- Dotted.
- Long Dash.
- Dash Dot Dot.

Choosing a line pattern is essentially the same for all zone layers; to be specific, the procedure for modifying mesh plots is given below. The procedure for changing the line pattern for other zone layers is similar.

To choose a mesh line pattern for a zone or zones:

1. From the Mesh page of the Zone Style dialog, select the zone or zones for which you want to specify a line pattern.
2. Click Line Pttrn. A drop-down appears containing six line pattern types.
3. Click on the desired line pattern option. If you choose any option besides Solid, you can also check the Pattern Length, and modify it as necessary. See Section 8.3.6, "Choosing a Pattern Length."

You can also choose the line pattern from the Quick Edit dialog for zones chosen interactively in the workspace. Figure 7-9 shows the line pattern region of the Quick Edit dialog, which contains one option for each of Tecplot's six line patterns. When you choose a line pattern from the Quick Edit dialog, the line pattern changes for all visible layers for the selected zones.

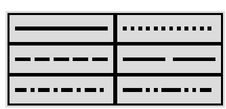
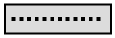
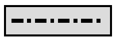


Figure 7-9. The line pattern region of the Quick Edit dialog.

To choose a line pattern for a zone or zones from the Quick Edit dialog:

1. In the workspace, use the Selector tool to select the zone or zones for which you want to assign a line pattern.
2. Call up the Quick Edit dialog from the sidebar.
3. Click on the option with the desired line pattern:

   -  Chooses a solid line.

-  Chooses a dotted line.
-  Chooses a dashed line.
-  Chooses a long dashed line.
-  Chooses an alternating dash-and-dot line.
-  Chooses an alternating dash-and-two-dots line.

7.3.6. Pattern Length

The pattern length determines the cycle length for your line pattern, that is, how long the pattern appears before repeating. In practice, this determines the length of dashed lines, and the spaces between dots and dashes. The pattern length has no effect on solid lines. You specify the pattern length as a percentage of the frame height.

Choosing a pattern length is essentially the same for all zone layers; to be specific, the procedure for modifying mesh plots is given below. The procedure for changing the pattern length for other zone layers is similar.

To choose a mesh pattern length for a zone or zones:

1. From the Mesh page of the Zone Style dialog, select the zone or zones for which you want to specify a pattern length.
2. Click Pttrn Lngth. A drop-down appears containing five pre-set lengths and an Enter option.
3. Click on the desired option. If you select Enter, an Enter Value dialog appears.
4. (Enter option only) Enter a percentage of the frame height in the Enter Value dialog.

You can also choose the pattern length from the Quick Edit dialog for zones chosen interactively in the workspace. When you choose a pattern length from the Quick Edit dialog, the pattern length changes for all visible zone layers for the selected zones.

To choose a pattern length for a zone or zones from the Quick Edit dialog:

1. In the workspace use the Selector tool to select the zone or zones for which you want to specify the pattern length.
2. Call up the Quick Edit by choosing Quick Edit... from the Edit menu.
3. Click Pttrn Length. A drop-down appears containing five pre-set lengths and an Enter option.
4. Click on the desired option. If you select Enter, an Enter Value dialog appears.
5. (Enter option only) Enter a percentage of the frame height in the Enter Value dialog.

7.3.7. Line Thickness

For all field layers except Shade zone layer, you can specify a line thickness independently for each zone. The line thickness is independent for each zone layer. The thickness is specified as a percentage of the frame height (see Figure 7-10). Differing line thicknesses can be drawn on the screen, but are not supported on all printers. In particular, HP-GL print files do not support varying line thicknesses. The minimum screen line thickness is one pixel.

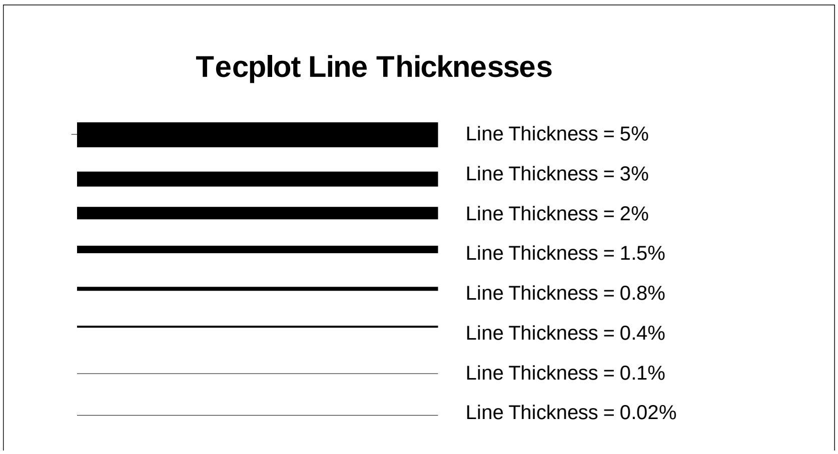


Figure 7-10. Varying line thicknesses in Tecplot.

Choosing a line thickness is essentially the same for all zone layers; to be specific, the procedure for modifying mesh plots is given below. The procedure for changing the line thickness for other zone layers is similar.

To choose a mesh line thickness for a zone or zones:

1. From the Mesh page of the Zone Style dialog, select the zone or zones for which you want to specify a line thickness.
2. Click Line Thck. A drop-down appears containing five pre-set widths and an Enter option.
3. Click on the desired option. If you select Enter, an Enter Value dialog appears.
4. (Enter option only) Enter a percentage of the frame height in the Enter Value dialog.

You can also choose the line thickness from the Quick Edit dialog for zones chosen interactively in the Tecplot workspace. When you choose a line thickness from the Quick Edit dialog, the line thickness changes for all visible zone layers for the selected zones.

To choose a line thickness for a zone or zones from the Quick Edit dialog:

1. In the workspace, use the Selector tool to select the zone or zones for which you want to specify the line thickness.
2. Call up the Quick Edit dialog by choosing Quick Edit... from Tecplot's Edit menu.

3. Click Line Thcknss. A drop-down appears containing five pre-set lengths and an Enter option.
4. Click on the desired option. If you select Enter, an Enter Value dialog appears.
5. (Enter option only) Enter a percentage of the frame height in the Enter Value dialog.

7.4. Data Point and Cell Labels

You can label all or some of the data points, or nodes, in your field plots with either the index value(s) of the data point or the value of some specified variable at each point. You can also label each cell, or element, of the data, with its index (which for finite-element data is its element number).

For example, Figure 7-11 shows a finite-element data set with each node labeled with its node number.

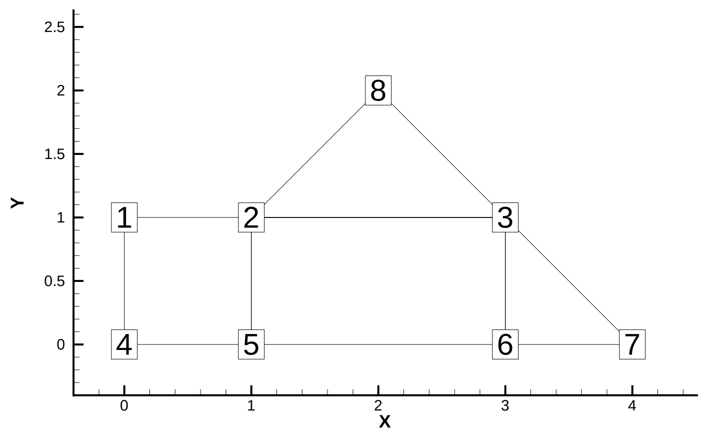


Figure 7-11. Finite-element data with data labels.

To create data labels:

1. From the Plot menu, choose Label Points and Cells... The Label Points and Cells dialog appears, as shown in Figure 7-12.
2. To label the data points, or nodes, select the Show Node Labels check box. If you select this check box, choose one of the two option buttons Show Index Value or Show Variable Value. If you select the Show Variable Value option, choose a variable from the drop-down immediately to the right of the option.
3. To label the cells, or elements, select the Show Cell Labels check box.

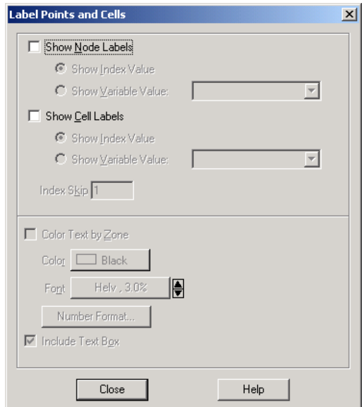


Figure 7-12. The Label Points and Cells dialog.

4. To label all nodes or cells, confirm that the Index Skip text field contains the default value 1. To label only some of the nodes or cells, enter a larger number in the text field. A value of 2 labels every other point, a value of 3 labels every third point, and so on.
5. Specify the format of the data labels using the following controls:
   - **Color Text by Zone:** Color the text with the outline color of the scatter symbols for each zone. You may change this on the Scatter page of the Zone Style dialog.
   - **Color:** Choose a color from Tecplot's Select Color dialog.
   - **Font:** Choose font options from Tecplot's Select Font dialog.
   - **Number Format:** If you are using a variable value as the data label, you can specify the format of the labels using the controls in Tecplot's Specify Number Format dialog. The available formats are the same as for tick mark labels; see Section 17.5.2, "Tick Mark Labels," for details.
   - **Include Text Box:** Select this check box to include a filled box around each data label.
6. Redraw your plot to see the data labels.

7.5. Two-Dimensional Plotting Order

In 2-D plots, by default, each zone layer is drawn for all zones before the next layer is drawn. Sometimes, you will want to plot the data zone by zone instead of layer by layer. To do this, choose 2D Draw Order from the Plot menu and select the By Zone check box.

7.6. Three-Dimensional Plot Control

You can view any type of data as a 3-D plot. By default, only IJK-ordered data and finite-element volume data are displayed in 3-D, but you can view other data in 3-D simply by selecting 3D Cartesian in the plot type menu on the sidebar. Three-dimensional plots can be rotated in space, allowing you to look at your data from any angle. This rotation is probably the most common control you will exercise over your 3-D plots, but Tecplot gives you control over a number of other 3-D plotting attributes that determine precisely how your plot is displayed. This control is necessary because 3-D plots need to provide an illusion of depth in a two-dimensional screen display. The available controls are as follows:

- **3D Rotation:** Control the 3-D orientation of the plot. See Section 7.6.1, "Three-Dimensional Rotation," for details.
- **3D View Details:** Set the specifications for a variety of parameters affecting the 3-D display of your plot, including the perspective, field of view, angular orientation of the plot, and view distance. See Section 7.6.2, "Three-Dimensional View Details," for details.
- **3D Orientation Axis:** Allows you to control the optional 3-D orientation axis, which shows the current orientation of the three axes. See Section 7.6.6, "Three-Dimensional Orientation Axis," for details.
- **Reset 3D Axes:** Allows you to reset the 3-D axis sizes and the 3-D origin of rotation. See Section 7.6.7, "Reset 3-D Axes," for details.
- **3D Axis Limits:** Allows you to control the data and axis aspect ratios for 3-D plotting. See Section 7.6.8, "Three-Dimensional Axis Limits," for details.
- **Light Source:** Control the light source position, as well as the intensity of the light, the background light, and the surface color contrast. See Section 15.1.2, "Lighting Effects," for more details.
- **Advanced 3D Control:** Specify the default lift fraction for 3-D lines, symbols, and tangent vectors, as well as the 3-D sorting algorithm for the plot. See Section 7.6.4, "Three-Dimensional Sorting," for more details.

7.6.1. Three-Dimensional Rotation

Tecplot allows you to rotate your data in a variety of different ways. Choose one of the six 3-D rotation mouse modes, then drag the pointer in the workspace to rotate your 3-D image. The six rotation mouse modes can be entered by selecting the appropriate sidebar tools, as follows:

- **Spherical**  **:** Drag the mouse horizontally to rotate about the Z-axis; drag the mouse vertically to control the tilt of the Z-axis.
- **Rollerball**  **:** Drag the mouse in the direction to move with respect to the current orientation on the screen. In this mode, your mouse acts much like a rollerball.
- **Twist**  **:** Drag the mouse clockwise around the image to rotate the image clockwise. Drag the mouse counterclockwise around the image to rotate the image counterclockwise.
- **X-axis**  **:** Drag the mouse to rotate the image about the X-axis.
- **Y-axis** **:** Drag the mouse to rotate the image about the Y-axis.
- **Z-axis** **:** Drag the mouse to rotate the image about the Z-axis.

Once you have chosen a rotation mouse mode, you can quickly switch to any of the others using the following keyboard shortcuts:

- **s:** Spherical.
- **r:** Rollerball.
- **t:** Twist.
- **x:** X-axis.
- **y:** Y-axis.
- **z:** Z-axis.

7.6.2. Three-Dimensional View Details

The angular orientation of the plot is defined by three spherical rotation angles:

- ψ **(Psi):** Tilt of eye origin ray away from Z-axis.
- θ **(Theta):** Rotation of the eye origin ray about the Z-axis.
- α **(Alpha):** Twist about the eye origin ray.

The eye origin ray is a line from the origin of the 3-D object to your eye. The eye origin ray is perpendicular to the plane of the computer screen. These angles define a unique view. These angles are shown in Figure 7-13.

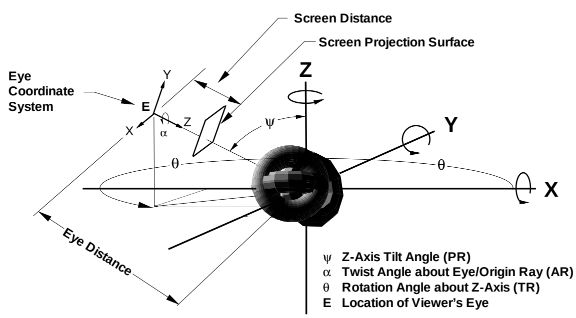


Figure 7-13. The 3-D angles and 3-D projection.

When rotating an object, there is a center of rotation about which the rotation takes place. This is called the 3-D origin, and it should not be confused with the actual XYZ-origin of the data. The default 3-D origin is approximately the centroid of all the data in the active zones.

Besides being able to set the 3-D rotate origin via the 3D Rotate dialog, you may also set the origin by positioning the rotation cursor over your data, then pressing the letter O. The rotation origin will then be shifted to the point on the closest surface underneath the cursor.

From the 3D Rotate dialog, you can also choose from four pre-set views or precisely specify the desired orientation of your 3-D plot by entering exact values for the three spherical angles Psi, Theta, and Alpha. You can also define the origin of the 3-D rotation.

7.6.2.1. Rotate About the Viewer Position. In addition to the rotation capabilities described above, you may use the Alt key and mouse to rotate about the viewer (instead of rotating the object). Although you may use this feature while in orthographic projection, it is best suited for when perspective projection is being used. The Alt key and your middle mouse button may be used to simulate fly-through type motion. You may move closer to the object using the Alt key and middle mouse button (or Ctrl-Alt-right mouse button), then turn your head using the Alt key and left mouse button.

7.6.2.2. The Rotate Dialog. You may also rotate your plots using the 3D Rotate dialog under the View menu, shown in Figure 7-14. At the top of this dialog, there are three options specifying three rotation modes—XYZ-Axis, Spherical, and Rollerball. Depending on the rotation mode chosen, the array of buttons to the right of the options will vary. To rotate the image, click these options. Each click rotates the image by the number of degrees specified in the Rotation Step Size text field.

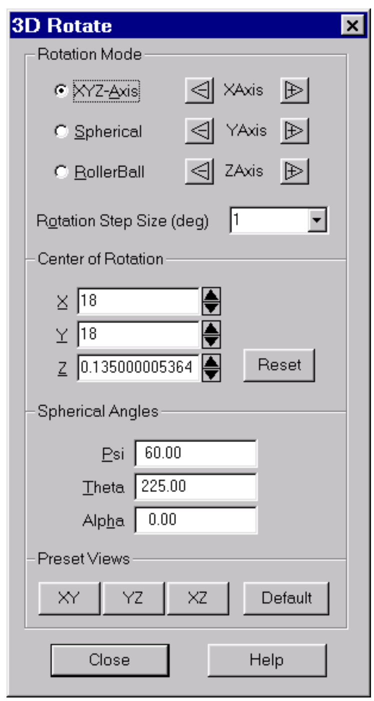


Figure 7-14. The Rotate dialog.

7.6.3. Three-Dimensional Zooming and Translating

Just as in all other plots, you may zoom and translate your plot using the mouse. This may be done using either the Zoom or Translate tools. For most sidebar tools you may also use your middle and right mouse buttons (or Ctrl-right mouse button) to zoom and translate.

When the plot projection is orthographic, zooming with the middle mouse button magnifies the plot. When the plot projection is perspective, zooming with the middle mouse button changes the viewer angle, making the plot appear larger or smaller. If you want to change the viewer's position by moving closer to or further away from an object hold the Alt key down while using the middle mouse button.

Working with very large data sets may result in slow zooming, rotating and translating. If this is the case you may use an approximate plot of the data instead of the full plot during these operations.

To change the transient drawing behavior:

1. In the sidebar, click the Performance button. This displays a dropdown that allows you to choose the level of approximation to plot during zooming, rotating and translating.
2. Click Draw Level for 3D View Changes, then select Trace from the drop-down.

If performance remains slow you may also want to turn off the caching of graphics in display lists. This option is also in the Display Performance dialog, which you may show by clicking Options... in the Performance dropdown.

If you are working with multiple frames you may increase performance by choosing to approximate all non-current frames. The option is also on the Display Performance dialog.

See Section 30.2, "Interactive Customization," for further information on customizing the display.

7.6.4. Three-Dimensional Sorting

For some 3-D plots, Tecplot uses a painter's algorithm. The data objects are divided into smaller objects. The smallest object is usually a cell, finite-element, vector, or scatter symbol. These objects are sorted based upon the distance from viewer. Tecplot draws the image starting with the objects farthest from the viewer and working forward.

In Tecplot, 3-D sorting occurs whenever you use translucency in a 3-D plot, or whenever you print or print preview a 3-D plot. A quick sorting algorithm is used by default. This does not detect problems such as intersecting objects. The 3-D sorting for each frame is controlled by the Perform Extra 3D Sorting check box on the Advanced 3D Control dialog, shown in Figure 7-15. If the Perform Extra 3D Sorting check box is selected, a slower, more accurate approach is used to detect problems for you. Call up the Advanced 3D Control dialog by selecting the Plot menu's Advanced 3D Control option.

Note: All of the settings in the Advanced 3D Control dialog are specific to the current frame.

There are instances when Tecplot cannot sort correctly. For example, consider elements A, B, and C, where element A overlaps part of element B which overlaps part of element C which overlaps part of element A. Since Tecplot draws only whole elements, one of these elements will be drawn last and (incorrectly) cover a portion of another element. If this occurs while printing or exporting, choosing an image format will often resolve the problem.

If you specify lift fractions for 3-D lines, tangent vectors, or scatter symbols, plotted objects of the appropriate type are lifted slightly towards you by this fraction so that they lie on top of surface elements. The lift fraction is the fraction of the distance from the 3-D origin of the object to your eye. You may specify lift fractions with the Advanced 3D Controls dialog.

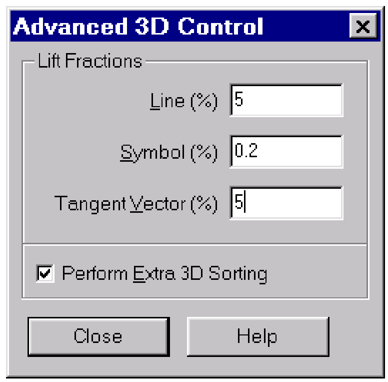


Figure 7-15. The Advanced 3D Control dialog.

7.6.5. Three-Dimensional Projection

The image you see on the screen is a 2-D representation of a 3-D image. Three-dimensional data is projected onto a 2-D plane, your screen. Tecplot offers two projection methods: orthographic and perspective. Orthographic projection is used as the default.

In the 3D View Details dialog, shown in Figure 7-16, you can choose either of the following two types of 3-D projection:

- **Orthographic:** The shape of the objects is independent of distance. This is sometimes an “unrealistic” view, but it is often used for displaying physical objects when preserving the true lengths is important (such as drafting).
- **Perspective:** The shape of the objects is dependent on the field of view angle. The larger the angle the larger the perspective effects. From the 3D View Details dialog you can control the field of view angle and the viewer position. As a convenience you can also change the viewer position by moving closer to or further from the object by changing the view distance.

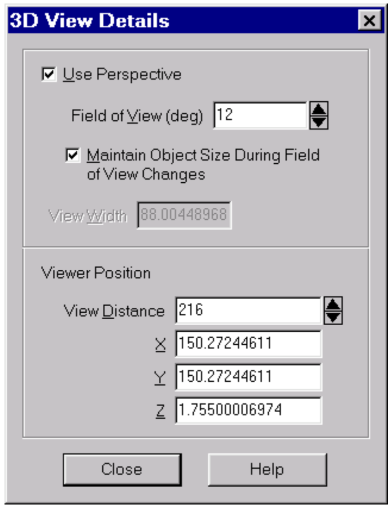


Figure 7-16. The 3D View Details dialog.

7.6.6. Three-Dimensional Orientation Axis

Depending on the view, it may be difficult to determine the current orientation of your 3-D axes. The 3-D orientation axis is a (usually small) representation of your axes that shows you the orientation immediately. By default, all 3-D plots show the 3-D orientation axis in the upper right of the frame. Using the 3D Orientation Axis dialog under the Plot menu, you can control whether the 3-D orientation axis is shown in your plot, and if so, its color, size, line thickness, and the position of the axis origin. You can also position the 3-D orientation axis simply by clicking on it and dragging it to the desired location in the frame.

7.6.7. Reset 3-D Axes

By default, the 3-D axes are calculated so that they just surround the data. If you alter your data to expand or contract the overall data size, the axes do not automatically adjust to the new size. For example, if you multiply your X-variable by four, the data will extend four times the length of the X-axis.

You can use the Reset 3D Axes option under the Plot menu to reset the axes so that they once again just surround the data.

The Reset 3D Axes option also resets the 3-D origin, that is, the origin of 3-D rotation. If you have modified your 3-D origin using the 3D Rotate dialog (see Section 7.6.1, “Three-Dimensional Rotation,” for details), the Reset 3D Axes option will reset it to approximately the centroid of the data.

7.6.8. Three-Dimensional Axis Limits

In a 3-D plot, whenever you read a data file or manipulate the values of variables assigned to axes or change variables assigned to the axes, Tecplot examines the data and determines how to plot it. The data may require scaling in one or more axis directions, a change of the axis dependency, an adjustment of the space between the data and the axis box, and/or an adjustment of the shape of the axis box.

For example, suppose you read into Tecplot X-Y-Z data that defined a pencil: a long and thin shape. You may want Tecplot to plot this true to scale (that is, a scale factor of one for each axis) in the dependent axis mode, with a long, thin axis box adapted closely to the pencil. Or you may want the data plotted true to scale, but with an axis box nearly cubic in shape. Or perhaps you want the data scaled so the pencil actually appears short and stubby and fills a nearly cube-shaped axis box. Tecplot can plot all of these variations.

Because there are many valid forms in which the data could be plotted, Tecplot requires some user input to determine how to automatically configure the plot the way you want. There are several parameters used by Tecplot to determine when and how the axes are rescaled and resized which are described below. These parameters make up the 3-D axis limit options described below.

You control the allowable shape of your data and axes using the 3D Axis Limits dialog, shown in Figure 7-17. From this dialog accessed from the Plot menu, you can set an aspect ratio and reset limits for both your data and your axes.

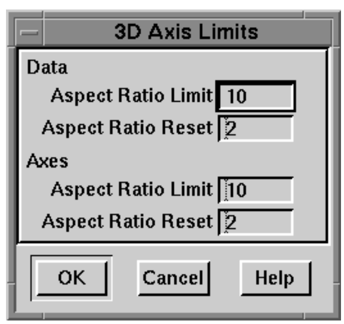


Figure 7-17. The 3D Axis Limits dialog.

The data aspect ratio is the ratio of the range of the variable assigned to one axis (multiplied by the axis size factor), divided by the range of the variable assigned to another axis (multiplied by that axis size factor).

For example, if the variable assigned to the X-axis ranged from -2 to 2 (range of four), the X-axis scale factor was one, the variable assigned to the Y-axis ranged from 100 to 500 (range of 400), and the Y-axis scale factor was 0.2, the data aspect ratio would be 20, which is [400*0.2]/[4*1].

The Data Aspect Ratio Limit is the ratio used when Tecplot is automatically resetting data. When the data aspect ratio of any two axes exceeds the Data Aspect Ratio Limit, Tecplot automatically rescales the longer axis (that which has the larger value of range multiplied by scale factor) so that the new data aspect ratio is equal to the Data Aspect Ratio Reset value.

If your plots are usually unscaled, such as plots of real physical objects, you should set the data aspect ratio maximum to a large number like 30. This allows you to plot data that is thirty times longer than it is wide without Tecplot automatically resizing it. If your plots are usually scaled, then you should set the data aspect ratio to a small number like two.

The Data Aspect Ratio Reset value should be equal to or smaller than the data aspect ratio limit. For scaled plots, a reset value of one is reasonable, making the two scale axes equal in length when automatic rescaling takes place.

The Axes Aspect Ratio Limit works like the Data Aspect Ratio Limit, except that it deals with the shape and size of the axes box. For example, if you are viewing a long, slender, physical object, you may want a high data-aspect-ratio limit to keep from rescaling the physical object, but you might also want a low axis-aspect-ratio limit to keep the axes box in a reasonable shape.

The Axes Aspect Ratio Reset value works like Data Aspect Ratio Reset, except that it deals with the shape and size of the axes box. This is the ratio used when Tecplot is automatically resetting the axes box. When the data aspect ratio of any two axes exceeds the Data Aspect Ratio Limit, Tecplot also automatically rescales the longer axis of the axes box (which has the larger value of range multiplied by scale factor) so that the new axes box aspect ratio is equal to the Axes Aspect Ratio Reset value.

Sometimes maintaining the same size factor for each axis is not desirable, especially if the 3-D data do not represent a real physical object. For example, a carpet plot is a 3-D surface where the height (Z) is a single-valued function of width (X) and depth (Y). The data points are usually arranged in an IJ-ordered rectangular array (although this is not required by Tecplot). The variable assigned to the Z-axis could have a range of values that differs greatly from the ranges on the X- and Y-axes. If you plotted this data with equal size factors for each of the three axes, the plot could be long and thin (or very flat). Usually the desired axes for carpet plots are nearly equal in length, but with different size factors. You can have Tecplot automatically rescale the axes for you by changing the Data Aspect Ratio Limit.

CHAPTER 8 *XY and Polar Line Plots*

A line plot is a graph of one or more series of data points. The data points consist of a dependent value and an independent value. Tecplot provides two different types of line plots: XY and Polar.

In the XY Line plot type, the independent and dependent values represent X and Y (in either order) and are plotted within Cartesian axes. You can create plots with lines, symbols, bars, and/or error bars. Using just bars, you can create bar charts.

In the Polar Line plot type, the independent and dependent values represent Theta and R (in either order) and are plotted within polar axes. You can create plots with lines and/or symbols.

Each series of data points in a line plot is referred to as a mapping. Tecplot dialogs often condense this as maps. Each mapping associates two variables. In XY Line, one variable is associated with an X-axis and one variable with a Y-axis. In Polar Line, one variable is associated with the Theta-axis and one variable with the R-axis.

For example, you may create a line plot of speed versus angle. In XY Line, the angle might be on the horizontal axis (the X-axis), while the speed might be on the vertical axis (the Y-axis). In Polar Line, one choice is to have the angle on the angular axis (the Theta-axis) and the speed on the radial axis (the R-axis).

To create this plot, you need only one mapping to associate angle with one axis and with another axis. In this case, Tecplot creates the mapping for you. Line plots of this data in XY and Polar Line are shown in Figure 8-1.

To create a line plot such as in Figure 8-1, perform the following steps:

1. From the Data menu, select Create Zone.
2. From the Create Zone menu, select Enter Values. The Enter Values to Create a Zone dialog appears.
3. On the Enter Values text field enter the value pairs, one pair per line; separate the values with one or more spaces. The first value is treated as X in XY Line plot types, or as Theta in Polar Line plot types. The second value is treated as either Y or R as appropriate.

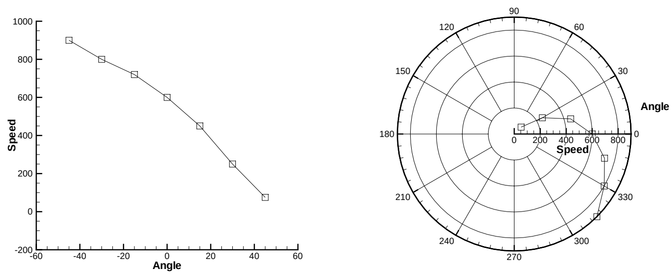


Figure 8-1. A plot of speed versus angle in Tecplot's XY Line (left) and Polar Line (right) plot types.

4. If you would like to specify a data type for the data (integer, float, double, byte, bit), select the desired data type from the Destination Data Type drop-down. For a basic plot, do not change this.
5. After entering all of the values, click Create, then Close.
6. An XY Line plot of your data appears. If you desire a Polar Line plot, select Polar Line from the Plot Type drop-down on the sidebar.
7. For Polar Lines, the Theta values are treated as angles in degrees. If your angle values are not in degrees, change settings via the Axis Details dialog. See Section 17.3.3.2, "Theta Mode," for a discussion of how to change the units of the Theta-axis.

You can also create this plot using a data file. For example, consider the data file **`line-data.dat`** (found in the **`examples/dat`** directory in your Tecplot home directory). This file lists the values of temperature measured at 20 locations unequally spaced along a wire. Each line of the file contains a single data point of location and temperature. Tecplot assigns an index (I) to identify each data point. Thus each data point can be identified uniquely by its I-index. Data point number 1 is addressed as I=1, data point number 2 is addressed as I=2, and so on up to I=20. Data that can be addressed in this way is called I-ordered data.

To plot this data, read the data file into Tecplot. You are prompted for an initial plot type. Choose either XY Line (default) or switch to Polar Line using the Initial Plot Type drop-down.

For XY Line, Tecplot (by default) plots the first variable (V1) on the X-axis versus the second variable (V2) on the Y-axis. For Polar Line, the defaults are V1 on the Theta-axis and V2 on the R-axis. Use the Mapping Style dialog to reassign variables to the axes. If you assign a different variable to the X-axis in XY Line, that variable is used for the Theta-axis if you switch plot types to Polar Line, and vice versa. Y- and R-axis variables behave similarly.

Tecplot initially sets the view of the plot and the ranges on the axes such that you can see all of your data points, text, and geometries. For Polar Line, the initial view also shows the entire circle of the axes. If you change the variable or variables assigned to a mapping, the axis range or view may need to be reset.

For XY Line, the default axis mode is independent; the scales on the X- and Y-axes are unrelated. This, and many other axis settings, can be changed using the Axis Details dialog.

When you create a line plot, Tecplot assigns colors, symbol types, and line patterns to each mapping. These and other line plot attributes can be changed using the pages of the Mapping Style dialog. To bring up the Mapping Style dialog, go to the Plot menu and select Mapping Style, or click the Mapping Style button on the sidebar.

The initial behavior of line plots can be changed in the Tecplot configuration file to create different default settings. See Section 30.1, "Configuration Files," for details on how to change settings in the Tecplot configuration file.

8.1. Line Plot Data

Line plots are usually created from one-dimensional data in the I-ordered structure. Tecplot also allows you to create line plots from two- or three-dimensional data in the IJ- or IJK-ordered structure, or finite-element data. Finite-element data sets are treated as I-ordered; the connectivity list is ignored. IJ-ordered data sets are a family of J-sets of I-ordered data; I-ordered data can be thought of as IJ-ordered data with J=1. IJK-ordered data sets are K-planes of J-families of lines; I-ordered data can be thought of as IJK-ordered data with J=K=1.

Use the Indices page of the Mapping Style dialog to select different ranges and skip intervals for the I-, J-, and K-indices. See Section 8.9, "I-, J- and K-Indices," later in this chapter for more information.

The data used for line plots must have at least two variables defined at each data point. The same number of variables must be defined at each data point. You may have up to 32,700 variables defined at each data point. For example, you could be recording the pressure at a set of 500 pressure probes every minute for two hours (120 time samples). The data could be organized into an array of numbers with 120 rows and 501 columns. The first column is the time, and columns 2 through 501 are the pressure measurements. Each row represents a data point. Each data point has 501 variables. In Tecplot, you can plot all or some of the pressure probes versus time on one plot. Since you can select any variable for any axis, you could create an XY Line plot with the pressure at probe 5 on the X-axis plotted against the pressure at probe 59 on the Y-axis. Or you could create a Polar Line plot with the Theta-axis as time and the R-axis as the pressure from various probes.

8.2. Map Creation

Line plots in Tecplot are composed of the graphs of one or more pairs of variables (XY pairs in XY Line plots or Theta-R pairs in Polar Line plots). These pairs and their dependency relations are defined in Tecplot as mappings. Mappings are defined for each frame; the same data set can have a different set of mappings in each frame it is attached to.

Mappings can be displayed as a combination of one or more of two basic plot styles:

- **Lines:** Can be drawn as linear segments that connect the data points in order, or as a curve that represents a fit of some mathematical function to the data.
- **Symbols:** Each data point is represented by a symbol (for example, a circle or a square).

For XY Line plots, there are two additional plots styles:

- **Bars:** Each data point is represented by a vertical or horizontal bar, according to whether the dependent variable of the XY-pair is the X- or the Y-variable.
- **Error Bars:** From each data point, a line is drawn a specific distance either vertically, horizontally, or both. The distance is determined by a third variable.

For XY Line, each mapping can be assigned to one of five X-axes and to one of five Y-axes. For Polar Line, there is only one Theta-axis and only one R-axis, and these axes are used by all mappings.

Mappings are defined using the Plot menu's Mapping Style dialog (active when the sidebar plot type drop-down is set to XY Line or Polar Line).

To define a new mapping:

1. From the Plot menu, select Mapping Style. The Mapping Style dialog appears. You may also open this dialog by clicking Mapping Style on the sidebar, or double clicking any mapping in the plot. Figure 8-2 shows the Mapping Style dialog for each line plot type when the demo file **`rainfall.plt`** is loaded.

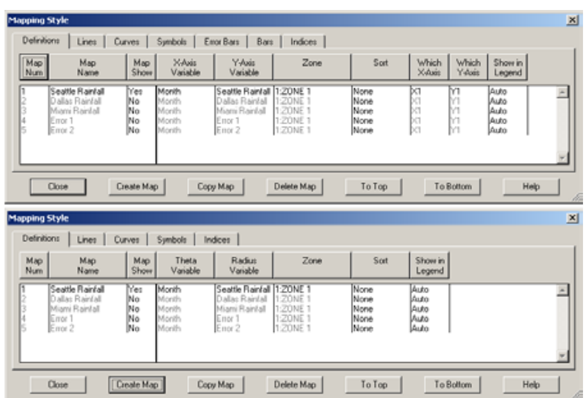


Figure 8-2. The Definitions page of the Mapping Style dialog for XY Line plots (top) and for Polar Line plots (bottom).

2. Click Create Map. The Create Mappings dialog appears. The Create Mappings dialog for each line plot type is shown in Figure 8-3.
3. Choose what sort of mapping (or mappings) to add. In XY Line plots, you have the following options:

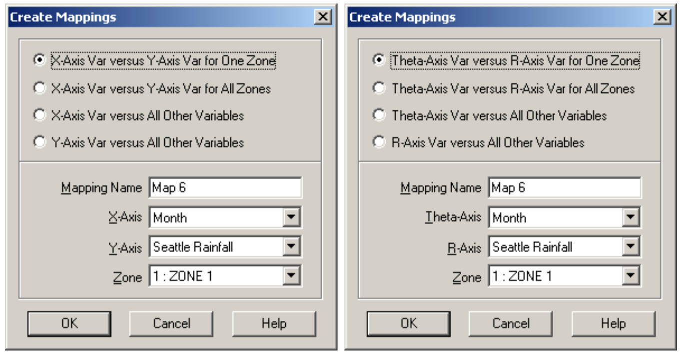


Figure 8-3. The Create Mappings dialog for XY Line plots (left) and for Polar Line plots (right).

X-Axis Var versus Y-Axis Var for One Zone (Default): Click to add a single mapping with one X- and one Y-variable for one zone. If you select this option, continue with Step 4.

X-Axis Var versus Y-Axis Var for All Zones: Click to add one mapping for each zone. You choose one X- and one Y-variable; Tecplot creates a mapping with those variables for each zone. If you select this option, continue with Step 5.

X-Axis Var versus All Other Variables: Click to create a new set of mappings using one variable as the X-variable and each of the other variables as Y-variables. When you choose this option, you must also specify the zone for which this set of mappings is defined. If you select this option, continue with Step 5 and then skip Step 6.

Y-Axis Var versus All Other Variables: Click to create a new set of mappings using one variable as the Y-variable and each of the other variables as X-variables. When you choose this option, you must also specify the zone for which this set of mappings is defined. If you select this option, continue with Step 6.

In Polar Line plots, you have the following options:

Theta-Axis Var versus R-Axis Var for One Zone (Default): Click to add a single mapping with one Theta- and one R-variable for one zone. If you select this option, continue with Step 4.

Theta-Axis Var versus R-Axis Var for All Zones: Click to add one mapping for each zone. You choose one Theta- and one R-variable; Tecplot creates a mapping with those variables for each zone. If you select this option, continue with Step 5.

Theta-Axis Var versus All Other Variables: Click to create a new set of mappings using one variable as the Theta-variable and each of the other variables as R-variables. When you choose this option, you must also specify the zone for which this set of mappings is defined. If you select this option, continue with Step 5 and then skip Step 6.

R-Axis Var versus All Other Variables: Click to create a new set of mappings using one variable as the R-variable and each of the other variables as Theta-variables. When you choose this option, you must also specify the zone for which this set of mappings is defined. If you select this option, continue with Step 6.

(Optional): Enter a name for the mapping in the Mapping Name text field. The default name is "Map *n*," where *n* is the number of the mapping to be created.

4. For XY Line plots, choose an X-axis variable using the X-Axis Var drop-down. For Polar Line plots, choose a Theta-axis variable using the Theta-Axis Var drop-down. In either case, the default is the first variable (V1).
5. For XY Line plots, choose a Y-axis variable using the Y-Axis Var drop-down. For Polar Line plots, choose an R-axis variable using the R-Axis Var drop-down. In either case, the default is the second variable (V2).
6. Choose a zone using the Zone drop-down. The default is the first zone.

When you first read an ordered data set, Tecplot defines some mappings for you. If your data set has more than two variables, Tecplot creates mappings that associate the first variable with each of the other variables for the first zone only. If your data set has only two variables, Tecplot creates mappings which associate the first variable with second variable for each zone. In XY Line plots, each of these mappings assigns the first variable to the first X-axis (X1) and the other variable to the first Y-axis (Y1). In Polar Line plots, each of these mapping assigns the first variable to the Theta-axis and the other to the R-axis. In either case, Tecplot activates the first mapping; the other mappings are left inactivated.

8.3. Mapping Definitions

Existing mappings are edited with the Plot menu's Mapping Style dialog. Choose Mapping Style from the Plot menu, the sidebar, or double-click on a line plot.

From the Definitions page of the Mapping Style dialog, you can perform the following tasks: modify names; activate and deactivate mappings; assign axis variables; assign zones; sort data points in a mapping; control the mappings appearance in the line plot legend; and for XY Line plots, assign particular X- and Y-axes.

In general, select mappings you want to change, and then select the appropriate button above the list of mappings. Some buttons call up drop-downs; others call up dialogs. You may change mapping whether they are shown on the plot or not (activated or deactivated).

8.3.1. Map Name Modification

Tecplot assigns each mapping a name. The nature of the name varies with the type of data used to create the mapping. If your data has only one dependent variable, the default is to use the zone name for the mapping. If your data has multiple dependent variables, then the default is to use the dependent variable name for the mapping. You can modify any mapping's name using the Enter Mapping Name dialog. This dialog is accessible from the Mapping Style dialog by selecting Edit Name from the Map Name drop-down.

To modify a mapping name:

1. On the Definitions page of the Mapping Style dialog, select the appropriate mappings.
2. Click Map Name, then select Edit Name. The Enter Mapping Name dialog appears, as shown in Figure 8-4.

Figure 8-4. The Enter Mapping Name dialog.

3. Enter a new name for the selected mappings, or construct a new name from text you enter and/or one or more of the pre-defined options:

   Zone Name: Adds the string "`&ZN&`" to the Map Name field, which is then replaced with the actual name of the zone assigned to that mapping.

   Zone Number: Adds the string "**`&Z#&`**" to the Map Name field, which is then replaced with the actual number of the zone assigned to the mapping.

   Map Number: Adds the string **`"&M#&"`** to the Map Name field, which is then replaced with the actual number of the mapping.

   Independent Var: Adds the string "**`&IV&`**" to the Map Name field, which is then replaced with the actual name of the independent variable assigned to that mapping.

   Independent Var Number: Adds the string "**`&I#&`**" to the Map Name field, which is then replaced with the actual number of the independent variable assigned to the mapping.

Dependent Var: Adds the string "**&DV&**" to the Map Name field, which is then replaced with the actual name of the dependent variable assigned to that mapping.

Dependent Var Number: Adds the string "**&D#&**" to the Map Name field, which is then replaced with the actual number of the dependent variable assigned to the mapping.

X-Axis Num: Adds the string "**&X#&**" to the Map Name field, which is then replaced with the actual number of the X-axis assigned to that mapping for XY Line plots. For Polar Line plots, this option is not available.

Y-Axis Num: Adds the string "**&Y#&**" to the Map Name field, which is then replaced with the actual number of the Y-axis assigned to that mapping for XY Line plots. For Polar Line plots, this option is not available.

In addition to the above items, any dynamic text item can be added to the Map Name field. Dynamic text for the mapping name is expanded when drawn in the line legend.

To add an insert, click on its button or type the associated string into the Map Name field.

4. When the Map Name field is as desired, click **OK**.

8.3.2. Mapping Activation

Each mapping may be shown on the plot (called activated) or not shown on the plot (called deactivated).

To activate or deactivate mappings:

1. On the Definitions page of the Mapping Style dialog, select the appropriate mappings.
2. Click Map Show, then select one of these options:

   Activate: Turns selected mappings on.

   Deactivate: Turns selected mappings off.

   Show Selected Only: Turns on selected mappings, and turns off all other mappings.

   Invert: Simultaneously turns on the selected mappings that are off, and turns off those selected mappings that are on.

Active mappings have the word Yes in the column under Map Show; inactive mappings have the word No.

8.3.3. Axis Variable Selection

The choice of variables is the heart of the mapping. Each mapping is defined by two variables: X and Y in XY Line plots; Theta and R in Polar Line plots. You may change the variables assigned to a mapping using the Mapping Style dialog. For XY Line plots, click X-Axis Variable or Y-Axis Variable. For Polar Line plots, click Theta-Axis Variable or R-Axis Variable.

To change a variable for a mapping or a group of mappings:

1. On the Definitions page of the Mapping Style dialog, select the appropriate mappings.

2. Click the appropriate Variable option. The Select Variable dialog appears. Choose a variable from the drop-down.

8.3.4. Zone Selection

Each mapping uses variable values from a specified zone. If your data set has multiple zones, you may want to change this specification.

To change the zone for a mapping or a group of mappings:

1. In the Mapping Style dialog, select the appropriate mappings.
2. Click Zone. The Select Mapping Zone dialog appears. Choose the desired zone from the drop-down.

8.3.5. Data Point Sorting

Certain line types and curve types use the points in a mapping in a specific order. Other curve-fits do not use this option. (See Section 8.6, "Curve-Fits and Splines," for details on curve and line types.)

By default mappings are sorting by the order they occur in the data file. You can change this order with the Sort option on the Mapping Style dialog.

To change the sorting behavior for mappings:

1. In the Mapping Style dialog, select the appropriate mappings.
2. Click Sort. The drop-down offers four options:

   None: Default behavior of sorting by the order in the data file.

   By Independent Variable: Points are sorted in ascending order of the values of the independent variable.

   By Dependent Variable: Points are sorted in ascending order of the values of the dependent variable.

   By Specify Variable: The Select Variable dialog appears. Select a variable. The points of the selected mappings are sorted in ascending of the values of this variable.

Regardless of the sorting options, most curve-fit types result in the same curve, and splines are always sorted by the independent variable. Of the standard curve types, only Line Segment and ParaSpline show a noticeable difference for changing this setting. See Section 8.6, "Curve-Fits and Splines," for more information on curve types.

8.3.6. Line Plot Legend Control

By default, all active mappings appear in the line legend, but the legend only lists mappings with identical entries once. (See Section 8.10, "Line Legend," for details on the Line Plot Legend.)

To change the line plot legend behavior of mappings:

1. In the Mapping Style dialog, select the appropriate mappings.
2. Click Show in Legend. The drop-down presents three options:

Always: The mapping appears in the legend even if the mapping is turned off (deactivated) or its entry in the table looks exactly like another mapping's entry.

Never: The mapping never appears in the legend.

Auto: The mapping appears in the legend only when the mapping is turned on. If two mapping would make the same entry in the legend, only one entry is shown.

8.3.7. XY Line Plot Axis Assignment

XY Line plots support five X-axes (X1-X5) and five Y-axes (Y1-Y5). Newly created mapping use the X1- and Y1-axes. You can change these assignments, using the Which X-Axis and Which Y-Axis fields on the Mapping Style dialog.

To change the axis assignments for mappings:

1. From the Mapping Style dialog, select the appropriate mappings.
2. Click Which X-Axis or Which Y-Axis, and select the desired axis from the drop-down.

For more information on working with multiple X- and Y-axes, see Section 8.5.4, "Multiple X- and Y-Axes for XY Line Plots."

8.4. Mapping Style

Style is broadly defined as the total look of the plot. The style of a line plot includes each mapping's color, curve type (such as line segment, spline), line pattern, symbol type, symbol size and many other attributes. One graph is drawn for each active mapping.

For example, an XY Line plot could have many mappings all of which use the same zone and have variable V1 as the X-variable V1 and variable V2 as the Y-variable, but the style of the mapping could be entirely different. On mapping might display symbols and a linear curve-fit while another shows a cubic spline. They could be sorted differently, as shown in Section 8.3.5, "Data Point Sorting," use different X- or Y-axes, as shown in Section 8.5.4, "Multiple X- and Y-Axes for XY Line Plots," or draw from different indices in the data, as shown in Section 8.9, "I-, J- and K-Indices.").

Each graph is composed of one or more map layers. The main map layers in line plots are Lines and Symbols. XY Line plots also contain Bars (bar charts) and Error Bars as map layers. By default, only the Lines map layer is turned on, so no symbols are plotted.

Style attributes in line plots are assigned on a per-mapping basis. Each created mapping is given an initial color to distinguish it from others. All mappings are assigned line segments and square symbols by default.

Use the Mapping Style dialog to set attributes for lines, symbols, and in XY Line plots, bar charts and error bars. You can also make many of these changes using the Quick Edit dialog (accessible from the Edit menu or the sidebar). You can set the style of any mapping independently of all other mappings, and regardless of whether that mapping is activated or deactivated.

8.4.1. Map Layer Display

Changing map layers is the quickest and most visually striking means of changing line plot style. Switching from a line plot to a symbol plot dramatically alters the style. You can also combine layers to create striking visuals—for example in XY Line plots, a curve-fit together with a bar chart can clearly outline the trend in the data. The same data is plotted as a polar plot with symbols and a curve-fit on right side of Figure 8-5.

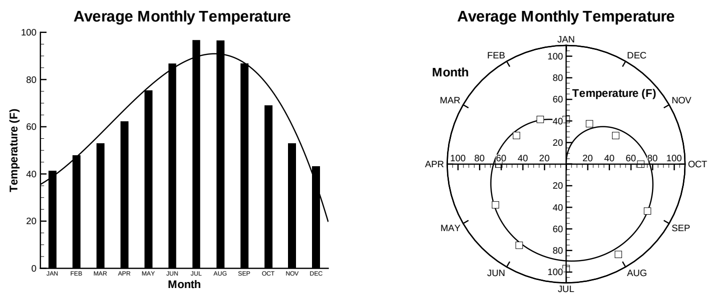


Figure 8-5. Using multiple layers: An XY Line plot showing a bar chart combined with a curve-fit (left), and a Polar Line plot showing symbols and a curve-fit.

To activate or deactivate a map layer, click on the map layer's check box on the sidebar. The map layer areas of the sidebar for both XY and Polar Lines plots with both the Lines and Symbols map layers active is shown in Figure 8-6.

Figure 8-6. The map layers area of the sidebar for XY Line plots (left) and for Polar Line plots (right).

8.4.2. Line Attributes

The Lines page of the Mapping Style dialog, with information from the demo file **`chem.plt`** is shown in Figure 8-7. The Mapping Style dialog for XY Line plots is the same for Polar Line plots except for the Error Bars and Bars pages.

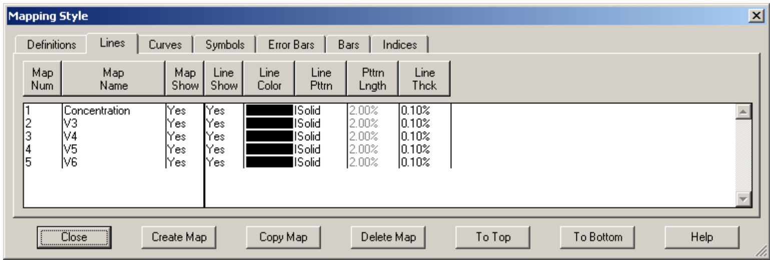


Figure 8-7. The Lines page of the Mapping Style dialog.

The first two columns, Map Num and Map Name, list the mapping number and name. The Map Show field shows which mappings are currently active. A mapping must be active for it to be displayed, although an active mapping is not always visible. The remaining columns of the Lines page of the Mapping Style dialog contain specific line attributes, as follows: Line Show; Line Color; Line Pattern; Pattern Length; Line Thickness.

The Quick Edit dialog for XY Line and Polar Line plots is shown in Figure 8-8. It allows you

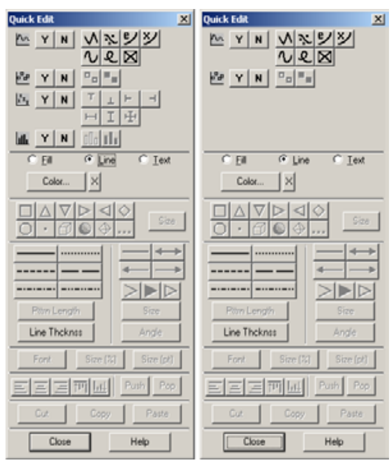


Figure 8-8. The Quick Edit dialog for XY Line plots (left) and for Polar Line plots (right).

to control all of the line attributes. The Quick Edit dialog can be accessed via the Edit menu or the sidebar.

8.4.2.1. Line Visibility. You can specify whether lines are shown for individual mappings. This option allows you to turn off selected mapping lines, keeping both the selected mappings and the Lines map layer active. You might want to do this, for example, if you want to plot two mappings, one as a line and the other as symbols. In this case, you would set Line Show to No for the symbol plot, and Symb Show to No for the line plot. See Section 8.4.3.1, "Symbol Visibility," for details on using Symb Show.

To turn on or off the line portion of a mapping or group of mappings:

1. From the Lines page of the Mapping Style dialog, select the appropriate mappings.
2. Click Line Show. Select Yes from the drop-down to show the line for the selected mappings; select No to turn off the line.

Alternatively, you may:

1. Click on the appropriate mapping in the plot. Either the lines or the symbols of the mapping work fine.
2. On the Quick Edit dialog, next to the Lines map layer icon, click Y to show the line for the selected mappings, N to turn off the line.

8.4.2.2. Line Color. You can set line color for line plots using the Line Color button on the Lines page of the Mapping Style dialog, or via the Quick Edit dialog. Note that there is a difference between the two methods. The Quick Edit dialog changes the line color for all selected objects (for example, if there are symbols as well as lines drawn, the symbol color would change with the line color). If you use Line Color, only the lines change (unless you have set style linking; see Chapter 18, "Frame Linking," for more information).

To change the line color from the Mapping Style dialog:

1. Select the mapping or mappings for which you want to assign a new color.
2. On Lines page of the Mapping Style dialog, click Line Color; the Select Color dialog appears.
3. Click the desired color; the dialog closes automatically.

To change the line color using the Quick Edit dialog:

1. Click on the appropriate mapping in the plot.
2. On the Quick Edit dialog, select the Line option immediately above the Color area. The Color area of the Quick Edit dialog is shown in Figure 8-9.

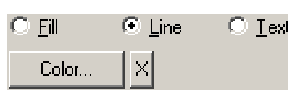


Figure 8-9. The Color area of the Quick Edit dialog.

3. Click Color on the Quick Edit dialog. The Select Color dialog appears. Click on the desired color.

8.4.2.3. Line Pattern. Set line patterns for line plots using the Line Pttrn drop-down on the Mapping Style dialog, or the Line Pattern area of the Quick Edit dialog, shown in Figure 8-10.

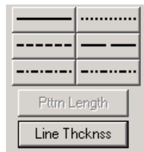


Figure 8-10. Line Pattern, Pttrn Length and Line Thickness on the Quick Edit dialog.

The choices are Solid, Dashed, DashDot, Dotted, LongDash, and DashDotDot. The line pattern setting affects the line or curve drawn for a plot, but not the symbols. Also, in XY Line plots, line pattern also has no effect on bar charts or error bars. Symbols, bar charts, and error bars are always drawn using solid lines.

8.4.2.4. Pattern Length. Set the pattern length for patterned lines using either the Pttrn Lngth drop-down on the Lines page of the Mapping Style dialog, or the Pttrn Length drop-down menu on the Quick Edit dialog. Pattern length is measured as a percentage of the frame height for one complete cycle of the pattern.

8.4.2.5. Line Thickness. Set the thickness of lines using the Line Thck drop-down on the Lines page of the Mapping Style dialog, or the Line Thcknss drop-down on the Quick Edit dialog. You can choose from pre-set widths, or enter an arbitrary width. Line thickness is measured as a percentage of the frame height. If you use the Quick Edit dialog, the new line thickness affects all displayed attributes of the selected mappings, so it also affects symbol, error bar, and bar chart line thickness. Using the Mapping Style dialog, the values can be set independently.

8.4.3. Symbol Attributes

The Symbols page of the Mapping Style dialog, with information from the demo file **`rainfall.plt`**, is shown in Figure 8-11. The dialog is the same for XY Line plots except for two additional pages: Error Bars and Bars.

The first two columns list the mapping number and name. The Map Show field lists currently active mappings. A mapping must be active for it to be displayed, although an active mapping need not be displayed. The remaining columns of the Symbol page of the Mapping Style dialog contain specific attributes: Symb Show; Symb Shape; Outline Color; Fill Mode; Fill

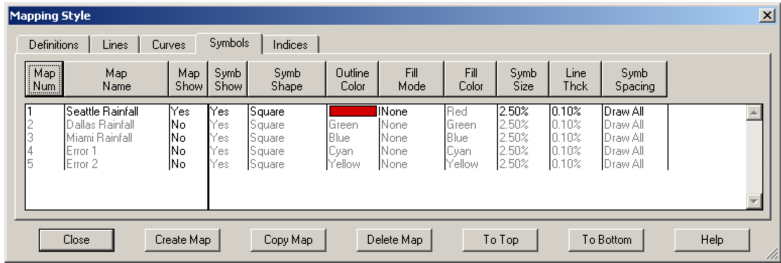


Figure 8-11. The Symbols page of the Mapping Style dialog.

Color; Symb Size; Line Thck; Symb Spacing. Each of these attributes can also be modified using the Quick Edit dialog, shown in Figure 8-8.

8.4.3.1. Symbol Visibility. You can specify whether symbols are shown for individual mappings. This option allows you to turn off selected mapping symbols, while keeping both the selected mappings and the Symbols map layer active. You might want to do this, for example, if you want to plot two mappings, one as a line and the other as symbols. In this case, you would set Symb Show to No for the line plot, and Line Show to No for the symbol plot. See Section 8.4.2.1, "Line Visibility," for details on using Line Show.

To turn on or off the symbol portion of a mapping or group of mappings:

1. From the Symbols page of the Mapping Style dialog, select the appropriate mappings.
2. Click Symb Show. Select Yes from the drop-down to show symbols for the selected mappings; select No to turn symbols off.

Alternatively, you may:

1. Click on the appropriate mapping in the plot. Either the lines or the symbols of the mapping work fine.
2. On the Quick Edit dialog, next to the Symbols map layer icon, click Y to show the symbols for the selected mappings, N to turn off the symbols.

8.4.3.2. Symbol Shape. Use the Symb Shape drop-down on the Symbols page of the Mapping Style dialog or use the Symbol area of the Quick Edit dialog, to select the symbol type for each mapping. The Symbol area of the Quick Edit dialog is shown in Figure 8-12.

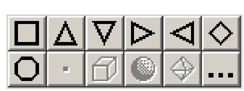


Figure 8-12. The Symbol area of the Quick Edit dialog.

There are seven pre-defined symbols: Square; Delta (equilateral triangle pointing up); Gradient (equilateral triangle pointing down); Right Triangle (equilateral triangle pointing to the right);

Left Triangle (equilateral triangle pointing to the left); Diamond; Circle. (The symbol types Point, Cube, Sphere, and Octahedron are not available for line plots, and thus grayed out in Figure 8-12.)

In addition to the predefined symbols, you can choose as a symbol any ASCII character in the following Tecplot fonts: Helvetica-Bold, Math, Greek, User-Defined.

To change the symbol shape using the Symb Shape drop-down on the Symbols page of the Mapping Style dialog:

1. Select the appropriate mappings.
2. Click Symb Shape. Select the desired symbol shape from the drop-down.
3. If you select Other, the Enter ASCII Character dialog appears, as shown in Figure 8-13.

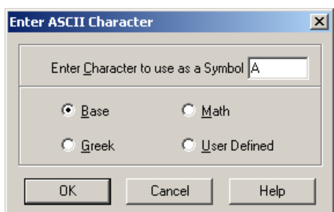


Figure 8-13. The Enter ASCII Character dialog.

Enter the ASCII character to use as a symbol, and select a font from which to display the symbol. When you are done, click OK.

To change the symbol shape using the Quick Edit dialog:

1. Click on the appropriate mapping in the plot. Either the lines or the symbols of the mapping work fine.
2. In the Symbol area of the Quick Edit dialog, click on the desired symbol.
3. If you select ..., the Enter ASCII Character dialog appears, as shown in Figure 8-13. Enter the ASCII character to use as a symbol, and select a font from which to display the symbol. When you are done, click OK.

8.4.3.3. Symbol Outline Color. Symbols can be filled or unfilled; by default they are unfilled. The symbol's outline color is the color of the unfilled symbol. You can choose an outline color using the Outline Color button on the Symbols page of the Mapping Style dialog, or using the Color area of the Quick Edit dialog. There is a difference between the two methods. The Quick Edit dialog changes the line color for all selected objects. For example, if there are lines as well as symbols drawn, the line color changes as well as the symbol color. If you use the Outline Color field to change the symbol color, only the symbols change (unless you have set style linking; see Chapter 18, "Frame Linking," for more information).

To change the symbol outline color using the Mapping Style dialog:

1. Select the appropriate mappings.
2. On the Symbols page of the Mapping Style dialog, click Outline Color.
3. The Select Color dialog appears. Click the desired color; the dialog closes automatically.

To change the symbol outline color using the Quick Edit dialog:

1. Click on the appropriate mapping in the plot.
2. On the Quick Edit dialog, select the Line option immediately above the Color area. Figure 8-9 shows the Color area of the Quick Edit dialog.
3. Click Color on the Quick Edit dialog. The Select Color dialog appears. Click on the desired color; the dialog closes automatically.

8.4.3.4. Filled Versus Outline Symbols. Symbols may be either filled or unfilled. Filled symbols are outlined using the specified outline color, and then filled with a fill color. You can specify filled symbols either by selecting an option from the Fill Mode button on the Symbols page of the Mapping Style dialog, or by specifying a fill color in the color area of the Quick Edit dialog. The methods are slightly different, however: the Fill attribute on the Symbols page affects only plotting symbols, while the fill color on Quick Edit affects all selected objects. Thus, you could end up filling any selected geometries, as well as filling mapping symbols when you use the Quick Edit approach. Also, there are more options when you use the Mapping Style dialog approach.

To change the symbol filling using the Fill Mode drop-down on the Symbols page of the Mapping Style dialog:

1. Select the appropriate mappings.
2. Click the Fill Mode button. The drop-down presents four options:

   None: The symbols are not filled. The symbol are just outlines.

   Use Line Color: The symbols are filled with the same color specified in Outline Color and appear as a a solid color.

   Use Back Color: The symbols are filled with background color of the grid area, and appear hollow, blotting out objects behind the symbol (such as grid lines or other mappings).

   Use Specific Color: The symbols are filled with the color specified in Fill Color, and can appear in two different colors.

To choose filled symbols using the Quick Edit dialog, or to turn off fill if you already have filled symbols:

1. Click on the appropriate mapping in the plot.

2. Click  for filled symbols (equivalent to Use Specific Color in the Mapping Style dialog), or  for symbols that are not filled. The Quick Edit dialog does not have options equivalent to Use Line Color and Use Back Color.

If you have filled symbols, and you select the Line option in the Color area of the Quick Edit dialog, choosing X makes the symbol outline color the same as the symbol fill color. If you select the Fill option in the Color area of the Quick Edit dialog, choosing X gives the symbol have a Fill Mode of None.

8.4.3.5. Symbol Fill Color. If the symbol is filled with a specific color, see Section 8.4.3.4, “Filled Versus Outline Symbols,” for more information, you set the fill color with either the Symbols page of the Mapping Style dialog or with the Quick Edit dialog.

To change the symbol fill color using the Mapping Style dialog:

1. Select the appropriate mappings.
2. On the Symbols page of the Mapping Style dialog, make sure the mappings have Fill Mode set to Use Specific Color. If not, see Section 8.4.3.5, “Symbol Fill Color.”
3. Click Fill Color. The Select Color dialog appears. Click the desired color; the dialog closes automatically.

To change the symbol fill color using the Quick Edit dialog:

1. Click on the appropriate mapping in the plot.
2. On the Quick Edit dialog, select the Fill option immediately above the Color area. Figure 8-9 shows the Color area of the Quick Edit dialog.
3. Click Color on the Quick Edit dialog. The Select Color dialog appears. Click the desired color; the dialog closes automatically.

If you have unfilled symbols, selecting a fill color on the Quick Edit dialog makes the symbols use the Fill Mode of Use Specific Color.

8.4.3.6. Symbol Size. Select the symbol size for your line plotting symbols using either the Symb Size drop-down on the Symbols page of the Mapping Style dialog, or the Size drop-down next to the Symbol area on the Quick Edit dialog. Symbol size is measured as percentage of the frame height.

8.4.3.7. Symbol Line Thickness. To specify the thickness of lines used to draw the plotting symbols, use either the Line Thck drop-down on the Symbols page of the Mapping Style dialog, or the Line Thcknss drop-down on the Quick Edit dialog. The Line Thck drop-down on the Symbols page affects only line plotting symbols, while the Line Thcknss drop-down on the Quick Edit dialog affects all selected objects (including lines for the same mapping if they are turned on). Line thickness is measured as a percentage of the frame height.

8.4.3.8. Symbol Spacing. If you are plotting many points, you might not want every point to show a symbol. To specify the spacing between symbols, use the Symb Spacing drop-down on the Symbols page of the Mapping Style dialog. The spacing is specified either as a percentage of the frame height or as a number of indices to skip.

You may either enter a value or use one of the following pre-set values:

- **Draw All:** All symbols are drawn at every data point.
- **ISkip=2:** Symbols are drawn every other data point.
- **ISkip=3:** Symbols are drawn every third data point.
- **ISkip=4:** Symbols are drawn every fourth data point.
- **Distance=1%:** Symbols are drawn at the first data point and subsequently at data points that are at least one percent of the frame height distant from the previously plotted data point.
- **Distance=2%:** Symbols are drawn at the first data point and subsequently at data points that are at least two percent of the frame height distant from the previously plotted data point.
- **Distance=3%:** Symbols are drawn at the first data point and subsequently at data points that are at least three percent of the frame height distant from the previously plotted data point.

To specify the symbol spacing:

1. On the Symbols page of the Mapping Style dialog, select the appropriate mappings.
2. Click Symb Spacing, and then select the desired option.
3. If you selected Enter Index or Enter Distance, an Enter Value dialog appears. For Enter Index, Enter the index skip between symbols. For Enter Distance, enter the distance between symbols as a percentage of the frame height.

8.5. Line Plot Axis

Line plots support many varied axis options, including two features unique to line plots: log scale and multiple axes. See Chapter 17, “Axes,” for a complete description of all axis options.

8.5.1. XY Line Axis Range Control

Controlling the axis range is a very common action in XY Line plots. You may want to modify the range, for example, to include additional text or geometries in your axis area; or you may want to have the axes begin and end at round numbers. You control the range of your X- and Y-axes using the Range page of the Axis Details dialog, shown in Figure 8-14.

To call up this page, select Axis from the Plot menu to bring up the Axis Details dialog, and then click on the Range button.

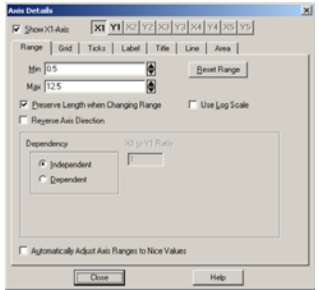


Figure 8-14. The Range page of the Axis Details dialog.

For XY Line plot types, the default range for both X and Y is the range of the X- and Y-variables for the initial active mappings. You can alter the range of any active axis. For more details on modifying the axis range, see Section 17.3, “Axis Range Modification.”

If you simply want the values on the axes to being and end at round numbers, you can choose Make Current Values Nice from the View menu.

8.5.2. Polar Line Axis Control

For Polar Line plots, Tecplot by default treats Theta values as angles in degrees. If your angle values are not in degrees, you may change the settings. From the Plot menu, select Axis. This will bring up the Axis Details dialog. Half-way down the Range page, there is a Reset button, which allows you to reset the polar axis to radians or arbitrary angle units.

You may also change the range on Theta to create a quarter circle; or change the range on R to create a “hole” in the center of your polar axis grid area. For more information on polar axis options, see Section 17.7.2, “Polar Axes Positioning.”

8.5.3. Log Axes

The X- and Y-axes of XY Lines plot types and the R-axis of Polar Line plot types can have a linear scale (default) or a logarithmic scale. The logarithmic scale is only available only for line plots (and not for Theta-axes).

When Auto Spacing is selected with logarithmic scale, large numbers are displayed in scientific notation (that is, **`3.48x10`**^{5}). It is strongly recommended that you use Auto Spacing with log axes.

To specify a log axis:

1. From the Plot menu, choose Axis. The Axis Details dialog appears.
2. On the Range page of the dialog, select the check box labeled Use Log Scale for each axis you wish to have a log scale.

3. On the Ticks or Labels page of the dialog, confirm that the check box labeled Auto Spacing is selected (Auto Spacing is turned on by default).

Two line plots using logarithmic scales are shown in Figure 8-15.

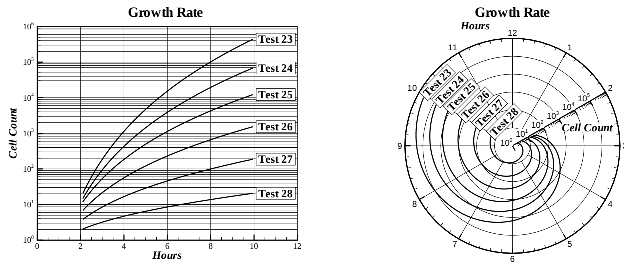


Figure 8-15. Logarithmic scale in an XY Line plot (left) and a Polar Line plot (right).

8.5.4. Multiple X- and Y-Axes for XY Line Plots

You may want to display mappings which have greatly differing scales on a single plot. For example, you might want to plot a function and its derivative together, or see a plot of both altitude and engine temperature versus elapsed time. To plot this type of data, XY Line plot types provide for multiple X- and Y-axes. You can assign each mapping to a separate Y-axis and use the natural scale of the variables assigned to that mapping.

For example, consider the data in the example Tecplot demo data file **`rainfall.plt`**; it includes monthly rainfall observations for three U.S. cities, along with two error measurements. The rainfall observations and the error measurements have very different scales.

To plot the Seattle rainfall observations and the second error measurement, do the following:

1. Read in the data file **`rainfall.plt`** from the **`demo/xy`** sub-directory of your Tecplot home directory.
2. Select XY Line Plot on the Select Initial Plot dialog. Tecplot creates a default line plot of the Seattle rainfall observations (V2) on the Y-axis versus the Month (V1) on the X-axis.
3. Bring up the Mapping Style dialog, select the Definitions page.
4. Select the mapping Error 2, click Map Show, and choose Activate.
5. Click Which Y-Axis, then choose Y2. The resulting plot shown in Figure 8-16.

By default, Tecplot places axis X1 at the bottom of your axis grid area, and subsequent X axes at the top. Similarly, it places axis Y1 at the left of your axis grid area and subsequent Y-axes at the right. Thus, in Figure 8-16, the Seattle rainfall observations are shown along axis Y1 at the left of the axis grid area, while the error observations are shown along Y2 at the right.

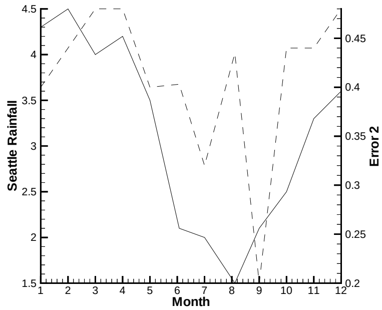


Figure 8-16. An XY Line plot using two Y-axes.

You can also use multiple axes to cycle through mappings with different ranges or axis settings. You may find it convenient to assign different mappings to different axes so that you can set axis ranges, axis positions, or other axis attributes independently for each mapping. Then as you activate and deactivate your mappings, each appears with the settings you have previously set.

8.6. Curve-Fits and Splines

By specifying a curve type, you control whether Tecplot simply connects the data points or performs a more sophisticated analysis to determine the drawn curve. Curve-fitting is an important data analysis tool because it lets you discover hidden trends in seemingly random scatters of data. Curve-fits can also help you determine if experimental results match theoretical predictions.

8.6.1. Curve Types

Tecplot offers a variety of curve-fits and spline fits. You set the type of curve plotted for a mapping using the Curve Type drop-down on the Curves page of the Mapping Style dialog, or by using the Line on the Quick Edit dialog.

Tecplot offers the following curve types (the names are as shown under the Curve Type drop-down; the buttons as shown in the Quick Edit dialog):

- **Line Segments** **:** A series of linear segments connect adjacent data points. In XY Line plots, these will be line segments. See Section 8.12, “Polar Drawing Options,” for a discussion of Line Segments in Polar lLine plots.
- **Linear Fit:** A linear function is fit to the data points. In XY Line plots, this will be a straight line. (Linear fit is not available on Quick Edit dialog.)

- **Polynomial Fit :** A polynomial of order N is fit to the data points (where 1 <= N <= 10, for N=1 a Linear Fit is done).
- **Exponential Fit :** An exponential curve fit that finds the best curve of the form $Y=e^{b*X+c}$ (equivalent to $Y=a*e^{b*X}$, where $a = e^c$). To use this curve type, Y-values for this variable must be all positive or all negative. If the function dependency is set to *X=f(Y)* all X-values must be all positive or all negative. Dependent values are discussed in Section 8.6.10, “Dependent and Independent Variables.”
- **Power Fit :** A power curve fit that finds the best curve of the form $Y=e^{b * \ln X + c}$ (equivalent to $Y=a*X^b$, where $a = e^c$). To use this curve type, Y-values for this variable must be all positive or all negative; X-values must be all positive. If the function dependency is set to *X=f(Y)*, X-values must be all positive or all negative, and the Y-values must all be positive.
- **Spline :** A smooth curve is generated that goes through every point. The spline is drawn through the data points after sorting the points into increasing values of the independent variable, resulting in a single-valued function of the independent variable. The spline may be clamped or free. With a clamped spline, you supply the derivative of the function at each end point; with a non-clamped (natural or free) spline, these derivatives are determined for you. In XY Line plots, specifying the derivative give you control over the initial and final slopes of the curve.
- **ParaSpline :** Creates a smooth curve as with a spline, except the assumption is that both variables are functions of the index of the data points. (For example in XY Line plot, ParaSpline fits *x=f(i)* and *y=g(i)* where *f()* and *g()* are both smooth.) No sorting of the points is performed; the order of the data points from the data file is used. This spline may result in a multi-valued function (of either or both axis variables).
- **Extended :** Uses a curve-fit supplied by an add-on. These curve-fits may be provided by Amtec, a third party, or written by users. The functionality of each extended curve-fit is defined by its creator. If you wish to write an extended curve fit add-on, see the *Add-On Developer’s Kit User’s Manual* for more information.

Linear Fit, Polynomial Fit, Exponential Fit, and Power Fit are all determined by using a least squares algorithm. Examples of each of Tecplot’s curve-fit types are shown in Figure 8-17.

The Curves page contains fields for controlling the following attributes:

- **Dependent Variable:** The Dependent Variable drop-down controls how Tecplot interprets curve fits and splines. Dependent Variable has no effect on mappings of the Line Segment curve type. For the XY Line plot type, the choices are *y=f(x)* or *x=f(y)*. For Polar Line plot types, the choices are *R=f(Theta)* and *Theta=f(R)*.
- **Curve Points:** Controls the number of points used to draw curve fits and splines. Raising the number of points increases the accuracy of curve but also increases plotting time and the size of print files.

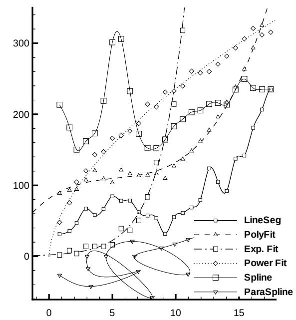


Figure 8-17. Tecplot's curve-fit types.

- **Curve Settings:** Control options specific to the curve type. For example, weighting for curve fits, or starting derivatives for splines.

The coefficients used to draw curve fits and splines may be output to a file, as can the actual points used to draw curve fits and splines.

8.6.2. Linear Fits

Tecplot can fit a linear function to data using the standard least-squares algorithm. It calculates the function for which the sum of the squared differences from the data points is a minimum. For the XY Line plot type, the linear function is a straight line.

To fit a linear function to your data:

1. From the Curves page of the Mapping Style dialog, select the mappings for which you want to fit a linear function.
2. Click Curve Type, and select Linear Fit from the drop-down.
3. By default, linear fits use all the points in the mapping, weighting them equally. Use the Curve Fit Settings dialog to specify different settings. The dialog is shown in Figure 8-18.
4. Select the appropriate mappings in the Mapping Style dialog and click Curve Settings. The Curve Fit Settings dialog appears as shown in Figure 8-19.
5. To limit the points used in the mapping, select Use Only Points Within Range, and enter minimum and maximum values.
6. To assign a curve-weighting variable, select Use Weighting Variable, and select the variable from the drop-down. For more information on curve weighting, see Section 8.6.11, "Curve-Fit Weighting Variables."
7. Polynomial Order is shown on the dialog, but should always be one for a linear fit. If you change this away from one, Tecplot changes the curve type to Polynomial Fit. See Section 8.6.3, "Polynomial Curve-Fits," for more information.

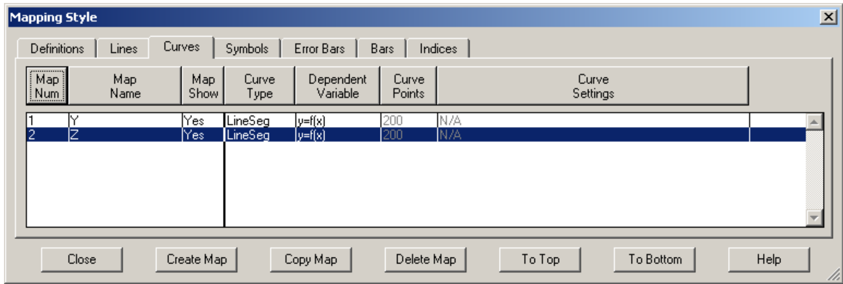


Figure 8-18. The Curves page of the Mapping Style dialog.

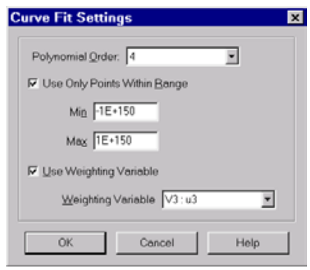


Figure 8-19. The Curve Fit Settings dialog.

To choose the dependent and independent variables for this fit, see Section 8.6.10, "Dependent and Independent Variables."

8.6.3. Polynomial Curve-Fits

Tecplot can fit a polynomial function to data using the standard least-squares algorithm. You specify the order of the polynomial (from one to ten), and Tecplot calculates the polynomial for which the sum of the squared differences from the data points is a minimum.

To fit a polynomial to your data:

1. From the Curves page of the Mapping Style dialog, select the mappings for which you want to fit a polynomial function.
2. Click Curve Type, and select Polynomial Fit from the drop-down.
3. You can also switch to a polynomial by selecting a mapping in the plot and clicking  on the Quick Edit dialog.

By default, this option fits a cubic polynomial, using all the points in the mapping and weighting them equally. Use the Curve Fit Settings dialog, shown in Figure 8-19, to specify different settings:

1. Select the appropriate mappings in the Mapping Style dialog and click Curve Settings. The Curve Fit Settings dialog appears.
2. To change the polynomial order, use the Polynomial Order drop-down. Select the desired polynomial order (one to ten). An order of two is a quadratic polynomial, an order of three is a cubic polynomial, etc. If you select one, the curve type is set to Linear Fit as a polynomial of order one is a linear function. (See Section 8.6.2, “Linear Fits.”)
3. To limit the points used in the mapping, select Use Only Points Within Range, and enter minimum and maximum values.
4. To assign a curve-weighting variable, select Use Weighting Variable, and select the variable from the drop-down. For more information on curve weighting, see Section 8.6.11, “Curve-Fit Weighting Variables.”

To choose the dependent and independent variables for this fit, see Section 8.6.10, “Dependent and Independent Variables.”

8.6.4. Exponential Curve-Fits

Tecplot can fit an exponential function to data using the standard least-squares algorithm. The dependent-variable values must be either all positive or all negative. Tecplot finds the best curve (in the least-squares sense) of the form $Y=e^{b*X+c}$ (equivalent to $Y=a*e^{b*X}$ where $a=e^c$). If Theta is the independent variable, the equation is $R = \pm e^{(b\theta + c)}$ or $R = \pm a e^{b\theta}$, and similarly for other independent variables.

To fit an exponential curve to your data:

1. From the Curves page of the Mapping Style dialog, select the mappings for which you want to fit an exponential function.
2. Click Curve Type, and select Exponential Fit from the drop-down.

You can also switch to an exponential fit by selecting a mapping in the plot and clicking  on the Quick Edit dialog.

By default, this option uses all the data points in the mapping, weighting them equally. Use the Exponential Fit Settings dialog to specify different settings. The dialog is shown in Figure 8-20.

To specify different settings:

1. Select the appropriate mappings in the Mapping Style dialog and click Curve Settings. The Exponential Fit Settings dialog appears.
2. To limit the points used in the mapping, select Use Only Points Within Range, and enter minimum and maximum values.

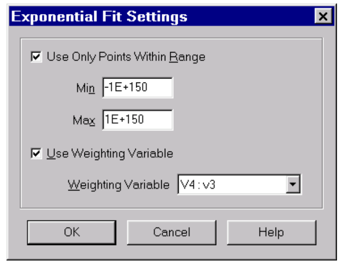


Figure 8-20. The Exponential Fit Settings dialog.

3. To assign a curve-weighting variable, select Use Weighting Variable, and select the variable from the drop-down. For more information on curve weighting, see Section 8.6.11, "Curve-Fit Weighting Variables."

To choose the dependent and independent variables for this fit, see Section <<8.6.10>>, "Dependent and Independent Variables."

8.6.5. Power Curve-Fits

Tecplot can fit a power function to data using the standard least-squares algorithm. The dependent-variable values must be either all positive or all negative, and the independent values should be all positive (although Tecplot will ignore data points with zero or negative independent values). Tecplot finds the best curve (in the least-squares sense) of the form $Y{=}e^{b*lnX+c}$ (equivalent to $Y{=}a*X^b$ where $a{=}e^c$). If Theta is the independent variable, the equation is $R = \pm e^{b\ln(\Theta)+c}$ or $R = \pm a\Theta^b$, and similarly for other independent variables.

To fit a power curve to your data:

1. From the Curves page of the Mapping Style dialog, select the mappings for which you want to fit a power function.
2. Click Curve Type, and select Power Fit from the drop-down.

You can also switch to a power fit by selecting a mapping in the plot and clicking  on the Quick Edit dialog.

1. From the Curves page of the Mapping Style dialog, select the mappings for which you want to fit a power function.
2. By default, this option uses all the data points in the mapping, weighting them equally. Use the Power Fit Settings dialog to specify different settings. The dialog is shown in Figure 8-21.
3. To limit the points used in the mapping, select Use Only Points Within Range, and enter minimum and maximum values.

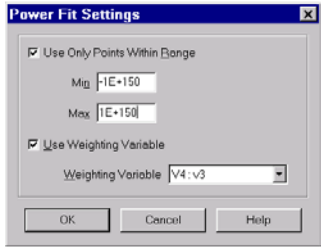


Figure 8-21. The Power Fit Settings dialog.

4. To assign a curve weighting variable, select Use Weighting Variable, and select the variable from the drop-down. For more information on curve weighting, see Section 8.6.11, “Curve-Fit Weighting Variables.”

To choose the dependent and independent variables for this fit, see Section 8.6.10, “Dependent and Independent Variables.”

8.6.6. Splines

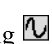
A spline is a mathematical function defined to link a specified set of points in such a way that the resulting function is continuous and smooth (differentiable) at every point. The most common type of spline, the cubic spline, is defined using a set of cubic polynomials, one for each interval between the data points.

Splines can be natural or clamped; natural splines are twice-differentiable at the end points and the second derivative is zero at those points, while clamped splines need not be twice-differentiable, but have known first-derivatives at the boundary points.

Before plotting the spline, Tecplot sorts the data points in increasing value along the independent axis. The Sort option of the Definitions page of the Mapping Style dialog has no effect on splines (see Section 8.3.5, “Data Point Sorting,” for a discussion of sorting).

To fit a spline to your data:

1. Use the Curves page of the Mapping Style dialog to select the mappings for which you want to fit a spline.
2. Click Curve Type, and select Spline from the drop-down.

You can also switch to a spline by selecting a mapping in the plot and clicking  on the Quick Edit dialog.

By default, this option fits a natural cubic spline. To specify a clamped spline:

1. Select the appropriate mappings in the Mapping Style dialog and click the Curve Settings button. The Spline Settings dialog appears as shown in Figure 8-22.

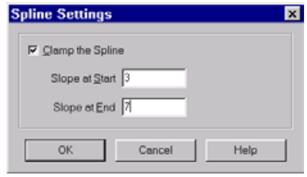


Figure 8-22. The Spline Settings dialog.

2. Select Clamp the Spline, and enter values for the derivative at the start and end of the spline.

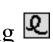
8.6.7. Parametric Splines

Tecplot's cubic spline fit assumes that the spline function is a single-valued function of the independent variable. Sometimes, however, you have data that curves back upon itself, but you would still like to have a spline-like curve fit to it. Parametric splines solve this problem by presuming that both variables (`X&Y` or `Theta&R`) are functions of the data-point index. The spline is then defined by two single-valued functions of the data-point index.

Unlike cubic splines, parametric splines are plotted in the order set in the Sort option of the Definitions page of the Mapping Style dialog. By default, the points are unsorted, and thus the spline is drawn in order the data points appear in the data file. See Section 8.3.5, "Data Point Sorting," for a discussion of sorting.

To fit a parametric spline to your data:

1. Use the Curves page of the Mapping Style dialog to select the mappings for which you want to fit a parametric spline.
2. Click Curve Type, and select ParaSpline from the drop-down.

You can also switch to a parametric spline by selecting a mapping in the plot and clicking  on the Quick Edit dialog.

By default, this option fits two natural cubic splines to the data point index. To specify a clamped spline:

1. Select the appropriate mappings in the Mapping Style dialog and click Curve Settings. The Parametric Spline Settings dialog appears as shown in Figure 8-23.
2. Select Clamp the Spline, and enter values for the derivative at the start and end of the spline.

For the XY Line plot type, the derivatives are either *dy/dx* or *dx/dy* depending on the Function Dependency for the mapping. Tecplot calculates *dx/ds* and *dy/d* from these values (where *s* is the parametric variable). For the Polar Line plot type, the derivatives are either *dR/dTheta* or *dTheta/dR* (depending on the Function Dependency for the mapping), and Tecplot calculates *dx/ds* and *dy/ds* from these values (where *s* is the parametric variable). See Section 8.6.10,

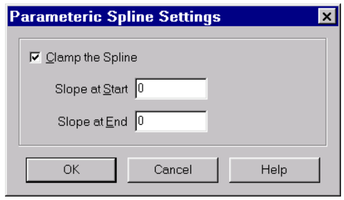


Figure 8-23. The Parametric Spline Settings dialog.

“Dependent and Independent Variables,” for a full description of the Function Dependency option.

8.6.8. Extended Curve-Fit

Tecplot add-ons can provide new curve-fit types. These curve types are called extended curve-fits. These curve-fits may be provided by Amtec, a third party, or written by users. The functionality of each extended curve-fit is defined by its creator. (If you wish to write an extended curve-fit add-on, see the *Add-On Developer’s Kit User’s Manual* for more information.)

To fit an extended curve to your data:

1. 1. Use the Curves page of the Mapping Style dialog to select the mappings for which you want to apply an extended curve-fit.
2. 2. Click Curve Type, and select Extended from the drop-down.
3. 3. The Choose Extended Curve Fit dialog appears with a list of optional curve fits, as shown in Figure 8-24. Select the desired curve fit.

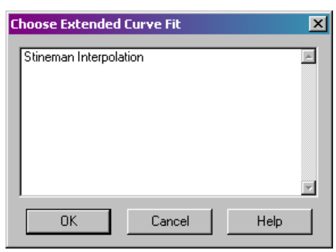


Figure 8-24. The Choose Extended Curve-Fit dialog.

You may also switch to an extended curve-fit by selecting a mapping in the plot, then clicking  on the Quick Edit dialog. This calls up the Choose Extended Curve Fit dialog, from which you may select the desired curve-fit.

Some curve-fits allow optional parameters. These can be accessed by selecting the Curve Settings option on the Curves page of the Mapping Style dialog. When the curve-fit includes optional settings, a dialog for that curve-fit appears. For example, the options available for the Stineman Interpolation curve-fit are shown in Figure 8-25.

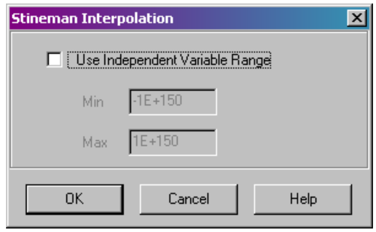


Figure 8-25. Stineman Interpolation curve-fit options.

8.6.9. Line Segments (No Curve-Fit)

By default, Tecplot draws a series of linear segments between each set of points. For the XY Line plot type, these will be line segments. See 8.12, "Polar Drawing Options," for a discussion of Line Segments in Polar Line plots. This is not really a curve-fit, but it is a good fall back if other fits are not appropriate.

To turn off curve fits for your data and just use linear segments between points:

1. From the Curves page of the Mapping Style dialog, select the mappings which you want to show as linear segments.
2. Click Curve Type. Select Line Segments from the drop-down.

You can also switch to line segments by selecting a mapping in the plot and clicking  on the Quick Edit dialog.

Line Segments are plotted in the order set in the Sort option of the Definitions page of the Mapping Style dialog. By default, the points are unsorted, and thus the lines segments are drawn in order the data points appear in the data file. See Section 8.3.5, "Data Point Sorting," for a discussion of sorting.

8.6.10. Dependent and Independent Variables

Every mapping has a dependent variable and an independent variable. The dependency relationship determines the shape of your plot for most curve types. This dependency has no effect on line segment curve types, and for parametric splines, the dependency is only used to determine starting derivatives for clamped parametric splines. Extended curve-fits are free to use or not use this dependency depending on the type of curve-fit supplied.

You specify the dependency relationship between your axis variables using the Dependent Variable drop-down on the Curves page of the Mapping Style dialog.

For the XY Line plot type, the default setting is *y=f(x)*, and you may change the value to *x=f(y)*. With *y=f(x)*, the X-axis variable is treated as the independent variable and the Y-axis variable as the dependent variable. With *x=f(y)*, the Y-axis variable is treated as the independent variable and the X-axis variable as the dependent variable. Two polynomial curve-fits of the same data using different dependency settings are shown in Figure 8-26.

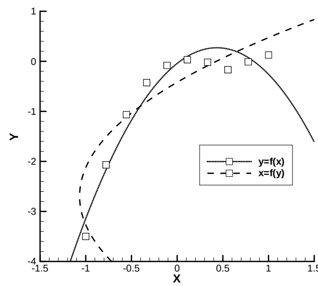


Figure 8-26. An XY Line plot type dependencies.

Similarly for Polar Line plots, the default setting is *R=f(Theta)*, and you may change the value to *Theta=f(R)*. With *R=f(Theta)*, the Theta-axis variable is treated as the independent variable and the R-axis variable as the dependent variable. With *Theta=f(R)*, the R-axis variable is treated as the independent variable and the Theta-axis variable as the dependent variable.

To change the dependency setting:

1. From the Curves page of the Mapping Style dialog, select the mappings to change.
2. Dependent Variable and choose the appropriate option.

For the XY Line plot type, the dependency setting determines the direction of bar charts. To create a vertical bar chart set the dependency to *y=f(x)*; to create a horizontal bar chart set the dependency to *x=f(y)*. See Section 8.8, “XY Line Bar Charts,” for information on bar charts.

8.6.11. Curve-Fit Weighting Variables

Linear, polynomial, exponential, and power fits allow you to specify a weighting variable. By default each data point is weighted equally. With the weighting variable, individual points can be given more or less weight. Relatively larger numbers in the curve weighting variable mean more significance for a given point. If the curve-weighting variable is zero at a data point, that data point has no effect upon the resulting curve. If the curve weighting variable is two at a data point and one at all the other points, that data point will have twice the effect upon the resulting curve. If the curve weighting variable is much larger at one point than the others, the weighting of that point pulls the resulting curve close to that point. The weighting coefficients must be integers in the range of zero to 9,999. Tecplot truncates weighting coefficients defined as floating-point numbers (that is, a weighting coefficient of 1.99 is truncated to 1.0).

For example, consider the distance-temperature data in the example data file **simpxy.dat** (found in the **examples/dat** directory in your Tecplot home directory). There is a small cluster of points centered about Distance=0.1 and Temperature=550. If we add the following weighting variable to the original data file **sympxy.dat**, we can omit this cluster from our analysis:

```
1 1 0 0 0 0 0 0 1 1 1 1 1 1 1 1 1 1 1 1
```

The data file **sympxy2.dat** (also found in the **examples/dat** directory in your Tecplot home directory) contains this additional variable as variable 6, Weight4. (There are other potential weightings in variables 3, 4 and 5.)

To assign the curve weighting variables, see the instructions for Linear Fit, Polynomial Curve-Fit, Exponential Curve-Fit or Power Curve-Fit.

The left side of Figure 8-27 shows an XY Line plot with weighted linear fit with the cluster of points omitted. For comparison, the original data points and the un-weighted least-squares fit are also plotted. The right side shows the same data in a Polar Line plot.

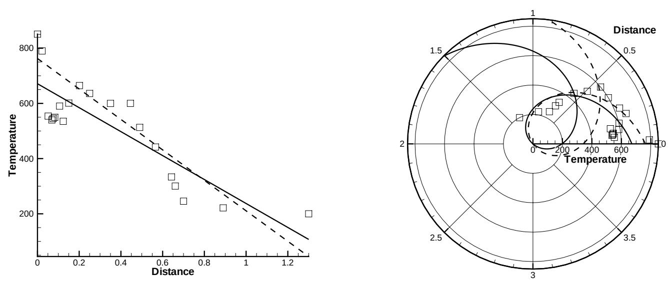


Figure 8-27. Weighted linear fits.

To create this plot, following these steps:

1. Read in the data set **simpxy2.dat** from the **examples/dat** directory of your Tecplot home directory.
2. Call up the Mapping Style dialog. On the Definitions page, create two copies of the distance-temperature mapping, and activate the new mappings.
3. Name the first mapping "Temperature," the second mapping "Weighted Temperature" and the third mapping "Data Points."
4. Turn on both the Lines and Symbols map layers.
5. On the Lines Page, turn off Line Show for the "Data Points" mapping.
6. Set the line pattern for the Weighted Temperature to be dashed.
7. On the Curves Page, set the Line Type for Temperature and Weighted Temperature to Linear Fit.

8. For the Weighted Temperature mapping, click Curve Settings on the Curves Page. Turn on Use Weight Variable, and select a Weighting Variable of Weight4. Click OK.
9. On the Symbols page, turn off Symb Show for the same two mappings.
10. Add the line plot legend. See Section 8.10, "Line Legend." .

8.6.12. Curve-Fit Coefficients

You can view information about curve-fits and splines using the Data menu's Curve Information dialog and write that information to a file.

To view information about curve-fits and splines in your plot:

1. From the Data menu, select Curve Info. The Curve Information dialog appears, as shown in Figure 8-28.

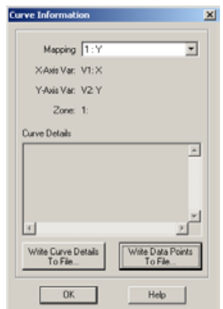


Figure 8-28. The Curve Information dialog.

2. Select a mapping from the Mapping drop-down.
3. The curve-fit coefficients for the selected mapping are shown in the Curve Details section of the Curve Details dialog.
4. To create an ASCII file with the curve-fit coefficients, click Write Curve Details to File.

The information presented in the Curve Details section and in the coefficient file is dependent on the curve type selected. For example, the dialog shown in Figure 8-28, shows the information for a linear fit. For extended curve-fits, the documentation for the extended curve-fit add-on supplies any necessary information on the format used.

8.6.13. Curve Data Point Extraction

Using the Curve Information dialog, you can save the calculated data points along the curve for further analysis in later sessions.

To create an ASCII data file of the points of the curve fits:

1. From the Data menu, select Curve Info. The Curve Information dialog appears.
2. Select a mapping from the Mapping drop-down.

3. Click Write Data Points to File.

The data file contains one zone for each line in the mapping. For mappings made from I-ordered zones, there is one zone. See Section 8.9, “I-, J- and K-Indices,” for details on mappings using IJ- and IJK-ordered data.

Each zone in the data file is I-ordered with the number of points equal to the active curve points setting (set via the Curve Points option on the Curves page of the Mapping Style dialog). The data file has two variables: one for the independent variable and one for the dependent variable. This is a valid Tecplot ASCII data file that can be read into another frame.

8.7. XY Line Error Bars

In the XY Line plot type, you can assign one or more variables to be used to compute error bars for another variable using the Error Bars page of the Mapping Style dialog, shown in Figure 8-29.

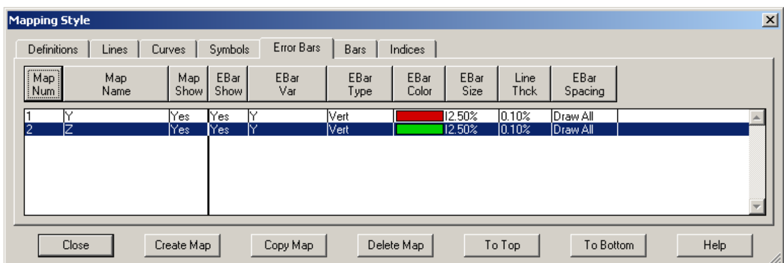


Figure 8-29. The Error Bars page of the Mapping style dialog.

Error bars are not available for the Polar Lines plot type. Each mapping has an associated error bar variable; if you want to assign multiple error bar variables to a mapping, you can create one copy of the mapping for each error bar variable.

You can use any variable in your data set as an error bar variable. However, for them to be meaningful, they should have the same units as the axis along which they are drawn. Sometimes, error variables are part of your original data file. At other times, you may create error variables using Tecplot’s data manipulation utilities. For example, if you know that the values of some measured variable are accurate only to within ten percent, you may create a new variable to use as the error bar variable by multiplying the measured variable by 0.10.

To add error bars to a plot:

1. Select the Error Bars check box on the sidebar. The Select Variable dialog appears; select the desired error bar variable. This variable is assigned as the error bar variable for each mapping.

2. No error bars appear on your plot, because they are off by default for each mapping. To turn on error bars for a mapping, select the appropriate mapping on the Error Bars page of the Mapping Style dialog, and choose EBar show.
3. To change the error bar variable for a mapping, select the appropriate mapping on the Error Bars page of the Mapping Style dialog, and choose EBar Var. The Select Variable dialog appears; select the desired error bar variable.

8.7.1. Error Bar Type

There are seven types of error bars:

- **Top:** Extends upward for positive values (and downward for negative values) of the error bar variable.
- **Bottom:** Extends downward for positive values (and upward for negative values) of the error bar variable.
- **Left:** Extends to the left for positive values (and to the right for negative values) of the error bar variable.
- **Right:** Extends to the right for positive values (and to the left for negative values) of the error bar variable.
- **Horizontal:** Extends left and right.
- **Vertical:** Extends up and down. (This is the default value.)
- **Cross:** Extends up, down, left, right.

To select an error bar type:

1. From the Error Bars page, select the appropriate mappings.
2. Click EBar Type, and select the desired error bar type.

Although the values are called Left, Right, Up and Down, the direction is determined by the direction of positive values in your plot. If you reverse the direction of an axis (using the Reverse Axis Direction option on the Range page of the Axis Details dialog), the error bars point in the opposite direction. For example, if you reverse the X-axis, left error bars will point to the right for positive values and to the left for negative values of the error bar variable.

You can use several variables as error bars. Each assignment, however, requires a separate mapping. For example, you could assign one variable for the left error bar, copy the mapping, and then assign another for the right error bar. You can even assign different variables with the same error bar type. For example, you could assign two variables to be vertical error bars resulting in two vertical error bars at the data point. An example plot with error bars is shown in Figure 8-30.

Variable V5 (called “Error 1”) is assigned as a vertical error bar for one mapping. Variable V6 (called “Error 2”) is assigned as a left error bar for a copy of the same mapping. These vertical and left error bars are plotted on the curve for variable V2 (called “Seattle Rainfall”). The data

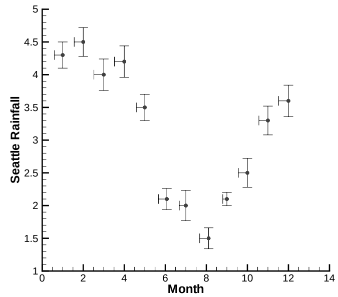


Figure 8-30. An XY Line plot with symbols and error bars.

is in the demo data file **`rainfall.plt`**. The Error Bars page for this plot is shown in Figure 8-29.

8.7.2. Error Bar Attributes

You can modify most of the attributes with which error bars are drawn—their color, their thickness, their spacing, and the width of the endpoint crossbars. You can make these changes from the Error Bars page of the Mapping Style dialog, or for some settings you can use the Quick Edit dialog.

8.7.2.1. Error Bar Color. The line color of the error bars is separate from the color of the lines or symbols of the mapping. You can change the error bar line color using either the Mapping Style dialog or the Quick Edit dialog.

To change the line color using the Mapping Style dialog:

1. On the Error Bars page of the Mapping Style dialog, select the mappings to assign a new error bar color.
2. Click EBar Color. The Select Color dialog appears; click the desired color.

To change the Error Bar color using the Quick Edit dialog:

1. Click on the appropriate mapping in the plot.
2. If the Line option in the color edit area is not already selected, select it.
3. Click Color. The Select Color dialog appears. Select the desired color.

If you change the line color of the error bars using the Quick Edit dialog, all other lines shown for that mapping will also change. So if the line layer is shown, the lines will be affected as will the outline of symbols (if shown). For individual control, use the Mapping Style dialog.

8.7.2.2. Crossbar Size. While the length of the error bar is determined by the error bar variable, the size of the crossbar is set on the Error Bars page of the Mapping Style dialog. Crossbar size is measured as a percentage of frame height.

To specify the crossbar size for error bars:

1. On the Error Bars page of the Mapping Style dialog, select the mappings to change the crossbar size.
2. Click EBar Size. Select the desired size from the drop-down.
3. If you selected Enter, the Enter Value dialog appears. Enter the desired value.

8.7.2.3. Error Bar Line Thickness. The line thickness of the error bars is separate from the thickness of the lines or symbols of the mapping. You can change the error bar line thickness using either the Mapping Style dialog or the Quick Edit dialog. The error bar line thickness is measured as a percentage of frame height.

To change the line thickness using the Mapping Style dialog:

1. On the Error Bars page of the Mapping Style dialog, select the mappings to assign a new error bar line thickness.
2. Click Line Thck. Select the desired line thickness from the drop-down.
3. If you selected Enter, an Enter Value dialog appears. Enter the desired value.

To change the Error Bar thickness using the Quick Edit dialog:

1. Click on the appropriate mapping in the plot.
2. Click Line Thcknss on the Quick Edit dialog.
3. Select the desired line thickness from the drop-down.
4. If you selected Enter, an Enter Value dialog appears. Enter the desired value.

If you change the line thickness of the error bars using the Quick Edit dialog, all other lines shown for that mapping also change. So if the Lines layer is shown, the lines are affected as is the outline of symbols (if shown). For individual control, use the Mapping Style dialog.

8.7.2.4. Error Bar Spacing. If you are plotting many points, you might not want every point to show an error bar. You can control the spacing between error bars using the EBar Spacing drop-down on the Error Bars page of the Mapping Style dialog. The spacing is specified either as a percentage of the frame height or as a number of indices to skip. You may either enter a value or use one of the following pre-set values:

- **Draw All:** Error bars are drawn at every data point.
- **ISkip=2:** Error bars are drawn every other data point.
- **ISkip=3:** Error bars are drawn every third data point.
- **ISkip=4:** Error bars are drawn every fourth data point.
- **Distance=1%:** Error bars are drawn at the first data point and subsequently at data points that are at least one percent of the frame height distant from the previously plotted data point.

- **Distance=2%:** Error bars are drawn at the first data point and subsequently at data points that are at least two percent of the frame height distant from the previously plotted data point.
- **Distance=3%:** Error bars are drawn at the first data point and subsequently at data points that are at least three percent of the frame height distant from the previously plotted data point.

To specify the error bar spacing:

1. On the Error Bars page of the Mapping Style dialog, select the appropriate mappings.
2. Click EBar Spacing, and then select the desired option.
3. If you selected Enter Index or Enter Distance, an Enter Value dialog appears. For Enter Index, Enter the index skip between error bars. For Enter Distance, enter the distance between error bars as a percentage of the frame height.

8.8. XY Line Bar Charts

A bar chart is an XY Line plot that uses vertical or horizontal bars placed along an axis to represent data points. (There is no equivalent to bar charts in Polar Line plots.) You create bar charts by activating the Bars map layer on the sidebar. You can use the Bars map layer alone to create a pure bar chart, or add symbols, lines, or error bars to create useful visual effects. An example plot with bar charts is shown in Figure 8-5.

To create a pure bar chart:

1. Read in a data file.
2. Select the initial plot type to be XY Line.
3. Select the Bars map layer check box. Deselect the check boxes for all other plot layers.

The style of the bar chart is controlled on the Bars page of the Mapping Style dialog, shown in Figure 8-31.

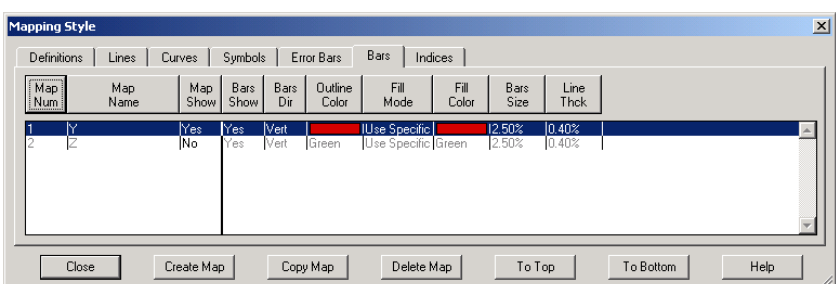


Figure 8-31. The Bars page of the Mapping Style dialog.

8.8.1. Bar Chart Direction

By default, bars are vertical. Change between vertical or horizontal bars:

1. On the Bars page of the Mapping Style dialog, select the mappings to change the bar direction.
2. Click Bars Dir, then choose Horizontal or Vertical.

Changing the direction of the bars changes the dependent variable attribute used for line curves (either *y=f(x)* or *x=f(y)*), and vice versa. By default, all mappings use *y=f(x)* and appear as vertical bar charts. If a mapping uses horizontal bars, the mapping will also use *x=f(y)* for curve fits. Of course, this only matters if you plot bars and curve-fits for the same mapping. For more information about dependency, see Section 8.6.10, “Dependent and Independent Variables.”

To modify other attributes (Bars Show, Outline Color, Fill Mode, Fill Color, Bar Size, Line Thck), use the Bars page, following the same procedures used to set symbol attributes. See Section 8.4.3, “Symbol Attributes,” for details. There is no skipping option for the Bars layer.

8.9. I-, J- and K-Indices

Whether you are making a Polar or XY Line plot, if your data is IJ- or IJK-ordered, a mapping consists of a family of lines; each line varies along one index while the other indices are held constant. By default, Tecplot displays the I-varying family of lines. For example, read in the demo data file cylinder.plt and choose the XY Line plot type as the initial plot type. You see the family of I-varying lines for Zone 1 of the data, as shown in Figure 8-32.

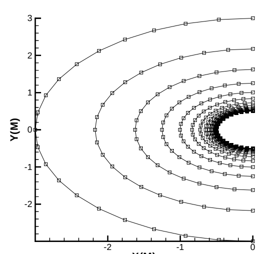


Figure 8-32. A family of I-varying lines for the cylinder data.

Each mapping can show either I-, J-, or K-varying families of lines. By default, Tecplot displays the I-varying family of lines. You can change the family of lines using the Indices page of the Mapping Style dialog as shown in Figure 8-33. The Mapping Style dialog for Polar Line plot type is shown; the dialog is the same for the XY Line plot type except for two additional pages: Bar Chart and Error Bars.

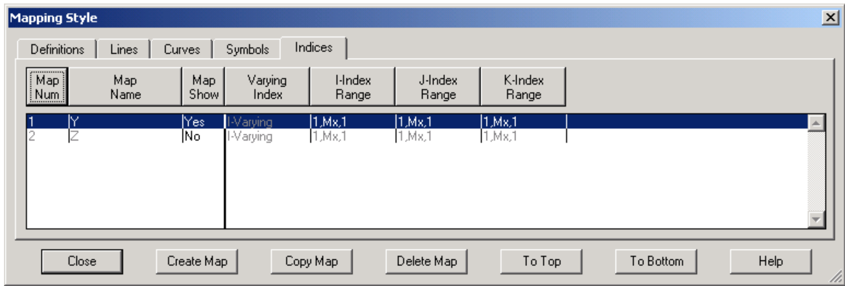


Figure 8-33. The Indices page of the Mapping Style dialog.

You can also choose which members of the family are drawn (and using which data points), by specifying index ranges for each of I, J, and K. The index range for the varying index tells Tecplot which points to include in each line, and the index ranges for the other indices tell Tecplot which lines in the family to include. Thus, you may use this option for selecting a subset of an I-ordered zone to plot.

8.9.1. Varying Index

To choose the varying index, and thus specify the family of lines to be drawn:

1. On the Indices page of the Mapping Style dialog, select the mapping for which you want to change the varying index. This mapping should be using an IJ- or IJK-ordered zone.
2. Click on Varying Index, and choose the desired family (I-, J-, or K-varying). K-varying is only available if the mapping is using an IJK-ordered zone.

For example, if you read in the **`cylinder.plt`** data file and change to use the J-varying index, the plot will appear as shown in Figure 8-34.

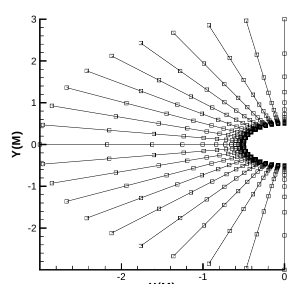


Figure 8-34. A family of J-varying lines for the cylinder data.

You may define families of lines in IJ- and IJK-ordered zones to have curve types of splines or curve fits. This means that you can plot the I-, J-, or K-varying lines of an IJ- or IJK-ordered zone using these curve types. For example, suppose you create a circular zone with a small number (seven for example) of points around the circumference. A two-dimensional plot or a straight line segment line plot of the J-lines of this zone will show a polygon-shaped zone. Using the Paraspline curve type for the J-lines results in splined mesh lines that reveal the circular shape of the zone.

8.9.2. Index Ranges

By default, the entire range of points is plotted in your mapping. For IJ- and IJK-ordered data, you may want to specify an index range to limit the number of lines drawn. Or, for any type of data, you may want to limit the points drawn to a select range.

To specify the index ranges:

1. On the Indices page of the Mapping Style dialog, select the mapping for which you want to specify an index range.
2. Click on one of I-, J-, or K-Index range. The Enter Range dialog appears.
3. Enter a starting index in the Begin field, an ending index in the End field, and a skip factor in the Skip field. A skip of one means “use every point in the range,” a skip of two means “use every other point,” and so on.

If you change index range of the varying index of the mapping, you limit the points in each line drawn in that mapping. For example, for the cylinder data, if you change the I-Index Range End to 15 while displaying I-varying lines, you obtain the plot shown in Figure 8-35. Compare

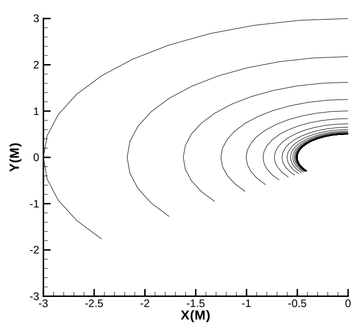


Figure 8-35. XY Line plot with a family of I-varying lines and the I-index ending at 15.

this to Figure 8-32 which shows I-varying lines for the entire I-index Range.

If you change index range of the non-varying indices of the mapping, you limit lines drawn for that mapping. For example, for the cylinder data, if you change the J-Index Range End to 15 while displaying I-varying lines, you obtain the plot shown in Figure 8-36. Compare this to Figure 8-32 which shows I-varying lines for the entire J-index Range.

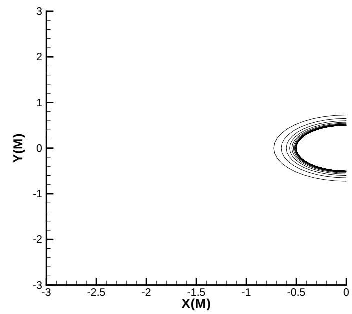


Figure 8-36. XY Line plot with a family of I-varying lines and the J-index ending at 15.

8.10. Line Legend

You can generate a legend that shows the line and symbol attributes of the mappings. In XY Line plots, this legend includes the bar chart information. The legend can be positioned anywhere within the line plot frame.

The mappings that are shown in the legend are selected on the Definitions page of the Mapping Style dialog. By default, all mappings are shown, but Tecplot removes redundant entries. That is, if two entries in the legend would look exactly alike, only one is printed. See Section 8.3.6, "Line Plot Legend Control," for more details.

To include the line plot legend:

1. From the Plot menu, choose Line Legend. The Line Legend dialog appears, as shown in Figure 8-37.
2. Select the check box labeled Show Line Legend.

Mapping names are included in the legend by default. If you do not want to see the mapping names, uncheck the Show Mapping Names check box.

The Line Legend dialog also allows you to format the text for the legend by choosing a color and font, and specifying the text height, and to enter a line spacing between entries in the dialog.

The legend is automatically placed for you. You may specify the position of the legend by entering values in the X (%) and Y (%) text fields. Enter X as a percentage of the frame width and Y as a percentage of the frame height.

You may also specify the anchor location of the legend with the Anchor button. This is the point on the legend that corresponds to the entered position. The anchor is also the point on the legend that does not move when the frame is resized. By default, the legend is anchored in the top right.

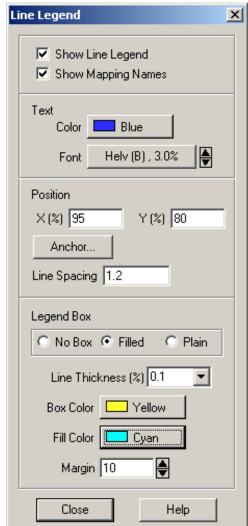


Figure 8-37. The Line Legend dialog.

The Line Legend can include a box. By default, the box is set to Plain, which is just an outline, but you can change to No Box or a Filled box using the appropriate check boxes. If the legend is Plain or Filled, the box attributes may be changed with the following controls:

- **Line Thickness:** Specify the line thickness as a percentage of frame height.
- **Box Color:** Choose a color for the legend box outline.
- **Fill Color:** Choose a color for the legend box fill. This option is only available if the Legend box is filled.
- **Margin:** Specify the margin between the legend text and legend box as a percentage of the text height.

8.11. Data Point Labels

You can label all or some of the data points, or nodes, in your line plots with either the index of the data point, the value of the dependent variable at the point, or the both the values (**`X&Y`** or **`Theta&R`**) for the data point. For example, Figure 8-38 shows an XY Line plot with each data point labeled with its X-Y value pair, while Figure 8-39 shows a Polar Line plot with each point labeled with its Theta-R value pair.

To add data labels to your plot:

1. From the Plot menu, choose Label Points. The Label Points dialog appears, as shown in Figure 8-40.
2. Select Show Node Labels.

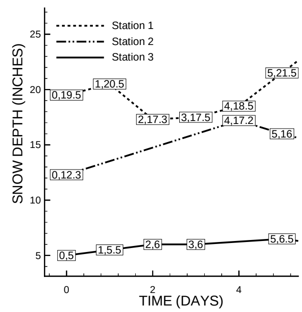


Figure 8-38. An XY Line plot with data labels.

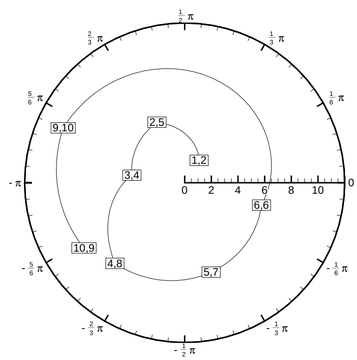


Figure 8-39. A Polar Line plot with each data point labeled.

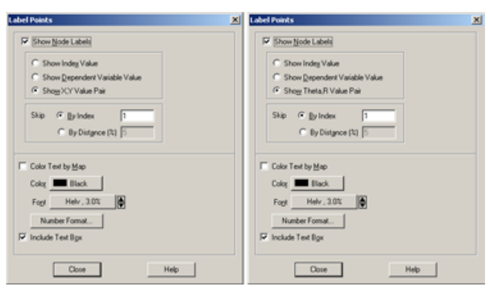


Figure 8-40. The Label Points dialog.

3. Choose one of the three toggles: Show Index Value, Show Dependent Variable Value, or Show X,Y Value Pair. For Polar Line, the third option is Show Theta-R Value Pair.

By default, all points of all mappings are labeled. If you want to label only some points, you can choose to skip labels by index or by distance. If you click the By Index check box, you can enter an index skip in the adjacent text field: a value of one shows a label at all points, a value of two labels every other point, and so on. If you click the By Distance (%) check box, you can enter a frame distance in the adjacent text field; data points closer than the specified distance to the last labeled point are not labeled.

The style of labels can be altered with the following controls:

- **Color Text by Map:** If this check box is checked, the labels will be the same color as the outline color used for symbols for the mapping. This color is set on the Symbols page of the Mapping Style dialog, and will be used even if symbols are turned off.
- **Color:** Choose a text color from the Select Color dialog. This option is only available if Color Text by Map is off (which by default, it is).
- **Font:** Select a font and size from the Select Font dialog.
- **Include Text Box:** Add a filled box around each data label. The box is always filled with the background color of the frame and outlined with the text color.
- **Number Format:** If you are using a variable value as the data label (as opposed to the index value), you can specify the format of the labels using the Specify Number Format dialog.

8.12. Polar Drawing Options

Options specific to the Polar Line plot type are found in the Polar Drawing Options dialog shown in Figure 8-41. These settings apply to all mappings in the frame.

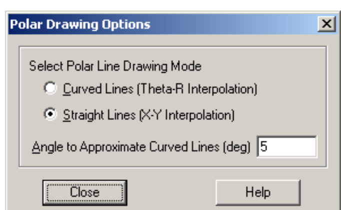


Figure 8-41. The Polar Drawing Options dialog.

In the Polar Line plot type, a line between two points may be drawn in one of two ways: they may be drawn as a straight line between the two points, or they may be drawn as an interpolation of the Theta-R values. In the latter case, the connection between the two points is a smooth curve.

By default, lines are drawn straight. This works for plots where the angular differences between consecutive points are not great. If the angular differences are great, or if you prefer a Theta-R interpolation over the default X-Y interpolation, you may change the drawing mode in the Polar Drawing options dialog. However, this can result in slow performance with large data sets.

The difference between the two Polar Line Drawing Modes is shown in Figure 8-42.

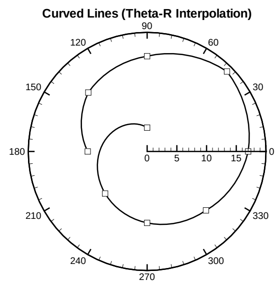


Figure 8-42. The Polar Drawing Options dialog.

With Theta-R interpolation, Tecplot will stop drawing a line that involves too many revolutions around the circle. For example, if adjacent points have angle values of 0 degrees and then 36000 degrees, the plot would involve 100 complete revolutions around the origin. If this is the case, Tecplot will draw only ten revolutions. If you need that many revolutions, create a new zone that has points interpolated in between the two points.

The Polar Drawing Options dialog also allows you to control the approximate used to draw arcs in Polar Line plots. This includes the axes, grid lines, and lines drawn in Curved Line mode. All arcs are drawn as a series of lines with a maximum angular difference. You can change this difference for more accurate Polar Line plots, but at a cost of reduced performance.

CHAPTER 9 *Mesh Plots and Boundary Plots*

A mesh plot is a field plot of the lines connecting neighboring data points within a zone. For I-ordered data, the mesh is a single line connecting all of the points in order of increasing I-index. For IJ-ordered data, the mesh consists of two families of lines connecting adjacent data points of increasing I-index and increasing J-index. For IJK-ordered data, the mesh consists of three families of lines, one connecting points of increasing I-index, one connecting points of increasing J-index, and one connecting points of increasing K-index. By changing the zone's Surfaces to Plot on the Surfaces page of the Zone Style dialog, you may limit the plotted mesh to the exterior surface (exposed cell faces) or to selected I-, J-, and K-grid planes. For finite-element zones, the mesh is a plot of all edges of all elements which are defined by the connectivity list for the node points. Mesh lines are straight lines between adjacent points.

A boundary plot is a field plot of the boundaries of ordered-data zones or 2-D finite-element zones. Boundary plots are frequently combined with other plot types.

Because the default field plot type consists of the Mesh and Boundary layers, we have already seen numerous examples of these plots in Chapter 7, "Field Plots." This chapter concentrates on those aspects of mesh and boundary plots unique to those plot types.

9.1. Mesh Plot Modification

Once you have read in your data, you can modify your mesh plot attributes using either the Mesh page of the Zone Style dialog or the Quick Edit dialog. You can control any of the following attributes from the Mesh page of the Zone Style dialog, shown in Figure 9-1:

- Which zones are active. See Section 7.3.2, "Zone Display."
- Whether the mesh is visible for each active zone. See Section 7.3.3, "Zone Layer Display."
- The mesh type. See Section 9.2, "Mesh Plot Types," below.
- The mesh color. See Section 7.3.4, "Color Choice."
- The mesh line pattern. See Section 7.3.5, "Line Pattern."
- The mesh line pattern length. See Section 7.3.6, "Pattern Length."
- The mesh line thickness. See Section 7.3.7, "Line Thickness."

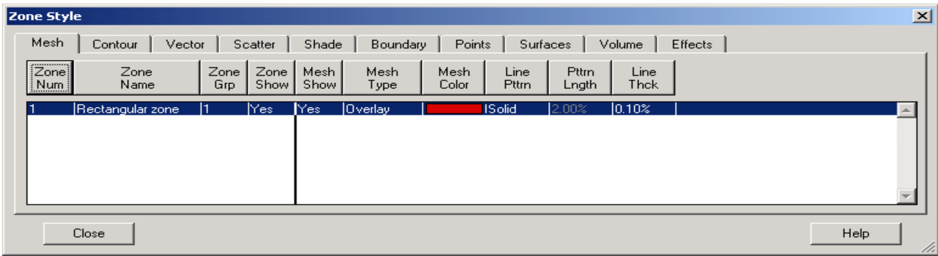


Figure 9-1. The Mesh page of the Zone Style dialog.

9.2. Mesh Plot Types

Tecplot has three distinct mesh types:

- **Wire Frame:** Mesh lines are drawn underlying all other field plots. Wire frame meshes are drawn below any other field plots (such as contours or vectors) on the same zone. In 3D Cartesian plots, no hidden lines are removed. For 3-D volume zones (finite-element volume or IJK-ordered), the full 3-D mesh consisting of all the connecting lines between data points is not generally drawn because the sheer number of lines would make it confusing. The mesh drawn will depend upon your choice of Surfaces to Plot on the Surfaces page of the Zone Style dialog. See Section 20.1, “Surfaces to Plot,” for further details. By default, only the mesh on exposed cell faces is shown.
- **Overlay:** Like Wire Frame, except that mesh lines are drawn over all other field-plot types except vectors and scatter symbols. For example, if you have flooded contours on a zone, you do not see a wire frame mesh (which would be underneath the contour flooding), but you can see an overlay mesh. In 3D Cartesian plots, the area behind the cells of the plot is still visible (unless some other plot type such as contour flooding prevents this). As with Wire Frame, the mesh drawn is dependent upon your choice of Surfaces to Plot in the Volume page of the Zone Style dialog. See Section 20.1, “Surfaces to Plot,” for further details.
- **Hidden Line:** Like Overlay above, except hidden lines are removed from behind the mesh. In effect, the cells (elements) of the mesh are opaque. Surfaces and lines that are hidden behind another surface are removed from the plot. For 3-D volume zones, using this plot type obscures everything inside the zone. If you choose this option for 3-D volume zones, then choosing to plot every surface (using the Volume page of the Zone Style dialog) has the same effect as plotting only exposed cell faces, but is much slower. You may want to use a different Surfaces to Plot option or use a wire frame mesh instead. If this option is set and the plot type is not 3D Cartesian, you get a mesh identical to that for Overlay. The opaque surfaces crated by Hidden Line are not affected by the Lighting zone effect (there is no light source shading).

Figure 9-2 shows the available mesh plot types, along with the effects of choosing Overlay and Wire Frame in combination with contour flooding.

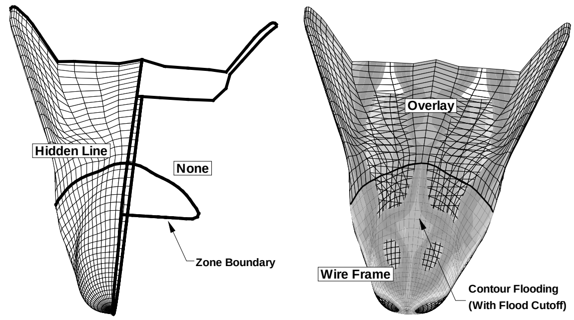


Figure 9-2. Mesh plot types.

To choose a mesh plot type for a zone or zones:

1. From the Mesh page of the Zone Style dialog, select the zone or zones for which you want to specify a plot type.
2. Click Mesh Type. The options Wire Frame, Overlay, and Hidden Line appear.
3. Click on the desired type.

You can also choose the mesh type from the Quick Edit dialog for zones chosen interactively in the workspace.

To choose a mesh type for a zone or zones from the Quick Edit dialog:

1. In the workspace, use the Selector tool to select the zone or zones for which you want to specify a plot type.
2. Call up the Quick Edit dialog from the sidebar or by choosing Quick Edit from the Edit menu.
3. Click  in the Mesh display area for a wire frame plot, click  for an overlay plot, or click  for a hidden line plot.

9.3. Boundary Plot Modification

The Boundary layer controls the boundary lines for zones. Zone boundaries exist only for ordered zones, or 2-D finite-element zones. They appear as lines around the edges of the zone. Figure 9-3 shows a boundary plot of multiple zones.

Three-dimensional finite-element zones do not have boundaries, although you may use the Extract FE Boundary dialog to create a zone that is the outer boundary or surface of a finite-element zone. See Section 19.4, “Boundary Extraction of Finite-Element Zones,” for details.

You can control any of the following attributes from the Boundary page of the Zone Style dialog, shown in Figure 9-4:

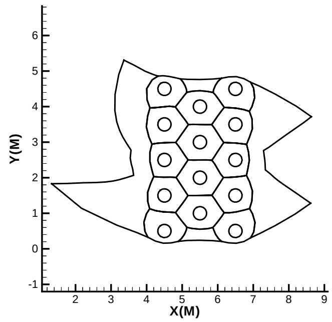


Figure 9-3. The boundary of a multiple-zone data set.

- Which zones are active. See Section 7.3.2, “Zone Display.”
- Whether the boundary is visible for each active zone. See Section 7.3.3, “Zone Layer Display.”
- Which boundaries are displayed for each zone. See Section 9.4, “Boundary Display,” below.
- The boundary color for each zone. See Section 7.3.4, “Color Choice.”
- The boundary line thickness for each zone. See Section 7.3.7, “Line Thickness.”

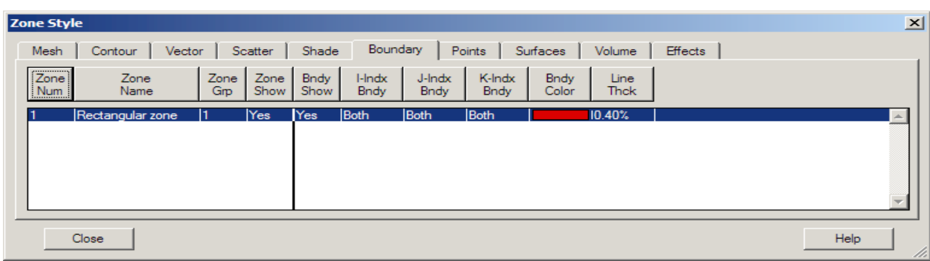


Figure 9-4. The Boundary page of the Zone Style dialog.

9.4. Boundary Display

For IJ-ordered zones, the available boundaries are the lines *I=1*, *I=IMax*, *J=1*, and *J=JMax*. When the Surfaces to Plot option is set to Boundary Cell Faces, Exposed Cell Faces, or Every Surface for IJK-ordered zones**,** the available boundaries are the boundaries of the surface areas, forming a “box” that contains the data. Surfaces to Plot can be set on the Surfaces page of the Zone Style dialog. When the Surfaces to Plot option is set to one of the planes options, such as I-, J-, or K-planes, for IJK-ordered zones the boundaries are the boundaries of each plane (I-, J-

, or K-plane). By default, all available boundaries are drawn when the Boundary layer is active. You can specify which of the available boundaries are plotted using either the Zone Style dialog or the Quick Edit dialog.

To specify which boundaries to draw:

1. From the Boundary page of the Zone Style dialog, select the zone or zones for which you want to specify which boundaries to draw.
2. Click I-Indx Bndy (or J-Indx Bndy or K-Indx Bndy). The options None, Min Only, Max Only, and Both appear.
3. Click on the desired option.

To specify which boundaries to draw using the Quick Edit dialog:

1. In the workspace, use the Selector tool to select the boundary of the zone for which you want to specify the boundaries to be drawn. When you select a zone boundary, an extra selection handle shows which of the zone's available boundaries is currently selected. Click on any other available boundary to select that boundary.
2. Call up the Quick Edit dialog from the sidebar or by selecting Quick Edit... from the Edit menu.
3. Click on one of the following buttons in the Boundary display area:
   -  To display all available boundaries for the zone.
   -  To display the currently selected boundary.
   -  To turn off the currently selected boundary.
   -  To turn off all boundaries except the currently selected boundary.

CHAPTER 10 Contour Plots

Contour plots show the variation of one variable across the data field. Contour lines and contour flooding are examples of contour plots, as are multicolored mesh, scatter, and vector plots. In this chapter, however, we restrict our attention to plots having contour lines and/or contour flooding.

Plotting contours allows you to add an extra dimension to a plot, and thus contour plots may require one more variable than mesh plots of the same dimension. Where a 2-D mesh plot shows two variables, a contour plot can show three, and a 3-D contour plot can show four. This extra variable is called the contour variable.

The procedure for creating a contour plot is:

1. Read in a data set.
2. Select the Contour check box on the sidebar to activate the Contour zone layer. If no contour variable is currently assigned, the Contour Details dialog appears with the default contour variable selected. You can either choose a different contour variable, or click Close to accept the default. Many other options can be set on the Contour Details dialog. you can also choose to use more than one contour variable in a plot. See 10.2, "Contour Groups."
3. Deselect the Mesh check box on the sidebar. This turns off the Mesh zone layer, which is on by default.

The default contour plot is a flood contour plot with contour levels spanning a range calculated by Tecplot from the range of your contour variable is shown in Figure 10-1.

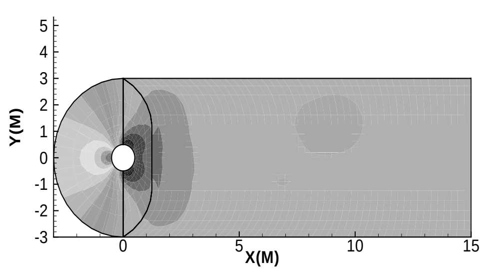


Figure 10-1. A contour plot of the cylinder data.

10.1. Contour Plot Settings

You can modify the attributes of your contour plot using either the Contour page of the Zone Style dialog or the Quick Edit dialog. You can control any of the following attributes from the Contour page, shown in Figure 10-2.

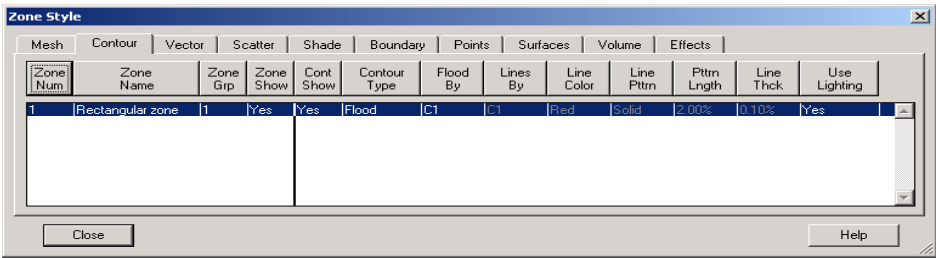


Figure 10-2. The Contour page of the Zone Style dialog.

- Which zones are active. See Section 7.3.2, "Zone Display."
- Whether the contours are visible for each active zone. See Section 7.3.3, "Zone Layer Display."
- The contour type. See Section 10.1.1, "Contour Plot Type."
- The contour group used for flooding. See Section 10.2, "Contour Groups."
- The contour group used for multi-color lines. See Section 10.2, "Contour Groups."
- The contour line color. See Section 7.3.4, "Color Choice."
- The contour line pattern. See Section 7.3.5, "Line Pattern."
- The contour line pattern length. See Section 7.3.6, "Pattern Length."
- The contour line thickness. See Section 7.3.7, "Line Thickness."
- Whether the lighting effect for contour flooding should be active for this zone. See Section 11.3.3, "Lighting Effects and Contour Flooding."

Options such as contour labels, contour legends and special settings for contour bands or contour lines are set by the selected contour group. See Section 10.2, "Contour Groups." The color map is set globally. See Section 10.4, "Global Color Map."

10.1.1. Contour Plot Type

Tecplot allows you to create contour plots of five different types:

- **Lines:** Draws lines of constant value of the specified contour variable.
- **Flood:** Floods regions between contour lines with colors from the global color map.
- **Both Lines and Flood:** Combines above two options.
- **Average Cell:** Floods cells or finite-elements with colors from the global color map according to the average value of the contour variable over the data points bounding the cell.

- **Primary Value:** Floods cells or finite-elements with colors from the global color map according to the primary value of the contour variable for each cell. If the variable is cell centered, the primary value is the value assigned to the cell. If the variable is node located, the primary value comes from the lowest index node in the cell.

An example of each contour plot type is shown in Figure 10-3.

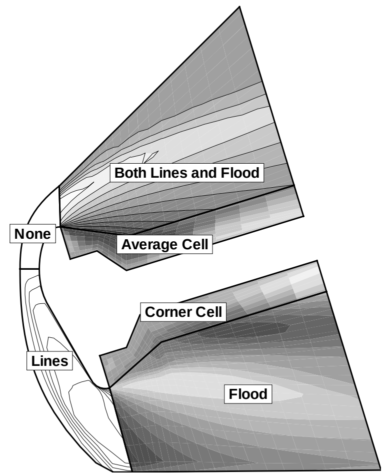


Figure 10-3. Contour plot types.

To modify the contour plot type:

1. Choose Zone Style on the sidebar, or from the Plot menu. Click Contour on the Zone Style dialog.
2. Select the zone or zones for which you want to modify the plot type.
3. Click Contour Type. A drop-down appears listing the available plot types. Click on the desired plot type.

To modify the contour plot type from the Quick Edit dialog:

1. From the sidebar, click Quick Edit. The Quick Edit dialog appears.
2. In the workspace, use the Selector tool to select the zone or zones for which you want to modify the plot type.
3. On the Quick Edit dialog, click on the appropriate button, as follows:
   -  Lines.
   -  Flood.
   -  Both Lines and Flood.

-  Average Cell.
-  Corner Cell.

A flooded contour plot is a contour plot in which the area between contour levels is filled (that is, flooded). Flooded contour plots give an immediate visual impression of how the contour variable is changing. All contour types except Lines use flooding.

The distribution of colors used for contour flooding may be banded or continuous. When banded distribution is used for flooding a solid color is used between contour levels. If continuous color distribution is used the flood color will vary linearly in all directions. See Section 10.2.4, “Color Distribution Methods,” for details on Tecplot’s color distribution methods.

To create a flooded contour plot from the Contour page of the Zone Style dialog, set the contour plot type to either Flood or Both Lines and Flood. The area between adjacent contour levels is colored according the value of the contour variable, the color distribution method, the number of contour levels (banded distribution only), and the active color map. If you select the contour plot type Both Lines and Flood, the flooding is displayed along with the contour lines. (If the contour lines are multi-colored by the same contour group, you will not be able to see them against the contour flooding.)

Each object in a field plot, such as a zone layer for a particular zone, the mesh or contour layer for streamtraces or iso-surfaces, or any of the layers for slices, can have its own setting for coloring by selecting one of the contour groups or RGB coloring. See Section 10.2, “Contour Groups,” and Section 10.3, “RGB Coloring.”

In addition to the standard contour flooding, Tecplot supports two other types of flooded contour plots: average cell and primary value. Unlike the standard flooding, which floods the regions between adjacent contour levels, these options flood individual cells or elements. Coloring will be affected by the location (nodal or cell-centered) of the contour variable. The following table shows how Tecplot creates average cell and primary value contour plots:

Contour Variable Location	**Average Cell**	**Primary Value**
Nodal	Values at the nodes are averaged.	The value of the lowest indexed node is the cell is used. When plotting IJK-ordered, FE-brick or -tetra cells, each face is considered independently of the other faces. You may get different colors on the different faces of the same cell.
Cell-Centered	Cell-centered values are averaged to the nodes and the nodes are then averaged.	The cell-centered value is used directly. When plotting I-, J-, or K-planes in 3-D, the cell on the positive side of the plane supplies the value, except in the case of the last plane, where the cell on the negative side supplies the value.

10.1.2. Flood By Options

For any Contour Type except Lines, the flooding of a zone is set to one of the contour groups or to RGB coloring. By default, the first contour group, C1, is used for flooding. To choose a different contour group (C2, C3, or C4) or RGB coloring (RGB), click Flood By, then select the desired option from the drop-down. If you choose a contour group for which no variable has been specified, the Contour Details dialog in its condensed form appears. Click the arrow and choose a variable from the drop-down. See Section 10.2, “Contour Groups.” If you choose to color by RGB and no variables have been selected, the Select Variables dialog appears, allowing you to specify variables for R, G, and B. See Section 10.3, “RGB Coloring.”

10.1.3. Lines By Option

When the Contour Type is set to Lines or Both Flood and Lines, the Lines are positioned by one of the contour groups. By default, the contour levels of the first contour group, C1, are used for placing the lines. To choose a different contour group (C2, C3, or C4), click Lines By, then select the desired option from the drop-down. If you choose a contour group for which no variable has been specified, the Contour Details dialog in its condensed form appears. Click the arrow and choose a variable from the drop-down. See Section 10.2, “Contour Groups.”

10.1.4. Use Lighting Option

Lighting effects on 3-D plots may distort the surface color, depending on the orientation of the surface with respect to the light source. You may want to turn off the lighting effects when creating flooded contour plots in 3-D to assure the surface colors match those in the contour legend. The easiest way to do this for zones is via the Lighting Zone Effects check box on the sidebar. Another option is to keep Lighting active on the sidebar, but set Use Lighting to No for any selected zone.

10.2. Contour Groups

Although most settings for contour plots can be unique for each zone, some settings are linked by a contour group. Zone layers or other objects that reference the same group for an attribute show the same plot style for that attribute. Up to four different contour groups can be set. Each contour group has its own settings for the following attributes:

- The contour variable.
- Contour levels.
- Coloring options (banded or continuous color distribution, or flood cutoff).
- Control of band coloring, such zebra shading.
- The contour line mode.
- Contour labeling.
- The contour legend.

Each of these attributes can be set in the Contour Details dialog and is explained in the following sections. The Contour Details dialog in its condensed form is shown in Figure 10-4.

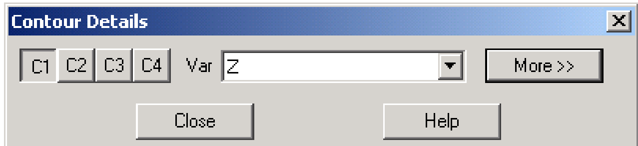


Figure 10-4. The Contour Details dialog in its condensed form.

10.2.1. Contour Variable Selection

You choose a contour variable from the Contour Details dialog. This dialog comes up automatically when you turn on the Contour zone layer for the first time in a frame, or can be accessed via the Contour option on the Plot menu. The Contour Details dialog is shown in Figure 10-4. Choose one of the data set's variables from the Var drop-down.

10.2.2. Contour Levels

A contour level is a value at which contour lines are drawn, or for banded contour flooding, the border between different colors of flooding. The range of contour levels is the interval between the minimum contour level and the maximum contour level.

You control contour levels using the Levels page of the Contour Details dialog. Call up this dialog by choosing Contour Levels option from the Field menu. From this dialog, you can perform any of the following tasks:

- Specify a new range or number of contour levels.
- Add levels to an existing set of contour levels.
- Remove selected contour levels.

The Levels page of the Contour Details dialog is shown in Figure 10-5.

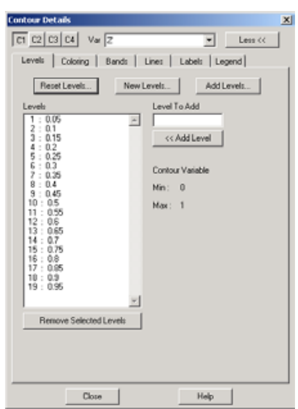


Figure 10-5. The Levels page of the Contour Details dialog.

10.2.2.1. New Contour Level Specification. You may specify a new set of contour levels via the Reset Levels or New Levels options on the Contour Details dialog. The Reset Levels dialog only asks you to supply an approximate number of levels to use. Tecplot estimates a starting point, end and increment which may use slightly more or less levels than you specified.

If you want more control over the exact values generated for contour levels, click New Levels. This calls up the Enter Contour Level Range dialog. The dialog is shown in Figure 10-6.

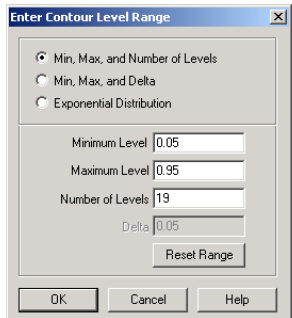


Figure 10-6. The Enter Contour Level Range dialog.

You can specify the range and number of levels in any of three ways:

- As a minimum and maximum level value, together with the number of levels to be distributed equally through the range. Choose this option (the default) by selecting the check box labeled Min, Max, and Number of Levels, then filling in the appropriate text fields.
- As a minimum and maximum level value, together with a delta, that is, the difference or change in contour value between two adjacent contour levels. You choose this option by selecting the check box labeled Min, Max, and Delta, then filling in the appropriate text fields. Tecplot generates contour levels beginning at the minimum level specified, then at intervals of Delta, until adding Delta would result in a level higher than the maximum level specified. Thus the highest actual contour level is usually somewhat less than the maximum level unless Delta is chosen precisely.
- As a minimum and maximum level value, together with the number of levels to be distributed exponentially through the range. You choose this option by selecting the check box labeled Exponential Distribution, then filling in the appropriate text fields.

In specifying a range, you can refer to the main Contour Levels dialog, which displays the minimum and maximum range of the contour variable. You can specify contour levels that extend beyond the range of the contour variable; any levels outside the range of the data are not displayed.

10.2.2.2. Contour Level Addition. You can add contour levels to an existing set of contours. You might do this to get a clearer idea of how the contour variable is varying in areas with few visible contour lines.

You can add new levels in any of three ways:

- Add a new range of contour levels to the existing set by clicking Add Levels on the Contour Levels dialog, then using the Enter Contour Level Range dialog as described in Section 10.2.2.1, "New Contour Level Specification."
- Enter a value in the Level To Add text field in the Levels page of the Contour Details dialog and then clicking Add Level.
- Choose  from the sidebar, then click at any location in the contour plot where you would like a new contour level. Tecplot adds a new contour level that goes through the specified point. By holding down the mouse button you can drag and interactively position the new contour level until you release the button. While using this tool you may also press the "-" key to switch to the Remove Contour tool.

10.2.2.3. Contour Level Removal. You can remove contour levels in either of two ways:

- In the Levels scrolled list in the Levels page of the Contour Details dialog, select one or more contour levels, then click Remove Selected Levels.
- Choose  from the sidebar, then click on any contour line in your contour plot. Tecplot deletes the specified contour level, or the nearest contour level to the specified point. While using this tool you may also press the "+" key to switch to the Add Contour tool.

10.2.2.4. Contour Level Adjustment. You can interactively adjust a contour level with the  tool from the sidebar. Hold down the Ctrl key; then click and drag the contour level you want to adjust. Move the contour to the desired location and release the mouse button. The value of the contour level will change as can be viewed in the Contour Levels dialog.

10.2.3. Contour Coloring

Although the color map is global, affecting all frames, there are some adjustments you can make that apply only a contour group in the current frame. These adjustments allow you to customize the look of a single contour plot without changing the global color map. Adjustments to the color map for individual frames are made from the Coloring page of the Contour Details dialog, shown in Figure 10-7.

- The color distribution method may be continuous or banded.
- The color map may be cut off such that coloring outside of a given range is not done at all.
- The color map can be reversed.
- The color map can contain multiple cycles.

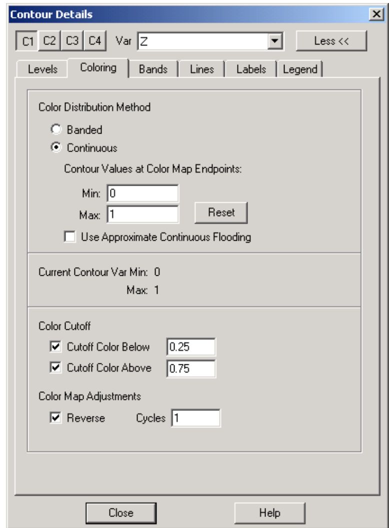


Figure 10-7. The Coloring page of the Contour Details dialog.

10.2.4. Color Distribution Methods

The color distribution method determines how to look up the colors in the global color map. There are two options; Banded and Continuous.

Banded color distribution is closely tied to the contour levels. A solid color is assigned for all values within the band between two levels. For contour flooding this means the area between two levels is filled with a single solid color. For multi-color scatter symbols or vectors this means all scatter symbols or vectors that have a contouring value between two contour levels are assigned the same color.

When banded coloring is in effect the following options are available:

- Color bands can be zebra shaded. This in effect colors every other band with a specific color (or no color at all).
- Specific contour bands can be assigned a unique basic color. This is useful for forcing a particular region to use blue, for example, to designate an area of water. You can define up to 16 color overrides.

The Bands page of the Contour Details dialog is shown in Figure 10-8.

Continuous color distribution assigns linearly varying colors to all multi-colored objects or contour flooded regions. You can vary the default assignment of colors by entering a Min or Max value for Color Map Endpoints. For example, if your contour variable has values ranging from -2 to ten, but you are only interested in the values less than two, setting Max to two determines that the entire color map is used for values from -2 to two.

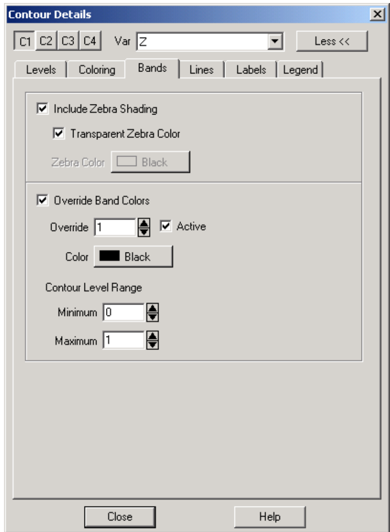


Figure 10-8. The Bands page of the Contour Details dialog.

The Use Approximate Continuous Flooding option, if selected, causes each cell to be flooded using the RGB values at each node, without considering values at adjacent nodes. When the transition from a color at one node to another node crosses over the boundary between control points in the color spectrum, approximate flooding may produce colors not in the spectrum. Leaving this option unchecked is slower but more accurate.

10.2.5. Color Cutoff

Color cutoff lets you specify a range within which contour flooding and multi-colored objects, such as scatter symbols, are displayed. For example, you may specify that only contour flooding in the range of -4.5 to 4.5 should be displayed; contour flooding outside this range is not plotted.

To use color cutoff:

1. Select the Contour tool from the sidebar, or choose Contour from the Plot menu. Go to the Coloring page of the Contour Details dialog, as shown in Figure 10-7.
2. To set a minimum color cutoff select the Cutoff Color Below check box and enter a value in the text field.
3. To set a maximum color cutoff select the Cutoff Color Above check box and enter a value in the text field.

A flooded contour plot before and after color cutoff has been applied is shown in Figure 10-9.

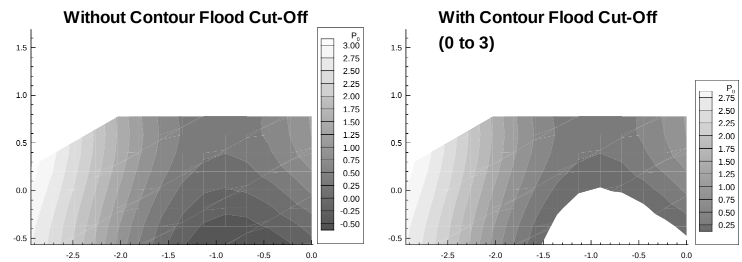


Figure 10-9. Flooded contour plot without and with flood cutoff.

10.2.6. Reversed Color Map

You can reverse the color map by selecting the check box at the bottom of the Contour Coloring Options dialog. Two plots, one with the color map going in the default direction, and one with the color map reversed, are shown in Figure 10-10.

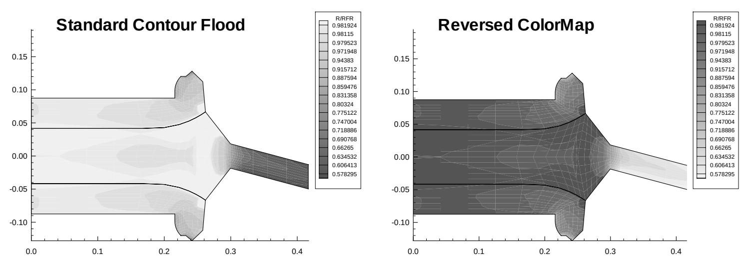


Figure 10-10. Flooded contour plot with a standard and a reversed color map.

10.2.7. Color Map Cycles

You may choose to cycle the color map. This is useful if you have data where there is a great deal of activity in multiple ranges of the contour variable and you want to cycle through all colors in each region. A plot with the color map cycled two times is shown in Figure 10-11.

10.2.8. Contour Bands

When Coloring Distribution for a group is set to Banded, the following options are available:

- Color bands can be zebra shaded. This in effect colors every other band with a specific color (or no color at all).
- Specific contour bands can be assigned a unique basic color. This is useful for forcing a particular region to use blue, for example, to designate an area of water. You can define up to 16 color overrides.

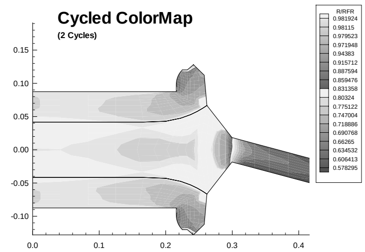


Figure 10-11. Flooded contour plot with the color map cycled two times.

For these selections, click the Bands tab on the Contour Details dialog. The Bands page of the dialog appears, as shown in Figure 10-8.

10.2.9. Contour Lines

You can control the color and thickness of the contour lines on a zone-by-zone basis, using the procedures of Section 7.3, "Field Plot Modification." You can also specify the line pattern and pattern length on a zone-by-zone basis, but whether those settings are used depends on the chosen contour group's line mode in that frame. The contour line settings determine how contour lines are drawn for all zones in the current frame's data set.

To specify the contour line mode:

1. On the Contours page of the Zone Style dialog, set Lines By to the desired contour group
2. Open the Contour Details dialog by clicking Contours on the sidebar, or by selecting Contours from the Plot menu. Choose the appropriate contour group, C1, C2, C3 or C4.
3. Go to the Lines page of the Contour Details dialog, as shown in Figure 10-12.

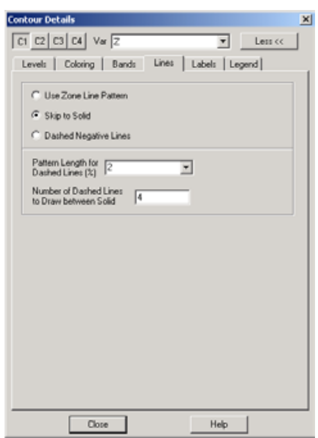


Figure 10-12. The Lines page of the Contour Details dialog.

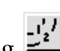
4. Choose one of the following options:
   - **Use Zone Line Pattern:** For each zone, draw the contour lines using the line pattern and pattern length specified in the Contour page of the Zone Style dialog.
   - **Skip to Solid:** Draw *n* dashed lines between each pair of solid lines, where *n* is an integer you enter in the text field Number of Dashed Lines to Draw between Solid Lines.
   - **Dashed Negative Lines:** Draw lines of positive contour variable value as solid lines and lines of negative contour variable value as dashed lines.
5. If you choose Skip to Solid or Dashed Negative Lines, specify a pattern length for the dashed lines.
6. If you choose Skip to Solid, enter the number of dashed lines to draw between solid lines.

10.2.10. Contour Labels

Contour labels are labels that identify particular contour levels either by number or by value. You can place contour labels interactively, or have Tecplot create them for you automatically. You can also have Tecplot create a set of contour labels automatically, then interactively add contour labels to this saved set. You control contour labels with the Contour Labels dialog under the Field menu, and with the Add Contour Label mouse mode tool from the sidebar.

To add contour labels to your plot, you can use either of the following procedures. The first describes using the Add Contour Label tool; the second describes using the Contour Labels dialog to have Tecplot automatically generate the contour labels. If you are using the Add Contour Label tool, you still use the Contour Labels dialog to specify whether contour level numbers or values are used and to specify formatting and alignment options for the labels. Contour labels show the value or number of the nearest contour level. In other words, if you add a label between two contour lines, the label will show the value or number of the nearest line. This can be misleading when contour labels are far away from contour lines.

To add contour labels interactively using the Add Contour Label tool:

1. Create a plot with an active Contour zone layer, where at least one zone has a contour type of Lines or Both Lines and Flood, with Lines By set to the appropriate contour group.
2. From the sidebar, choose the Add Contour Label tool by clicking .
3. In the workspace, click on the location at which you want a contour label to appear. By default, Tecplot uses contour values and aligns the labels with the contour line. You can modify the following options using the Contour Labels dialog.
   - The label color.
   - The label font.
   - The label size.
   - The numeric format. The available formats are described in Section 17.5.2, “Tick Mark Labels.”

- Whether there is a color fill behind the labels, and if so, its color and the margin of fill around the labels.
- Label alignment. If the Align Next User-Positioned Label is selected, the next label placed is aligned with the contour line. Otherwise, the label is written with normal, upright text.

To have Tecplot generate contour labels automatically with each redraw:

1. Create a plot with an active Contour zone layer, where at least one zone has the contour type of Lines or Both Lines and Flood, with Lines By set to the appropriate contour group.
2. On the Contour Details dialog, choose the right contour group, then select Labels. The Labels page of the dialog appears, as shown in Figure 10-13.

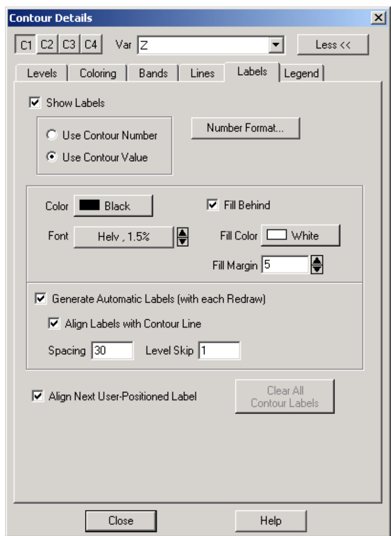


Figure 10-13. The Labels page of the Contour Details dialog.

3. Select the check box labeled Show Labels.
4. Choose whether contour numbers or contour values are used in the labels by selecting the appropriate option.
5. If you want the labels aligned with the contour lines, select the Align Labels check box.
6. Specify the spacing between labels in frame units.
7. Specify the number of levels to skip in the Level Skip text field. The default of one labels all contour labels.
8. Select the Generate Automatic Labels (with each Redraw) check box.

At each Redraw, Tecplot creates a new set of contour labels. At any time, you can deselect the Generate Automatic Labels (with each Redraw) check box, and Tecplot retains the last set of labels generated. Thus you can create labels for several redraws, then save a set to which you can interactively add more labels.

When Generate Automatic Labels is deselected, you can click Clear All Contour Labels to erase the current set of contour labels.

10.2.11. Contour Legend Creation

A contour legend is a key to the contour flooding in your flooded contour plot. It relates the displayed colors to the actual values of the contour variable. The contour legend can also relate contour level numbers to contour values. This is useful if you are using contour number labels. To create a contour legend, select the Legend page of the Contour Details dialog for the appropriate contour group. The Legend page of the Contour Details dialog is shown in Figure 10-14.

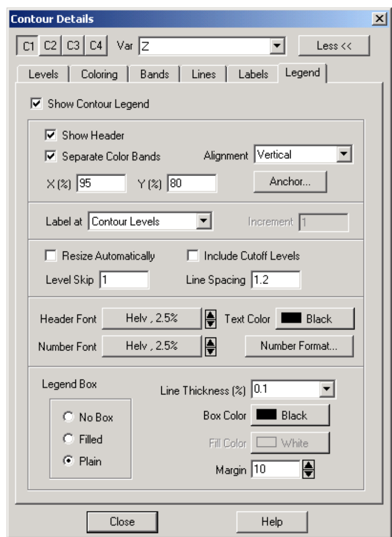


Figure 10-14. The Legend page of the Contour Details dialog.

The following options are available:

- Show Contour Legend.
- Show Header to include the name of the contour variable.
- Separate Color Bands.
- Alignment: Vertical or Horizontal.
- **Position:** X (%) and Y (%) as percentages of the frame width and height. (You can also move the legend interactively.)

- **Anchor:** Specify which part of the legend is anchored in the selected position. The Anchor Alignment dialog is shown in Figure 10-15.

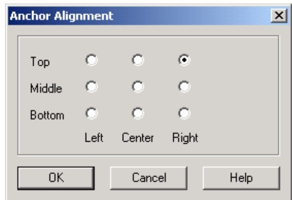


Figure 10-15. The Anchor Alignment dialog.

- **Label Placement:** If you have chosen Continuous Color Distribution on the Coloring page of the Contour Details dialog, you have three options for placement of labels on the legend:
  - **Label at Contour Levels:** This option places one label for each contour level. See Section 10.2.2, “Contour Levels.”
  - **Label at Specified Increment:** Enter a value in the Increment text field when selected.
  - **Label at Color map Divisions:** Places one label for each control point on the global color map. See Section 10.4, “Global Color Map.”
- **Resize Automatically:** Legend is roughly 50 percent of the frame width (for horizontal legends) or height (for verticals).
- **Include Cutoff Levels:** Color bands and labels for levels affected by Color Cutoff are shown in the legend. See Section 10.2.5, “Color Cutoff.”
- **Line Spacing:** Adjusts the height of individual color bands on the legend.
- Font and height for the legend header or the legend labels.
- **Color:** Affects all text in the legend.
- **Number Format:** The available options are the same as for axis tick mark labels; see Section 16.5.3 Tick Mark Label Formats.
- **Legend Box (No Box, Filled, Plain):** If you choose Filled or Plain, format the box using the following controls:
  - **Line Thickness:** Specify the line thickness as a percentage of frame height.
  - **Box Color:** Choose a color for the legend box outline.
  - **Fill Color (Filled only):** Choose a color for the legend box fill.
  - **Margin:** Specify the margin between the legend text and legend box as a percentage of the text height.

10.3. RGB Coloring

RGB coloring occurs when Red, Green, and Blue values are supplied at each vertex. It may be used to create special flooding such as for Oil/Water/Gas plots. RGB coloring may be used for each field plot object in Tecplot: zone layers, the mesh or contour layer for streamtraces or iso-surfaces, or any of the layers for slices. This affects multi-coloring for that object as well as any contour flooding. With RGB coloring, multi-colored objects such as vectors or scatter symbols have their color determined based on the RGB components of the field variables at their location. Multi-colored mesh and contour lines use the average value across the mesh line.

Note: *RGB Coloring Limitation in Exported Vector-Based Files.* Vector-based export files such as WMF do not show continuous RGB flooding. Objects that use RGB flooding are reduced to contain average cell flooding where each cell is flooded a solid color based on the averages of the RGB values at each vertex. The user is warned before such output is generated. The RGB coloring legend uses a resolution defined by a configuration file entry. The default resolution is usually be sufficient.

10.3.1. RGB Variable Assignment

The first time RGB coloring is assigned in a plot, the Select Variables dialog appears. Assign one variable for R, one for G, and one for B. If you have only two of these variables in your data set, use the RGB Coloring Options dialog to set them.

10.3.2. RGB Coloring Options

If your data has only two RGB variables, or if the sum of the variables is not normalized, you can adjust the settings using the RGB Coloring Options dialog. From the Plot menu, select RGB Coloring, then Variables/Range. The RGB Coloring Options dialog appears, as shown in Figure 10-16.

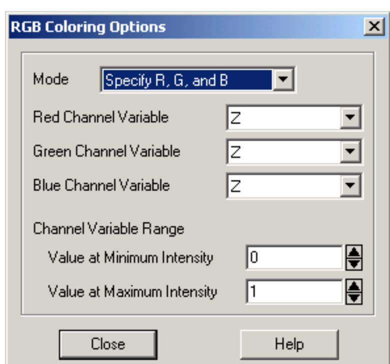


Figure 10-16. The RGB Coloring Options dialog.

10.3.2.1. RGB Mode. You can choose to:

- Assign variables to R, G, and B.
- Assign variables to two of R,G,B entries and leave the third to be calculated, where *f(R)+f(G)+f(B)=1.0* (assuming *f()* is a function that maps R,G,B values into [0,1.0]).

10.3.2.2. Channel Variables. Assign the variables which supply the values for the color components, as specified in the RGB Mode. For example, if the Mode says G is calculated, only variables for R and B are assigned.

10.3.2.3. Channel Variable Range. By default, it is assumed that the minimum value for any of the Channel Variables is zero, the maximum is one, and the sum of the three variables is one at every point. If the sum is not normalized, you can set a new minimum and maximum. For example, if your variables sum to 100 at every point, you can enter 100 in the field for Value at Maximum Intensity.

10.3.3. RGB Legend

To create an RGB legend, perform the following steps:

1. From the Plot menu, select RGB Coloring, then Legend. The RGB Legend dialog appears.
2. Select Show RGB Coloring Legend.
3. Specify the position of the anchor point with X (%) and Y (%) as percentages of the frame width and height. (You can also move the legend interactively.)
4. Specify the height of the legend in frame units.
5. Choose the Orientation of the Legend from the drop-down. The first channel listed is shown on the lower left corner, the second on the lower right, and the third at the top.
6. To specify which part of the legend is anchored to the selected position, click Anchor. The Anchor Alignment dialog appears.
7. To show labels, click Show Text Labels.
8. Specify the text color.
9. Choose the text font and height.
10. Each channel can be labeled by the name of the assigned variable, or by text you enter. To choose a new label for a channel, click Specify, and type in the alternate label. When a channel has been calculated (no variable assigned), no label is shown unless the user enters text.
11. Select which kind of box you want drawn around the legend (No Box, Filled, or Plain). If you choose Filled or Plain, format the box using the following controls:
    - **Line Thickness:** Specify the line thickness as a percentage of frame height.
    - **Box Color:** Choose a color for the legend box outline.
    - **Fill Color (Filled only):** Choose a color for the legend box fill.
    - Margin: Specify the margin between the legend text and legend box as a percentage of the text height.

Notes: When two channels are selected (and one is calculated), the legend is a simple ternary plot with each corner labeled with variable name or a user-specified text string. It is possible to have a plot that may contain colors not displayed in the legend. This occurs if you supply all three channel variables, and for one or more points, the sum of the variables values does not equal 1.0 or the specified Value at Maximum Intensity. See Section 10.3.2, “RGB Coloring Options.”

10.4. Global Color Map

The colors used in flooded contour plots are determined by the global color map, controlled in the Workspace menu, or by frame-specific color options controlled in the Plot menu. This section discusses the global color map.

By default, Tecplot uses a color map called Small Rainbow, which is a rainbow of colors from blue to cyan to green to yellow to red. This default color map is called Small Rainbow to distinguish it from another color map, Large Rainbow, which adds purple and white beyond the red. The color map is used by all frames; if you change the color map to modify the look of one frame, all frames with contour flooding or any form of multi-colored are modified as well.

To choose a color map:

1. From the Workspace menu, choose Color Map. The ColorMap dialog appears, as shown in Figure 10-17.

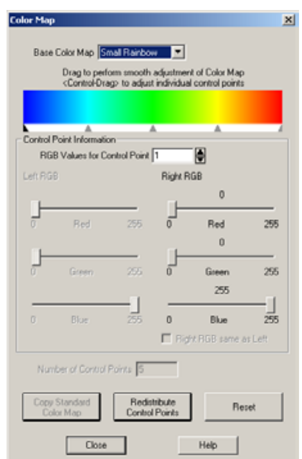


Figure 10-17. The Color Map dialog.

2. From the Base Color Map drop-down, choose one of the following color maps:
   - **Small Rainbow:** Five color spectrum from blue to cyan to green to yellow to red.

- **Large Rainbow:** Seven color spectrum from blue to cyan to green to yellow to red to purple to white.
- **Modern:** Seven color spectrum; within each color band colors change in intensity from dark to light.
- **GrayScale:** Color spectrum from black to white.
- **Wild:** random Color spectrum. Wild is different each time you select it.
- **Two Color:** A two-color spectrum.
- **User-Defined:** A customizable version of one of the first four options above. You can add or delete control points, as well as change RGB values for each control point. See Section 10.4.2, "User-Defined Color Map," for details on modifying a User Defined color map.
- **Raw User-Defined:** A customizable version of one of the first four options above. To customize the color map, however, you must save your Raw User-Defined map to a file using the Copy Color Map to File option in the Workspace menu. See Section 10.4.3, "Raw User-Defined Color Map," for details on modifying a Raw User Defined color map.

You can modify any color map, except the Raw User-Defined color map, using the controls in the Color Map dialog.

10.4.1. Standard Color Map Adjustments

You can modify the standard color maps either by altering the position of the color map control points or by modifying the RGB values of the control points. Altering the position of the control points allows you to alter the proportions of colors in the spectrum. Modifying the RGB values of the control points changes the spectrum itself. All of the control points except the first and last are two-sided; they have both a left- and a right-RGB value. This allows you to define color maps such as Modern, which has sharp demarcations between color bands. To define smoothly varying color maps, select the Right RGB Same as Left check box for each two-sided control point. Except for the Wild and Modern color maps (and, if suitably modified, the Raw User-Defined color map), all color maps vary smoothly between control points.

To alter the position of control points:

1. Move the pointer into the color bar in the Color Map dialog.
2. Drag a point on the color bar. The entire color bar, including the control points, is distorted relative to the point where the drag started. If the point is dragged to the right, control points to the left of the drag point become more widely spaced, while control points to the right of the drag point are moved closer together. An XORed line shows the point being dragged. To move just a single control point, which does not affect the position of other control points, Ctrl-drag on the color bar to select the nearest control point, then drag as before. To return the control points to their original positions, click Redistribute Control Points.

To modify the RGB values for a control point:

1. Select the control point for which you want to modify RGB values. You can do this either by entering a value (or using the up and down arrows to choose a value) in the field labeled RGB Values for Control Point, or by Ctrl-clicking on the control point in the color bar.
2. Use the three sliders under the heading Left RGB to specify the left-RGB colors for the control point. (These sliders are disabled for the left-most control point.)
3. Use the three sliders under the heading Right RGB to specify the right-RGB colors for the control point, or select the check box Right RGB Same as Left to use the same values you just set for the left-RGB colors. (The Right RGB sliders are disabled for the right-most control point, or if the check box Right RGB Same as Left is selected.)

To reset the RGB values to their original values (and also reposition the control points in their original locations), click Reset.

10.4.2. User-Defined Color Map

You can modify a User-Defined color map in the same way you modify a standard color map, by altering the position of control points or modifying the RGB values of the control points, but you have the added ability to modify the number of control points.

To change the number of control points in a user defined color map:

1. Choose the User Defined color map. By default, the User-Defined color map has the same settings as the standard Small Rainbow color map.
2. Enter a value from two through nine in the Number of Control Points field.

If you choose a number greater than the current value, the new control points are added to the right of the existing control points; click Redistribute Control Points to see all your control points. If you choose a number less than the current value, the control points are removed rightmost first.

10.4.3. Raw User-Defined Color Map

You can modify a Raw User-Defined color map only by saving it to a file and then editing the resulting file, which consists of RGB triplets for every color in the spectrum. You can modify these RGB triplets as you want, using any ASCII text editor. In most cases, you want to use the User-Defined color map rather than the Raw User-Defined, since you cannot edit the raw color map in Tecplot.

10.4.4. Color Map Files

The position of color map control points and their RGB values can be stored in color map files; you can then edit the color map files to modify either the position or RGB values of the control points.

To create a color map file:

1. From the menu, choose Copy Color Map to File. The Write Color Map dialog appears.

2. Specify a file name for the color map.
3. Click OK.

To use the saved color map in a new plot, choose Paste Color Map from File on the workspace menu. The color map file is a Tecplot macro file with a limited set of commands (only **$!COLORMAP** and **$!COLORMAPCONTROL** commands are allowed). The first part of the color map file generated from the Large Rainbow color map is shown below:

```
#!MC 1000
$!COLORMAP
 CONTOURCOLORMAP = LGRAINBOW
$!COLORMAPCONTROL RESETTOFACTORY
$!COLORMAP
 LGRAINBOW
    {
    CONTROLPOINT 1
      {
      COLORMAPFRACTION = 0
      LEADRGB
        {
        R = 0
        G = 0
        B = 255
        }



      TRAILRGB
        {
        R = 0
        G = 0
        B = 255
        }
      }
. . .
```

CHAPTER 11 Vector Plots

Vector plots are field plots of the direction and or magnitude of vector quantities. The vector quantities can be displacements, velocities, forces, or anything else that can be represented by vectors. You create vector plots by activating the Vector layer in the Tecplot sidebar, and, if you have not done so already, specifying two or three vector component variables.

One important use of vector components in Tecplot is to allow you to compute the trajectories of massless particles in a steady-state velocity field. These trajectories are called streamtraces. Streamtraces do not need to be created in vector plots, even though they require vector components. For this reason, they are discussed in a separate chapter. See Chapter 12, "Streamtraces."

11.1. Vector Plot Creation

When you select the Vector check box in the Tecplot sidebar, Tecplot checks to see whether vector components have been assigned for the current data set in the current plot type, whether 2D or 3D Cartesian. If you have not assigned vector components, the Select Variables dialog appears. In 2D Cartesian plots, the Select Variables dialog allows you to choose two vector components, as shown in Figure 11-1.

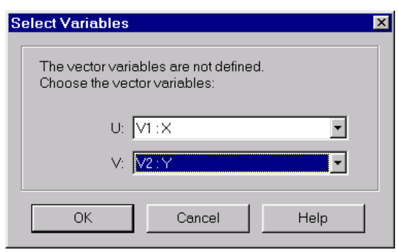


Figure 11-1. The Select Variables dialog for assigning 2-D vector components.

Choose variables by selecting the desired U-, V-, and in 3D Cartesian plots, W-variables from their respective drop-downs. You may select any of the current data set's variables as any component. You can change the component variables at any time by choosing Vector Variables from the Vector sub-menu of the Plot menu.

Once you have selected the Vector check box and have chosen your vector components your vector plot will appear as shown in Figure 11-2 for the cylinder data. If vectors are not visible, see 11.5, "Vector Length."

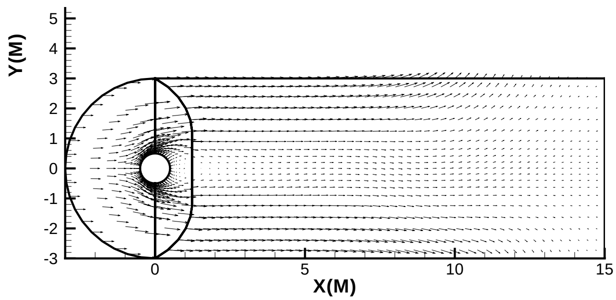


Figure 11-2. A vector plot of the cylinder data.

11.2. Vector Plot Modification

You can modify your vector plot attributes using either the Vector page of the Zone Style dialog or the Quick Edit dialog. You can control any of the following attributes from the Vector page of the Zone Style dialog (Figure 11-3):

- Which zones are active. See Section 7.3.2, “Zone Display.”
- Whether the vectors are visible for each active zone. See Section 7.3.3, “Zone Layer Display.”
- The vector plot type. See Section 11.3, “Vector Types.”
- The arrowhead style. See Section 11.4.1, “Arrowhead Style.”
- Whether 3-D vectors are tangent vectors or regular vectors. See Section 11.7.1, “Tangent Vectors.”
- The vector color. See Section 7.3.4, “Color Choice.”
- The vector line pattern. See Section 7.3.5, “Line Pattern.”
- The vector line pattern length. See Section 7.3.6, “Pattern Length.”
- The vector line thickness. See Section 7.3.7, “Line Thickness.”
- The vector spacing. See Section 11.6, “Vector Spacing.”

The following attributes are assigned on a frame-by-frame basis, rather than zone-by-zone:

- Vector lengths. See Section 11.5, “Vector Length.”
- Arrowhead angle and size. See Section 11.4, “Vector Arrowheads.”
- The optional reference vector. See Section 11.8, “Reference Vectors.”

11.3. Vector Types

Tecplot allows you to plot vectors of four different types:

- **Tail at point:** Vectors are drawn with the tail of the vector positioned at the data point, which for ordered data is a corner of the cell.
- **Head at point:** Vectors are drawn with the head of the vector positioned at the data point.

- **Anchor at midpoint:** Vectors are drawn with the midpoint of the vector positioned at the data point.
- **Head only:** A vector arrowhead (but no tail) is drawn with the head of the arrow centered at the data point.

By default, Tecplot uses the “Tail at Point” type. Figure 11-4 shows examples of each of the vector plot types.

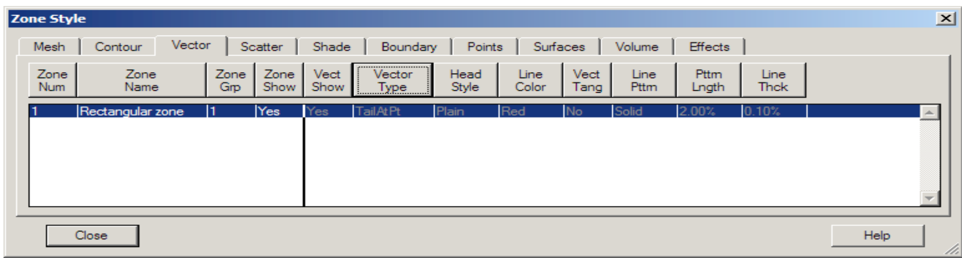


Figure 11-3. The Vector page of the Zone Style dialog.

To modify the vector plot type:

1. Click the Zone Style button in the sidebar. The Zone Style dialog appears.
2. Display the Vector page, as shown in Figure 11-3.

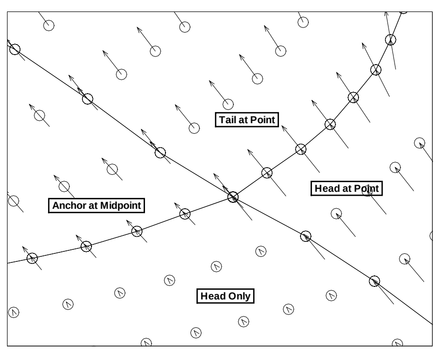


Figure 11-4. Vector plot types.

3. Select the zone or zones for which you want to modify the plot type.
4. Click Vector Type. A drop-down appears listing the available types.
5. Click on the desired type.

To modify the vector type from the Quick Edit dialog:

1. On the sidebar, click Quick Edit, or select Quick Edit... from the Edit menu. The Quick Edit dialog appears.
2. In the workspace, use the Selector tool to select the zone or zones for which you want to modify the plot type.
3. On the Quick Edit dialog, click on the appropriate button, as follows:
   -  Tail at point.
   -  Head at point.
   -  Anchor at midpoint.
   -  Head only.

11.4. Vector Arrowheads

Tecplot allows you a good deal of control over your vector arrowheads. You can control the style of the arrowhead, its size, and the angle it makes with the vector. The style of the arrowhead can change from zone to zone; the size and angle are global attributes affecting vectors in all zones.This section explains how to perform these tasks.

11.4.1. Arrowhead Style

You can assign arrowhead styles on a zone-by-zone basis. Tecplot arrowheads come in three styles:

- **Plain:** Line segments drawn from the head of the vector.
- **Filled:** Filled isosceles triangles with apex at the head of the vector.
- **Hollow:** Hollow isosceles triangles with apex at the head of the vector.

By default, Tecplot draws plain arrowheads. Figure 11-5 shows an example with all three arrowhead styles.

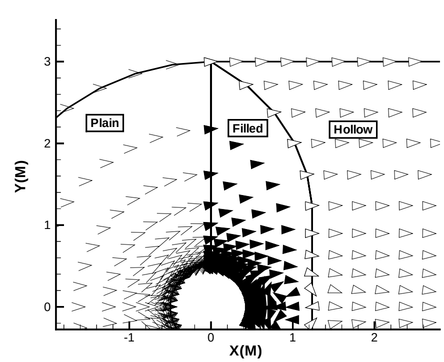


Figure 11-5. Examples of Tecplot's three arrowhead styles.

To modify the arrowhead style for a zone or zones:

1. Click the Zone Style button in the sidebar. The Zone Style dialog appears.
2. Display the Vector page.
3. Select the zone or zones for which you want to modify the arrowhead style.
4. Click Head Style. A drop-down appears containing the available choices.
5. Click on the desired arrowhead style.

or

1. In the workspace, use the Selector tool to select the zone or zones for which you want to modify the arrowhead style.
2. On the sidebar, click Quick Edit, or select Quick Edit... from the Edit menu. The Quick Edit dialog appears.
3. In the Quick Edit dialog, click on the button for the desired arrowhead style, as follows:
   -  Plain arrowhead style.
   -  Filled arrowhead style.
   -  Hollow arrowhead style.

11.4.2. Arrowhead Size

You can specify arrowhead sizes as either a fraction of the vector length or in frame units (that is, as a percentage of the frame height). Arrowhead size is a global attribute; it applies to all arrowheads in all zones in the current frame. By default, Tecplot specifies size as a fraction of the vector length.

To modify the arrowhead size:

1. Choose Arrowheads from the Vector sub-menu of the Plot menu. The Vector Arrowheads dialog appears as shown in Figure 11-6.

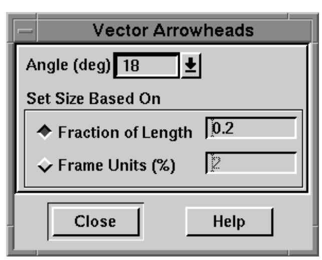


Figure 11-6. The Vector Arrowheads dialog.

2. Select one of the two option buttons in the box labeled Set Size Based On: either Fraction of Length or Frame Units (%).

3. Enter a value in the text field to the right of the selected option button. Use a fraction (a decimal value from zero to one) in the Fraction of Length text field. Use a percentage (a decimal value from zero to 100) in the Frame Units (%) field. Entering a value of zero in either field effectively “turns off” the vector arrowhead in the current frame.

11.4.3. Arrowhead Angle

Arrowhead angle is another global attribute; you assign one angle for all vector arrowheads in all zones in the current frame. The arrowhead angle is the angle (in degrees) that one side of the arrowhead makes with the vector; thus, the apex angle is twice the arrowhead angle. For example, to create hollow equilateral triangles as arrowheads, specify an arrowhead angle of 30 globally in conjunction with an arrowhead style of hollow for all zones.

To specify the arrowhead angle:

1. Choose Vector Arrowheads from the Vector sub-menu of the Plot menu. The Vector Arrowheads dialog appears.
2. In the field labeled Angle (deg), either enter a value from 1 to 90, or choose a value from the drop-down, indicated by the down-arrow button.

11.5. Vector Length

You can specify the length of vectors in any of three different ways. In the first two, the length of any given vector is proportional to the vector magnitude, while in the third, all vectors have the same length. The difference between the first two methods is the units used to specify the relative size of the vectors, either grid units or screen centimeters. Vector length is a global attribute — it applies to all zones in the current frame.

To specify the vector length:

1. Choose Length from the Vector sub-menu of the Plot menu. The Vector Length dialog appears as shown in Figure 11-7.

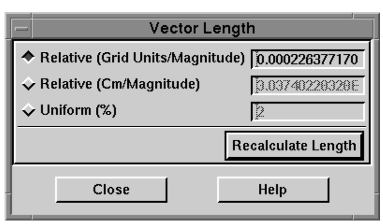


Figure 11-7. The Vector Length dialog.

2. Select one of the three option buttons:
   - **Relative (Grid Units/Magnitude):** specify the vector length as the number of grid units per unit of vector magnitude.

- **Relative (Cm/Magnitude):** specify the vector length as the number of centimeters per unit of vector magnitude.
- **Uniform (%):** specify the vector length as a percentage of frame height.

3. Enter a value in the text field to the right of the selected option button. For either of the “Relative” options, the value you specify is a scale factor which is multiplied by the vector magnitude to determine the length of the vector.

By default, Tecplot calculates a reasonable default based on the size of the longest vector. You can have Tecplot recalculate this default length by clicking Recalculate Length.

11.6. Vector Spacing

You can draw every vector or specify a skip factor that lets you plot only every *n*th vector (in I-, J-, or K- coordinates). Spacing the drawn vectors is useful in situations where you have an extensive vector field, and plotting all the vectors makes the general flow of the vector field difficult to discern. For example, Figure 11-8 shows the cylinder data with every other vector shown in the I direction and every third vector shown in the J direction. Compare this to the vector plot shown in Figure 11-2.

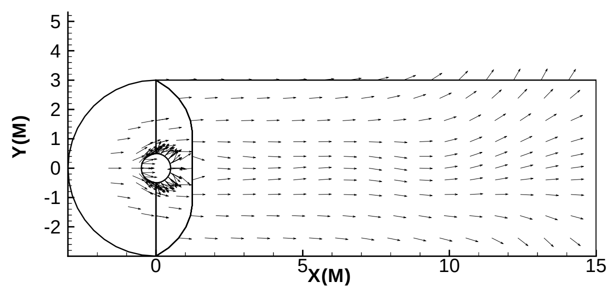


Figure 11-8. A vector plot with Index Skip specified.

To specify the spacing of points for vectors (this also affects point spacing for scatter symbols):

1. Click the Zone Style button in the sidebar. The Zone Style dialog appears.
2. Display the Points page.
3. Select the zone or zones for which you want to specify the vector spacing.
4. Click Index Skip. A drop-down appears with the options No Skip and Enter Skip. (No Skip is the default.)
5. Click on the desired option. If you click on Enter Skip, the Enter Index Skipping dialog appears, as shown in Figure 11-9 below.
6. (Enter Skip only) Enter the desired values of I-Skip, J-Skip, and K-Skip. Figure 11-8 was created with the following settings: *I-Skip=2*, *J-Skip=3*, *K-Skip=1*.

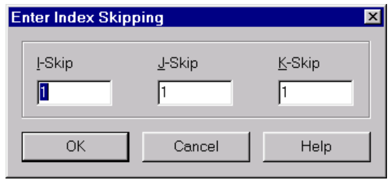


Figure 11-9. The Enter Index Skipping dialog.

For irregular and finite-element data, only the I-Skip has an effect, skipping through nodes in the order they are listed in the data file.

11.7. Three-Dimensional Vector Plots

To create a 3-D vector plot, simply read a data set into Tecplot, choose the 3D Cartesian plot type, and select the Vector check box in the Tecplot sidebar. Three-dimensional vector plots require three vector components, U, V, and W, for the three axes X, Y, and Z. The Select Variables dialog appears asking you to specify vector variables.

11.7.1. Tangent Vectors

In 3D Cartesian plots, Tecplot allows you to display 3-D surface tangent vectors. These are 3-D vectors that have been projected onto the 3-D surface; in other words, the component of the 3-D vectors normal to the surface is removed, leaving only the component parallel to the surface. Figure 11-10 shows how tangent vectors compare to regular vectors.

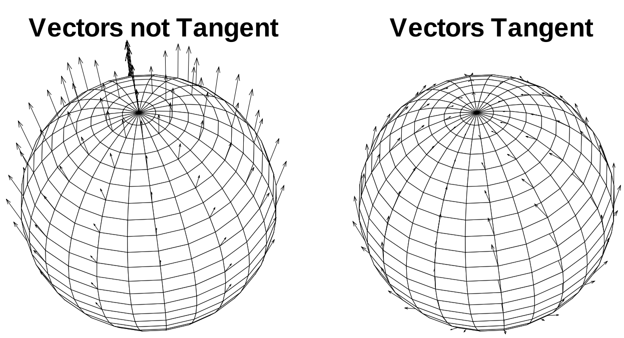


Figure 11-10. Regular vectors compared to tangent vectors.

To select tangent vectors for a zone:

1. Click the Zone Style button in the sidebar. The Zone Style dialog appears.
2. Display the Vector page.
3. Select the zone or zones for which you want to draw tangent vectors.

4. Click Vect Tang. A drop-down appears with the choices Yes and No.
5. Click Yes.

Tangent vectors are drawn on 3-D surfaces only where it is possible to determine a vector normal to the surface. A plot where multiple surfaces intersect each other using common nodes is a case where tangent vectors are not drawn because there is more than one normal to choose from. An example of this would be a volume IJK-ordered zone where both the I- and J-planes are plotted. If tangent vectors cannot be drawn then regular vectors are plotted instead.

11.7.2. Three-Dimensional Vector Lengths

Since 3-D vectors are plotted in the plane of the screen, a 3-D vector's length will depend on both the vector length settings and the orientation of the vector. The length may be distorted even further if the vector length setting is Relative and the 3-D projection is Perspective.

11.8. Reference Vectors

A reference vector is a vector of specified magnitude placed on the plot as a measure against all other vectors. You can specify whether to show a reference vector, and if so, its color, orientation, line thickness, magnitude, and position.

To display a reference vector:

1. From the Vector sub-menu of the Plot menu, choose Reference Vector. The Reference Vector dialog appears as shown in Figure 11-11.

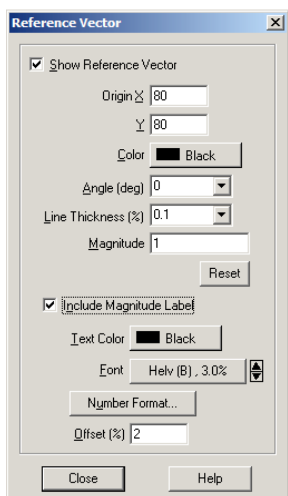


Figure 11-11. The Reference Vector dialog.

2. Select the Show Reference Vector check box.

3. Modify any of the following options, as desired:
   - **Origin (%):** Enter the coordinates of the starting point of the reference vector, as a percentage of the frame width (X) and frame height (Y).
   - **Color**: Choose a color from the Select Color dialog. Multi-color and RGB coloring are not available.
   - **Angle (deg):** Enter the orientation of the vector in degrees from horizontal, or choose a value from the drop-down.
   - **Line Thickness (%):** Enter the desired line thickness or choose a value from the drop-down.
   - **Magnitude:** Enter the magnitude of the reference vector. The units correspond to those of the vector components.
4. If you want the magnitude of the reference vector labeled, select the Include Magnitude Label toggle and modify any of the following options:
   - **Text Color:** Choose a color from the Select Color dialog. Multi-color and RGB coloring are not available.
   - **Font:** Click the button to choose the font typeface and size from the Select Font dialog, or lick the up and down arrows to adjust the size alone.
   - **Number Format:** Click the button to specify how the number will be formatted. See 17.5.2.2. "Tick Mark Label Formats" on page 314 for a discussion of this dialog.
   - **Offset:** Choose the spacing between the label and the reference vector as a percentage of frame height.
5. Click OK to close the dialog.

Figure 11-12 shows a plot with a reference vector.

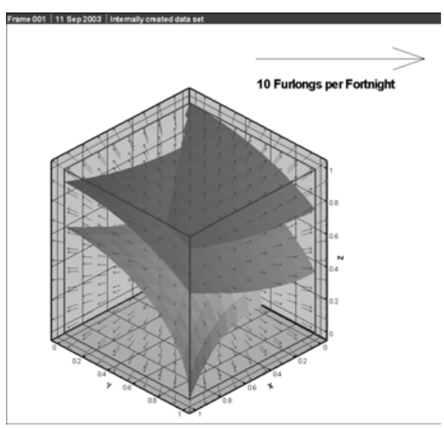


Figure 11-12. Vector plot with reference vector.

CHAPTER 12 Streamtraces

A streamtrace is the path traced by a massless particle placed at an arbitrary location in a steady-state vector field. Tecplot's streamtrace features may be used to illustrate the nature of the vector field flow in a particular region of the plot.

Because streamtraces are dependent upon a vector field, you must define vector components before creating streamtraces in Tecplot. You do not, however, have to activate the Vector zone layer. You may place streamtraces on any of Tecplot's field layers, or even on bare axes.

There are two main categories of streamtraces:

- Surface line streamtraces, or streamlines.
- Volume streamtraces.

Surface streamtraces are confined to the surface on which they are placed. They can only be placed in zones displayed as a 2- or 3-D surface. If you try to place streamlines in a zone displayed as a 3-D volume, an error dialog appears, and no streamlines are drawn.

Volume streamtraces can be created only in 3-D volume zones (IJK-ordered or FE-volume zones). Volume streamtraces themselves fall into three categories:

- Volume Lines, or volume streamlines.
- Volume Ribbons, or streamribbons.
- Volume Rods, or streamrods.

You can add streamtraces to your plot either singly or in a rake, which is a set of streamtraces with starting positions along a defined line. Once the rake is added, the individual streamtraces of the rake are identical to singly placed streamtraces.

12.1. Surface Streamlines

Surface streamlines include all 2-D streamtraces and 3-D surface streamtraces, which are confined to the surface on which they are placed. You can place streamlines one at a time, or in groups called rakes. A streamline rake is a set of streamlines with starting points along a given line.

You can place streamlines and streamline rakes using either the mouse or the Streamtrace Details dialog. The mouse enables you to place streamtraces quickly, but the Streamtrace Details dialog gives you precise control over the starting points.

To create a streamline or streamline rake using the mouse:

1. On the sidebar, choose the Place Streamtrace tool, represented by .
2. If you have not already assigned vector components you will be prompted for them.
3. Move the pointer into the workspace. The pointer changes to a cross-hair.
4. To place a single streamline, click at the desired starting point for the streamline, or Ctrl-click to begin the streamline at the data point nearest to the cross-hair.
5. To create a rake of streamlines, click-and-drag from one end point of the desired rake starting line to the other, then release.

Before placing streamlines, you can change the streamtrace direction or the number of streamtraces per rake using the Streamtrace Details dialog as discussed below.

To create a streamline or rake of streamlines using the Streamtrace Details dialog:

1. From the Plot menu, choose Streamtraces. If you have not yet assigned vector components for the current plot type, the Vector Variables dialog appears for you to assign them. Otherwise, the Position page of the Streamtrace Details dialog appears, as shown in Figure 12-1.

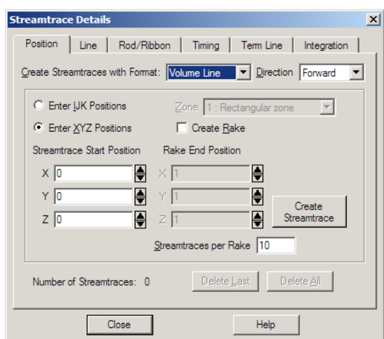


Figure 12-1. The Streamtrace Placement dialog.

2. Choose a direction for the streamline integration from the Direction drop-down:
   - **Forward:** The streamline is calculated downstream, that is, in the direction of the flow.
   - **Backward:** The streamline is calculated upstream, that is, against the flow.
   - **Both:** Both the forward and backward streamlines are calculated.

   No matter which direction is chosen for the integration, the arrowheads still point in the forward, “downstream” direction.

3. In the remainder of the Position page, select either the Use IJK or Use XYZ option. (Use XYZ is the default). If you select Use IJK, also pick a zone from the drop-down labeled Zone.

   If you select Use IJK, the fields under Start Position and Rake End Position are labeled I, J, and K. Otherwise, they are labeled X, Y, and Z.
4. Enter the starting position by specifying either a set of IJK-indices or XYZ-coordinates. For finite-element zones, only enter I. The J and K fields will not be available.
5. To place a single streamline, click Place Streamtrace(s).

   To place a rake of streamlines, select the Enter Rake Positions check box and enter the end positions for the rake as either a set of IJK indices or XYZ-coordinates. Then click Place Streamtrace(s). By default, Tecplot draws ten streamlines per rake. To change this, enter a new value in the Streams per Rake field.

   Streamtraces will be terminated at the edge of any cell which is all or partially value-blanked.

Figure 12-2 shows some surface line streamtraces and a streamtrace rake on the cylinder data.

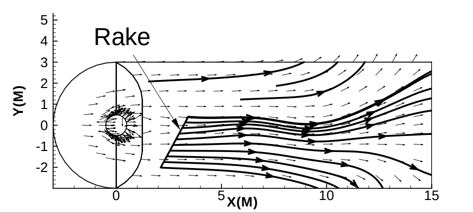


Figure 12-2. Surface line streamtraces and a streamtrace rake.

12.2. Volume Streamtraces

You can place volume streamtraces using either the mouse or the Streamtrace Details dialog. The mouse enables you to place streamtraces quickly, but the Streamtrace Details dialog gives you precise control over the starting points.

Volume streamtraces may only be drawn in 3-D volume zones so you must have at least one volume zone active. It does not matter what plot style you choose for the zone, but you must choose a setting that includes streamtraces on the Volume page of the Zone Style dialog. The default is to allow streamtraces to be drawn in all zones.

There are a number of different ways to place volume streamtraces. It is best to have some knowledge of the general direction of flow for the velocity field in the zones, so that you may place your streamtraces in a location of interest. Often it is a good idea to start streamtraces from somewhere within the volume, as opposed to starting on the outer surface of the volume. One of the best ways to start a volume streamtrace is to create a slice through the volume, then place the streamtrace starting position on the slice. A slice through a volume zone and a rake of streamtraces starting from the slice is shown in Figure 12-3.

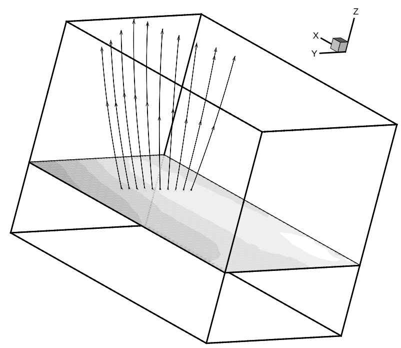


Figure 12-3. A slice through a volume zone with a rake of streamlines starting from the slice.

To place volume streamtraces on a slice perform the following steps:

1. Add a slice plane to your volume zone using the Slice tool  from the sidebar. Click on the volume zone to add a slice. You may press I, J, K, X, Y, or Z on your keyboard first to orient the slice in one of the constant I-, J-, K-, X-, Y-, or Z-planes.
2. Click on the Place Streamtrace tool  on the sidebar.
3. Press D on your keyboard to create streamrods, R for streamribbons, or V to create volume streamlines. You may also choose streamtrace types using the Streamtrace Details dialog from the Plot menu.
4. Place a single streamtrace by holding down Alt on your keyboard while clicking on the slice, or place a rake of streamtraces by holding down Alt and clicking-and-dragging. Alt will make sidebar tools operate on any objects except for zones, which in this case is the slice.
5. If you want to place streamtraces starting on the outer boundary of your zone, perform steps 2, 3, and 4 without holding down Alt.

To create a volume streamtrace or a rake of volume streamtraces using the Streamtrace Details dialog:

1. From the Plot menu, choose Streamtraces. If you have not yet assigned vector components for the 3D Cartesian plot type, the Vector Variables dialog appears for you to assign vector components. Otherwise, the Position page of the Streamtrace Details dialog appears, as shown in Figure 12-1.
2. Choose a format for the streamtrace from the drop-down labeled Format. You can choose Surface Line, Volume Line, Volume Ribbon, or Volume Rod. The default is Volume Line for 3-D volume zones, and Surface Line for all other zones.
3. Choose a direction for the streamtrace integration from the Direction drop-down:
   - **Forward:** The streamtrace is calculated "downstream," that is, in the direction of the flow.
   - **Backward:** The streamtrace is calculated "upstream," that is, against the flow.
   - **Both:** Both the forward and backward streamtraces are calculated. You should use this only for surface and volume lines. For ribbons and rods, Tecplot always integrates one step from the starting position before actually plotting or extracting the streamtrace.

   No matter which direction is chosen for the integration, the arrowheads still point in the forward, "downstream" direction.
4. Select either the Use IJK or Use XYZ option. (Use XYZ is the default). If you select Use IJK, also pick a zone from the drop-down labeled Zone. If you select Use IJK, the fields under Start Position and Rake End Position are labeled I, J, and K. Otherwise, they are labeled X, Y, and Z.
5. Enter the starting position by specifying either a set of IJK-indices or XYZ-coordinates.
6. To place a single streamtrace, click Place Streamtrace(s). To place a rake of streamtraces, select the Enter Rake Positions check box and enter the end positions for the rake as either a set of IJK-indices or XYZ-coordinates. Then click Place Streamtrace(s). By default, Tecplot draws ten streamtraces per rake. To change this, enter a new value in the Streams per Rake field.

Figure 12-4 shows several examples of volume streamtraces.

12.3. Streamtrace Plot Appearance

You can control the style of your streamtraces using the Streamtrace Details dialog. These style attributes affect all streamtraces in the current frame, including those already placed. They do not affect extracted streamtrace zones, discussed in Section 12.7, "Streamtrace Extraction as Zones," because these are now ordinary ordered zones, not streamtraces at all.

12.3.1. Streamlines

The following attributes may be set with the Line page of the Streamtrace Details dialog, shown in Figure 12-5.

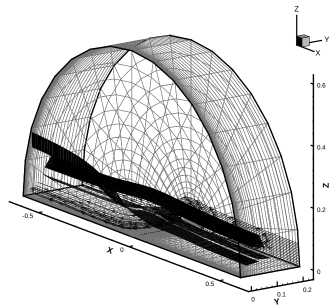


Figure 12-4. Volume streamlines, volume ribbons, and volume rods.

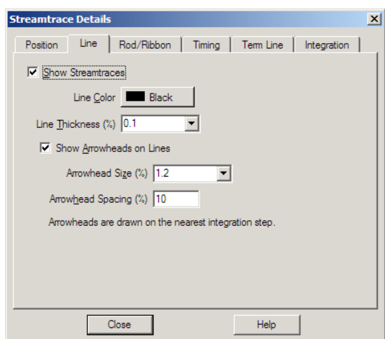


Figure 12-5. The Line page of the Streamtrace Details dialog in Motif.

- **Whether streamtraces are displayed:** Select the Show Streamtraces check box if you want streamtraces drawn on your plot. By default, this check box is selected.
- **Line Color:** Enter the color for all streamtraces. You may set the color to Multi-Color to color the streamtraces by the chosen contour group variable in the same manner as color flooding. (If the contour variable is not currently defined, the Contour Variable dialog appears so that you can define it.) You can use the Multi-Color option, for example, to color the streamtraces by the local temperature or by the velocity magnitude. You can also specify RGB coloring.

The following attributes affect surface and volume streamlines only:

- **Line Thickness:** Either enter a value for the streamline thickness (as a percentage of the frame height for 2-D lines and as a percentage of the median axis length for 3-D surface lines and volume lines), or choose a pre-set value from the drop-down.
- **Arrows:** Select the Show Arrowheads on Lines check box to display arrowheads along all streamlines (surface and volume) in the current frame. Arrows are not shown on volume ribbons or volume rods. You can also control the following attributes of the displayed arrows:
  - **Arrowhead Size:** Either enter a value for the arrowhead size (as a percentage of the frame height), or choose a pre-set value from the drop-down.
  - **Arrowhead Spacing:** Enter the distance between arrowheads in terms of Y-frame units. A value of ten percent will space arrowheads approximately ten percent of the frame height apart from each other along each streamline.

12.3.2. Streamrods and Streamribbons

The following attributes may be set with the Rod/Ribbon page of the Streamtrace Details dialog, shown in Figure 12-6. They affect volume ribbons and volume rods only:

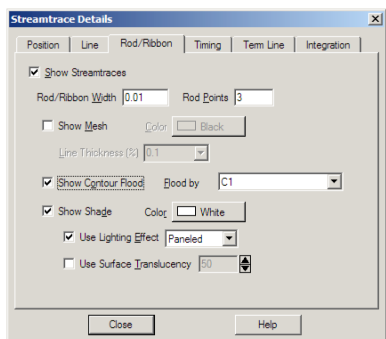


Figure 12-6. The Rod/Ribbon page of the Streamtrace Details dialog in Motif.

- **Rod/Ribbon Width:** Enter a width for the volume ribbons and volume rods. The width is expressed in grid units. If you want two sets of streamtraces with different widths, you must create one set and then extract them as zones, then configure a new set of streamtraces with the second width.
- **Rod Points:** Volume rods have a polygonal cross-section; this parameter tells Tecplot what that cross-section should be. Three is an equilateral triangle; four is a square; five, a regular pentagon; and so on. If you want two sets of volume rods with different cross-sections, you must create one set and then extract them as zones, then configure a new set of streamtraces with the second cross-section.
- **Show Mesh:** Select this check box to display a mesh.

- **Mesh Color:** Select a mesh color from the drop-down, or choose a custom color or multi-color.
- **Mesh Line Thickness:** Select a line thickness from the drop-down, or enter your own number in the text field.
- **Show Contour Flood:** Select this check box to display contour flooding.
- **Flood by:** Select the contour group by which to flood.
- **Show Shade:** Select this check box to display shading.
- **Shade Color:** Select a shade color from the Select Color dialog. Multi-Color and RGB coloring are not available—use contour flooding instead.
- **Use Lighting Effect:** Select this check box to enable the lighting effect drop-down where you may choose Paneled or Gouraud shading.
- **Use Surface Translucency:** Select this check box to enable the surface translucency text field, where you may set the surface translucency from one (opaque) to 99 (translucent).

12.4. Streamtrace Deletion

You can delete streamtraces, either individually or in groups, by first selecting them and then either pressing Delete or choosing Clear from the Edit menu. You may delete all streamtraces at once by clicking Delete All on the Position page of the Streamtrace Details dialog. You may delete the last streamtrace placed by clicking Delete Last on this page.

12.5. Streamtrace Termination Lines

A streamtrace termination line is a polyline that terminates any streamtraces that cross it. The termination line is useful for stopping streamtraces before they spiral or stall. If a streamtrace and the termination line intersect on the screen, the streamtrace is considered to cross the termination line. This is true for volume streamtraces within a 3-D volume zone as well as for surface streamlines. Figure 12-7 shows the cylinder data with some streamtraces terminated with a 2-D streamtrace termination line.

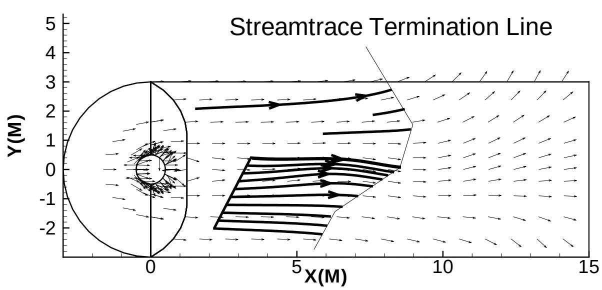


Figure 12-7. Surface streamlines and 2-D termination line.

Streamtraces are also terminated whenever any of the following occur:

- The maximum number of integration steps is reached.
- Any point in the streamtrace passes outside the available data.
- The streamtrace enters a cell for which the velocity magnitude is zero.

For more details, see Section 12.8, “Streamtrace Integration.”

12.5.1. Termination Line Creation

To add a streamtrace termination line:

1. From the sidebar, choose the Add Streamtrace Termination Line tool, represented by the  button, or from the Term Line page of the Streamtrace Details dialog, click Draw Stream Term Line.
2. Move the pointer into the workspace. The pointer becomes a cross-hair.
3. Click at the desired starting point for the termination line, then click at additional points to define the desired polyline.
4. To end the termination line, press Esc or select another tool from the sidebar.

Only one termination line can exist at any one time in a given frame. If you draw a second termination line, the first is automatically deleted.

In 2D Cartesian plots, the termination line is drawn in the grid coordinate system and moves with the data as you zoom and translate. In 3D Cartesian plots, unlike most Tecplot objects, the termination line is drawn in the eye coordinate system, which is essentially the plane of your computer screen. This coordinate system also moves with the data as you zoom and translate. If you rotate a 3-D data set after drawing a streamtrace termination line, streamtraces previously terminated by the termination line may be terminated at different places, or not terminated at all if the rotated streamtrace no longer intersects the termination line. Figure 12-8 shows a 3-D volume plot with streamribbons and a streamtrace termination line, and how the termination points vary as the plot is rotated. Notice that the termination line itself remains in place on the screen as the plot is rotated.

12.5.2. Termination Line Controls

You control the streamtrace termination line from the Term Line page of the Streamtrace Details dialog, shown in Figure 12-9.

From the Term Line page, you can control the following attributes of the termination line:

- **Active:** If the Active Termination Line check box is selected, the termination line is active, and any streamtraces that cross it are terminated. You can deselect the check box and redraw the plot to view the unterminated streamtraces.

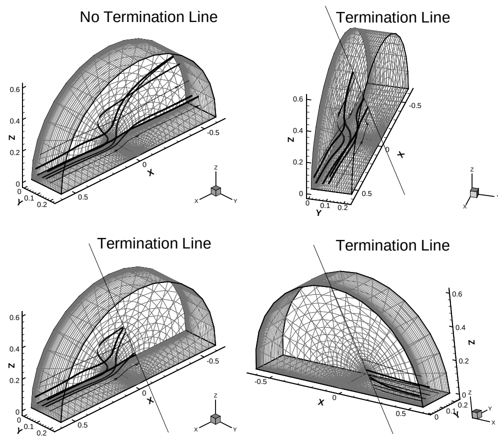


Figure 12-8. Terminating volume streamtraces with a termination line.

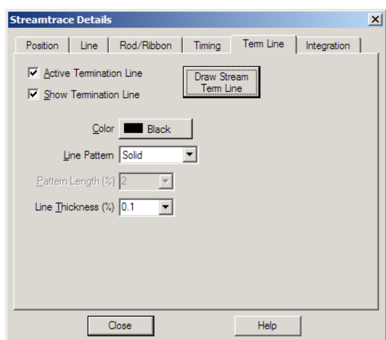


Figure 12-9. The Term Line page of the Streamtrace Details dialog.

- **Shown:** If the Show Termination Line check box is selected, the termination line is displayed. You can deselect the check box and redraw the plot so that only the terminated streamtraces are displayed, not the termination line.
- The color, line pattern, pattern length, and line thickness of the termination line.

You can select a termination line with the Selector or Adjustor tool. This allows you to interactively move the line (with the Selector), modify the line (with the Adjustor), or delete the line (with either tool).

12.6. Streamtrace Timing

Stream markers are symbols plotted along streamtrace paths to identify the positions of particles at certain times. Streamtrace dashes are another means of indicating the passage of time by causing a streamtrace to be “on” for a time interval, then “off” for another time interval.

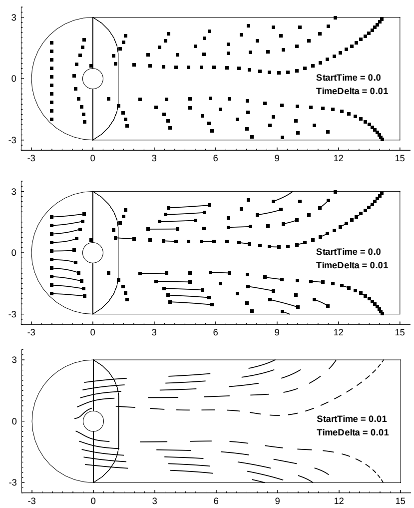


Figure 12-10. Streamtrace markers (top), dashes (bottom), and both (middle).

Figure 12-10 shows a plot with both streamtrace markers and dashes.

12.6.1. Stream Markers

Stream markers are drawn at timed locations along streamlines. The spacing between stream markers is proportional to the magnitude of the local vector field—that is, it is large in regions where the local magnitude is large, small in regions where the local magnitude is small. You can adjust the spacing between stream markers by specifying the time interval, or delta, between stream markers. Increasing the delta time will increase the space between stream markers and vice versa. The actual spacing is the product of the local vector magnitude and the specified delta.

You may also select the shape of your stream marker using the pre-set list under the Shape drop-down. Selecting Other from the list activates the Enter ASCII Character option. Clicking this will call up the Enter ASCII Character dialog, where you may enter an ASCII character to be used as your stream marker.

Stream markers are available only for streamlines (surface and volume); they are not available for volume ribbons or volume rods.

To place stream markers along your streamtraces:

1. From the Plot menu, choose Streamtraces. The Streamtrace Details dialog appears.
2. Click Timing (Timing tab in Windows). The Timing page of the Streamtrace Details dialog appears, as shown in Figure 12-11.

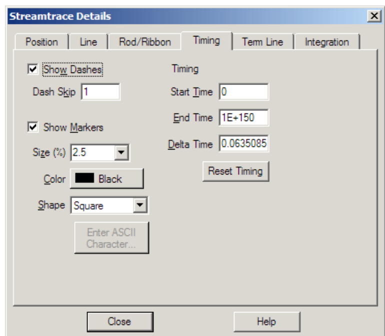


Figure 12-11. The Timing page of the Streamtrace Details dialog.

3. Select the check box labeled Show Markers. The three fields immediately below Show Markers become active, as do the fields grouped under the heading Timing.
4. Specify the size, color, and shape of the markers in the fields provided. For more information on marker shapes see Section 13.3. “Scatter Symbols” on page 252.
5. Specify the timing for the stream markers by entering values in the following fields:
   - **Start Time:** Enter the time at which the first marker is drawn. A start time of zero means that the first marker is drawn at the starting point. A start time of 2.5 means that the first stream marker is drawn 2.5 time units downstream of the starting point.
   - **End Time:** Enter the time after which no more stream markers are drawn.
   - **Delta Time:** Enter the time interval which measures the time between stream markers. The actual distance between markers is the product of this number and the local vector magnitude.
6. On the sidebar, click Redraw to see the stream markers on existing streamlines; subsequent streamlines are drawn automatically.

12.6.2. Stream Dashes

Stream dashes, unlike stream markers, are not restricted to streamlines; you can also apply stream dashes to volume ribbons and volume rods. A stream dash shows the streamtrace with a dashed line pattern. The streamtrace is "on" for a time interval, then "off" for a time interval, then "on" again, and so on. The lengths of the dashes and the spaces between them are controlled by the same time delta used for stream markers.

To use stream dashes:

1. From the Plot menu, choose Streamtraces. The Streamtrace Details dialog appears.
2. Click Timing (Timing tab in Windows). The Timing page of the Streamtrace Details dialog appears, as shown in Figure 12-11.
3. Select the check box labeled Show Dashes. The Dash Skip field immediately below Show Dashes becomes sensitive, as do the fields grouped under the heading Timing.
4. Enter the dash skip factor, which controls the number of time deltas used for the "off" sections of the streamtraces. Using a dash skip factor of one produces an on-off-on-off pattern. A skip factor of two produces an on-off-off-on-off-off pattern. The actual lengths of the dashes are computed as the product of the delta time and the local vector magnitude.
5. Specify the timing for the stream dashes by entering values in the following fields:
   - **Start Time:** Enter the time at which the first dash is drawn. A start time of zero means that the first dash is drawn at the starting point. A start time of 2.5 means that the first stream dash is drawn 2.5 time units downstream of the starting point.
   - **End Time:** Enter the time after which no more stream dashes are drawn.
   - **Delta Time:** Enter the time interval which controls the length of the dashes. The actual dash length is the product of this number and the local vector magnitude.
6. Click Redraw on the sidebar to see the stream dashes on existing streamtraces; subsequent streamtraces are drawn automatically.

12.7. Streamtrace Extraction as Zones

In Tecplot you should be able to assign any style you desire to streamtraces without further processing. Temporary streamtrace objects are created and drawn just like zones. However, if you need to make permanent objects from streamtraces you may extract them to zones.

To extract your streamtraces as zones:

1. Create a plot containing streamtraces. You may (if your data includes any 3-D volume zones) use multiple streamtrace formats.
2. From the Data menu, choose Extract, then choose Streamtraces. The Extract Streamtraces dialog appears, containing the single check box Concatenate Common Streamtraces into One Zone.
3. If you want all streamtraces of a given format extracted to a single zone, select the check box labeled Concatenate Common Streamtraces into One Zone. If you select this check box, Tecplot extracts all surface lines into one zone, all volume lines into another, all vol-

ume ribbons into a third, and all volume rods into a fourth. Tecplot uses value-blanking to blank out the intervals between streamtraces (and between stream dashes). This is discussed more fully later in this section.

4. If you do not select the check box, each streamtrace is extracted into its own zone.
5. Click Extract to extract the streamtraces to zones. A Working dialog appears while the extraction is proceeding; click Cancel to interrupt the extraction.

Once you have created these new zones, you may treat them as any other zone, and by default, that is what Tecplot does. If you have a mesh plot, you will see the mesh of your original data plotted with the mesh for each of the new zones. You will also see the original streamtraces, which may obscure the plotted streamtrace zones. Once you have extracted the zones, you can delete the original streamtraces by clicking Delete All Streamtraces in the Streamtrace Details dialog. Figure 12-12 shows some extracted volume ribbon zones, with the original streamtraces deleted.

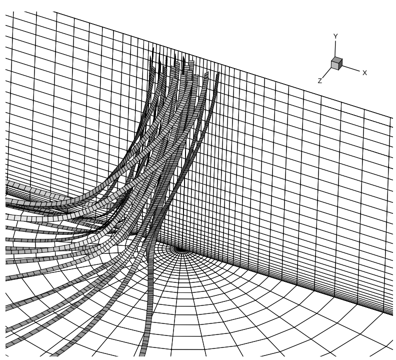


Figure 12-12. Extracted volume ribbons.

If timed dashes are active, all extracted streamtraces will be finite-element zones. Otherwise, all extracted streamline zones are I-ordered, and extracted volume ribbon and volume rod zones are IJ-ordered.

12.8. Streamtrace Integration

Tecplot uses a predictor-corrector integration algorithm to calculate streamtraces. The basic idea is to create the streamtrace by moving in a series of small steps from the starting point in the direction of, or in opposition to, the local vector field. Each step is only a fraction of a cell or element. Tecplot automatically adjusts the step size based on the local cell shape and vector field variation.

You can control the streamtrace integration by modifying the following parameters in the Integration page of the Streamtrace Details dialog, shown in Figure 12-13:

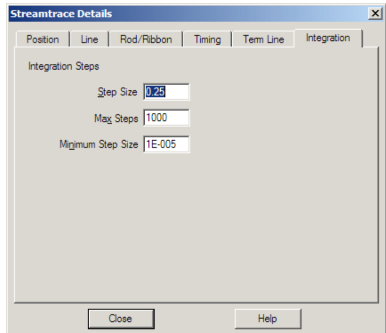


Figure 12-13. The Integration page of Streamtrace Details dialog.

- **Step Size:** Enter the initial and maximum step size Tecplot uses while integrating through the vector field, as a decimal fraction of the local cell or element width. A typical value (and the default) is 0.25, which results in four integration steps through each cell or element. The value for Step Size affects the accuracy of the integration. Setting Step Size too small can result in round-off errors, while setting it too large can result in errors due to missed cells.
- **Max Steps:** Enter the maximum number of steps before the streamtrace is terminated. This prevents streamtraces from spinning forever in a vortex, or from wandering aimlessly in a region where the vector components are very small, very random, or both. If you choose a small Step Size, you should enter a larger Max Steps.
- **Minimum Step Size:** The smallest step size for Tecplot to use. Setting this too small results in integration problems. Setting this greater than or equal to the Step Size results in a constant step size.

During the integration, a streamtrace is terminated if any of the following conditions occur:

- The maximum number of integration steps (Max Steps) have been taken.
- Any point in the streamtrace passes outside the available data. This is a particular concern with volume ribbons: a volume ribbon with a large width may terminate when one edge passes outside the vector field, even though the center is within the field. You can avoid this problem by entering a smaller Ribbon/Rod Width on the Rod/Ribbon page of the Streamtrace Details dialog.
- The streamtrace enters a cell for which the velocity magnitude is zero.
- The streamtrace crosses the stream termination line.

Streamtraces may terminate at a zone boundary even if there is an adjacent zone into which the streamtraces should proceed. This can happen if there is a small gap between the zones. Specifying face neighbors in the data file to connect the zones can alleviate this problem. increasing the minimum integration step size can also eliminate this problem.

CHAPTER 13 *Scatter Plots*

Scatter plots are plots of symbols at the data points in a field. The symbols may be sized according to the values of a specified variable, colored by the values of the contour variable, or may be uniformly sized or colored. Unlike contour plots, scatter plots do not require any mesh structure connecting the points, allowing you to make scatter plots of irregular data.

13.1. Scatter Plot Creation

A 2-D scatter plot plots the position of the Y-variable against the position of the X-variable. Thus, it is a representation of the location of data points in the 2-D field. Thought of somewhat differently, the scatter plot is a plot of the vertices of an ordered mesh or the nodes of a finite-element mesh.

To create a scatter plot in Tecplot, you activate the Scatter layer (and deactivate any active layers that you do not want to appear). For example, to create the scatter plot shown in Figure 13-1, do the following:

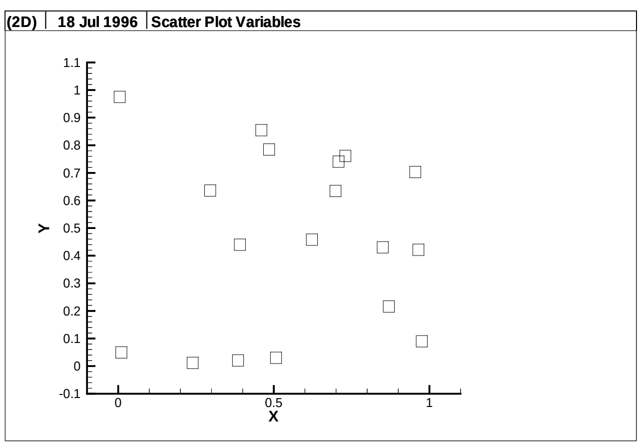


Figure 13-1. A scatter plot.

1. Read in the data file **`simpscat.plt`** from your Tecplot **`examples/dat`** directory. (If you currently have a data set in your Tecplot frame, choose to replace the data set and reset the frame style.)

2. From the sidebar, select the Scatter check box and deselect the Mesh check box (and any other active layer check box).

13.2. Scatter Plot Modification

Once you have read in your data, you can modify your scatter plot attributes using either the Scatter page of the Zone Style dialog or the Quick Edit dialog. You can control any of the following attributes from the Scatter page of the Zone Style dialog, shown in Figure 13-2:

- Which zones are active. See Section 7.3.2, “Zone Display.”
- Whether the scatter symbols are visible for each active zone. See Section 7.3.3, “Zone Layer Display.”
- The scatter symbol shape. See Section 13.3, “Scatter Symbols.”
- The scatter outline color. See Section 13.4.1, “Outline Colors.”
- The scatter fill color. See Section 13.4.2, “Filled Symbols and Fill Colors.”
- The scatter symbol size. See Section 13.5, “Scatter Symbol Sizes and Fonts.”
- The scatter line thickness. See Section 7.3.7, “Line Thickness.”
- The scatter symbol spacing (Index Skip). See Section 13.6, “Symbol Spacing.”

The following scatter attributes are assigned on a frame-by-frame basis:

- The scatter-size variable. See Section 13.5, “Scatter Symbol Sizes and Fonts.”
- The optional reference scatter symbol. See Section 13.5.4, “Reference Scatter Symbols.”
- The scatter legend. See Section 13.7, “Scatter Legends.”

13.3. Scatter Symbols

By default, Tecplot uses outlined squares for the scatter symbols. You can choose a different scatter symbol for each zone from any of Tecplot’s seven predefined scatter symbols, or any printable character in any of Tecplot’s four character sets. You can change the scatter symbol either from the Scatter page of the Zone Style dialog or from the Quick Edit dialog.

To choose a scatter symbol:

1. Go to the Scatter page of the Zone Style dialog, as shown in Figure 13-2.
2. Select the zone or zones for which you want to choose a scatter symbol.
3. Click Symbol Shape. A drop-down appears listing predefined scatter symbols together with an Other option. Figure 13-3 shows the predefined scatter shapes from the Quick Edit dialog. The options include:
   - **Square:** Plot with square plotting symbols.
   - **Delta:** Plot with upward pointing equilateral triangles.
   - **Gradient:** Plot with downward pointing equilateral triangles.
   - **Right Triangle:** Plot with rightward pointing equilateral triangles.

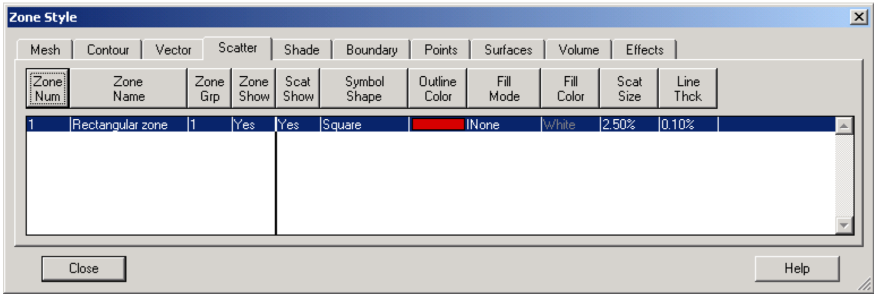


Figure 13-2. The Scatter page of the Zone Style dialog.

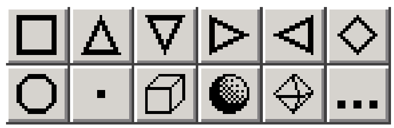


Figure 13-3. Pre-defined scatter symbols of the Quick Edit dialog.

- **Left Triangle:** Plot with leftward pointing equilateral triangles.
- **Diamond:** Plot with diamonds.
- **Circle:** Plot with circles.
- **Point:** Plot as points, always one pixel on screen and in exported images.
- **Cube:** Plot as cubes in a 3-D; reduced to Square for 2-D plots.
- **Sphere:** Plot as spheres in a 3-D; reduced to Circle for 2-D plots.
- **Octahedron:** Plot as octahedrons in a 3-D; reduced to Diamond for 2-D plots.
- **Other:** Plot with a specified ASCII character.

Spheres, Cubes, and Octahedrons are always light-source shaded. Spheres are Gouraud shaded, and Cubes and Octahedrons are Panel shaded. Cube edges are aligned with X-, Y-, and Z-axes. Octahedrons are oriented so one vertex points in the Z-direction and one vertex points in the X-direction. For best appearance of 3-D shapes, adjust the Light Source and use Specular Highlighting.

4. Click on the desired symbol shape. If you click on Other, the Enter ASCII Character dialog appears, as shown in Figure 13-4.
5. (Other option only) Enter a character to use as a symbol, and then specify the Tecplot character set from which to obtain the symbol: Base (the Courier, Helvetica and Times fonts are collectively referred to as English fonts), Greek, Math, or User Defined. Click OK to dismiss the Enter ASCII Character dialog.

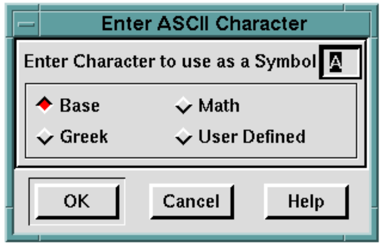
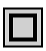
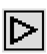
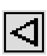
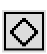
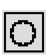
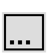
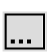


Figure 13-4. The Enter ASCII Character dialog.

To choose a scatter symbol from the Quick Edit dialog:

1. In the workspace, select the zone or zones for which you want to choose a scatter symbol.
2. On the sidebar, click on Quick Edit to call up the Quick Edit dialog.
3. In the Quick Edit dialog, click on the button for the desired symbol shape, as follows:
   -  Plot with square plotting symbols (Square).
   -  Plot with upward pointing equilateral triangles (Delta).
   -  Plot with downward pointing equilateral triangles (Gradient).
   -  Plot with rightward pointing equilateral triangles (Right Triangle).
   -  Plot with leftward pointing equilateral triangles (Left Triangle).
   -  Plot with diamonds (Diamond).
   -  Plot with circles (Circle).
   -  Plot with a specified ASCII character from one of Tecplot's four character sets (Other). When you select this option, the Enter ASCII Character dialog appears.
4. ( option only) Enter a character to use as a symbol, and then specify the Tecplot character set from which to obtain the symbol: Base (English Font), Greek, Math, or User Defined. Click OK to close the Enter ASCII Character dialog.

13.4. Symbol Color

By default, scatter symbols are drawn as outlined symbols, that is, unfilled geometric shapes in a single color. You control the basic color by choosing an outline color for each zone's scatter symbols. For 2-D symbols, you can also choose a fill mode for a zone's symbols, and specify a fill color. (If you fill a symbol chosen from the ASCII character set, you obtain a filled rectangle with the character drawn inside in the outline color; the perimeter of the box is also drawn in the outline color.)

For both outline and fill color, available options allow you to choose on of the Multi-Color contour groups or RGB coloring. See Section 10.2.1, "Contour Variable Selection," for information about multi-color options.

13.4.1. Outline Colors

For unfilled scatter symbols, the outline color is simply the symbol color. You can specify a different outline color for each zone using either the Scatter page of the Zone Style dialog or the Quick Edit dialog.

To specify an outline color:

1. Choose the Scatter page of the Zone Style dialog.
2. Select the zone or zones for which you want to choose an outline color.
3. Click Outline Color. A dialog appears listing Tecplot's basic colors together with a multi-color option.
4. Click on the desired color or option.

or

1. On the sidebar, click on Quick Edit to call up the Quick Edit dialog.
2. In the workspace, select the zone or zones for which you want to choose a scatter symbol.
3. On the Quick Edit dialog, click Line, then click Color. On the Select Color dialog, choose the desired option. If your symbols are filled, click X to make the line color match the current fill color.

If you select the Multi-Color contour group option, each plotting symbol is colored according to the value of the selected contour variable at that data point. If no contour variable is currently assigned, the Contour Details dialog appears with the default contour variable assignment. You can either accept the default or choose a new contour variable.

If you select the RGB coloring option, each plotting symbol is colored according to the values at that data point for the variables assigned to RGB. If no variables are currently assigned, the Select Variables dialog appears with the default variable assignments. You can either accept the default or choose new variables.

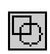
13.4.2. Filled Symbols and Fill Colors

The 3-D symbol shapes, Cube, Sphere, and Octahedron, are filled with the line color, but the other shapes have several optional fill modes. Symbols may be unfilled, filled with the background color or the outline color, or filled with a chosen color.

To control filling of a zone’s scatter symbols:

1. Go to the Scatter page of the Zone Style dialog.
2. Select the zone or zones for which you want to specify filled or unfilled symbols.
3. Click Fill Mode; the drop-down options are:
   - **None:** Lines or objects behind each symbol are seen.
   - **Use Line Color:** Symbols are filled with the outline color.
   - **Use Back Color:** Symbols are filled with the color of the frame background. No objects or lines behind them appear.
   - **Use Specific Color:** Default fill color is white.
4. 4. Choose the desired mode. You can change Specific Color using the Fill Color button.

You can optionally control whether a zone’s scatter symbols are filled, and if so, what color they are filled with, using the Quick Edit dialog.

1. On the sidebar, click Quick Edit to call up the Quick Edit dialog.
2. In the workspace, select the zone or zones for which you want to specify filled or unfilled symbols.
3. On the Quick Edit dialog, click on the  button for unfilled symbols; click on  for filled symbols. If you choose filled symbols and want them filled with a color other than the default white, you should also specify a fill color.
4. With Fill selected, click Color and choose a color or Multi-color option. You can click on X to turn off fill.

13.5. Scatter Symbol Sizes and Fonts

You can have Tecplot draw each zone’s scatter symbols at a specified size, or you can size each scatter symbol according to the value of a specified variable at that point. To specify fixed-size scatter symbols, you can use either the Scatter page of the Zone Style dialog, or Quick Edit dialog. To specify variable-size scatter symbols, you must use the Zone Style dialog.

13.5.1. Symbol Sizes

To specify a fixed symbol size:

1. Go to the Scatter page of the Zone Style dialog.
2. Select the zone or zones for which you want to specify the symbol size.

3. Click Scat Size. A drop-down appears listing five preset options, an Enter option, and a Size by Variable option. Click the appropriate option.
4. If you click Enter, the Enter Value dialog appears. Enter a percentage of the frame height in the Enter Value dialog.
5. If you click Size by Variable, the Scatter Size/Font dialog appears asking you to choose a scatter-sizing variable. (See Section 13.5.2, "Variable Symbol Sizes," for complete instructions for sizing scatter symbols by variable.) From the Scatter Size Variable drop-down, choose the variable you want to use to size the scatter symbols. Specify a size multiplier by selecting one of the two options labeled Grid Units/Magnitude or Cm/Magnitude, then enter a value in the text field. The Scatter Size/Font dialog is shown in Figure 13-5.

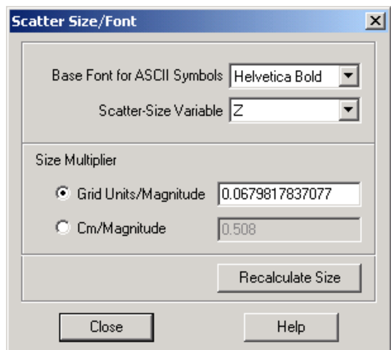


Figure 13-5. The Scatter Size/Font dialog.

For fixed size only:

1. On the sidebar, click Quick Edit to call up the Quick Edit dialog.
2. In the workspace, select the zone or zones for which you want to specify the symbol size.
3. On the Quick Edit dialog, click Size to the right of the Scatter Symbol buttons. A drop-down appears listing five preset options and an Enter option.
4. Click on the desired option.
5. If you click Enter, the Enter Value dialog appears. Enter a percentage of the frame height in the Enter Value dialog.

A sample scatter plot sized by a variable is shown in Figure 13-6.

13.5.2. Variable Symbol Sizes

When you choose Size by Variable for scatter plots, Tecplot multiplies the value of the scatter-size variable at each point by a specified multiplier to determine the actual size of the plotting symbol. You can modify this multiplier from the Scatter Size/Font dialog. You can also change the variable used for sizing, or the base font used for ASCII character symbols.

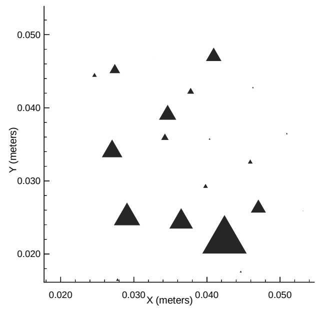


Figure 13-6. Scatter plot sized by a variable.

To modify the settings:

1. Choose Scatter Size/Font from the Scatter sub-menu of the Plot menu. The Scatter Size/Font dialog appears.
2. Select Base Font for ASCII Symbols and choose the desired font.
3. If you have not specified a scatter-size variable, or if you want to use a different variable, choose a variable from the Scatter Size Variable menu on the Scatter Size/Font dialog. The Size Multiplier region becomes active.
4. Choose the Grid Units/Magnitude or Cm/Magnitude option. By default, Tecplot uses Grid Units/Magnitude. Tecplot calculates and displays an initial size multiplier in the adjacent text field.
5. Enter the desired multiplier in the text field to the right of the selected option.

13.5.3. Variable Size Multiplier and Fonts

When you choose Size by Variable for scatter plots, Tecplot multiplies the value of the scatter-size variable at each point by a specified multiplier to determine the actual size of the plotting symbol. You can modify this multiplier from the Scatter Size/Font dialog.

You can change the base font used for ASCII character symbols to any of Tecplot's basic fonts (Helvetica, Courier Bold, etc.). To do this, choose Scatter Size/Font from the Field menu. The Scatter Size/Font dialog appears. Choose the desired font from the drop-down labeled Base Font for ASCII Symbols. The base font affects all ASCII symbols using the base font in the current frame.

To modify the scatter size multiplier and font:

1. Click the Zone Style button in the sidebar. The Zone Style dialog appears.
2. Display the Scatter page.
3. Select the zones for which you want to modify the multiplier for scatter symbols.

4. Click Scat Size. On the drop-down which appears, select Size by Variable.
5. Choose Scatter Size/Font from the Scatter sub-menu of the Field menu. The Scatter Size/Font dialog appears.
6. Select Base Font for ASCII Symbols and choose the desired font from the list shown.
7. If you have not specified a scatter-size variable, or if you want to use a different variable, choose a variable from the Scatter Size Variable menu on the Scatter Size/Font dialog. The Size Multiplier region becomes active.
8. Choose the Grid Units/Magnitude or Cm/Magnitude option. By default, Tecplot uses Grid Units/Magnitude. Tecplot calculates and displays an initial size multiplier in the adjacent text field.
9. Enter the desired multiplier in the text field to the right of the selected option.

13.5.4. Reference Scatter Symbols

If you are using a scatter-size variable, it is sometimes useful to create a reference scatter symbol that shows the size at which a data point of a given magnitude will be represented. You create the reference scatter symbol using the Reference Scatter Symbol dialog. The Reference Scatter Symbol appears only if a scatter size variable is defined; if you have not yet created one, select one by choosing Scatter Font/Size from the Field menu, then choosing a Scatter Size Variable from the drop-down.

To create a reference scatter symbol:

1. Choose Reference Scatter Symbol from the Scatter sub-menu of the Field menu. The Reference Scatter Symbol dialog appears, as shown in Figure 13-7.

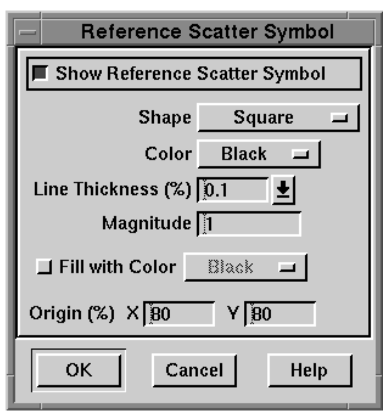


Figure 13-7. The Reference Scatter Symbol dialog.

2. Select the check box labeled Show Reference Scatter Symbol.

3. Enter an appropriate value in the Magnitude text box. It may be useful to use the Probe tool  on a symbol of appropriate size, then set the Reference Scatter Symbol magnitude accordingly.
4. Modify the shape, color, line thickness, fill color, and origin as desired.
5. Click OK.

Figure 13-8 shows a scatter plot with a reference scatter symbol.

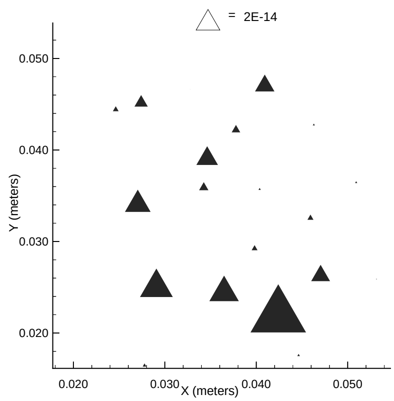


Figure 13-8. Scatter plot with reference scatter symbol.

13.6. Symbol Spacing

If your data consists of a dense mesh of points, a scatter plot may be too crowded to be of much use. You can “thin” the scatter plot by plotting only a certain subset of the data points.

You control the number of points plotted with the Index Skip attribute from the Points page of the Zone Style dialog. For IJ-ordered data, you can specify both an I-skip and a J-skip, while for IJK-ordered data, you can specify I-, J-, and K-skips. (For I-ordered data and finite-element data, only an I-skip is permitted; it allows you to plot every *n*th data point, using the natural order of nodes and data points in the original data set.)

For example, a typical scatter plot from a full-size mesh has so many points it is difficult to see individual symbols (shown in Figure 13-9). Figure 13-10, on the other hand, shows a “thinned” scatter plot of the same data with Index Skip specified, showing every third point in the I-direction and every fourth point in the J-direction.

To specify the symbol spacing:

1. Click the Zone Style button in the sidebar. The Zone Style dialog appears.

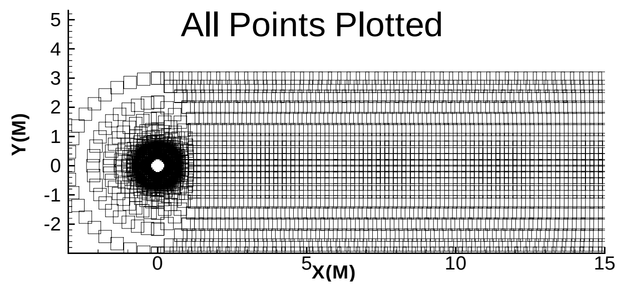


Figure 13-9. A crowded scatter plot, using the cylinder data.

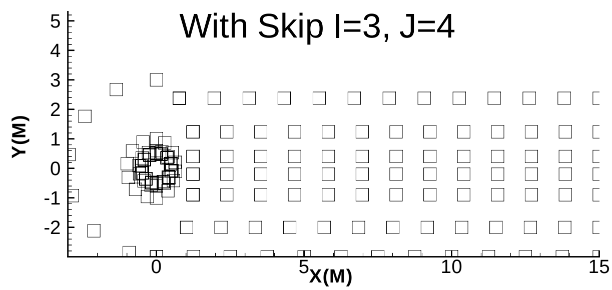


Figure 13-10. A scatter plot with Index Skip specified.

2. Display the Points page.
3. Select the zone or zones for which you want to specify the symbol spacing.
4. Click Index Skip. A drop-down appears with the options No Skip and Enter Skip. (No Skip is the default.)
5. Click the desired option. If you click Enter Skip, the Enter Index Skipping dialog appears.
6. (Enter Skip only) Enter the desired values of I-Skip, J-Skip, and K-Skip. Figure 13-10 was created with the following settings: *I-Skip=3*, *J-Skip=4*, *K-Skip=1*.

13.7. Scatter Legends

You can generate a legend that shows the style attributes of all scatter symbols. This legend can be positioned anywhere on the plot. You can elect to have the zone names included in the legend.

To create the scatter legend:

1. Choose Scatter Legend from the Scatter sub-menu of the Field menu. The Scatter Legend dialog appears.
2. Select the check box labeled Show Scatter Legend.

3. Select the Show Zone Names check box to include zone names in the legend.
4. Format the text for the legend by choosing a color and font, and specifying the text height as a percentage of the frame height. Enter the desired line spacing in the Line Spacing text field.
5. Specify the location of the upper left corner of the legend by entering values in the X (%) and Y (%) text fields. Enter X as a percentage of the frame width and Y as a percentage of the frame height.
6. Select which kind of box you want drawn around the legend (No Box, Filled, or Plain). If you choose Filled or Plain, format the box using the following controls:
   - **Line Thickness:** Specify the line thickness as a percentage of frame height.
   - **Box Color:** Choose a color for the legend box outline.
   - **Fill Color:** (Filled only) Choose a color for the legend box fill.
   - **Margin:** Specify the margin between the legend text and legend box as a percentage of the text height.

Figure 13-11 shows a scatter plot with a scatter legend.

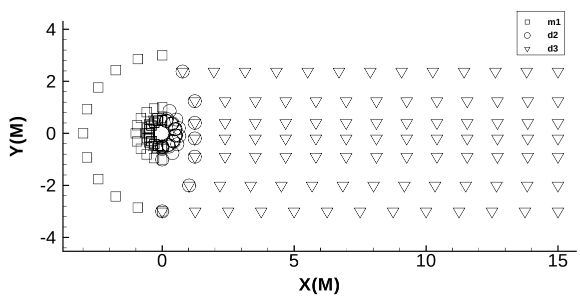


Figure 13-11. Scatter plot with legend.

CHAPTER 14 Shade Plots

Shade plots cover the surface of zones with a single color. In 2-D plots, each shaded zone will have one constant color. In 3-D plots, effects such as translucency and lighting will cause variation in color at different locations on the zone.

Shade plots require IJ- or IJK-ordered, or finite-element data. I-ordered, or irregular data, cannot be used to create shade plots.

14.1. Two-Dimensional Shade Plots

In 2D Cartesian plots, the only type of shading available is solid zone flooding. Each shaded zone is drawn as a uniform color.

To create a 2-D shade plot:

1. Select the Shade zone layer from the sidebar.

To control the 2-D shade color for a zone or zones:

1. Click the Zone Style button in the sidebar. The Zone Style dialog appears.
2. Display the Shade page. The Shade page appears as shown in Figure 14-1.

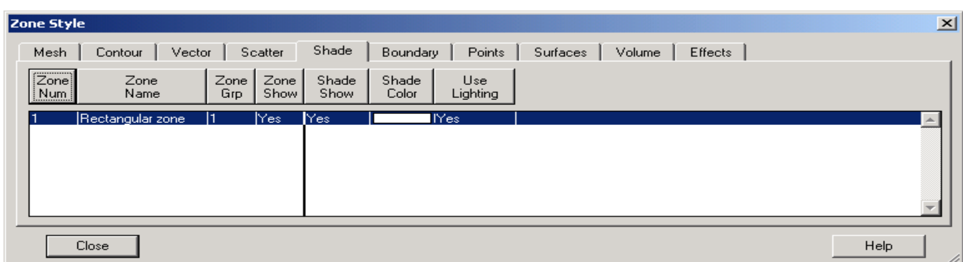


Figure 14-1. The Shade page of the Zone Style dialog.

3. In the Zone Style dialog, select the zone or zones for which you want to modify the shade color.
4. Click Shade Color. A drop-down appears containing Tecplot's basic colors.
5. Click on the desired color.

14.2. Three-Dimensional Surface Shade Plots

Three-dimensional shade plots may be created with surface or volume zones. Surface zones, such as IJ-ordered, finite-element triangle or finite-element quadrilateral, are loaded by default into Tecplot as 2-D plots. They may be viewed as 3-D surfaces by selecting the 3D Cartesian plot type from the sidebar. Volume zones, IJK-ordered, finite-element brick, and finite-element tetrahedral, by default will plot the outer surface with the field layer you select.

To create a 3-D shade plot:

1. Select the Shade zone layer from the sidebar.
2. Control which zones are shaded using the Shade Show option of the Shade page of the Zone Style dialog.
3. Control the shade color for each zone using the Shade Color option of the Shade page of the Zone Style dialog.
4. Choose which zones will have a 3-D shade effect using the Use Lighting option on the Shade page of the Zone Style dialog. To change the lighting effect, go to the Effects page of the Zone Style dialog.

Figure 14-2 shows a 3-D surface shade plot, with the Use Lighting option selected and the default Paneled lighting effect. The data file from which this plot was generated is delivered with Tecplot as **`demo/3d/spcship.plt`**.

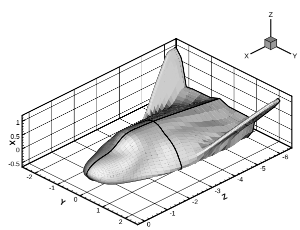


Figure 14-2. A 3-D surface shade plot.

You may specify zone colors for each plotted zone. When Use Lighting is set to No, the zone color is used to uniformly color the zone. For Paneled and Gouraud shading, the zone color is combined with the light source effect, as described in Section 15.1.2, “Lighting Effects.”

To specify the zone color, use the procedures for choosing colors in Section 7.3.4, “Color Choice.” (The Multi-Color option is not available for shade plots.)

CHAPTER 15 *Translucency and Lighting*

The 3-D Effects, Translucency and Lighting, are effective with shaded or contour flooded zones. Each must be selected on the sidebar to be available for zones with a plot. Streamtraces, slices and iso-surfaces all have separate controls from zones for lighting and translucency. The Effects page of the Zone Style dialog is shown in Figure 15-1.

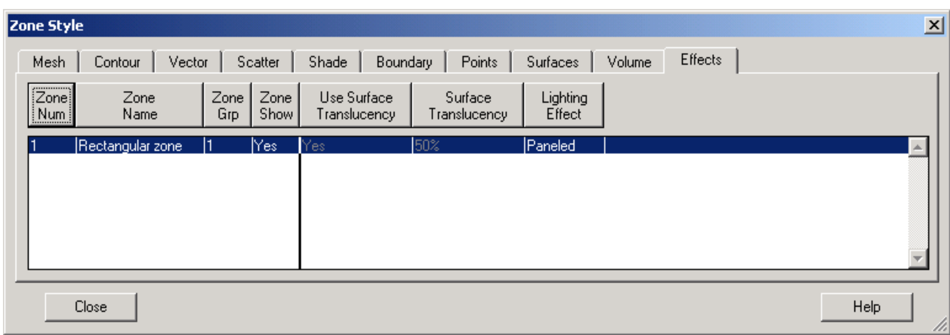


Figure 15-1. The Effects page of the Zone Style dialog.

15.1. Three-Dimensional Effects

When a zone is translucent, you may view objects inside or beyond the zone. You control the translucency of a zone using the Surface Translucency attribute in the Effects page of the Zone Style dialog. Translucency may be set to a value between one, nearly solid, and 99, nearly invisible. There are nine pre-set percentages ranging from ten to 90. You may also use the Enter option to define a percentage of your own. An example of a translucent plot is shown in Figure 15-2.

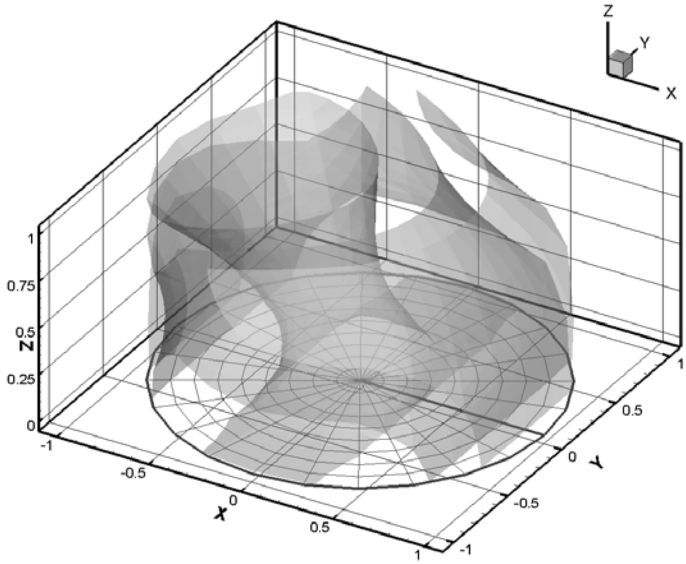


Figure 15-2. A plot with translucency.

To create a plot using translucency:

1. Create a 3-D shade or contour flooded plot.
2. Select the Translucency check box on the sidebar.
3. Click the Zone Style button in the sidebar. The Zone Style dialog appears.
4. Display the Effects page.
5. Select the zone or zones that you want to plot using translucency, then set Use Surface Translucency to Yes.
6. Click Surface Translucency, then select a percentage from the drop-down. Alternatively, you may click Enter to type in a value between one and 99.

15.1.1. Translucency

All surfaces in 3D Cartesian plots may be made translucent. A different translucency may be assigned to individual zones, and may also be assigned to derived objects such as slices, streamtrace ribbons or rods, and iso-surfaces. Please note that the Translucency check box on the side bar applies only to zones, not slices, streamtraces, or iso-surfaces. Translucency for those objects is controlled through their respective dialogs.

Plots with translucency cannot be printed. Translucency will only appear on your screen, or in exported bitmap images. See Appendix E, “Limits of Tecplot Version 10,” for more details.

15.1.2. Lighting Effects

There are two types of lighting effects, Paneled and Gouraud.

- **Paneled:** Within each cell the color assigned to each area by shading or contour flooding is tinted by a shade constant across the cell. This shade is based on the orientation of the cell relative to your 3-D light source.
- **Gouraud:** This plot type offers a more continuous and much smoother shading than Paneled shading, but also results in slower plotting and larger print files. Gouraud shading is not continuous across zone boundaries. Gouraud shading is not available for finite-element volume zones when blanking is included. A finite-element volume zone set to use Gouraud shading will revert to Paneled shading when blanking is included.

  IJK-ordered data with Surfaces to Plot set to Exposed Cell Faces, faces exposed by blanking will revert to Paneled shading.

Figure 15-3 shows two shade plots. The one on the left uses a Paneled lighting effect and the one on the right a Gouraud lighting effect.

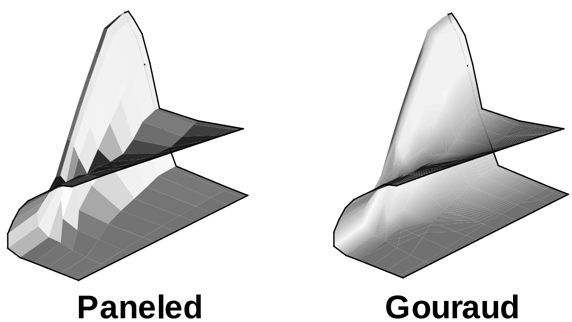
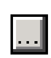


Figure 15-3. Plots showing Paneled and Gouraud lighting effects.

15.2. Three-Dimensional Light Source

Open the Light Source dialog by clicking the  next to Lighting zone effect check box on the sidebar, or by selecting Light Source from the Plot menu. The light source position is indicated by a dot over the origin of the 3-D orientation axes displayed in the Light Source Position control.

15.2.1. Light Source Position

The 3-D light source is a point of light infinitely far from the drawing area. The 3-D light source may be different for each frame. By default, the light source is positioned at infinity along the eye-origin axis. You can specify its location as a point on a hemisphere with a pole along the eye-origin axis. Control the location with the Light Source Position control on the Light Source dialog, shown in Figure 15-4.

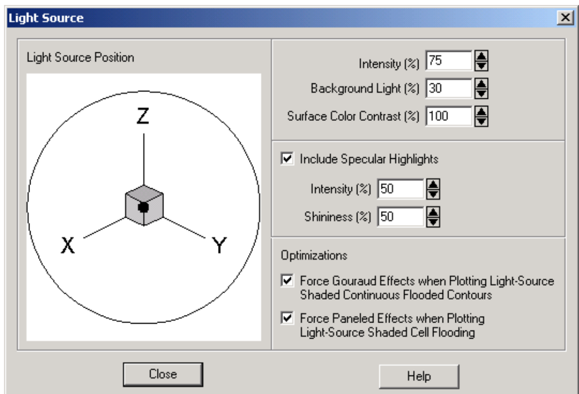


Figure 15-4. The Light Source dialog.

The following options are available:

In the Light Source Position area, click the mouse at the desired light source location. You may also drag the mouse to move the light source position. As the light source position moves away from the eye origin ray, its representation appears as an arrow. The length of the arrow indicates how far from the eye origin ray the light source position is. At the eye origin ray, the arrow is pointing directly into the screen, so just a dot is visible; at the horizon (on the circle surrounding the 3-D orientation axes in the Light Source Position control), the arrow is at its longest.

15.2.2. Light Source Attributes

The following options are available:

- **Intensity (%):** Controls the amount of lighting effect produced by the directional light source. An intensity of 100 produces the maximum contrast between lit and unlit areas, and fully lit areas use the full surface color. Lesser values produce less contrast between lit and unlit areas, and fully lit areas use darker colors. An intensity of zero means the light source produces no contrast between lit and unlit areas, and all areas are black.

- **Background Light (%):** Controls the amount of lighting effect applied to all objects regardless of the light source position. A background light of zero means that areas unlit by the directional light source receive no lighting at all and are entirely black, while areas lit by the directional light source get only the effect of that light. Larger values produce more lighting effect in areas not lit by the directional light source, making these areas show some of the surface color. A background light of 100 means that all areas are lit by the maximum amount, and areas unlit by the directional light source use the full surface color.

  Note: Intensity and Background Light are cumulative; they can add up to more than 100 and result in colors lightened beyond the base surface color. For example, reds will become pink and grays will become white.

- **Surface Color Contrast (%):** Controls the contrast of the color of light source shaded surfaces before applying lighting effects. A surface color contrast of 100 means that light source shaded surfaces use the full surface color for applying lighting effects. Lesser values mean that the surface color is blended with progressively more white, making light source shaded surface colors lighter. A surface color contrast of zero means that colors are pure white before applying lighting effects; the plot will only be shades of gray.

15.2.3. Specular Highlighting

Specular Highlighting adds the semblance of reflected light to 3-D shaded or flooded objects.

Options include:

- **Include Specular Highlighting:** Turns on/off specular highlight for all light-source shaded objects in the plot.
- **Intensity (%):** Controls intensity of specular highlights (that is, the amount of reflected light, which controls the amount of whiteness at the peak of the highlight).
- **Shininess:** Controls shininess of specular highlight (that is, roughly the size and spread of specular highlight).

15.2.4. Lighting Optimizations

Some combinations of lighting type and plot style may result in very slow redrawing of plots. Tecplot provides lighting optimizations to avoid such conditions and instead draws a similar, but less intensive plot. These optimizations are on by default. Turn them off if you need to see the exact effects you have specified. You may want to turn off the graphics cache before turning off those optimizations for plots with large amounts of data. (See Section 30.3.2, "Graphics Cache," for information on the graphics cache.)

CHAPTER 16 Text, Geometries and Images

You can enhance any plot, or create a drawing from scratch, using Tecplot's text and drawing tools. Tecplot provides tools for creating polylines, circles, ellipses, squares, and rectangles, in addition to a text tool for creating titles, labels, and any other text you want. In addition to adding text and geometries, you can insert BMP, JPEG, or PNG images to enhance your plot. You can annotate line and Cartesian plots.

Alternatively, pure sketches are created with the "Sketch" plot type. Figure 16-1 shows a sketch created with Tecplot drawing tools.

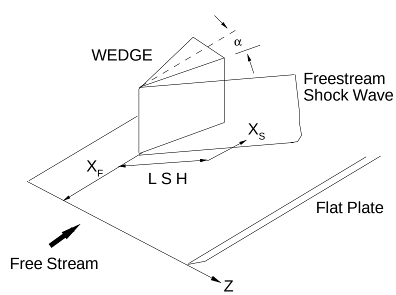


Figure 16-1. A sketch created with Tecplot.

16.1. Text

Text strings are used for plot titles and labels. Figure 16-2 shows a small sample of the types of text you can create with Tecplot.

To add text to your plot or sketch:

1. From the sidebar, choose the Text tool Ab, or from the Insert menu, choose Text. In either case, when you move the pointer into the workspace it becomes a cross-hair.

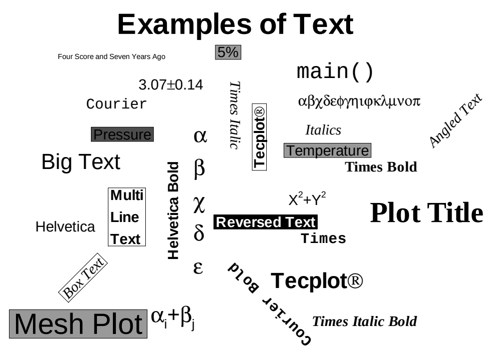


Figure 16-2. Text examples created with Tecplot.

2. Click anywhere in a frame to indicate the location of the text. The Text Details dialog appears, as shown in Figure 16-3.

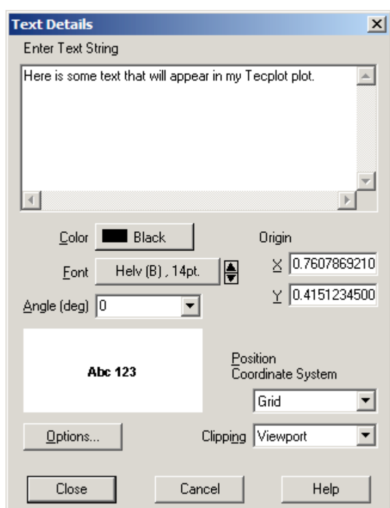


Figure 16-3. The Text Details dialog.

3. Enter the desired text in the text area labeled Enter Text String. As you type, the text you enter is echoed in the frame.
4. Modify the text color, font, angle, height, position, and clipping as desired.
5. Click Options to add a box around your text, modify the line spacing for multi-line text, or set a text anchor location.

6. Click Close to place the text and close the dialog, or click elsewhere in the work area to place additional text.

16.1.1. Text Editing

To edit text already placed:

1. From the sidebar, choose the Selector tool .
2. Double-click on the text you wish to edit. The Text Details dialog appears.
3. Make the desired modifications to the text.
4. To cancel your modifications, click Cancel.

To modify the text attributes of several text items at once, drag the mouse to create a group select rubber band box to enclose the items you want to edit. In the Group Select dialog, deselect all object type check boxes except for Text, and click OK. Then click Quick Edit on the sidebar, or from the Edit menu. The Quick Edit dialog appears. You can use the Quick Edit dialog to modify the font, size, and color of your text items.

16.1.2. Text Deletion

To delete text:

1. From the sidebar choose the Selector tool .
2. Select the text you wish to delete, then from the Edit menu, choose Clear, or press Delete. A confirmation dialog appears asking if you are sure you want to clear the selected text.
3. Click OK.

16.1.3. Text Fonts

You can create text using any of Tecplot's eleven built-in fonts. Samples of all eleven fonts are shown in Figure 16-4. Any of these eleven build-in fonts may be selected by using the Select Font dialog, which is displayed by pressing the Font button on the Text Details dialog. Eight of the fonts (Courier, Helvetica and Times) are collectively referred to as English fonts. You can embed Greek, Math, and User-Defined characters into English-font strings by enclosing them with special character sequences called text formatting tags, together with the keyboard characters that correspond to the desired characters in the chosen font, as shown in Figure 16-5.

The text formatting tags and their effects are as follows (format tags are not case sensitive and may be either upper or lower case):

- **...:** Draw the enclosed characters using bold (if possible).
- **<i>...</i>:** Draw the enclosed characters using italics (if possible).
- **<verbatim>...</verbatim>:** Draw the enclosed characters verbatim.
- **_{...}:** Draw the enclosed characters as subscripts.
- **^{...}:** Draw the enclosed characters as superscripts.

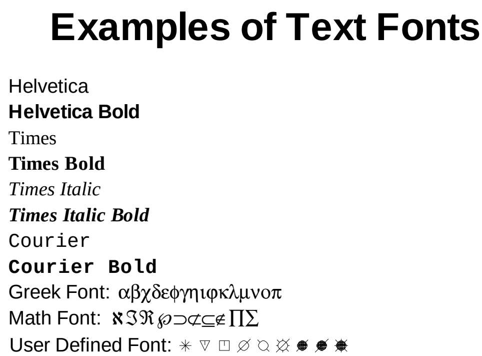


Figure 16-4. Examples of Tecplot's text fonts.

- **<greek>...</greek>:** Draw the enclosed characters using the Greek font.
- **$...$:** Draw the enclosed characters using the Math font.
- **<userdef>...</userdef>:** Draw the enclosed characters using the User-Defined font.
- **<helvetica>...</helvetica>:** Draw the enclosed characters using the Helvetica font.
- **<times>...</times>:** Draw the enclosed characters using the Times font.
- **<courier>...</courier>:** Draw the enclosed characters using the Courier font.
- **<userdef>...</userdef>:** Draw the enclosed characters from the User-Defined font.

For example, to insert the Greek letter Φ into a Tecplot text string, use the combination "**<greek>F</greek>**." A serif registered trademark symbol (such as that accompanying the word "Tecplot" in Figure 16-2) can be created with the combination "**R**."

Similarly, you can produce subscripts or superscripts by enclosing any characters with **_{...} or ^{...}**, respectively. Tecplot has only one level of superscripts and subscripts; expressions requiring additional levels, such as e^{x^2}, must be created by hand using multiple Tecplot text strings. If you alternate subscripts and superscripts, Tecplot positions the superscript directly above the subscript. Thus, the string **a_b^c** produces a_b^c. To produce consecutive superscripts, enclose all superscript characters in a single pair of tags. The string **x^(a+b)** produces $x^{(a+b)}$ in your plot.

Character Index	English Text	Greek	Math	User Defined	Character Index	English Text	Greek	Math	User Defined
32	(space)				80	P	Π	∠	⌀
33	!	!	ϒ		81	Q	Θ	∇	○
34	"	∀	′		82	R	Ρ	®	◇
35	#	#	≤		83	S	Σ	©	⌀
36	$	∃	⁄		84	T	Τ	™	☼
37	%	%	∞		85	U	Υ	∏	☼
38	&	&	ƒ		86	V	ς	√	☼
39	’	∍	♣		87	W	Ω	·	●
40	(	(	♦		88	X	Ξ	¬	●
41	)	)	♥		89	Y	Ψ	∧	●
42	*	∗	♠		90	Z	Ζ	∨	●
43	+	+	↔		91	[	[	⇔	
44	,	,	←		92	\	∴	⇐	
45	-	−	↑		93	]	]	⇑	
46	.	.	→		94	^	⊥	⇒	
47	/	/	↓		95	_	_	⇓	
48	0	0	°		96	‘	‾	◊	
49	1	1	±		97	a	α	〈	✸
50	2	2	″		98	b	β	®	✸
51	3	3	≥		99	c	χ	©	✸
52	4	4	×		100	d	δ	™	✸
53	5	5	∝		101	e	ε	∑	✸
54	6	6	∂		102	f	φ	⎛	⏀
55	7	7	•		103	g	γ	⎜	
56	8	8	÷		104	h	η	⎝	
57	9	9	≠		105	i	ι	⎡	
58	:	:	≡		106	j	ϕ	⎢	
59	;	;	≈		107	k	κ	⎣	
60	<	<	…		108	l	λ	⎧	
61	=	=	⏐		109	m	μ	⎨	
62	>	>	—		110	n	ν	⎩	
63	?	?	↵		111	o	ο	⎪	
64	@	≅	ℵ		112	p	π		
65	A	Α	ℑ		113	q	θ	〉	
66	B	Β	ℜ	+	114	r	ρ	∫	
67	C	Χ	℘	×	115	s	σ	⌠	
68	D	Δ	⊗	✳	116	t	τ	⎮	
69	E	Ε	⊕	△	117	u	υ	⌡	
70	F	Φ	∅	▽	118	v	ϖ	⎞	
71	G	Γ	∩	◫	119	w	ω	⎟	
72	H	Η	∪	◇	120	x	ξ	⎠	
73	I	Ι	⊃	⊙	121	y	ψ	⎤	
74	J	ϑ	⊇	⊙	122	z	ζ	⎥	
75	K	Κ	⊄	★	123	{	{	⎦	
76	L	Λ	⊂	∘	124	|	|	⎫	
77	M	Μ	⊆	+	125	}	}	⎬	
78	N	Ν	∈	○	126	~	~	⎭	
79	O	Ο	∉	⌀	127				

Character Index	Extended Character	Character Index	Extended Character
160		208	Ð
161	¡	209	Ñ
162	¢	210	Ò
163	£	211	Ó
164	¤	212	Ô
165	¥	213	Õ
166	¦	214	Ö
167	§	215	×
168	¨	216	Ø
169	©	217	Ù
170	ª	218	Ú
171	«	219	Û
172	¬	220	Ü
173	-	221	Ý
174	®	222	Þ
175	¯	223	ß
176	°	224	à
177	±	225	á
178	²	226	â
179	³	227	ã
180	´	228	ä
181	µ	229	å
182	¶	230	æ
183	·	231	ç
184	¸	232	è
185	¹	233	é
186	º	234	ê
187	»	235	ë
188	¼	236	ì
189	½	237	í
190	¾	238	î
191	¿	239	ï
192	À	240	ð
193	Á	241	ñ
194	Â	242	ò
195	Ã	243	ó
196	Ä	244	ô
197	Å	245	õ
198	Æ	246	ö
199	Ç	247	÷
200	È	248	ø
201	É	249	ù
202	Ê	250	ú
203	Ë	251	û
204	Ì	252	ü
205	Í	253	ý
206	Î	254	þ
207	Ï	255	ÿ

Figure 16-5. Character indices in Tecplot.

To insert a tag into text literally, precede the first angle bracket with a backslash ("\"). To insert a backslash in the text, just type two backslashes ("\\"). In ASCII input files, the number of backslashes must be doubled (two to precede a special character, four to create a backslash) because the Preplot program also requires a backslash to escape special characters.

16.1.4. European Characters

Tecplot supports the ISO-Latin one-character encodings. Characters in the ASCII ordinal range from 160-255 are now available, providing support for most of the major European languages.

Figure 16-5 shows the characters supported by Tecplot. Note that the two right-hand columns represent the extended European characters. Using text formatting tags to produce Greek, Math, or User-Defined characters only works with characters in the range 32-126 and is not available for the extended European characters.

If your keyboard is configured to produce European characters, then the European characters should appear and print automatically with no further setup.

16.1.5. European Characters and Character Codes

If your keyboard is not configured to produce a specific European character you can generate it by including the sequence ****_nnn_ in your text where _nnn_ the character index value is taken from the character index table found in Figure 16-5. For example, if your keyboard will not generate the é and you want to show the word "latté," you would enter:

```
latt\233
```

You may need to redraw to get the characters to display clearly.

16.1.6. Text Size and Position

Text can be specified using either of two coordinate systems: frame or grid. In the frame coordinate system, text is positioned relative to the frame, but not to any data that might be in the frame. Thus, if you change the view of the data (for example, by zooming, or translating, or rotating), the frame coordinate text does not move. In the grid coordinate system, on the other hand, the text does move as you alter the view by zooming or translating. If you want to annotate individual data points, it is wise to use grid coordinates.

Once you have chosen a coordinate system, you can choose the units and height for your text.

16.1.6.1. Text Size. You specify the text size and font by using the Select Font dialog, which is launched by pressing the Font button on the Text Details dialog. For text using the Frame coordinate system, you may specify your text size in point units or as a percentage of the frame height. For text in the grid coordinate system, you may also specify your text size in the same units as the data in your frame.

To change the size of your text:

1. In the workspace, double-click on the text for which you want to change the size. The Text Details dialog appears.
2. On the Text Details dialog, press the Font button. The Select Font dialog appears.
3. Specify the units in which you want to express the size of the text. The options are Point and Frame(%). For grid mode text an additional option, Grid Units, is available.
4. In the Height text field, specify the desired height.

If you simply want to change the text height, without changing the units in which the height is specified, you can simply click on the up/down arrows to the right of the Font button.

16.1.6.2. Text Position. You specify the anchor position for each piece of text by clicking at the desired location in the frame. You have some guidance in this by using the workspace rulers, and you can gain some specific control by using the Snap to Paper or Snap to Grid sidebar options. However, for complete control over the position of your anchor point, you can specify exact coordinates for the anchor position, or origin of the text, using the Origin controls in the Text Details dialog.

To specify an exact position for the text anchor position:

1. In the workspace, double-click the text for which you want to specify an origin. The Text Details dialog appears.
2. Enter a value for the X-position in the text field labeled X and a value for the Y-position in the text field labeled Y. For grid mode geometries in the Polar plot type, values are expressed in units of Theta and R. Values are expressed in the coordinate system specified for the text, either frame units or grid units.

The text anchor can be at any of nine locations with respect to the text: vertically, one of Headline, Midline, or Baseline, and horizontally, one of Left, Center, or Right. The anchor location determines whether text is centered about the text origin, or right, left, top, or bottom justified.

To specify the anchor location:

1. In the workspace, double-click the text for which you want to specify the anchor position. The Text Details dialog appears.
2. Click Options. The Text Options dialog appears as shown in Figure 16-6.
3. Under the heading Text Anchor Location, select one of the nine option buttons corresponding to the allowable anchor locations.

16.1.7. Text Boxes

You can put a plain or filled box around any piece of text. A filled text box obscures all portions of the plot under the text box; a plain box is a transparent outline.

To add a text box:

1. In the workspace, double-click the text for which you want to specify a text box. The Text Details dialog appears.

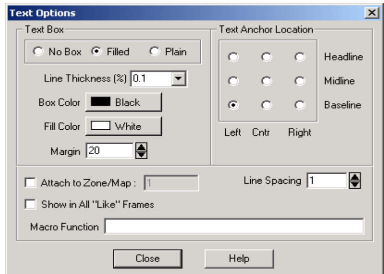


Figure 16-6. The Text Options dialog.

2. Click Options. The Text Options dialog appears.
3. Choose one of the option buttons Filled or Plain.
4. Specify the line thickness and box outline color, and for a filled box, select a fill color.
5. Specify the margin around the text as a percentage of the text character height.

16.1.8. Text Scope

By default, text is displayed only in the frame in which it is created. You can, however, choose to have the text appear in all frames using the same data set as the one in which the text was created. Such frames are called like frames.

To propagate text to all like frames:

1. In the workspace, double-click the text that you want to appear in like frames. The Text Details dialog.
2. Click Options. The Text Options dialog appears.
3. Select the check box labeled Show in All Like Frames.
4. Click Close.

16.1.9. Text Attachment to Zones or Mappings

By default, text is always displayed, regardless of which zones or mappings are currently active. Sometimes, however, you use text to highlight a particular feature of a specific zone or mapping, and that text is meaningless unless the zone or map is displayed as well. In such cases, you can control the display of the text by attaching the text to the zone or map.

To attach text to a zone or map:

1. In the workspace, select the text that you want to attach to a zone or map.
2. On the sidebar, click Object Details. The Text Details dialog appears.
3. Click Options. The Text Options dialog appears.
4. Select the check box labeled Attach to Zone/Map.

5. Enter a zone or mapping number in the text field immediately to the right of the Attach to Zone/Map check box.

Once attached to a zone or map, the text is displayed when the zone or map is active, and not displayed when the zone or map is inactive. In line plots, this option is controlled by just the mapping number. In 2- and 3-D plots, it is controlled by just the zone number.

16.1.10. Text Clipping

Clipping refers to displaying only that portion of an object that falls within a specified clipping region of the plot. If you have specified your text position in the Frame coordinate system, the text will be clipped to the frame—any portion of the text that falls outside the frame is not displayed. If you have specified the Grid coordinate system, you can choose to clip your text to the frame or the viewport. If you choose Viewport clipping, the text is clipped to the viewport. The size of the viewport depends on the plot type as follows:

- **3D Cartesian:** The viewport is the same as the frame, so viewport clipping is the same as frame clipping.
- **2D Cartesian/XY Line:** The viewport is defined by the extents of the X and Y axes. You can modify this with the Area page of the Axis Details dialog.
- **Polar Line/Sketch:** By default, the viewport is the same as the frame. You can modify this with the Area page of the Axis Details dialog.

16.1.11. Dynamic Text

You can add special placeholders to text that changes with the data or the display environment. For example, you can add a date placeholder that Tecplot will replace with the current date at each Redraw. Similarly, you can add a zone name or variable name placeholder.

The complete list of placeholders is as follows:

Variables	Notes
&(AUXDATASET:*name***)**	The named auxiliary data attached to the current frame.
&(AUXFRAME:*name***)**	The named auxiliary data attached to the data set of the current frame.
&(AUXZONE[*nnn***]:***name***)**	The named auxiliary data attached to the data set of the current frame.
&(AXISMAXA)	Maximum value of current Theta-axis range.
&(AXISMAXR)	Maximum value of current R-axis range.
&(AXISMAXX)	Maximum value of current X-axis range.
&(AXISMAXY)	Maximum value of current Y-axis range.

Variables	Notes
&(AXISMAXZ)	Maximum value of current Z-axis range.
&(AXISMINA)	Minimum value of current Theta-axis range.
&(AXISMINR)	Minimum value of current R-axis range.
&(AXISMINX)	Minimum value of current X-axis range.
&(AXISMINY)	Minimum value of current Y-axis range.
&(AXISMINZ)	Minimum value of current Z-axis range.
&(BYTEORDERING)	Platform's byte ordering: "**INTEL**" or "**MOTOROLA**"
&(COLORMAPDYNAMIC)	Returns one if the color map is dynamic, zero if static.
&(DATE)	Replaced with the current date in the format *dd Mon yyyy*.
&(DATASETFNAME[*nnn***])**	Data set file name of the *nth* file associated with the current data set. If *n* is omitted then all data set file names are show, separated by new lines.
&(DATASETTITLE)	Replaced with the current data set title.
&(ENDSLICEPOS)	Replace with the position of the ending slice plane.
&(EXPORTISRECORDING)	Returns "YES" if currently recording, otherwise returns "NO."
&(FRAMENAME)	Replaced with the current plot.
&(INBATCHMODE)	A value of one if Tecplot is in batch mode, zero if interactive.
&(ISDATASETAVAILABLE)	A value of one if a data set exists for the current frame, zero if nonexistent.
&(ISOSURFACELEVEL[*nnn***])**	Replace with the value of the contour variable on the *nnn*th iso-surface.
&(LAYOUTFNAME)	Replaced with the name of the current layout file.
&(LOOP)	Innermost loop counter.
&(MACROFILEPATH)	Path to the directory containing the most recently opened macro file.
&(MAXA)	Maximum value for the Theta variable. The value is calculated from the zone assigned to the lowest numbered active line mapping.

Variables	Notes
`&(MAXB)`	Maximum value for blanking variable. If the plot type is 2D or 3D Cartesian, the value is calculated from the current set of active zones. If the plot type is XY or Polar Line, the value is calculated from the zone assigned to the lowest numbered active line mapping.
`&(MAXC)`	Maximum value for contour variable. If the plot type is 2D or 3D Cartesian, the value is calculated from the current set of active zones. If the plot type is XY or Polar Line, the value is calculated from the zone assigned to the lowest numbered active line mapping.
`&(MAXI)`	I-dimension for the lowest numbered active zone for 2D and 3D Cartesian plot types. For line plots this represents the maximum I-value for the zone assigned to the lowest numbered active line mapping. For finite-element data, this represents the number of nodes in the lowest numbered active zones.
`&(MAXJ)`	J-dimension for the lowest numbered active zone for 2D and 3D Cartesian plot types. For line plots this represents the maximum J-value for the zone assigned to the lowest numbered active line mapping. For finite-element data, this shows the number of elements in the lowest numbered active zone.
`&(MAXK)`	K-dimension for the lowest numbered active zone for 2D and 3D Cartesian plot types. For line plots this represents the maximum K-value for the zone assigned to the lowest numbered active line mapping. For finite-element data, this shows the number of nodes per element for the lowest numbered active zone.
`&(MAXR)`	Maximum value for the R variable. The value is calculated from the zone assigned to the lowest numbered active line mapping.
`&(MAXS)`	Maximum value for scatter sizing variable for the currently active zones.
`&(MAXU)`	Maximum value for variable assigned to the X-vector component for the currently active zones.

Variables	Notes
&(MAXV)	Maximum value for variable assigned to the Y-vector component for the currently active zones.
&(MAXVAR[*nnn*])	Maximum value of variable *nnn*.
&(MAXW)	Maximum value for variable assigned to the Z-vector component for the currently active zones.
&(MAXX)	Maximum value for variable assigned to the X-axis. For 2D or 3D Cartesian plots, the value is calculated from the current set of active zones. For line plots, the value is calculated from the zone assigned to the lowest numbered active line mapping.
&(MAXY)	Maximum value for variable assigned to the Y-axis. For 2D or 3D Cartesian plots, the value is calculated from the current set of active zones. For line plots, the value is calculated from the zone assigned to the lowest numbered active line mapping.
&(MAXZ)	Maximum value for variable assigned to the Z-axis for the currently active zones.
&(MINA)	Minimum value for the Theta variable. The value is calculated from the zone assigned to the lowest numbered active line mapping.
&(MINB)	Minimum value for blanking variable. For 2D or 3D Cartesian plots, the value is calculated from the current set of active zones. For line plots, the value is calculated from the zone assigned to the lowest numbered active line mapping.
&(MINC)	Minimum value for contour variable. For 2D or 3D Cartesian plots, the value is calculated from the current set of active zones. For line plots, the value is calculated from the zone assigned to the lowest numbered active line mapping.
&(MINR)	Minimum value for the R variable. The value is calculated from the zone assigned to the lowest numbered active line mapping.
&(MINS)	Minimum value for scatter sizing variable for the currently active zones.
&(MINU)	Minimum value for variable assigned to the X-vector component for the currently active zones.

Variables	Notes
&(MINV)	Minimum value for variable assigned to the Y-vector component for the currently active zones.
&(MINVAR[*nnn*])	Minimum value of variable *nnn*.
&(MINW)	Minimum value for variable assigned to the Z-vector component for the currently active zones.
&(MINX)	Minimum value for variable assigned to the X-axis. For 2D or 3D Cartesian plots, the value is calculated from the current set of active zones. For line plots, the value is calculated from the zone assigned to the lowest numbered active line mapping.
&(MINY)	Minimum value for variable assigned to the Y-axis. For 2D or 3D Cartesian plots, the value is calculated from the current set of active zones. For line plots, the value is calculated from the zone assigned to the lowest numbered active line mapping.
&(MINZ)	Minimum value for variable assigned to the Z-axis for the currently active zones.
&(NUMFRAMES)	Number of frames.
&(NUMPLANES)	Returns number of graphics bit-planes
&(NUMVARS)	Number of variables in current data set.
&(NUMXYMAPS)	Number of XY-maps assigned to the current frame.
&(NUMZONES)	Number of zones in current data set.
&(OPSYS)	Returns **1=UNIX**, **2=DOS**.
&(PAPERHEIGHT)	Paper height in inches.
&(PAPERWIDTH)	Paper width in inches.
&(PLATFORM)	Platform name (such as "**SGI**" or "**WINDOWS**").
&(PLOTTYPE)	Plot type for the current frame: Zero for Sketch, one for XY Line, two for Cartesian 2D, three for Cartesian 3D, and four for PolarLine.
&(PRINTFNAME)	Replaced with the name of the current print file.
&(SLICEPLANETYPE)	Replace with the type of slice plane (X-, Y-, Z-, I-, J- or K-planes).
&(STARTSLICEPOS)	Replace with the position of the starting slice plane.
&(STREAMSTARTPOS[*nnn*])	Starting position (X, Y, Z) of the *nnn*th streamtrace.

Variables	Notes
&(STREAMTYPE[*nnn***])**	Type (Surface Line, Volume Line, Volume Ribbon, Volume Rod) of the *nnn*th streamtrace.
&($*string***)**	Replaced with the value of the system environment variable *string*.
&(TECHOME)	Path to the Tecplot home directory.
&(TECPLOTVERSION)	Returns Tecplot Version. (Currently returns "**100**.")
&(TIME)	Replaced with the current time in the format *hh:mm:ss.*
&(VARNAME[*nnn***])**	Replaced with the variable name for variable *nnn*.
&(ZONEMESHCOLOR[*nnn***])**	Color of the mesh for the *nnn*th zone.
&(ZONENAME[[ACTIVEOFF-SET=]*nnn***])**	Replaced with the zone name for zone *nnn*. If ACTIVEO-FFSET= is used, the integer value indicates the *nnn*th active zone for field plots and the zone associated with the *nnn*th active line mapping for line plots.

The placeholders must be typed exactly as shown, except that the *nnn* in the zone name and variable name placeholders should be replaced by the actual number of the zone or variable, such as **&(ZONENAME[3])** or **&(VARNAME[2])**.

You can, of course, embed the dynamic text strings in text records in a Tecplot-format data file, as in the following example:

```
TEXT CS=FRAME HU=POINT T="&(DATE)"
```

System environment variables can be accessed directly from Tecplot by using the following: **&($***string***)**, where *string* is the name of your environment variable. Using environment variables within Tecplot can add another degree of flexibility by taking advantage of your customized environment. If an environment variable is missing, the environment variable name itself will appear on the screen.

16.2. Geometries

Geometries in Tecplot are simply line drawings. Geometries include polylines (a set of line segments), circles, ellipses, rectangles, and squares. Images are also considered geometries, but they are discussed below in Section 16.3, "Images." Polylines may include arrowheads at either or both ends. Figure 16-7 shows some examples of geometries.

16.2.1. Geometry Creation

You create geometries by drawing them in a frame using one of the following sidebar tools:

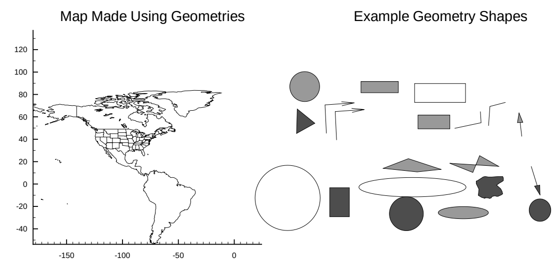


Figure 16-7. Sample geometries.

- The button to draw polylines. You can also choose this tool by choosing Polyline from the Insert menu.
- The button to draw circles. You can also choose this tool by choosing Circle from the Insert menu.
- The button to draw ellipses. You can also choose this tool by choosing Ellipse from the Insert menu.
- The button to draw squares. You can also choose this tool by choosing Square from the Insert menu.
- The button to draw rectangles. You can also choose this tool by choosing Rectangle from the Insert menu.

After choosing any of these tools, when you move the pointer into the workspace, it becomes a cross-hair. Click once in a frame to set the anchor position for the geometry. To complete the geometry, follow the instructions for each geometry type as follows:

- **For polylines**: Move the mouse (without dragging) to the desired end point of the first line segment, then click the left mouse button. Move the pointer to the next end point, click, and so on. After placing the last segment, double-click on the final end point or press Esc on your keyboard. To draw a horizontal or vertical line segment, press the H or V keys, respectively, while drawing the segment. After you place the segment's end point, the horizontal or vertical restriction is lifted. To lift the horizontal or vertical line segment restriction without placing the end point, press A on your keyboard. You can draw unconnected line segments in a single polyline; press U on your keyboard to "lift the pen." You can then move the pointer to the start of the next line segment.

- **For circles**: Click at the desired center point of the circle; drag the mouse until the circle is the desired radius, then release.
- **For ellipses**: Click at the desired center point of the ellipse; drag the mouse until the ellipse is the desired size and shape, then release.
- **For squares**: The anchor point of the square is either the lower left-hand corner or the upper right corner of the square. Drag the mouse to the right of the anchor to create a square with the anchor at lower left; drag the mouse to the left to create a square with the anchor at upper right. Release when the square is the desired size.
- **For rectangles**: Drag the mouse until the rectangle is the desired size and shape. In contrast to squares, rectangles can propagate in any direction.

16.2.2. Geometry Modification

You can modify the outline color, line pattern, line thickness, fill color, and position of your geometries, along with other attributes that affect the relationship of the geometry to the frame and the data set (if any). Individual geometry types have their own specific attributes.

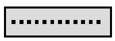
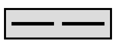
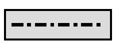
16.2.2.1. Geometry Colors. Geometries may be filled or unfilled. Unfilled geometries have a single color attribute: the line color. Filled geometries have both a line color and a fill color. You can control the colors from either the Geometry dialog or the Quick Edit dialog.

To specify a geometry's colors:

1. In the workspace, select the geometry or geometries for which you want to modify the color.
2. On the sidebar, click Object Details. If you have selected a single geometry, the Geometry dialog appears. Continue with Step 3. If you have selected multiple geometries, the Quick Edit dialog appears. Continue with Step 4.
3. (Geometry dialog) Specify the geometry's outline color by pressing the Color button next to the Outline Color label. The Select Color dialog appears, allowing you to select one of Tecplot's 64 colors. If you want the geometry filled, select the check box labeled Fill Color, then select a fill color by clicking on the Fill Color button and selecting the desired color from the Select Color dialog.
4. (Quick Edit dialog) Specify the outline color for all selected geometries by selecting the Line option button, then clicking the Color button. If you want the geometries filled, select the Fill option button, then click on one of the basic colors. The X button causes fill to be turned off (when Fill option button is selected), or causes line color to match fill color (when Line option button is selected, and fill is present). If no fill is present, the X button also has no effect.

16.2.2.2. Geometry Line Patterns. The outline of a geometry, or any polyline, can be drawn in any of Tecplot's line patterns. You control the line pattern from either the Geometry dialog or the Quick Edit dialog.

To specify a geometry's line pattern:

1. In the workspace, select the geometry or geometries for which you want to modify the line pattern.
2. On the sidebar, click Object Details. If you have selected a single geometry, the Geometry dialog appears. Continue with Step 3. If you have selected multiple geometries, the Quick Edit dialog appears. Continue with Step 4.
3. (Geometry dialog) Specify the geometry's line pattern by choosing one of Tecplot's six line patterns (Solid, Dashed, Dotted, Dash Dot, Long Dash, and Dash Dot Dot) from the Line Pattern menu. Specify a pattern length by either choosing a pre-set value from the drop-down or entering a percentage of the frame height in the text field.
4. (Quick Edit dialog) Specify the line pattern for all selected geometries by selecting the appropriate line pattern button, as follows:
   -  Chooses a solid line.
   -  Chooses a dotted line.
   -  Chooses a dashed line.
   -  Chooses a long dashed line.
   -  Chooses an alternating dot-and-dash line.
   -  Chooses an alternating dash-and-two-dots line.

   Specify a pattern length by clicking on the Pttrn Length button, and choosing either one of the pre-set values or Enter. If you choose Enter, an Enter Value dialog appears; enter the desired pattern length as a percentage of frame height and click OK.

16.2.2.3. Geometry Line Thickness. You can control the thickness of polylines and the outlines of other geometries. You control the line thickness from either the Geometry dialog or the Quick Edit dialog.

To specify a geometry's line thickness:

1. In the workspace, select the geometry or geometries for which you want to modify the line thickness.
2. On the sidebar, click Object Details. If you have selected a single geometry, the Geometry dialog appears. Continue with Step 3. If you have selected multiple geometries, the Quick Edit dialog appears. Continue with Step 4.
3. (Geometry dialog) Specify the geometry's line thickness by either choosing a pre-set value from the drop-down or entering a percentage of the frame height in the Line Thickness (%) field.

4. (Quick Edit dialog) Specify a line thickness by clicking Line Thcknss, and choosing either one of the pre-set values or Enter. If you choose Enter, an Enter Value dialog appears; enter the desired line thickness as a percentage of frame height and click OK.

16.2.2.4. Geometry Coordinate Systems. Geometries, like text, can be positioned using either frame coordinates or grid coordinates. Grid coordinates are used by default. Unlike text, however, geometries are sized using the units of the coordinate system in which they are placed. That is, geometries that use frame coordinates use frame units to specify both the position and the size, while geometries that use grid coordinates use grid units to specify both the position and size. When frame coordinates are used, the geometry is locked at a particular location within the frame, and actions which modify the view (such as zooming, translating, and rotating) have no effect on the geometry. When grid coordinates are used, the geometry is part of the view, subject to change when the view changes. For example, a circle in frame coordinates retains its size and shape when you zoom into the plot, while a circle in grid coordinates grows as you zoom in. You specify the coordinate system using the Geometry dialog.

To specify a geometry's coordinate system:

1. In the workspace, select the geometry for which you want to specify a coordinate system.
2. On the sidebar, click Object Details. The Geometry dialog appears.
3. Select either Frame or Grid from the Coordinate System drop-down list.

16.2.2.5. Geometry Position. You specify the anchor position for a new geometry by clicking anywhere in the frame. You have some guidance in this by using the workspace rulers, and you can gain some specific control by using the Snap to Paper or Snap to Grid sidebar options. Users can also move a geometry to the front or the back of a group of geometries by selecting it, then choosing Edit, Push or Edit, Pop. However, for complete control over the position of your anchor point, you can specify exact coordinates for the anchor position, or origin of the geometry, using the Origin controls in the Geometry dialog.

To specify an exact position for a geometry's origin:

1. In the workspace, select the geometry for which you want to specify an origin.
2. On the sidebar, click Object Details. The Geometry dialog appears.
3. Enter a value for the X-position in the text field labeled X, a value for the Y-position in the text field labeled Y, (and, for 3-D line geometries brought in from data files, a value for the Z-position in the text field labeled Z). Values are expressed in the coordinate system specified for the geometry, either frame units or grid units. For grid mode geometries in the Polar plot type, values are expressed in units of Theta and R.

16.2.2.6. Geometry Clipping. Clipping refers to displaying only that portion of an object that falls within a specified clipping region of the plot. If you have specified your geometry position in the Frame coordinate system, the geometry will be clipped to the frame—any portion of the geometry that falls outside the frame is not displayed. If you have specified the Grid coordinate system, you can choose to clip your geometry to the frame or the viewport. If you choose Viewport clipping, the geometry is clipped to the viewport. The size of the viewport depends on the plot type as follows:

- **3D Cartesian:** The viewport is the same as the frame, so viewport clipping is the same as frame clipping.
- **2D Cartesian/XY Line:** The viewport is defined by the extents of the X and Y axes. You can modify this with the Area page of the Axis Details dialog.
- **Polar Line/Sketch:** By default, the viewport is the same as the frame. You can modify this with the Area page of the Axis Details dialog.

16.2.2.7. Geometry Draw Order. Geometries can be draw either before the data, or after the data. If a geometry is drawn before the data, the plot layers, such as mesh, contour lines, etc. will be drawn on top of the geometry. If a geometry is drawn after the data, the geometry will be drawn last, obscuring the data. The draw order setting is particularly useful for Images, which are discussed in Section 16.3. “Images”

To change the draw order for a geometry:

1. In the workspace, double-click the geometry for which you want to change the draw order. The Text Details dialog appear.
2. In the Draw Order field, select either Before Data or After Data.

16.2.2.8. Geometry Scope. By default, a geometry is displayed only in the frame in which it is created. You can, however, choose to have the geometry appear in all frames using the same data file as the one in which the geometry was created. Such frames are called like frames.

To propagate a geometry to all like frames:

1. In the workspace, select the geometry that you want to appear in like frames.
2. On the sidebar, click Object Details. The Geometry dialog appears.
3. Select the check box labeled Show in All Like Frames.

16.2.2.9. Geometry Attachment to Zones or Mappings. By default, geometries are always displayed, regardless of which zones or mappings are currently active. Sometimes, however, you draw a geometry to highlight a particular feature of a specific zone or mapping, and that geometry is meaningless unless the zone or map is displayed as well. In such cases, you can control the display of the geometry by attaching the geometry to the zone or map.

To attach a geometry to a zone or map:

1. In the workspace, select the geometry that you want to attach to a zone or map.
2. On the sidebar, click Object Details. The Geometry dialog appears.
3. Select the check box labeled Attach to Zone/Map.

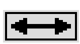
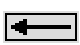
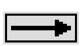
4. Enter a zone or mapping number in the text field immediately to the right of the Attach to Zone/Map check box.

Once attached to a zone or map, the geometry is drawn when the zone or map is active, and not drawn when the zone or map is inactive.

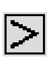
16.2.2.10. Arrowheads and Polylines. You can add arrowheads to either or both ends of any polyline, and control the arrowhead style, size, and angle. You control polyline arrowheads from either the Geometry dialog or the Quick Edit dialog.

To control polyline arrowheads:

1. In the workspace, select the polyline or polylines for which you want to specify arrowhead attachment.
2. On the sidebar, click Object Details. If you have selected a single polyline, the Geometry dialog appears. Continue with Step 3. If you have selected multiple polylines, the Quick Edit dialog appears. Continue with Step 4.
3. (Geometry dialog) Specify where to place arrowheads for the geometry by selecting none, one, or both of the check boxes labeled Start and End beside the label Attachment. If you select one or both of Start and End, specify a style by selecting one of the option buttons labeled Plain, Filled, or Hollow. Specify a size by either choosing a pre-set value or entering a value in the Size (%) field. Specify an angle by either choosing a pre-set value or entering a value in the Angle (deg) field.
4. (Quick Edit dialog) Specify where to place arrowheads by clicking on one of the following:

   -  No arrowheads.
   -  Arrowheads at both ends.
   -  Arrowhead at start of polyline.
   -  Arrowhead at end of polyline.

   Specify a style by clicking on one of the following buttons:

   -  Plain arrowhead style.
   -  Filled arrowhead style.
   -  Hollow arrowhead style.

   Specify a size for the arrowheads by clicking Size, located beneath the Arrowhead Style options, then choosing either one of the pre-set sizes, or Enter. If you choose Enter, an Enter Value dialog appears. Enter the desired size as a percentage of the frame height, then click OK.

Specify an angle for the arrowheads by clicking Angle, then choosing either one of the preset values, or Enter. If you choose Enter, an Enter Value dialog appears. Enter the desired arrowhead angle, which is the angle that one side of the arrowhead makes with the polyline, in degrees.

16.2.2.11. Circle Attributes. Using the Geometry dialog, you can modify the radius of a circle and also the number of line segments used to approximate the circle.

To modify a circle's radius:

1. In the workspace, select the circle for which you want to modify the radius.
2. On the sidebar, click Object Details. The Geometry dialog appears.
3. Enter a new value in the text field labeled Radius.

To modify the number of line segments used to approximate the circle:

1. In the workspace, select the circle for which you want to specify a different number of line segments.
2. On the sidebar, click Object Details. The Geometry dialog appears.
3. Enter a new value in the text field labeled Approximated by Number of Sides.

16.2.2.12. Ellipse Attributes. Using the Geometry dialog, you can modify the horizontal and vertical axes of an ellipse and also the number of line segments used to approximate the ellipse.

To modify an ellipse's axes:

1. In the workspace, select the ellipse for which you want to modify the axes.
2. On the sidebar, click Object Details. The Geometry dialog appears.
3. Enter a new value for the horizontal axis length in the text field labeled Horizontal Axis.
4. Enter a new value for the vertical axis length in the text field labeled Vertical Axis.

To modify the number of line segments used to approximate the ellipse:

1. In the workspace, select the ellipse for which you want to specify a different number of line segments.
2. On the sidebar, click Object Details. The Geometry dialog appears.
3. Enter a new value in the text field labeled Approximated by Number of Sides.

16.2.2.13. Square Attributes. The only specific attribute of a square that you can modify is the side length.

To modify the size of a square:

1. In the workspace, select the square for which you want to modify the size.
2. On the sidebar, click Object Details. The Geometry dialog appears.

3. Enter a new value in the text field labeled Size.

16.2.2.14. Rectangle Attributes. The only specific attributes of a rectangle that you can modify are the width and height.

To modify the size of a rectangle:

1. In the workspace, select the rectangle for which you want to modify the size.
2. On the sidebar, click Object Details. The Geometry dialog appears.
3. Enter a new value in the text field labeled Width.
4. Enter a new value in the text field labeled Height.
5. Click Close.

16.2.2.15. Individual Point Modification. With the Adjustor tool, you can move one or more points from polyline geometries, as follows:

1. On the sidebar, choose the Adjustor tool by clicking .
2. Click the point you want to move. If you want to move more than one point, you can drag a box around them to select them as a group, or Shift-click each point in turn. Handles appear on the selected points. (The points do not all have to be in the same polyline.)
3. Drag the selected points to move them. The points move as a group.

As you move the points, you can use the V and H keys on your keyboard to restrict motion to the vertical and horizontal directions, respectively. Press A to allow movement in all directions (the default).

16.2.3. Three-Dimensional Line Geometries

Three-dimensional line geometries cannot be created interactively; they must be created in a data file. For example, the following data file includes two 3-D line geometries representing the trajectories of two bugs in a cubic room (10 x 10 x 10). Two zones are used to represent the ceiling and floor of the room. (In order to display 3-D geometries, you must either include at least one zone in the data file with the 3-D geometries or read the 3-D geometries in, using the Add to Current Data Set option, after having first read a data set into the frame.) A plot of this data set (included in your Tecplot distribution as **`examples/dat/3dgeom.dat`**) is shown in Figure 16-8.

```
TITLE = "EXAMPLE: 3D GEOMETRIES"
 VARIABLES = "X", "Y", "Z"
 ZONE T="Floor", I=3, J=3, F=POINT
 0. 0. 0.
 5. 0. 0.
 10. 0. 0.
 0. 5. 0.
 5. 5. 0.
```

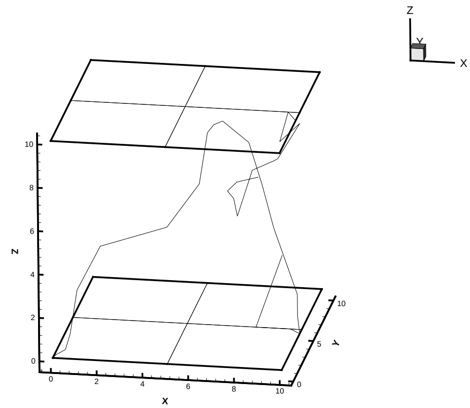


Figure 16-8. Three-dimensional line geometries.

```
10. 5. 0.
0. 10. 0.
5. 10. 0.
10. 10. 0.
ZONE T="Ceiling", I=3, J=3, F=POINT
0. 0. 10.
5. 0. 10.
10. 0. 10.
0. 5. 10.
5. 5. 10.
10. 5. 10.
0. 10. 10.
5. 10. 10.
10. 10. 10.
GEOMETRY X=0, Y=0, Z=0, CS=GRID, C=BLUE, T=LINE3D, F=POINT
1
12
10. 4.0 10.
9.5 5.0 10.0
9.3 4.0 9.0
10. 5.0 9.5
9.2 4.0 8.2
8.8 5.3 7.6
7.8 5.6 7.0
7.2 5.2 5.0
6.9 6.0 5.5
6.9 4.5 6.4
```

```
 7.3 4.6 6.8
 8.2 4.8 7.0
 GEOMETRY X=0, Y=0, Z=0, CS=GRID, C=RED, T=LINE3D, F=BLOCK
 1
 20
 0 .1 .5 .7 1. 2. 4.9 6.3 6.5 6.6 6.9 8. 8.5 9. 10. 10. 10. 9.5
8. 9.
 0 .1 .3 .4 .5 .7 .8 1. 2. 3. 3.5 3.7 4. 4. 4. 4. 4.5 5. 5. 6.
 0 .1 .3 1. 3. 5. 6. 8. 10. 10. 10. 9. 7. 5. 2. 1. 0. 0. 0. 3.
```

16.3. Images

Tecplot can import images from JPEG, BMP, and PNG files. These images can be used as logos or as a backdrop to your plot.

16.3.1. Inserting Images

You create an image by using the insert image menu option.

To add an image to your plot:

1. From the Insert menu, choose Image.
2. Select an image file. Tecplot can read in images in JPEG, BMP, or PNG format.

When you insert an image, the image is initially centered in the frame at a preset size.

16.3.2. Modifying Images

The Image Geometry Details dialog is used to modify an image just like it is used to modify other geometries. The dialog also displays the file name of the image and its resolution (number of pixels in each direction) for informational purposes. The Image Geometry Details dialog is shown in Figure 16-9.

Images share many properties of other Tecplot geometries, including coordinate system, position, scope, clipping, draw order, and attachment to zones or mappings. See Section 16.2. “Geometries” for details.

16.3.2.1. Image Size. Using the geometry dialog, you can specify the width and height of the image.

To modify the size of an image:

1. Select the image on the screen and then select Object Details on the sidebar. Alternatively, double-click the image on the screen.
2. Enter a new value in one of the text fields labeled Width or Height. Units may be specified by typing them after the number. Use cm for centimeters, in for inches, or pix for pixels.
3. Click close on the Geometry dialog when you are done.

To interactively resize an image:

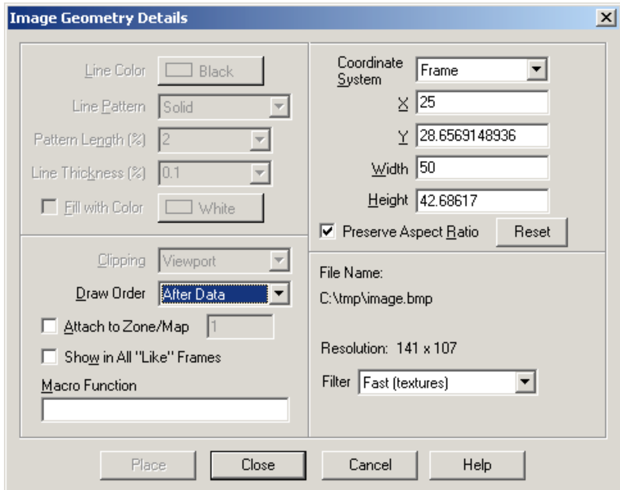


Figure 16-9. The Image Geometry Details dialog

1. Select the image on the screen. Pick handles appear on the corners and sides image.
2. To resize the image, drag the handle on the appropriate side or corner.
3. To drag the image to another location without resizing, click and drag anywhere else in the image.

16.3.2.2. Image Aspect Ratio. By default, the image maintains its shape. That is, if you change the width, the height changes so the image is the same shape; likewise, if you change the height, the width changes so the image is the same shape. You can change this behavior with the Preserve Aspect Ratio toggle.

When the Preserve Aspect Ratio toggle is off, the width and height of the image can be set independently. If the Preserve Aspect Ratio is then turned back on, the current image aspect ratio is used. That is, future changes to width or height change the other to maintain the image in its current shape. To return the image to its original shape, you can press the Reset button.

16.3.2.3. Image Filter. The image size on the screen is most likely not the same size as the original image. The Resize filter determines how the image is resized to fit the screen.

To change the resize filter:

1. Select the image on the screen and then select Object Details on the sidebar. Alternatively, double-click the image on the screen.
2. Select a new filter in the Filter drop-down.
3. Click close on the Geometry dialog when you are done.

The following filters are available:

- **Fast (textures)**: Tecplot uses OpenGL textures to resize the image. Assuming you have enough graphics memory, this is the fastest option, although the accuracy of the image may suffer, especially when reducing an image to a size much smaller than it was before. This is the default setting.
- **Pixelated:** Choose this option when the image is much larger than its original size and you want to see the individual pixels. This option is slower than the Fast (textures) for increasing the size of images.
- **Smooth:** There are seven smooth options, all producing slightly different effects. These options are slower, often dramatically slower than the Fast (textures), but produce better effects for highly reduced images. In general, use the Smooth (Lanczos2) option unless you have specific image processing needs.

The resize filter has no effect on vector-based output, only on the screen and for exported images.

16.3.3. Images and Tecplot Files

Unlike other geometries, images cannot be included in data files. When you save a data file, even if you specify to include geometries, any images in the plot are not saved.

In layout and style sheet files, the image is referenced from its original location. This reference can be a relative reference or an absolute (as with data files). See Section 5.3.2. “Layout Files” on page 92 for details.

For layout package files, images are included.

16.3.4. Example: Using Image as a Backdrop to Data

Sometimes, you want to import an image and use it behind your data. Figure 16-10 shows a crude mesh displayed on top of a simple black-and-white image.

If you know the size and position of the image with respect to the data, you can follow these steps:

1. Read in your data file.
2. Choose Cartesian 2D as the plot type.
3. Import the image file.
4. Double-click on the image. This brings up the Image Geometry Details dialog.
5. Change the coordinate system to Grid.
6. Enter the grid data position of the lower-left corner of your image in the X and Y text fields.
7. Enter the grid data width and height of the Width and Height text fields. You can easily calculate these if you know the grid data position of two opposing corners of your data. You may have to turn off the Preserve Aspect Ratio toggle if the image has a different aspect ratio than the data.

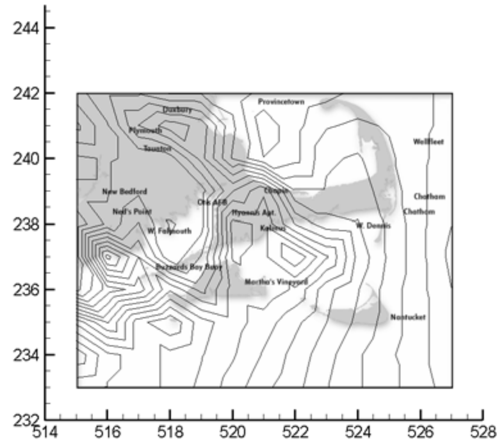


Figure 16-10. Using an Image as the backdrop to data

8. Change the draw order to Before Data. The image will not show up behind the data until you close the Image Geometry Dialog.
9. Close the Image Geometry Dialog.

If you do not know the size and position of the image with respect to the data, you can follow these steps:

1. Read in your data file.
2. Choose Cartesian 2D as the plot type.
3. Import the image file.
4. Double click on the image. This brings up the Image Geometry Details dialog.
5. Change the Draw Order to Before Data. The image will not show up behind the data until you close the Image Geometry Dialog.
6. Close the Image Geometry Dialog.
7. Zoom in and out of your plot and translate the plot until the data is right over the image.
8. Double click the image again. Be sure to get the image and not the data.
9. Change the image to Coordinate System to Grid.
10. Close the Image Geometry Dialog.

Now when you zoom or translate the plot, the image stays with the data.

16.4. Text and Geometry Pushing and Popping

You can place text and geometries in any order you like. Tecplot draws all geometries first, in the order in which they were placed, then all text. There are times, however, when you want to override this default order. Sometimes, for example, you may place one geometry, then draw a second geometry that partially obscures the first. You can pop the first geometry to the top of the draw stack, so that Tecplot draws it after the geometry that had partially obscured it.

Popping raises the object to the top of its draw stack. Pushing lowers the object to the bottom of its draw stack. The text stack is always drawn on top of the geometry stack.

To push a text or geometry object:

1. Select the object.
2. From the Edit menu, choose Push.

To pop a text or geometry object:

1. Select the object.
2. From the Edit menu, choose Pop.

16.5. Text and Geometry Alignment

When you have a number of text and geometries, you may want to align them after placing them. You can do this using the alignment tools in the Quick Edit dialog, shown in Figure 16-11.

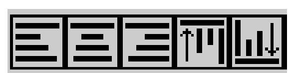


Figure 16-11. Alignment tools.

You can use these tools as follows:

1. On the sidebar, choose the Selector tool by clicking .
2. In the workspace, select a text or geometry with which you want to align other objects.
3. Drag the mouse to draw a rubber band box around the text and geometries you want to align. The Group Select dialog appears.
4. Select the Text and Geometries check boxes in the Group Select dialog, then click OK. Selection handles appear on the selected text and geometries.
5. On the sidebar, click Quick Edit to call up the Quick Edit dialog, if it is not already displayed.
6. Use the alignment buttons as follows:
   -  Left align the selected text and geometries with the original selected object.
   -  Center the selected text and geometries with the original selected object.
   -  Right align the selected text and geometries with the original selected object.

-  Top align the selected text and geometries with the original selected object.
-  Bottom align the selected text and geometries with the original selected object.

16.6. Text and Geometry Links to Macros

Each text or geometry you create can be linked to a macro function. This macro function is called whenever the user holds down the control key and clicks the right mouse button on the text or geometry.

For example, if you have pieces of text, each representing a different well, Ctrl-right-click on any piece could run a macro that brings up an XY-plot of that well's data.

Macro functions are specified with the “Macro Function” field in the Geometry dialog or in the Text Options dialog. If desired, the macro function may be listed with one or more parameters. See Chapter 27, “Macro Commands,” and the *Tecplot Reference Manual*, for more detailed information on using macros in Tecplot.

16.7. Custom Characters

You can create symbols, characters, and even custom fonts for use in Tecplot. See Section 30.5, “Custom Character and Symbol Definition,” for instructions.

CHAPTER 17 Axes

Tecplot creates axes automatically for all line plots, as well as 2- and 3-D field plots. For these automatically created axes, Tecplot determines good tick mark position and spacing, and creates reasonable tick mark and axis labels. You can modify your Tecplot configuration file to change the default behavior, and you can use the Axis Details dialog to exercise control over your axes.

You control each axis individually. You specify whether each axis is displayed, its color, position, range, length, tick mark spacing, and other attributes. You can also specify dependencies between axes that help control the shape of your data when you change the view, or individual axis ranges. For 3-D axes, these dependencies can be used to automatically rescale the axes when the X-, Y-, or Z-ranges of your data are significantly different.

Most axis controls are concentrated in the Axis Details dialog. Each page of the Axis Details dialog controls a different aspect of the axis, and each page is repeated for each axis. The Range page of the Axis Details dialog is shown in Figure 17-1.

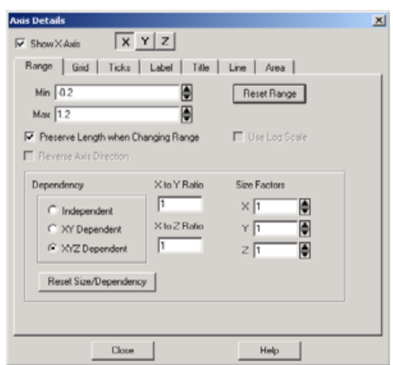


Figure 17-1. The Range page of the Edit Axis dialog for a 3-D plot.

Tecplot maintains five distinct sets of axes, one for each plot type. This means that modifying the color, say, of your X-axis line for 2-D plots will not affect 3-D plots. The 2-D X-axis and the 3-D X-axis are different objects.

To edit an axis from the Axis Details dialog, you must be sure you are working on the correct axis. At the top of the dialog there are buttons that indicate which axis you are working with. To edit a different axis, select a different axis by clicking a different axis button.

17.1. Axis Display

The most basic axis control is whether or not to display the axis. Showing an axis, by default, shows the axis line, tick marks, tick mark labels, and title for the axis. It is possible to disable any of these components separately, including the axis line. But if you choose not to show an axis, none of the plot components associated with that axis (line, tick marks, tick mark labels, title, or grid lines) is displayed. You can control whether an axis is shown from any page (except Area) of the Axis Details dialog, using the Show *a*-Axis check box, where *a* is Theta or R for Polar Line plots, X or Y for 2D Cartesian and Sketch plots, X1 – X5 or Y1 – Y5 for XY Line plots and X, Y, or Z for 3D Cartesian plots.

To control the display of an axis, perform the following steps:

1. From the Plot menu, choose Axis. The Axis Details dialog appears.
2. At the top of the Axis Details dialog, click X, Y, or Z (Theta or R for Polar Line plots) to select the axis you want to change. The check box to the left of the axis buttons is labeled Show *a*-Axis, where *a* is one of X, Y, or Z (Theta or R for Polar Line plots). For example, by default X is selected for all plot types except Polar Line, and the check box is labeled Show X-Axis.
3. Select the Show Axis check box to show the axis; deselect the check box to hide the axis.

You can display axes for Sketch plots as well as frames containing data. By default, sketch axes are not displayed. Follow the above procedure to display sketch axes.

17.2. Axis Variable Assignment

For 2-D axes, Tecplot initially assigns the first and second variables in the data set to the X- and Y-axes, respectively. For 3-D axes, Tecplot initially assigns the first three variables in the data set to the X-, Y-, and Z-axes respectively.

To change variable assignments for 2-D and 3-D axes:

1. From the Plot menu, choose Assign XY in 2-D or Assign XYZ in 3-D. A Select Variables dialog appears, with a drop-down for each available axis listing the variables in the data set.
2. Choose one variable for each axis.

For line plots, assigning axis variables is part of defining the mappings. See Chapter 7, “Line Plots,” for more information.

17.3. Axis Range Modification

The range of an axis specifies the minimum and maximum data values displayed along it. The length of an axis is its physical length on the screen or paper. The scale of an axis is the ratio of its length to its range. You can modify the range in two ways. In the first case, changing the range does not affect the length of the axis, and thus modifies the scale. In the second, changing the range preserves the scale, and thus modifies the length.

To change range and preserve axis length:

1. From the Plot menu, choose Axis. The Axis Details dialog appears.
2. Choose the Range page and the appropriate axis.
3. Select the check box labeled Preserve Length when Changing Range.
4. Enter the desired range in the text fields labeled Min and Max. You may also use the up and down arrows to increase or decrease the displayed values, or click Reset Range and choose from one of three options:

   Reset to Nice Values: Sets the range to slightly larger than the range of the axis variable in order to begin and end the axis at major axis increments.

   Reset to Var Min/Max: Sets the range to slightly larger than the range of the axis variable.

   Make Current Values Nice: Moves the axis range to the nearest major axis increments.

To change range and preserve axis scale:

1. From the Plot menu, choose Axis. The Axis Details dialog appears.
2. Choose the Range page and the appropriate axis.
3. Deselect the check box labeled Preserve Length when Changing Range.
4. Enter the desired range in the text fields labeled Min and Max. You may also use the up and down arrows to increase or decrease the displayed values, or click Reset Range and choose from one of three options:

   Reset to Nice Values: Sets the range to slightly larger than the range of the axis variable in order to begin and end the axis at major axis increments.

   Reset to Var Min/Max: Sets the range to slightly larger than the range of the axis variable.

   Make Current Values Nice: Moves the axis range to the nearest major axis increments.

The difference between the two methods of changing the range is shown in Figure 17-2.

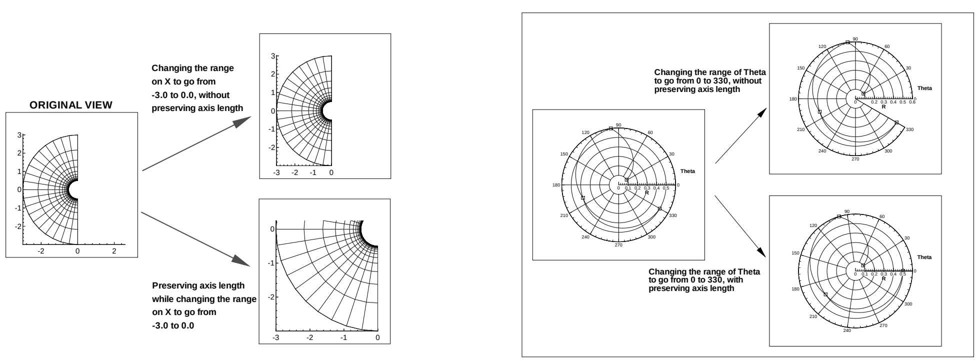


Figure 17-2. Preserving length versus preserving scale while changing range (left); preserving length versus preserving scale while changing range in a Polar Line plot (right).

In the 3D Cartesian plot type, you can change the length of axes using the X-, Y-, and Z-Size Factor fields, which also affects scale. If the 3-D axes are independent (see Section 17.3.2, “Axis Dependency,” for details), you can resize each axis independently. If the axes are XY-dependent, changing the X or Y-size factor changes the other. If the axes are XYZ-dependent, changing one size factor changes the other two.

For XY- and Polar Line plots, you can display the axis range using a logarithmic scale. For Polar plots, log axes are only available for the R-axis. See Section 7.5.2. “Log Axes” on page 135.

17.3.1. Axis Direction

You can reverse the direction of any axis. Normally, values along an axis increase as you move from bottom to top on the Y-axis and from left to right on the X-axis. When you reverse the axis direction, the value increase from top to bottom and right to left. For Polar axes, the Theta axis increases in the counter-clockwise direction, and the R axis increases from the middle of the plot to the outer circle. Reversing the Theta axis makes Theta increase in the clockwise direction, and by reversing the R axis, R increases from the outer circle to the center.

To reverse the axis direction for Sketch, XY-, Polar- or 2D Cartesian axes:

1. From the Plot menu, choose Axis. The Axis Details dialog appears.
2. Choose the Range page and the appropriate axis.
3. Select the check box labeled Reverse Axis Direction.

For 3-D axes, you can use negative size factors and/or negative X-to-Y and X-to-Z ratios to reverse the direction of an axis. This can create a left-handed coordinate system, or rotate the plot so that, for example, the Z-axis is pointed downward. If the axis mode is Dependent, changing the size factor for one axis changes it on all axes. If the axis mode is XY Dependent, changing the sign of either the X- or Y-axis size factor changes the sign of the other. If the axis mode is Independent, changing the sign of the size factor does not affect the signs of the size factors for the other axes, but it does change the sign of the corresponding X-to-Y and X-to-Z ratios.

To reverse the direction of your 3-D axis:

1. From the Axis menu, choose Edit. The Edit Axis dialog appears.
2. Choose the Range page and the appropriate axis.
3. Select the Independent axis mode.
4. Enter a negative number in the Size Factor text field for the axis you want to reverse.

17.3.2. Axis Dependency

Axis dependency is available for all plot types except Polar Line. When you modify the range on one axis, other axes may be affected as well, depending on the current dependency settings for the axes. Axes may be dependent or independent; in general, if axes are dependent, changing the range for one axis causes similar changes in the other axes, while changing the range of an independent axis has no effect on the other axes. As you can see, axis dependency within Tecplot is distinct from dependency relations within your data.

For Sketch, XY Line, and 2D Cartesian axes, the dependency relations Dependent and Independent are the only choices. For XY Line plots, independent axes are the default; for 2D Cartesian field plots, dependent axes are the default. You can effect the shape of the axes by using the X to Y Ratio field on the dialog. This field changes the relationship between your X and Y axes.

To control the dependency of your axes:

1. From the Plot menu, choose Axis. The Axis Details dialog appears; choose the Range page and the appropriate axis.
2. In the region labeled Dependency, choose Independent or Dependent.
3. If you choose dependent, enter a value in the X to Y Ratio text field.

For example, if you enter 2 in the X to Y Ratio text field and Dependent is checked, the Y axis will be twice as long as the X axis.

Axis dependency for 3D Cartesian plots is similar to XY Line and 2D Cartesian, however you can also change the dependency of the Z-axis.

For 3-D axes, there are three dependency modes, as follows:

- **Independent:** All axes are independent.
- **XY Dependent:** The X- and Y-axes are dependent upon each other just as for dependent 2-D axes. The Z-axis is independent.
- **XYZ Dependent:** Changing the scale on any axis results in a proportional change in scale on the other two axes, so that the specified X to Y Ratio and X to Z Ratio are preserved.

To control the dependency of your axes:

1. From the Plot menu, choose Axis. The Axis Details dialog appears; choose the Range page and the appropriate axis.
2. In the region labeled Dependency, choose Independent, XY Dependent, or XYZ Dependent.
3. For XY Dependent axis mode, enter the X to Y Ratio in the text field.
4. For XYZ Dependent mode, also enter the X to Z Ratio in the text field.

17.3.3. Polar Axis

Polar axes are different than any other axis type due to their cyclical nature. Polar axes are separated into the Theta- and R-axis. Each axis has very different settings, unlike XY- or XYZ-axes. For the Theta-axis you can change the Theta Mode, Theta Period, and orientation; for the R-axis you can change the origin; and for both axes you can clip the data to the axes. Below is a discussion of each of these features.

By default the Theta-axis is in degrees mode with a range of zero to 360. This default can be changed through the Tecplot configuration file. In the Tecplot configuration file there is a section dedicated to polar plots. See Section 30.1, “Configuration Files,” for more information on changing settings in the Tecplot configuration file.

17.3.3.1. Clip Data to Axes. For Polar Line plots, it is possible to have data that extends beyond the edges of the axes. Use this feature to eliminate data drawn outside of the range of the axes. Clipping data can be set independently for the axes. For example, you may clip data for the Theta-axis, yet see points extending beyond the R-axis. To change this setting, use the Clip Data to Axis toggle on the Range page of the Axis Details dialog. This feature is illustrated in Figure 17-3.

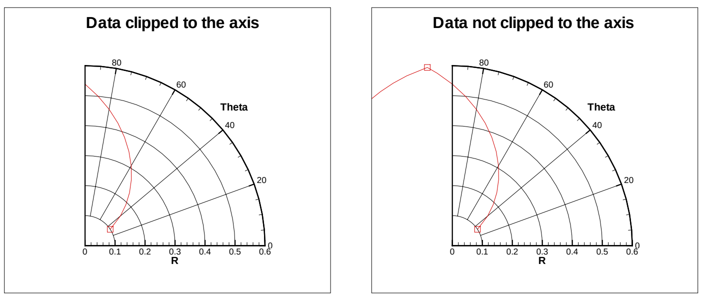


Figure 17-3. An example of clipping polar data to an axis.

17.3.3.2. Theta Mode. For the Theta axis you can change the mode in which the axis is drawn. There are three modes available: Radians, Degrees and Arbitrary. The options for the Theta Mode are 0 - 360 degrees, -180 – 180 degrees, 0 – 2 Pi radians, -Pi – Pi radians, and Fit to Var Min/Max. Selecting any of these options changes the Theta Mode, resets the Theta-axis range, and resets the Theta Period.

To change the Theta Mode:

1. Launch the Axis Details dialog. Select the Range page and the Theta-axis.
2. Click Reset (next to Theta Mode).
3. Select the appropriate Theta Mode.

When the Theta Mode is radians, Tecplot attempts to draw theta labels as fractional units of Pi. An example is shown in Figure 17-4.

17.3.3.3. Theta Period. The Theta Period specifies the theta range that is required to create a complete circle. If your Theta Mode is Degrees, the Theta Period is forced to 360; for Radians the period is 2 Pi. For Arbitrary you can set the period to any value.

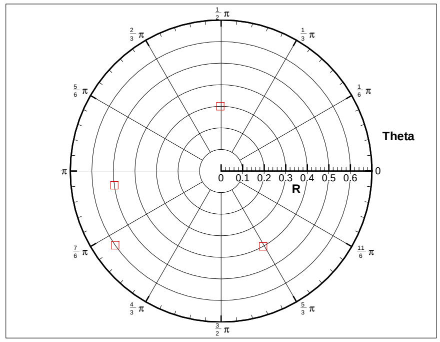


Figure 17-4. An example of polar radian labels.

17.3.3.4. Theta Value on Circle Right. The Theta Value on Circle Right setting changes the orientation of the Theta-axis. By default this value is zero, which means that the value zero (or equivalent value, 360 degrees, 720 degrees, and so forth) is displayed on the right hand side of the circle. You can change this value to change the orientation of the axis. For example, if your plot has a Theta Mode of Degrees and you set the Theta Value on Circle Right to be 270, the value 270 appears on the right hand side of the plot and zero is straight up. If the axis is reversed, zero is straight down.

17.3.3.5. R-Origin. The R-Origin can be changed on the Range page of the Axis Details dialog. This setting specifies what value of R is represented at the center of the axis. The R-axis has a range of 0.3 to 0.6 and the effect of changing the R-origin value may be seen in Figure 17-5.

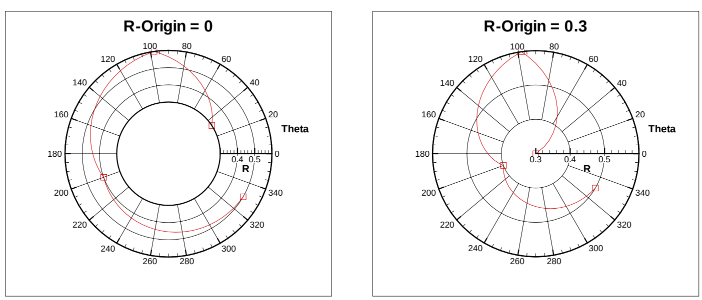


Figure 17-5. An example of changing the R-origin on a polar plot.

17.4. Axis Grids

The grid area is one or more regions defined and bounded by the axes. For XY Line and 2D Cartesian plot types, the grid area is the rectangle defined by the X- and Y-axes. For 3D Cartesian plot types, the grid area consists of the three rectangles defined by the X-, Y-, and Z-axes. For Polar Line plots, the grid area is defined by the annular segment of the axes.

Gridlines are a set of lines drawn from one or more axes. Gridlines extend from the tick marks on an axis, across the grid area. Minor gridlines extend from the minor tick marks. Gridlines make it easier to determine the values of individual data points. For polar axes, the R-grid lines form circular lines, while Theta-grid lines are rays emanating from the center of the grid outward.

The precise dot grid is a set of small dots drawn at the intersection of every minor gridline. In line plots, the axis assignments for the first active mapping govern the precise dot grid. The precise dot grid option is disabled for the 3D Cartesian plot type, and line plots when either axis for the first active line mapping uses a log scale.

You control the gridlines and precise dot grid from the Grid page of the Axis Details dialog, as shown in Figure 17-6.

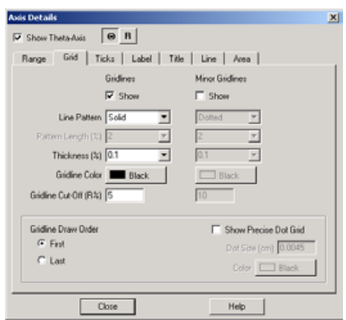


Figure 17-6. The Grid page of the Edit Axis dialog for a Polar Line plot.

17.4.1. Gridlines

You can control whether grid lines or minor gridlines are shown using the Grid page of the Axis Details dialog. This includes line pattern, pattern length, or line thickness. The spacing of gridlines is controlled by the tick mark spacing; see Section 17.5, “Tick Marks and Labels,” for more information.

To control grid lines and minor grid lines:

1. From the Plot menu, select Axis. The Axis Details dialog appears. Select the Grid page.
2. To show gridlines, select the Show check box under Gridlines. To show minor gridlines, select the Show check box under Minor Gridlines.

3. On the Grid page, specify a line pattern, line pattern length, and line thickness for both the gridlines and minor gridlines.

For the Theta-axis in the Polar Line plot type, you can also specify a gridline cutoff. The gridline cutoff is the point along the R-axis in which you want to stop drawing Theta-gridlines. In a Polar Line plot the abundance of gridlines at the center may obscure data, in which case you may want to adjust this setting. For an example of polar grid cutoff, see Figure 17-7.

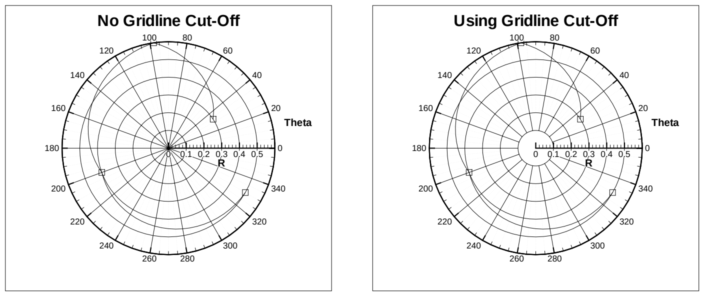


Figure 17-7. An example of polar grid cutoff.

17.4.2. Precise Dot Grid

From the Grid page of the Axis Details dialog, you can control whether the precise dot grid is shown, and if so, the size and color. The effect of turning on the precise dot grid is shown in Figure 17-8.

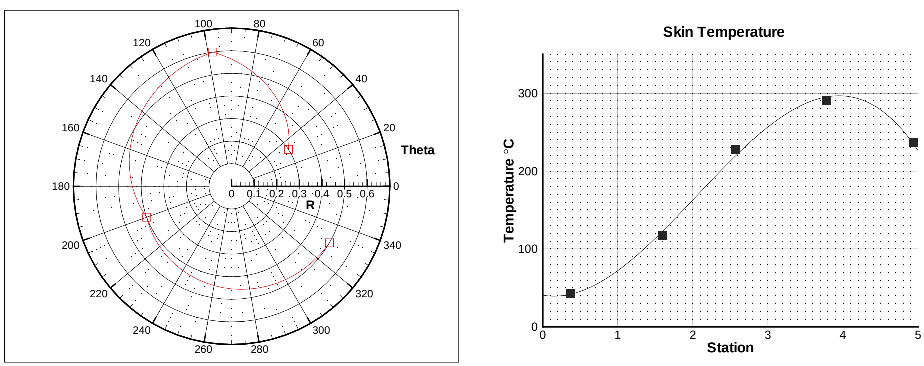


Figure 17-8. An example of polar plot showing the precise dot grid (left); the effect of turning on the precise dot grid (right).

To control the precise dot grid:

1. From the Plot menu, select Axis. The Axis Details dialog appears. Select the Grid page.
2. To show the precise dot grid, select the Show check box under Precise Dot Grid.

3. Specify dot size (measured in centimeters on the output) and color as desired.

17.4.3. Gridline Draw Order

For all axes except 3-D, you may specify a gridline draw order. Gridlines may be drawn either first, before any of the other plotting layers, or last, so they overlay any plotting layers. These different options are illustrated in Figure 17-9.

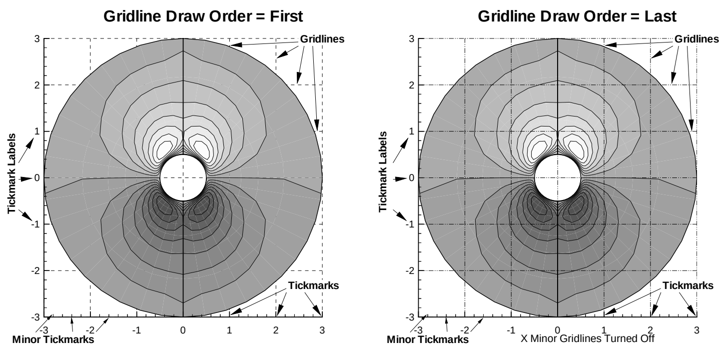


Figure 17-9. The grid plotting order.

You can also specify the gridline draw order by “pushing” or “popping” the axis grid area from the Edit menu. First select the axis grid area by clicking on a gridline, then choose Push from the Edit menu to plot the gridlines first, or choose Pop to plot the gridlines last.

17.5. Tick Marks and Labels

Each axis can be marked with tick marks, and those tick marks may or may not be labeled, either with numbers or with custom text strings. You control tick marks and their placement using the Ticks page of the Axis Details dialog. You control the tick mark labels using the Label page of the Axis Details dialog. You can control the tick mark spacing from either page.

17.5.1. Tick Marks

From the Ticks page of the Axis Details dialog, you can specify any of the following tick mark attributes independently for each axis:

- Whether tick marks are displayed.
- Length of tick marks and minor tick marks.
- Thickness of tick marks and minor tick marks.
- Number of minor tick marks between tick marks.
- Direction of the tick marks (into the plot, out of the plot, or centered on the axis line).
- Spacing and position of tick marks.

17.5.1.1. Tick Mark Display. You can show tick marks on any, all, or none of your axes.

To show tick marks for an axis:

1. From the Plot menu, choose Axis. The Axis Details dialog appears. Select the Ticks page, as shown in Figure 17-10.

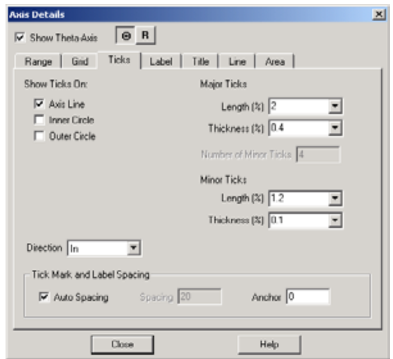


Figure 17-10. The Ticks page of the Axis Details dialog.

2. Choose the axis for which you want to show tick marks. Select one of the toggles below the Show Ticks On label.

Note: There is no separate control for showing minor tick marks, as there is for minor grid-lines. To show no minor tick marks, enter zero in the Number of Minor Ticks text field.

To hide tick marks, deselect the toggles below the Show Ticks On label.

For each plot type there are many different sections of the axis on which you can display tick marks (this description also applies to Labels and Titles).

Sketch, XY Line, and 2D Cartesian axes allow tick marks to be displayed in the following areas:

- Axis Line.
- Grid Border Left (Grid Border Bottom).
- Grid Border Right (Grid Border Top).

Where Axis Line is the line that represents the specified axis. By default the axis line and grid border left/bottom are in the same position. Grid Border Left is the left most position of the grid as defined by the viewport settings on the Area page of the Axis Details dialog. Grid Border Bottom, Right, and Top are similarly defined by the viewport settings.

Polar R-axis allows tick marks to be displayed in the following areas:

- Axis Line.
- All R-axes.

- Grid Border Start.
- Grid Border End.

Where Axis Line is the line that represents the R-axis. The option, All R-axes, is only available if the Draw Axis in Both Directions toggle or Draw Perpendicular Axis toggle is checked for the R-axis, see Section 17.7.2, "Polar Axes Positioning." The All R-axes setting will draw tick marks on the additional axes that are drawn if the aforementioned toggles are checked. Grid Border Start and Grid Border End are the start and end points of the polar grid area. These options are only available if the polar plot does not form a complete circle. For example a plot that only goes from zero to 90 degrees can have tick marks drawn at the grid border start/end. If the data forms a complete circle, there is no start or end point on which to draw the ticks.

Polar Theta-axis allows tick marks to be displayed in the following areas:

- Axis Line.
- Inner Circle.
- Outer Circle.

Where Axis Line is the line that represents the Theta-axis. Inner Circle is only available if the minimum value on the R-axis is greater than the R-Origin value; when this is the case, the center of the polar plot is a circle rather than a single point, therefore ticks can be drawn on the inner circle. Outer Circle is the outer edge of the polar grid area.

Three-dimensional axes allow tick marks to be displayed in the following areas:

- Axis Line.
- Opposite Edge.

Where Axis Line is the line that represents the specified axis. Opposite Edge is complimentary line that is opposite the axis line.

17.5.1.2. Tick Mark Length and Thickness. Tick mark length and thickness can be set independently for tick marks and minor tick marks using the Length and Thickness fields on the Ticks page of the Axis Details dialog.

17.5.1.3. Number of Minor Tick Marks. The number of minor tick marks can be set in the Number of Minor Ticks text field on the Ticks page of the Axis Details dialog. By selecting the Auto Spacing toggle at the bottom of the page, Tecplot will determine an optimal number of minor ticks for you.

17.5.1.4. Tick Mark Direction. You can specify the direction in which the tick marks are drawn.

The following options are available:

- **In:** Tick marks and minor tick marks are drawn from the axis toward the center of the plotting region.
- **Out:** Tick marks and minor tick marks are drawn from the axis away from the center of the plotting region.
- **Center:** Tick marks and minor tick marks are centered on the axis line.

17.5.2. Tick Mark Labels

Tick mark labels are the values that are displayed next to the axis tick marks and represent the data value at that tick mark. From the Labels page of the Axis Details dialog, you can specify attributes for tick mark labels for each axis.

The following options are available:

- Whether tick mark labels are displayed.
- The color, font, and size of the tick mark labels.
- The format of the tick mark labels.
- The offset of the tick mark labels—that is, the distance between the tick mark labels and the axis line (in frame units for Polar Line, XY Line, and 2D Cartesian axes, and as a percentage of the median axis length for 3D Cartesian axes).
- The orientation of the tick mark labels. This is the angle the labels make with the horizontal, or they can be placed parallel or perpendicular to the axis.
- Show label at axis intersection.
- Erase behind labels.
- A label skip for each axis. This is the number of major tick marks to skip. For example, with the label skip set to 2, every other major tick mark will be labeled.
- The position and spacing of the tick marks and tick mark labels

17.5.2.1. Tick Mark Label Display. Tick mark labels can be displayed on the same objects as tick marks themselves. For a discussion of where tick mark labels can be displayed see Section 17.5.1.1, “Tick Mark Display.”

There are however two other settings that affect the tick label display. These settings are Show Label at Axis Intersection and Erase Behind Labels. The Show Label as Axis Intersection toggle draws a label at the point where two axes intersect. When the toggle is unchecked, Tecplot does not draw labels at axis intersection points. Use this toggle if you have axis labels that are colliding, or stacked on top of one another a the intersection of two axes. This problem is common for polar axes, so the default for polar axes is to have this toggle turned off. The Erase Behind Labels toggle creates a rectangle behind the label so in some cases the labels are more easily seen.

17.5.2.2. Tick Mark Label Formats. You can choose several numeric formats for your tick mark labels, or specify a set of text strings to use as custom labels.

The following numeric formats are available:

- **Integer:** Tick marks are labeled in integer format (for example, 12). If this format is selected, tick mark labels with a decimal part are truncated.
- **Float:** Tick marks are labeled with floating-point numbers (for example, 10.2).
- **Exponent:** Tick marks are labeled using numbers in exponential format (for example, 1.02E-03).
- **Best Float:** Tecplot selects the best floating-point representation of the tick mark labels.
- **Range Best Float:** Tecplot selects the best floating-point representation of the tick mark labels, taking into account the range of values on the axis.
- **Superscript:** Tick marks are labeled with numbers in scientific notation (for example, 1.2x10-3).
- **Custom:** Uses the specified custom label set to label the axes.

Custom labels are text strings defined in your data file that allow you to print meaningful labels for variables that do not contain numeric data, such as variables that contain names or levels (such as Yes/No, Small/Medium/Large, or months of the year) of a categorical variable. Custom labels are defined using the **`CUSTOMLABELS`** record; each **`CUSTOMLABELS`** record corresponds to one custom set. When you choose custom labels for an axis, you also choose which custom set should be used for that axis.

To specify the format for your tick mark labels:

1. From the Labels page of the Axis Details dialog, choose the axis for which you want to specify the format of the tick mark labels.
2. Click Number Format on the Label page; the Specify Number Format dialog appears, ash shown in Figure 17-11.
3. Choose one of formats from the Format drop-down: Integer, Float, Exponent, Best Float, Range Best Float, Superscript, Custom.

   (Float, Exponent, or Superscript only): Enter a value in the Precision field to specify the number of digits to the right of the decimal point in the tick mark labels. The default is four.

   (Custom only): Enter a value in the Custom Set field to specify which set of custom labels to use on this axis.

Other options for changing the format of your labels include:

- **Show Decimal on Whole Numbers:** When this toggle is checked, whole numbers include a trailing decimal (that is, the number 2 is displayed as 2.).
- **Remove Leading Zeros:** When this toggle is checked leading zeros are removed from numbers (that is, 0.25 is displayed as .25).

- **Show Sign on Negative Numbers:** When this toggle is checked negative numbers show the negative sign. When unchecked the negative sign will be removed (that is, -1.43 is displayed as 1.43). This is useful if you have specified a special prefix or suffix for negative values.
- **Prefix and Suffix:** You can specify a custom prefix and/or suffix for numbers in Tecplot using the Prefix/Suffix text fields. Tecplot allows you to specify separate prefixes and suffixes for zero values and negative values as well.

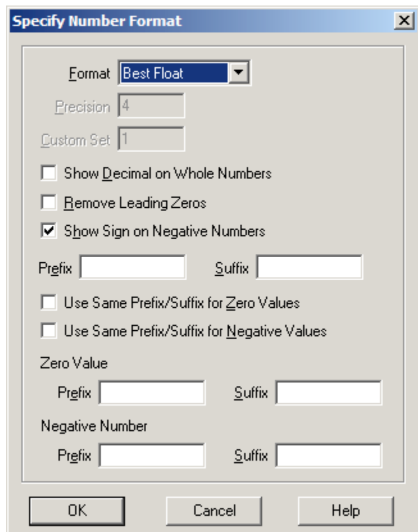


Figure 17-11. The Specify Number Format dialog.

17.5.2.3. Custom Labels. As a simple example of using custom labels, consider the following data file, containing data about attendance at two schools:

```
VARIABLES="SCHOOL", "ATTENDANCE"
CUSTOMLABELS "Cleveland", "Garfield"
                    ZONE T="1991"
                    1 950
                    2 640
                    ZONE T="1992"
                    1 1010
                    2 820
```

The numbers 1 and 2 represent the school number, and the **CUSTOMLABELS** record defines Cleveland as school one and Garfield as school two. Once you assign custom labels in Tecplot, the School axis is labeled with Cleveland and Garfield rather than 1 and 2.

To create a plot with custom labels:

1. Create a data file with one or more **CUSTOMLABELS** records, and one or more variables with ordered integer values 1, 2, 3, and so forth. The first string in the **CUSTOMLABELS** record corresponds to a value of 1, the second string to 2, and so on. Read the data file into Tecplot.
2. Create a plot. XY Line plots are the most likely to use custom labels, but you can use them anywhere.
3. From the Plot menu, choose Axis, and select the Label page of the Axis Details dialog.
4. Choose the axis for which you want to assign custom labels, click Number Format, and select Custom from the Format drop-down. Choose a set of custom labels for the axis from among all the **CUSTOMLABELS** records in the data file. For this example, edit the X-axis and choose custom set 1.
5. Go to the Ticks page of the Axis Details dialog. Deselect Auto Spacing, then set the spacing to one. (You may also want to set the number of minor ticks to zero.)
6. Go to the Range page of the Axis Details dialog. Set the Min and Max value to 0.5 and 2.5 respectively.
7. Close the Axis Details dialog, then go to the sidebar. From the plot layers, select the Bars, deselect the Lines check box.

The attendance data are plotted in Figure 17-12.

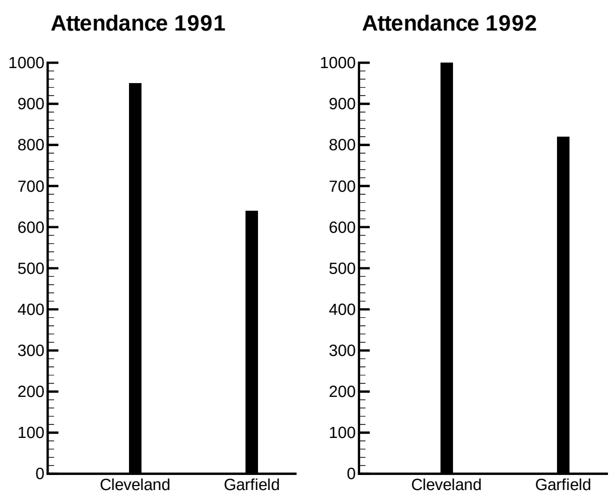


Figure 17-12. Bar charts with custom labels.

As another example, consider the following data file containing temperature and rainfall data:

```
VARIABLES="MONTH", "TEMPERATURE", "RAINFALL"
CUSTOMLABELS "Jan", "Feb", "Mar", "Apr", "May", "Jun"
                    "Jul", "Aug", "Sep", "Oct", "Nov", "Dec"
CUSTOMLABELS "Cold", "Cool", "Warm", "Hot"
CUSTOMLABELS "Dry", "Average", "Wet"
1 1 1
2 1 2
3 2 3
4 2 3
5 3 3
6 3 2
7 4 1
8 4 1
```

This weather data file is plotted in Figure 17-13.

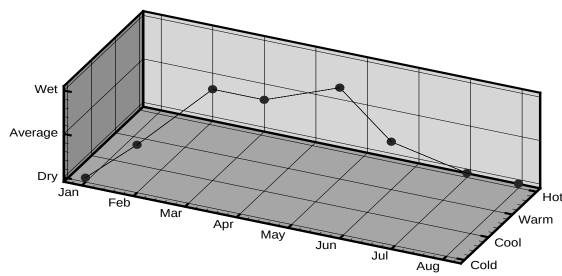


Figure 17-13. A 3-D plot with custom labels on each axis.

Custom labels are used cyclically. That is, if the variable assigned to the axis using custom labels goes over the number of custom labels, Tecplot starts with the first label again. This is useful for days of the week, months of the year, or other cyclical data. In the weather data set above, a value of 13 for the **`MONTH`** variable yields a tick mark label of “Jan.” Similarly, a value of five for **`TEMPERATURE`** yields a tick mark label of “Cold.”

17.5.3. Tick Mark and Label Spacing

You can control tick mark and tick mark label spacing directly, or use Auto Spacing (the default), to calculate an optimal spacing for tick marks and tick mark labels. As you change views, particularly in zooming, Tecplot recalculates the spacing. With Auto Spacing selected, Tecplot also calculates the number of minor tick marks for you.

The tick mark and label spacing values are shared between the tick marks and tick labels. If you change the spacing for your axis labels, the tick marks use the same spacing. You can change the spacing by adjusting the controls at the bottom of the Ticks or Label pages of the Axis Details dialog.

For a particular view, you may want to exercise manual control by specifying the spacing, and an anchor position, which specifies a particular value at which one tick mark should be drawn; additional tick marks are drawn at intervals determined by the spacing value.

To exercise manual control of tick marks and tick mark labels:

1. From the Plot menu, choose Axis. The Axis Details dialog appears. Click Ticks to display the Ticks page, or Label to display the Label page.
2. Choose the axis for which you want to specify tick mark and tick mark label spacing.
3. In the area labeled Tick Mark and Label Spacing, deselect Auto Spacing.
4. In the text field labeled Spacing, specify a spacing value in the units of the variable assigned to the axis.
5. In the text field labeled Anchor, specify a value at which you want one tick mark drawn. Additional tick marks are spaced according to the Spacing parameter away from this anchor tick mark.

17.6. Axis Titles

An axis title is a text label that identifies the axis. By default, Tecplot labels each axis with the name of the variable assigned to that axis.

From the Title page of the Axis Details dialog, you can specify the following attributes for each axis title:

- Whether axis titles are displayed, and if so, where, and what text to use for the title.
- The color, font, and size of the axis title.
- The offset of the axis title—that is, the spacing between the axis title and the axis line.
- The position of the axis title along the axis line.

17.6.1. Axis Title Display

Axis titles can be displayed on the same objects as tick marks and tick mark labels. For a discussion of where axis titles can be displayed see Section 17.5.1.1, “Tick Mark Display.”

If you choose to show the axis title, you can either specify to use the variable name that is assigned to the selected axis, or you can specify text string to use.

To specify the axis title:

1. From the Axis Details dialog select the Title page, as shown in Figure 17-14.
2. Select Use Variable Name or Use Text.
3. If you select Use Text, enter the desired axis title in the appropriate text field.

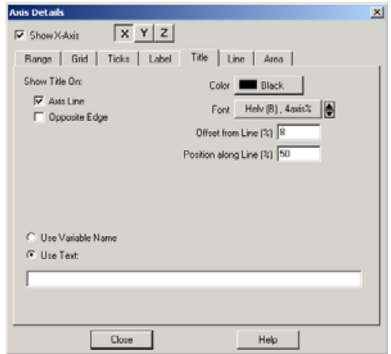


Figure 17-14. The Title page of the Axis Details dialog.

You may also specify the color and font of the axis title by using Color and Font on the Title page. To more easily adjust the font size of the axis titles, use the up/down arrows to the right of the Font option.

17.6.2. Axis Title Offset

The axis title offset prevents Tecplot from printing your axis title directly on top of the axis. You specify the offset as a percentage. You may specify a negative offset, which moves the axis title to the other side of the axis. Zero offset prints the edge of the axis title on the axis.

To specify an axis title offset:

1. From the Plot menu, choose Axis. The Axis Details dialog appears; click Title.
2. Choose the axis for which you want to modify the axis title offset.
3. Enter the desired offset in the Offset (%) field.
4. (Optional) You may also adjust the axis title offset using the Adjustor  tool from the sidebar.

17.6.3. Axis Title Position

Using the Position along Line field you can change where the axis title is positioned along the axis line. For example, in an XY Line plot, if you want the axis title to be on the left side of the X-axis, specify a position near zero. If you want the axis title to be near the right side of the axis, specify a position near 100.

To specify an axis title position:

1. From the Plot menu, choose Axis. The Axis Details dialog appears; click Title.
2. Choose the axis for which you want to modify the axis title position.
3. Enter the position as a percentage in the Position Along Line text field.

17.7. Axis Lines

The actual axis line is shown by default whenever the axis is shown, but it is just as much an optional part of the plot as the tick marks or tick mark labels. There may be situations when you want to see an axis represented simply by the tick marks or the gridlines, without an additional line for the axis itself. In such cases, you can hide the axis line without turning off the axis as a whole.

To show or hide the axis line:

1. From the Plot menu, choose Axis. The Axis Details dialog appears.
2. Click Line to call up the Line page of the Axis Details dialog, as shown in Figure 17-15.

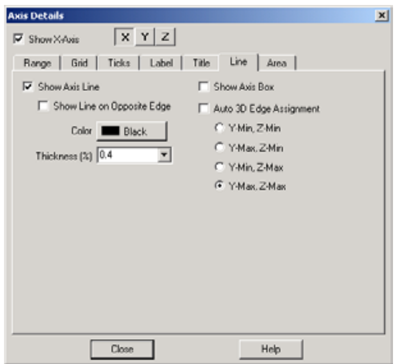


Figure 17-15. The Line page of the Axis Details dialog.

3. Choose the axis for which you want to show or hide the axis line.
4. To show the axis line, select the Show Axis Line check box. To hide the axis line, deselect the Show Axis Line check box.

For Sketch, XY Line, Polar Line, and 2D Cartesian axes you can also show the Grid Border. The grid border is defined as the extents of the grid area, which is defined in Section 17.8, “Grid Area.” For Polar Line plots, you may also show the Viewport border, which is defined on the Area page of the Axis Details dialog. For 3D Cartesian plots you may elect to show the line opposite the axis line. The axis line and line on opposite edge are always drawn if any of the faces adjacent to them have grid lines or are filled, or are part of the axis box; in this case the toggles have no effect.

For Sketch, XY Line, and 2D Cartesian axes you can specify a separate color and line thickness for the axis line and grid border. For Polar Line plots, the grid border color and line thickness is shared with the axis line, however the viewport border may have separate color and line thickness settings. In 3D Cartesian plots, the axis line, line on opposite edge, and axis box all share the same color and line thickness settings.

17.7.1. Sketch, XY Line, and 2D Cartesian Line Position

Sketch, XY, and 2D axes can be positioned in a couple different ways. The first, and easiest is to use the Adjustor  by selecting the axis you want to move and simply drag it into the desired position. When dragging the axis, it pins itself at the minimum or maximum of the grid area. Once the axis is pinned at the edge of the grid area, you may reselect it an move it beyond the edge of the grid area, but you cannot drag it beyond the edge of the frame. The second method for positioning the axis line is manually through the Line page of the Axis Details dialog.

To manually position the axis line follow these steps:

1. From the Plot menu, choose Axis. The Axis Details dialog appears; click Line.
2. Choose the axis for which you want to change the axis position.
3. Select the desired option from the Align Axis with drop-down.
4. Enter the desired offset or position in the sensitive text field.

17.7.2. Polar Axes Positioning

Polar axes, unlike Sketch, XY Line, and 2D Cartesian axes, cannot be positioned interactively. You must use the Axis Details dialog to change the position of a given axis line.

For the Theta axis, you can align the axis with the following objects:

- R-Value.
- Inner Circle.
- Outer Circle.

When aligning with an R-value you may enter an R-axis value to specify the position of the axis line. When aligning with the inner or outer circle, specify an offset. With a zero offset, the axis line is on the inner or outer circle, a positive offset moves the axis line outside the grid area. A negative offset moves the axis line within the grid area.

For the R-axis the alignment options are more numerous. The possible options are:

- **Theta Value:** Align the R-axis with a specific Theta-value. The axis is limited to the grid area.
- **Grid Border Start:** Align the R-axis with the start of the grid border. The axis is limited to the grid area.
- **Grid Border End:** Align the R-axis with the end of the grid border. The axis is limited to the grid area.
- **Specific Angle:** Align the R-axis with a specific screen angle. The axis is limited to the grid area.
- **Top of Grid Area:** Align the R-axis with the top of the grid area. The axis may be drawn outside the grid area.

- **Bottom of Grid Area:** Align the R-axis with the bottom of the grid area. The axis may be drawn outside the grid area.
- **Left of Grid Area:** Align the R-axis with the left side of the grid area. The axis may be drawn outside the grid area.
- **Right of Grid Area:** Align the R-axis with the right side of the grid area. The axis may be drawn outside the grid area.

In addition to setting the alignment of the R-axis, you may choose to extend the R-axis by drawing an axis line perpendicular or parallel to the existing axis line. When the Draw Axis in Both Directions toggle is selected, Tecplot extends the axis line so it spans the width of the grid area. If the Draw Perpendicular Axis toggle is selected, Tecplot draws an axis line perpendicular to the main axis line.

17.7.3. Three-Dimensional Axes Position

In three dimensions, a given axis line might appear in any of four locations relative to the other axes. By default, Tecplot automatically places the three axis lines so they will not interfere with the drawing of the plot.

You can override this automatic edge assignment for each axis line as follows:

1. From the Plot menu, choose Axis. The Axis Details dialog appears; click Line.
2. Choose the axis for which you want to change the edge assignment.
3. Deselect Auto 3D Edge Assignment. Four option buttons, labeled Min-Min, Max-Min, Min-Max, and Max-Max, become active.
4. Select the option yielding the desired edge assignment.

17.8. Grid Area

The grid area of your plot is the area defined by the axes. For Sketch, XY Line, and 2D Cartesian plots you can alter the size of the grid area by changing the extents of the viewport. (For these plot types the viewport and grid area are synonymous.) For Polar Line and 3D Cartesian plots, the grid area is altered by changes to the axis ranges. For 3-D axes, you can also specify an axis box padding, the minimum distance from the data to the axis box, and whether to light-source shade the axis planes. From the Area page of Axis Details, you control whether the grid area or viewport are color-filled. The Area page is shown in Figure 17-16.

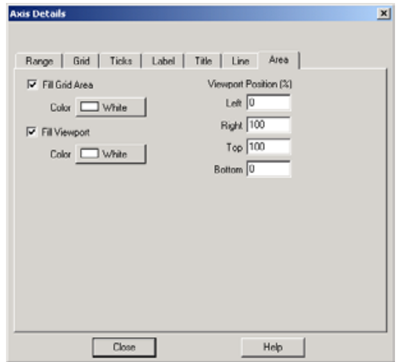


Figure 17-16. The Area page of the Axis Details dialog.

CHAPTER 18 Frame Linking

Tecplot's frame linking feature allows you to link specific style attributes either *between* frames or *within* a frame.

The Set Links for Current Frame dialog controls linking. To access the dialog, select Style Linking on the Plot menu.

18.1. Attribute Linking Between Frames

Linking between frames allows you to quickly make changes in one frame and propagate them through a number of other frames. For example, you might link the contour levels in one frame to the contour levels in another frame. If you change the contour levels in one of the linked frames, the resulting contour levels will also be used in the other linked frame.

18.1.1. Attributes Linkable Between Frames

The Between Frames page of the Set Links for Current Frame dialog is shown in Figure 18-1.

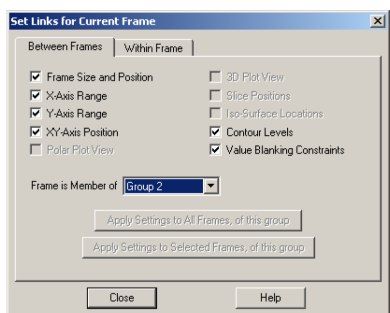


Figure 18-1. The Between Frames page of the Set Links for Current Frame dialog.

It allows you to link the following attributes:

- **Frame Size and Position:** Allows you to stack transparent frames. (See Section 2.1.4, "Frame Background Color Modification," for further information.)
- **X-Axis Range:** For XY Line and 2-D plots. This links not only the ranges on the X-axis but also the positioning of the left and right sides of the viewport.
- **Y-Axis Range:** For XY Line and 2-D plots. Links not only the ranges on the Y-axis but also the positioning of the top and bottom of the viewport.

- **XY-Axis Position:** For XY Line and 2-D plots. Links the positioning of the X- and Y-axes between frames. This includes the method used for positioning the axes, such as aligning with an opposing axis value. The axis positioning has no bearing on the range of values on the axis.
- **Polar Plot View:** Link views for frames using the Polar Line plot type.
- **3D Plot View:** Link the 3-D axes and 3-D view.
- **Slice Positions:** Link slice positions and slice planes for active slices (but not slice style).
- **Iso-Surface Locations:** Link iso-surface values (but not iso-surface plot style).
- **Contour Levels:** Link the values and number of contour levels for 2- and 3-D plots.
- **Value Blanking Constraints:** Link all value-blanking attributes.

18.1.2. Frame Linking Groups

In addition to setting which attributes to link between frames, you can also choose to which group the current frame belongs. The first step when linking attributes between frames is to identify which frames are going to be assigned to which group. If you only have two frames, then this is unimportant as both frames default to being in group 1 and there is no need to have any other groups. Attributes are only propagated to other frames that are members of the same group and that have the same link attributes selected.

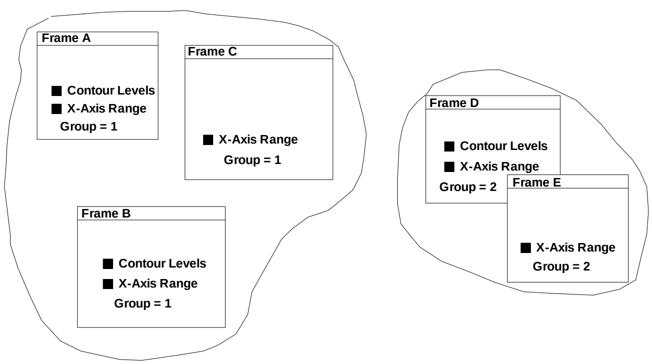


Figure 18-2. Five frames in two groups with different linking options.

Figure 18-2 shows five frames. Frames A, B, and C are in group 1. Frames D and E are in group 2. Note that all five frames have their X-axis ranges linked. If you change the X-axis range in frame A, the corresponding change will occur in frames B and C. It will not occur in frames D and E, as they are not in group 1. A change to the X-axis range in frame D will only be propagated to frame E.

Some frames in Figure 18-2 also have linked contour levels. Changes in the contour levels in frame A would propagate to frame C only. Setting the link for contour levels in frame D has no effect as frame E, the only other frame in group 2, does not have the same attribute linked.

New frames added to a group take on the characteristics of previous members of the group. For this reason, it is important to start the group with a frame that has the characteristics you want for that group (though, once frames are in a group, a change to a linked attribute of one frame changes that attribute for all frames in that group).

18.1.3. Between Frame Link Attributes Propagation to Other Frames

Once link attributes are set in a frame, you must set these same attributes in other frames for linking to occur. Each frame may have each of the attributes selected or not linked. If you want all or a select group of frames to have the same link attributes click the appropriate Apply Settings to All Frames button to quickly propagate the link settings. The alternative is to select each frame individually, making the same selections on the Set Links for Current Frame dialog.

18.1.4. Dependent Axes

When 2-D or XY Line frames have dependent axes and the axis ranges are linked, Tecplot makes a “best-fit” attempt to match the axis ranges between frames. Misalignments can occur when the aspect ratios for the lengths of the axes is not the same between two frames with linked X- and Y-axes. Setting the X- and Y-axes to be independent allows a precise match.

18.2. Attribute Linking Within A Frame

This section describes how to link attributes between similar objects within a frame. An example of *within* frame linking is to link the axis label style. Changing the font for the labels on the X-axis is automatically propagated to the other axes for the current frame.

18.2.1. Attributes Linkable Within Frames

The Within Frame page of the Set Links for Current Frame dialog is shown in Figure 18-3. It allows you to link the following attributes:

- **Axis Style:** Link activation, colors, line styles, font styles for objects associated with axes.
- **Gridline Style:** Link activation, colors, line styles for gridlines.
- **Zone/Map Color between Plot Layers:** Link the color of meshes, contour lines, and other zone layers for Cartesian plots, or link the color of lines, symbols and other map layers for line plots.
- **Zone Line Pattern between Plot Layers:** Link line pattern style and length for meshes, vector and contour lines for Cartesian plots.

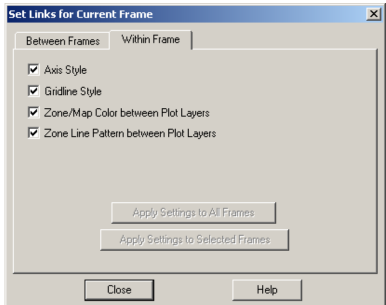


Figure 18-3. The Within Frame page of the Set Links for Current Frame dialog.

18.2.2. Within Frame Link Attribute Propagation to Other Frames

The Apply Settings buttons quickly propagate link settings from the current frame to other frames. The alternative is to visit each frames one by one, making the same selections on the Set Links for Current Frame dialog.

Keep in mind that Within-Frame linking only links attributes between similar objects *within* a frame. These attributes are *not* linked to *other* frames. The Apply Settings buttons only makes other frames have the same Within-Frame linking properties (the same objects within the other frames are linked to themselves).

CHAPTER 19 Finite-Element Data

A finite-element zone consists of a set of points (nodes) that are connected into polygonal or polyhedral units called elements. Associated with each element is a list of the nodes used by that element, in the order in which they are connected. A complete finite-element zone consists of the set of nodes plus the connectivity list for each element.

Tecplot supports both triangular and quadrilateral elements; these are collectively referred to as finite-element surface zones. Tecplot also supports tetrahedral or brick polyhedral elements, which are referred to as finite-element volume zones. Each Tecplot zone must be composed exclusively of one element type. You can, however, simulate zones with mixed element types by repeating nodes as necessary. Thus, a triangle element can be included in a quadrilateral zone by repeating one node in the element's connectivity list, and tetrahedral, pyramidal, and prismatic elements can be included in a brick zone by repeating nodes appropriately.

While finite-element data is usually associated with numerical analysis for modeling complex problems in 3-D structures, heat transfer, fluid dynamics, and electromagnetics, it also provides an effective approach for organizing data points in or around complex geometrical shapes. For example, you may not have the same number of data points on different lines, there may be holes in the middle of the data set, or the data points may be irregularly (randomly) positioned. For such difficult cases, you may be able to organize your data as a patchwork of elements. Each element can be virtually independent of the other elements, so you can group your elements to fit complex boundaries and leave voids within sets of elements. Figure 19-1 shows how finite-element data can be used to model a complex boundary.

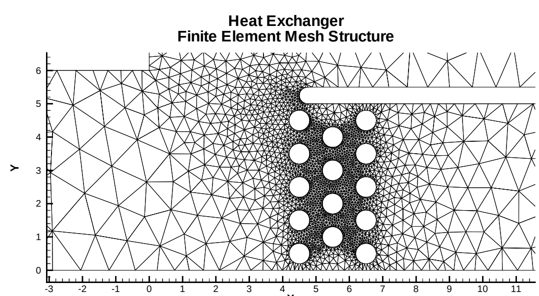


Figure 19-1. Finite-element data used to model a complex boundary.

19.1. Finite-Element Data Sets

Creating a finite-element data set is generally somewhat more work than creating a similar sized ordered data set, because in addition to specifying all the data points, you must also specify the connectivity list which describes how the data points are connected into elements. As an example, consider the data shown in Table 19-1.

Node	X	Y	P	T
A	0.0	1.0	100.0	1.6
B	1.0	1.0	150.0	1.5
C	3.0	1.0	300.0	2.0
D	0.0	0.0	50.0	1.0
E	1.0	0.0	100.0	1.4
F	3.0	0.0	200.0	2.2
G	4.0	0.0	400.0	3.0
H	2.0	2.0	280.0	1.9

Table 19-1. Finite-element data.

We can create a **POINT** Tecplot data file for this data set as follows (a 2-D mesh plot of this data set is shown in Figure 19-2):

```
TITLE = "Example: 2D Finite-Element Data"
VARIABLES = "X", "Y", "P", "T"
ZONE N=8, E=4, DATAPACKING=POINT, ZONETYPE=FEQUADRILATERAL
0.0 1.0 100.0 1.6
1.0 1.0 150.0 1.5
3.0 1.0 300.0 2.0
0.0 0.0 50.0 1.0
1.0 0.0 100.0 1.4
3.0 0.0 200.0 2.2
4.0 0.0 400.0 3.0
2.0 2.0 280.0 1.9
1 2 5 4
2 3 6 5
6 7 3 3
3 2 8 8
```

The **ZONE** record describes completely the form and format of the data set: there are eight nodes, indicated by the parameter **N=8**; four elements, indicated by the parameter **E=4**, and the elements are all quadrilaterals, as indicated by the parameter **ZONETYPE=FEQUADRILATERAL**.

The same data file can be written more compactly in **BLOCK** format as follows:

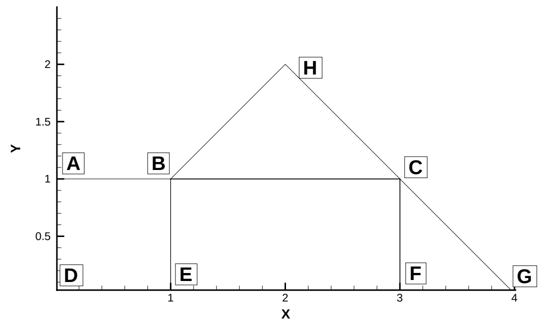


Figure 19-2. A mesh plot of 2-D finite-element data.

```
TITLE = "Example: 2D Finite-Element Data"
VARIABLES = "X", "Y", "P", "T"
ZONE N=8, E=4, DATAPACKING=BLOCK, ZONETYPE=FEQUADRILATERAL
0.0 1.0 3.0 0.0 1.0 3.0 4.0 2.0
1.0 1.0 1.0 0.0 0.0 0.0 0.0 2.0
100.0 150.0 300.0 50.0 100.0 200.0 400.0 280.0
1.6 1.5 2.0 1.0 1.4 2.2 3.0 1.9
1 2 5 4
2 3 6 5
6 7 3 3
3 2 8 8
```

In **`BLOCK`** format, all values for a single variable are written in a single block. The length of the block is the number of data points in the zone. In **`POINT`** format, all variables for a single data point are written in a block, with the length of the block equal to the number of variables. The connectivity list, however, is the same for both formats.

You can change the connectivity list to obtain a different mesh for the same data points. In the above example, substituting the following connectivity list yields the five-element mesh shown in Figure 19-3. (You must also change the **`E`** parameter in the zone control line to specify five elements.)

```
1 2 4 4
4 2 3 5
5 3 6 6
6 7 3 3
3 2 8 8
```

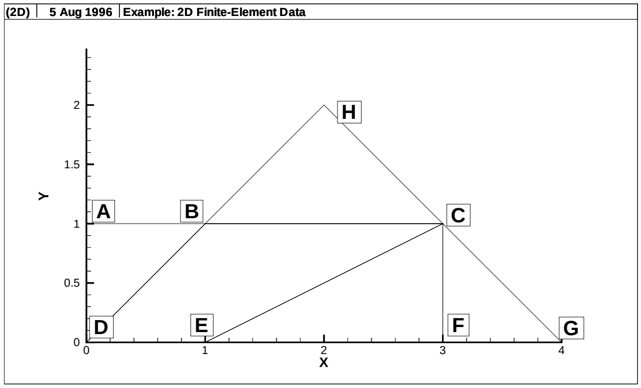


Figure 19-3. Finite-element data of Figure 19-2 with different connectivity list.

Finite-element surface data specify node locations in three dimensions. For example, consider the data in Table 19-2. Locations are listed for eleven nodes, each having only the three spatial variables X, Y, and Z. We would like to create an finite-element surface zone with this data set, where some of the elements are triangles and some are quadrilaterals. All the elements could be organized into one zone, of element type Quadrilateral, but as an illustration of creating 3-D surface data, create three zones: one triangular, one quadrilateral, and one a mixture (using quadrilaterals with repeated nodes for the triangles).

X	Y	Z
0.0	0.0	1.0
0.0	0.0	–2.0
1.0	0.0	–2.0
1.0	1.0	0.0
1.0	1.0	–1.0
1.0	–1.0	0.0
1.0	–1.0	–1.0
–1.0	1.0	0.0
–1.0	1.0	–1.0
–1.0	–1.0	0.0
–1.0	–1.0	–1.0

Table 19-2. Data set with eleven nodes and three variables.

A Tecplot data file for the data in Table 19-2 is shown below in **POINT** format and plotted in Figure 19-4:

```
TITLE = "Example: 3D FE-SURFACE
ZONES"
VARIABLES = "X", "Y", "Z"
ZONE T="TRIANGLES", N=5, E=4, DATAPACKING=POINT, ZONETYPE=FETRIANGLE
0.0 0.0 1.0
-1.0 -1.0 0.0
-1.0 1.0 0.0
1.0 1.0 0.0
1.0 -1.0 0.0
```

```
1 2 3
1 3 4
1 4 5
1 5 2
ZONE T="PURE-QUADS", N=8, E=4, DATAPACKING=POINT,
                    ZONETYPE=FEQUADRILATERAL
-1.0 -1.0 0.0
-1.0 1.0 0.0
1.0 1.0 0.0
1.0 -1.0 0.0
-1.0 -1.0 -1.0
-1.0 1.0 -1.0
1.0 1.0 -1.0
1.0 -1.0 -1.0
1 5 6 2
2 6 7 3
3 7 8 4
4 8 5 1
ZONE T="MIXED", N=6, E=4, DATAPACKING=POINT, ZONETYPE=FEQUADRILATERAL
-1.0 -1.0 -1.0
-1.0 1.0 -1.0
1.0 1.0 -1.0
1.0 -1.0 -1.0
0.0 0.0 -2.0
1.0 0.0 -2.0
1 5 2 2
2 5 6 3
3 4 6 6
4 1 5 6
```

19.2. Three-Dimensional Volume Data Files

Finite-element volume data in Tecplot is constructed from either tetrahedrons having four nodes or bricks having eight nodes. Bricks are more flexible, because they can be used (through the use of repeated nodes in the connectivity list) to construct elements with fewer than eight nodes and combine those elements with bricks in a single zone. Bricks, on the other hand, are harder to construct because care must be taken to make sure all faces in all bricks are planar.

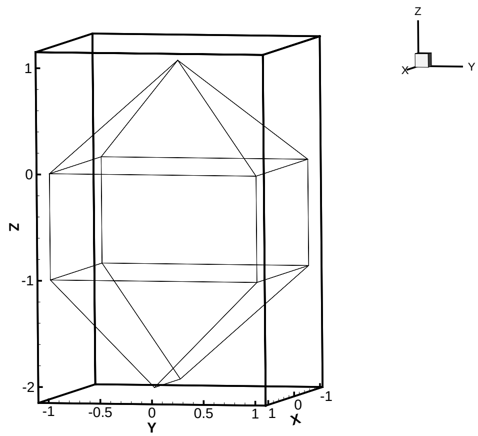


Figure 19-4. Three-dimensional mesh plot of finite-element surface data.

19.2.1. A Finite-Element Volume Brick Data Set

As a simple example of finite-element volume brick data, consider the data in Table 19-3. The data can be divided into five brick elements, each of which is defined by eight nodes.

X	Y	Z	Temperature
0.0	0.0	0.0	9.5
1.0	1.0	0.0	14.5
1.0	0.0	0.0	15.0
1.0	1.0	1.0	16.0
1.0	0.0	1.0	15.5
2.0	2.0	0.0	17.0
2.0	1.0	0.0	17.0
2.0	0.0	0.0	17.5
2.0	2.0	1.0	18.5
2.0	1.0	1.0	20.0
2.0	0.0	1.0	17.5
2.0	2.0	2.0	18.0
2.0	1.0	2.0	17.5
2.0	0.0	2.0	16.5

Table 19-3. Data with fourteen nodes and four variables.

In each element's connectivity list, Tecplot draws connections from each node to three other nodes. You can think of the first four nodes in the element as the "bottom" layer of the brick, and the second four nodes as the "top." Within the bottom or top layer, nodes are connected cyclically (1-2-3-4-1; 5-6-7-8-5); the layers are connected by connecting corresponding nodes (1-5; 2-6; 3-7; 4-8). Figure 19-5 illustrates this basic connectivity. When you are creating your

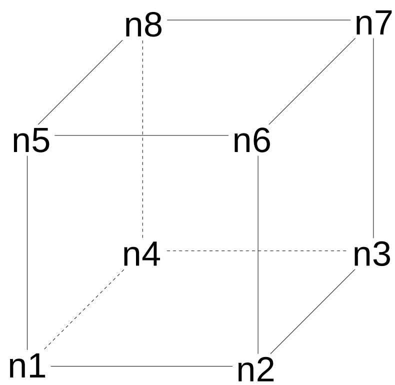


Figure 19-5. Basic connectivity for finite-element bricks.

own connectivity lists for brick elements, you must keep this basic connectivity in mind, particularly when using duplicated nodes to create pyramids and wedges. Tecplot lets you create elements that violate this basic connectivity, but the result will probably not be what you want.

The data file in **POINT** format is shown below:

```
TITLE = "Example: FE-Volume Brick Data"
VARIABLES = "X", "Y", "Z", "Temperature"
ZONE N=14, E=5, DATAPACKING=POINT, ZONETYPE=BRICK
0.0 0.0 0.0 9.5
1.0 1.0 0.0 14.5
1.0 0.0 0.0 15.0
1.0 1.0 1.0 16.0
1.0 0.0 1.0 15.5
2.0 2.0 0.0 17.0
2.0 1.0 0.0 17.0
2.0 0.0 0.0 17.5
2.0 2.0 1.0 18.5
2.0 1.0 1.0 20.0
2.0 0.0 1.0 17.5
2.0 2.0 2.0 18.0
2.0 1.0 2.0 17.5
2.0 0.0 2.0 16.5

1 1 1 1 2 4 5 3
```

```
2 4 5 3 7 10 11 8
4 4 5 5 10 13 14 11
4 4 4 4 9 12 13 10
2 2 4 4 7 6 9 10
```

The same data in **BLOCK** format is shown below:

```
TITLE = "Example: FE-Volume Brick Data"
VARIABLES = "X", "Y", "Z", "Temperature"
ZONE N=14, E=5, DATAPACKING=BLOCK, ZONETYPE=FEBRICK
0.0 1.0 1.0 1.0 1.0 2.0 2.0 2.0 2.0 2.0 2.0 2.0 2.0 2.0
0.0 1.0 0.0 1.0 0.0 2.0 1.0 0.0 2.0 1.0 0.0 2.0 1.0 0.0
0.0 0.0 0.0 1.0 1.0 0.0 0.0 0.0 1.0 1.0 1.0 2.0 2.0 2.0
9.5 14.5 15.0 16.0 15.5 17.0 17.0
17.5 18.5 20.0 17.5 18.0 17.5 16.5

1 1 1 1 2 4 5 3
2 4 5 3 7 10 11 8
4 4 5 5 10 13 14 11
4 4 4 4 9 12 13 10
2 2 4 4 7 6 9 10
```

Figure 19-6 shows the resulting mesh plot from the data set listed in this section.

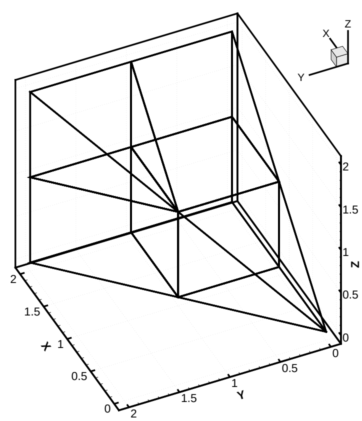


Figure 19-6. A finite-element brick zone.

19.2.2. Creating a Finite-Element Volume Tetrahedral Data Set

As a simple example of an finite-element volume data set using tetrahedral elements, consider the data in Table 19-4. The data set consists of thirteen nodes, with seven variables. The nodes are to be connected to form twenty tetrahedral elements, each with four nodes.

X	Y	Z	C	U	V	W
0	0	–95	–1	1	0	8
0	85	–42	0	–5	–3	9
81	26	–42	2	–22	80	8
50	–69	–42	–6	72	52	9
–50	–69	–42	14	67	–48	9
–81	26	–2	20	–30	–82	9
0	0	0	1	–2	–5	10
50	69	43	14	–68	48	11
81	–26	43	20	31	82	11
0	–85	43	0	84	–3	10
–81	–26	43	2	21	–80	11
–50	69	43	–6	–71	–51	11
0	0	96	1	0	–1	12

Table 19-4. Data with thirteen nodes and seven variables.

The data file in **POINT** format for the data in Table 19-4 is shown below, and plotted in Figure 19-7:

```
TITLE = "Example: FE-Volume Tetrahedral Data"
VARIABLES = "X", "Y", "Z", "C", "U", "V", "W"
ZONE N=13, E=20, DATAPACKING=POINT, ZONETYPE=FETETRAHEDRON
0 0 -95 -1 1 0 8
0 85 -42 0 -85 -3 9
81 26 -42 2 -22 80 8
50 -69 -42 -6 72 52 9
-50 -69 -42 14 67 -48 9
-81 26 -42 20 -30 -82 9
0  0 0 1 -2 -5 10
50 69 43 14 -68 48 11
81 -26 43 20 31 82 11
0 -85 43 0 84 3 10
-81 -26 43 2 21 -80 11
-50 69 43 -6 -71 -51 11
0  0 96 1 0 -1 12
```

```
1 2 3 7
1 3 4 7
1 4 5 7
1 5 6 7
1 6 2 7
2 8 3 7
3 9 4 7
4 10 5 7
5 11 6 7
6 12 2 7
12 2 8 7
8 3 9 7
9 4 10 7
10 5 11 7
11 6 12 7
12 8 13 7
8 9 13 7
9 10 13 7
10 11 13 7
11 12 13 7
```

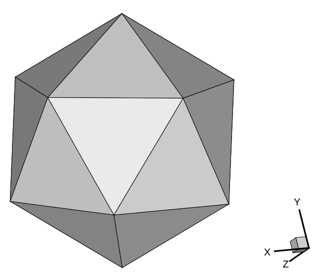


Figure 19-7. Finite-element volume tetrahedral data.

This data file is included in your Tecplot distribution's **`examples/data`** directory as the file **`fetetpt.dat`**. A block format version of the same data is included as the file **`fetetbk.dat`**.

19.3. Triangulated Data Sets

One common source of finite-element surface data is Tecplot's triangulation option. If you have 2-D data without a mesh structure, it is probably simplest to enter your data points as an I-ordered data set, then use Tecplot's triangulation feature to create a finite-element data set.

You can then edit the file, and particularly the connectivity list, to obtain the set of elements you want, rather than having to create the entire connectivity list by hand.

For example, consider again the data of Table 19-1. We can triangulate that data set as follows:

1. Enter the data as a simple ordered data file, as follows:

```
VARIABLES = "X", "Y", "P", "T"
0.0 1.0 100.0 1.6
1.0 1.0 150.0 1.5
3.0 1.0 300.0 2.0
0.0 0.0 50.0 1.0
1.0 0.0 100.0 1.4
3.0 0.0 200.0 2.2
4.0 0.0 400.0 3.0
2.0 2.0 280.0 1.9
```

2. Read the data file into Tecplot and switch the plot type to 2D Cartesian.
3. From the Data menu, choose Triangulate, then select the simple ordered zone as the source zone, and click Compute.
4. From the File menu, choose Write Data File. The Write Data File Options dialog appears.
5. Select the ASCII check box and the Point Format check box, then click OK.
6. Save to a file name of your choice. The result is the following finite-element surface zone (in addition to your original zone):

```
VARIABLES = "X"
"Y"
"P"
"T"
ZONE T="Triangulation"
 N=8, E=7,DATAPACKING=POINT ZONETYPE=Triangle
DT=(SINGLE SINGLE SINGLE SINGLE )
 0.000000000E+000 1.000000000E+000 1.000000000E+002 1.600000024E+000
 1.000000000E+000 1.000000000E+000 1.500000000E+002 1.500000000E+000
 3.000000000E+000 1.000000000E+000 3.000000000E+002 2.000000000E+000
 0.000000000E+000 0.000000000E+000 5.000000000E+001 1.000000000E+000
 1.000000000E+000 0.000000000E+000 1.000000000E+002 1.399999976E+000
 3.000000000E+000 0.000000000E+000 2.000000000E+002 2.200000048E+000
 4.000000000E+000 0.000000000E+000 4.000000000E+002 3.000000000E+000
 2.000000000E+000 2.000000000E+000 2.800000000E+002 1.899999976E+000
 2 3 5
 5 4 2
 4 1 2
 7 6 3
 5 3 6
 3 2 8
 8 2 1
```

Figure 19-8 shows a plot of the resulting data. With triangulation, we obtain more elements (seven) than when we created the data set by hand (four), and the elements are triangles (naturally) rather than quadrilaterals. However, when you have many data points, triangulation is the most reasonable approach.

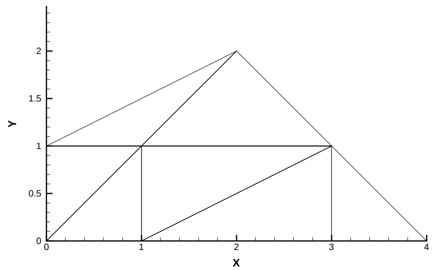


Figure 19-8. Triangulated data from Table 19-1.

19.4. Boundary Extraction of Finite-Element Zones

Boundary lines for finite-element data are similar to boundary lines in ordered data, with a few exceptions. For triangular and quadrilateral meshes a boundary is drawn along the edges of elements that have no neighboring element according to the element connectivity. In some cases finite-element data will be supplied to Tecplot where each element is independent of all the others, that is, elements do not share common nodes. For this type of data a boundary will be drawn around each element.

Finite-element volume data, such as tetrahedral and brick element types, will not plot boundary lines, as opposed to ordered volume data. With finite-element volume data there are, by definition, no boundary lines. However, some plot styles will draw on the outer surface of these zones, in effect they are just drawing on the boundary. Extracting the boundary of these zones extracts the outer surface.

To extract the boundary of a finite-element zone:

1. Choose the appropriate plot type for your data, either 2D or 3D Cartesian. If the zone for which you are extracting the boundary is a 3-D surface, make sure the plot type is set to 3D Cartesian. If you create the boundary zone in a 2D Cartesian plot, the Z-coordinate is not taken into account, and points that are not coincident in 3D Cartesian plots may become coincident in 2D plots. Tecplot eliminates coincident points in the final phase of the boundary extraction, so you could lose important boundary points.
2. From the Data menu, choose Extract, then choose FE-Boundary. The Extract FE-Boundary dialog appears as in Figure 19-9.
3. Choose the source zone, that is, the zone for which you want to extract the boundary zone.

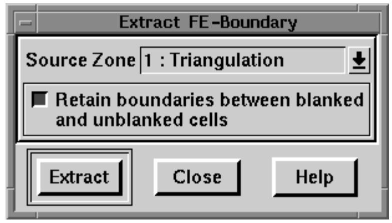


Figure 19-9. Extract FE-Boundary dialog.

4. If blanking is on, decide whether to include the boundary between blanked and un-blanked cells in the zone boundary. To include this boundary, select the check box labeled Retain boundaries between blanked and un-blanked cells.
5. Click Extract. The extracted boundary zone is an FE-surface zone with quadrilateral elements, but each element has two repeated nodes so that each element is a single line segment along the boundary.

19.5. Finite-Element Data Limitations

Working with finite-element data has some limitations, as follows:

- Finite-element data cannot be smoothed.
- Finite-element data cannot be mathematically differentiated.
- XY-plots of finite-element data treat the data as I-ordered; that is, the connectivity list is ignored. Only nodes are plotted, not elements, and the nodes are plotted in the order in which they appear in the data file.
- Index skipping in vector and scatter plots treats finite-element data as I-ordered; the connectivity list is ignored. Nodes are skipped according to their order in the data file.

CHAPTER 20 Three-Dimensional Volume Data

This chapter brings together descriptions of most of the Tecplot tasks involving 3-D volume data, whether IJK-ordered or finite-element. In this chapter, you will find descriptions of the following common 3-D volume tasks:

- Choosing which surfaces you want to plot from your volume data.
- Choosing which data points to use for vector and scatter plots.
- Interpolating 3-D volume irregular data.
- Extracting I-, J-, and K-planes from an IJK-ordered zone.
- Generating and extracting iso-surfaces.
- Generating and extracting planar slices.
- Extracting the outer surface of an FE-volume zone.
- Generating and extracting volumes with a plane.
- Creating specialized 3-D volume plots.

Other related topics such as IJK-blanking and animating IJK-planes are discussed in Chapter 25, “Blanking,” and Chapter 28, “Animation and Movies,” respectively.

20.1. Surfaces to Plot

There are many ways to divide volume data for plotting. One way to view volume data is to select surfaces from part of the data. For example, a typical plot would view a contour flooded plot drawn only on the outer surface of the volume data.

In Tecplot you may choose which surfaces to plot for volume zones from the Surfaces page of the Zone Style dialog. You can call up the Zone Style dialog by clicking Zone Style on the sidebar, or by double-clicking on a zone.

Figure 20-1 shows the Surfaces page of the Zone Style dialog.

The Surfaces to Plot option allows you to choose on of the following:

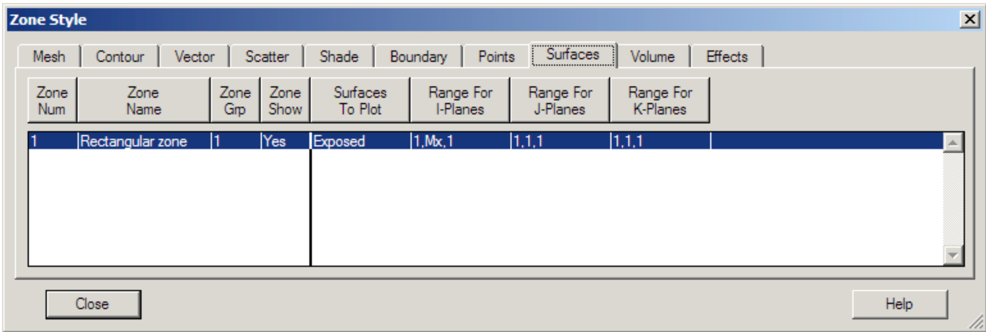


Figure 20-1. The Surfaces page of the Zone Style dialog.

- **Boundary Cell Faces:** This will plot all surfaces on the outside of the volume zone. For IJK-ordered data this amounts to plotting the minimum and maximum I-, J-, and K-planes. For finite-element volume data this will plot all faces that do not have a neighbor cell (according to the connectivity list). If blanking is turned on, the boundary cells in the blanked region will not be drawn and you will be able to see the interior of the volume zone. Figure 20-2 shows plots of a volume zone with surface to plot set to Boundary Cell Faces without blanking, with value blanking, and with IJK-blanking.
- **Exposed Cell Faces (Default):** This setting is similar to the Boundary Cell Faces setting, save for cases in which value blanking is turned on. When value blanking is used the outer surfaces are drawn, similar to results from the Boundary Cell Faces setting. In addition, the cells faces between blanked and non-blanked cells are drawn. Figure 20-3 shows a plot of a volume zone with Surfaces to Plot set to Exposed Cell Faces with and without value blanking.
- **Planes Settings (I-, J-, K-, IJ-, JK-, IK-, and IJK-planes):** These settings will plot the appropriate combination of I-, J-, and or K-planes. The planes are determined by the Range for columns to the right of the dialog. These settings are available only for IJK-ordered data. Figure 20-4 shows a number of examples of plotting I-, J-, and K-planes.
- **Every Surface (Exhaustive):** This setting will plot every face of every cell in volume data. It is not recommended for large data sets. Unless the surfaces are translucent, the plot will appear the same as for the Exposed Cell Faces setting.

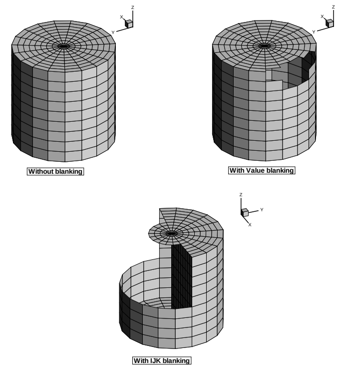


Figure 20-2. Boundary Cell Face plotting without blanking, with value-blanking, and with IJK-blanking.

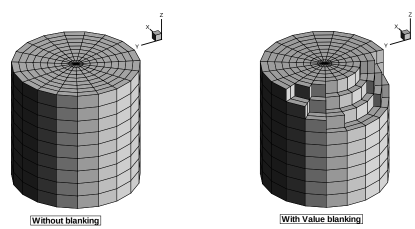


Figure 20-3. Examples of plots where Surfaces to Plot has been set to Exposed Cell Faces with (left) and without (right) value-blanking.

20.2. Points to Plot

You may select the source for the data points used to plot vectors and scatter symbols from the Points page of the Zone Style dialog. Your choices are Surface Nodes, All Nodes, All Connected, Cell Centers Near Surfaces, and All Cell Centers.

Choosing Surface Nodes will draw a vector or scatter symbol (when the appropriate zone layer is active) at every data point on all surfaces being plotted. To select the surface use the Surfaces to Plot option, discussed in the section above.

Choosing All will enable the plotting of vector or scatter symbols at every data point.

A plot where zone 1 is plotting scatter symbols only on one plane (*J=5*) and zone 2 is plotting all symbols is shown in Figure 20-5.

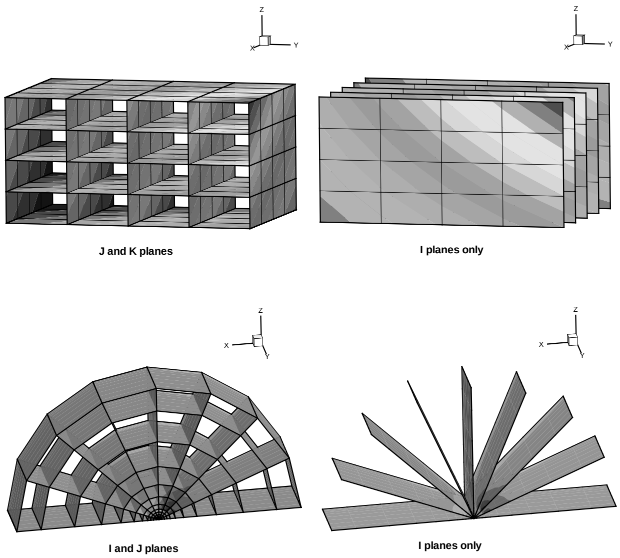


Figure 20-4. Examples of plotting I-, J-, and K-planes.

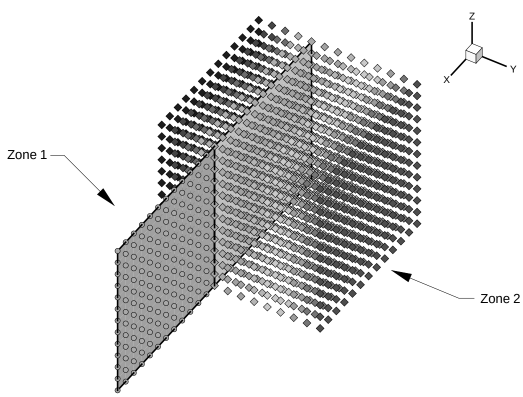


Figure 20-5. A plot showing two zones set to show only J-planes equal to five, with scatter symbols plotted on the surface in zone 1 and all symbols in zone 2.

In addition to selecting which surfaces to use to plot vector and scatter symbols, you may further limit these objects by setting the index skip on the Points pages of the Zone Style dialog. You can only set a skip value for ordered zones.

20.3. Derived Volume Object Plotting

Volume streamlines, volume streamribbons, volume streamrods, slices and iso-surfaces are all derived from volume data automatically. The data used to generate these objects only exists for the life of the frame they are plotted in. When you save a layout or the style of a frame only the instructions necessary to recreate these objects are saved.

From the Volume page of the Zone Style dialog you may include or exclude volume zones from consideration in the construction of volume objects. Figure 20-6 shows a plot with two

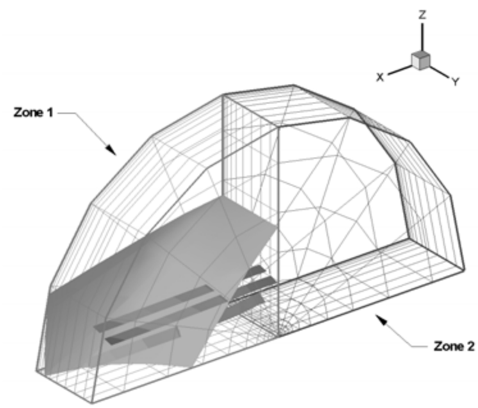


Figure 20-6. A plot where streamribbons and an iso-surface have been excluded from zone 2.

zones where streamribbons and an iso-surface have been excluded from zone 2.

20.4. Three-Dimensional Volume Irregular Data Interpolation

To use 3-D volume irregular data in Tecplot field plots, you must interpolate the data onto a regular, IJK-ordered zone. (Tecplot does not have a 3-D equivalent for triangulation.) To interpolate your data, perform the following steps:

1. Place your 3-D volume irregular data into an I-ordered zone in a data file.
2. Read in your data file and create a 3-D scatter plot.
3. From the Data menu, choose Create Zone, then choose Rectangular. The Create Rectangular Zone dialog appears.
4. Enter the I-, J-, and K-dimensions for the new zone; at a minimum, you should enter 10 for each dimension. The higher the dimensions, the finer the interpolation grid, but the longer the interpolating and plotting time.

5. Enter the minimum and maximum X, Y, and Z values for the new zone. The default values are the minimums and maximums of the current (irregular) data set.
6. Click Create to create the new zone, and close to dismiss the dialog.
7. From the Data menu, choose Interpolate, then choose Kriging. The Kriging dialog appears (alternatively, choose Inverse Distance).
8. Choose the irregular data zone as the source zone, and the newly created IJK-ordered zone as the destination zone. Set any other kriging parameters as desired (see Section 24.9.2, “Kriging,” for details).
9. Click Compute to perform the kriging.

Once Tecplot completes the interpolation, you can plot the new IJK-ordered zone as any other 3-D volume zone. You may plot iso-surfaces, volume streamtraces, and so forth. At this point, you may want to deactivate or delete the original irregular zone so as not to conflict with plots of the new zone.

Figure 20-7 shows an example of irregular data interpolated into an IJK-ordered zone, with iso-surfaces plotted on the resultant zone.

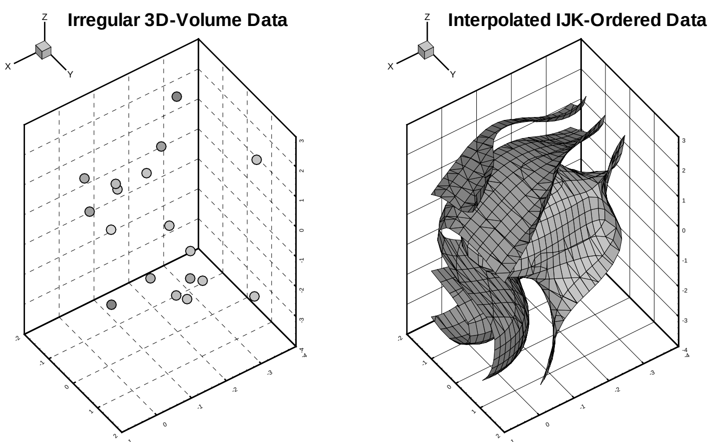


Figure 20-7. Irregular data interpolated into an IJK-ordered zone.

20.5. I-, J-, and K-Plane Extraction

Suppose you want to plot a collection of I-, J-, and K-planes that cannot be specified using the index range and index skip options of the Surfaces page of the Zone Style dialog. You can plot an arbitrary set of planes in Tecplot, but you must first extract each plane as a separate zone. Extracting planes is very simple using the Create SubZone dialog.

To extract a K-plane from an IJK-ordered zone, follow these steps:

1. From the Data menu, choose Create Zone, then choose SubZone. The Create SubZone dialog appears as shown in Figure 20-8.

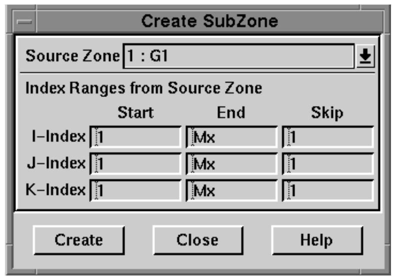


Figure 20-8. The Create SubZone dialog.

2. From the Source Zone drop-down, select the IJK-ordered zone.
3. In the K-Index fields, set Start and End to the same value: the number of the desired K-plane. Set Skip to 1.

   For example, to extract the *K=5* plane, set Start to 5, End to 5, and Skip to 1.
4. Click Create to extract the plane. Tecplot creates an IJ-ordered zone containing just the data points in the extracted plane.

Extracting I- and J-planes is similar.

20.6. Iso-Surface Generation and Extraction

An iso-surface is a surface having a constant value for the contour variable. Iso-surfaces require that your data contains volume zones, such as IJK-ordered, finite-element brick, or finite-element tetrahedral zones. In Tecplot you control iso-surfaces from the Iso-Surface Details dialog under the Plot menu, shown in Figure 20-9.

20.6.1. Iso-Surface Locating

The locations of iso-surfaces are set on the Definition page of the Iso-Surface Details dialog. The contour values where iso-surfaces are defined can either be associated with the current set of levels for the selected contour group, or you may specify up to three unique levels independent of the contour group levels. To enter unique levels, chose the 1, 2, or 3 Specified Values option from the Definition page of the Iso-Surface Details dialog. You can then either enter values in the appropriate text fields, or use the increase or decrease arrows.

If you choose the Contour Group Levels option to draw iso-surfaces, you may control the iso-surface positions using the Contour Details dialog from the Plot menu, or by using a contour level tool from the sidebar. The Contour Add and Contour Delete tools may also be used to add

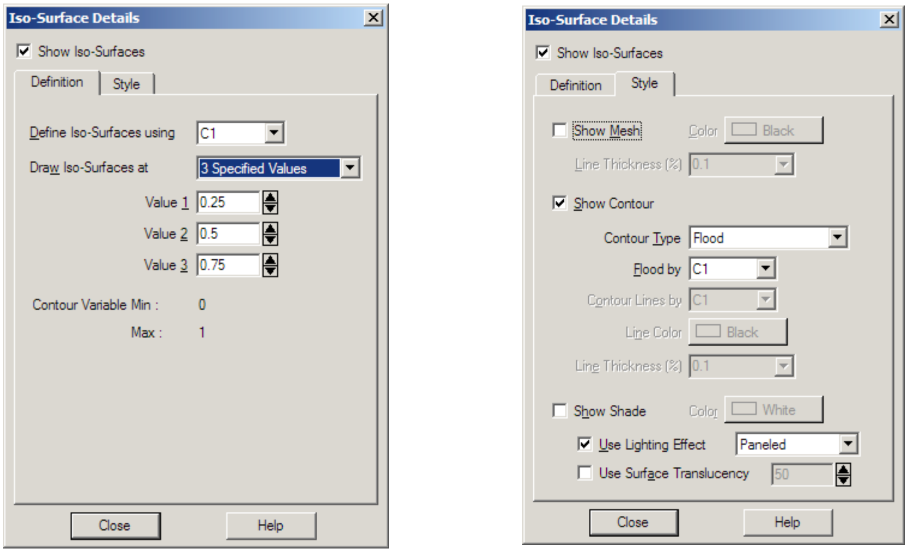


Figure 20-9. The Iso-Surface Details dialog.

or delete iso-surfaces because they add and delete contour levels. The Contour Add tool may also adjust existing levels if you hold down the Ctrl key while clicking and dragging.

20.6.2. Iso-Surface Style

Style settings for all iso-surfaces are handled through the Style page of the Iso-Surface Details dialog. (These are independent of the style assigned to zones by the Zone Style dialog.) The following options are available:

- **Show Iso-Surfaces:** Select this check box to display iso-surfaces.
- **Show Mesh:** Select this check box to display the mesh on iso-surfaces.
- **Mesh Color:** Select a mesh color from the drop-down, or choose a custom color or multi-color.
- **Mesh Line Thickness:** Select a mesh line thickness from the drop-down, or enter your own number in the text field.
- **Show Contour:** Select this check box to display contours on iso-surfaces.
- **Contour Type:** Select the contour display type.
- **Flood by:** If you chose contour flooding, select the contour group by which to flood the contours, or select RGB flooding.
- **Contour Lines by:** If you chose contour lines, select the contour group by which to draw lines.

- **Contour Line Color:** If you chose contour lines, click this button to display the Select Color dialog and choose the line color.
- **Line Thickness:** If you chose contour lines, select a contour line thickness from the drop-down, or enter your own number in the text field.
- **Show Shade:** Select this check box to display shading on iso-surfaces.
- **Shade Color:** Select a shade color from the drop-down, or choose a custom color.
- **Use Lighting Effect:** Select this check box to enable the lighting effect drop-down where you may choose Paneled or Gouraud shading.
- **Use Surface Translucency:** Select this check box to enable the surface translucency text field, where you may set the surface translucency from one (opaque) to 99 (translucent).

20.6.3. Iso-Surface Extraction

You may wish to extract existing iso-surfaces to Tecplot zones to retain these surfaces while switching the contour variable to generate a different set of iso-surfaces. Once extracted, the new zones may be plotted like any other zone in which case style is set with the Zone Style dialog instead of the Iso-Surface Details dialog.

To extract iso-surfaces to zones, perform the following steps:

1. Add iso-surfaces to your plot as described above.
2. From the Data menu, choose Extract, then choose Iso-Surfaces. The Extract Iso-Surfaces dialog appears, as shown in Figure 20-10.

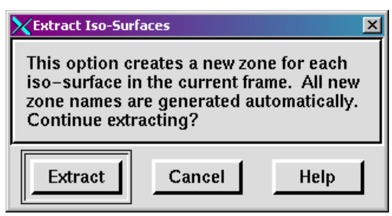


Figure 20-10. The Extract Iso-Surfaces dialog.

3. Click Extract to create the new iso-surface zones, one zone for each contour level. All of the variables in the data set are interpolated from the 3-D volume zones to the data points of the iso-surfaces.

Iso-surface zones are FE-surface quadrilateral element-type zones, regardless of the original 3-D volume zone types. The mesh of the iso-surfaces is derived from the mesh of the original zones, so that in regions where the original mesh was coarse, the iso-surface mesh is coarse, and where the original mesh was fine, the iso-surface mesh is fine.

After creating the new iso-surface zones, it is often a good idea to turn off or reconfigure the current settings for iso-surfaces because the new zones will occupy the same physical space as the iso-surfaces.

20.7. Three-Dimensional Data Slicing

There are two methods for creating slices:

1. Create slicing planes defined by a constant X-, Y-, or Z-location, or constant I-, J-, or K-index for IJK-ordered zones. These slices are created using either the Slice tool from the sidebar, or the Slices option of the Plot menu.
2. Extract an arbitrary slice using the Slice from Plane option on the Extract sub-menu of the Data menu. This option allows you to slice through 3-D surface as well as 3-D volume zones.

These operations are separate and each has unique advantages.

20.7.1. Slice Plane Definition

Slicing planes defined with the Slice Details dialog or the Slice tool become part of the style of your plot. They do not add to the data set used to create your plot unless you extract them. When you save a layout or stylesheet the information about where the slices are defined will be saved in your file.

Starting and ending slice positions may be defined. Intermediate slice positions between the start and end slice may also be activated. You may generate slices of constant X, Y, or Z, or, if you have IJK-ordered data, you may create slices of constant I, J, or K. Only volume zones may be sliced using this feature. The resulting slices are always 3-D surfaces.

Figure 20-11 shows the pages of the Slice Details dialog. Selecting the Show Slices check box activates the start slice.

20.7.1.1. The Position Page. Use the slider to move the start slice, or you may type in the slice position. Activate the end slice and move it with the end slice slider. You may also activate intermediate slices. Intermediate slices are distributed evenly between the start and end slices.

The following options are available:

- **Show Slices:** Select this check box to enable 3-D slicing.
- **Draw Slices at:** Select which plane to slice on.
- **Slice 1:** Your first slice.
- **Position:** Indicates the current location of your slice. Use the slider to select the position of your slice, or enter a value in the field.
- **Show Slice 2:** Select this check box to show a second slicing plane.

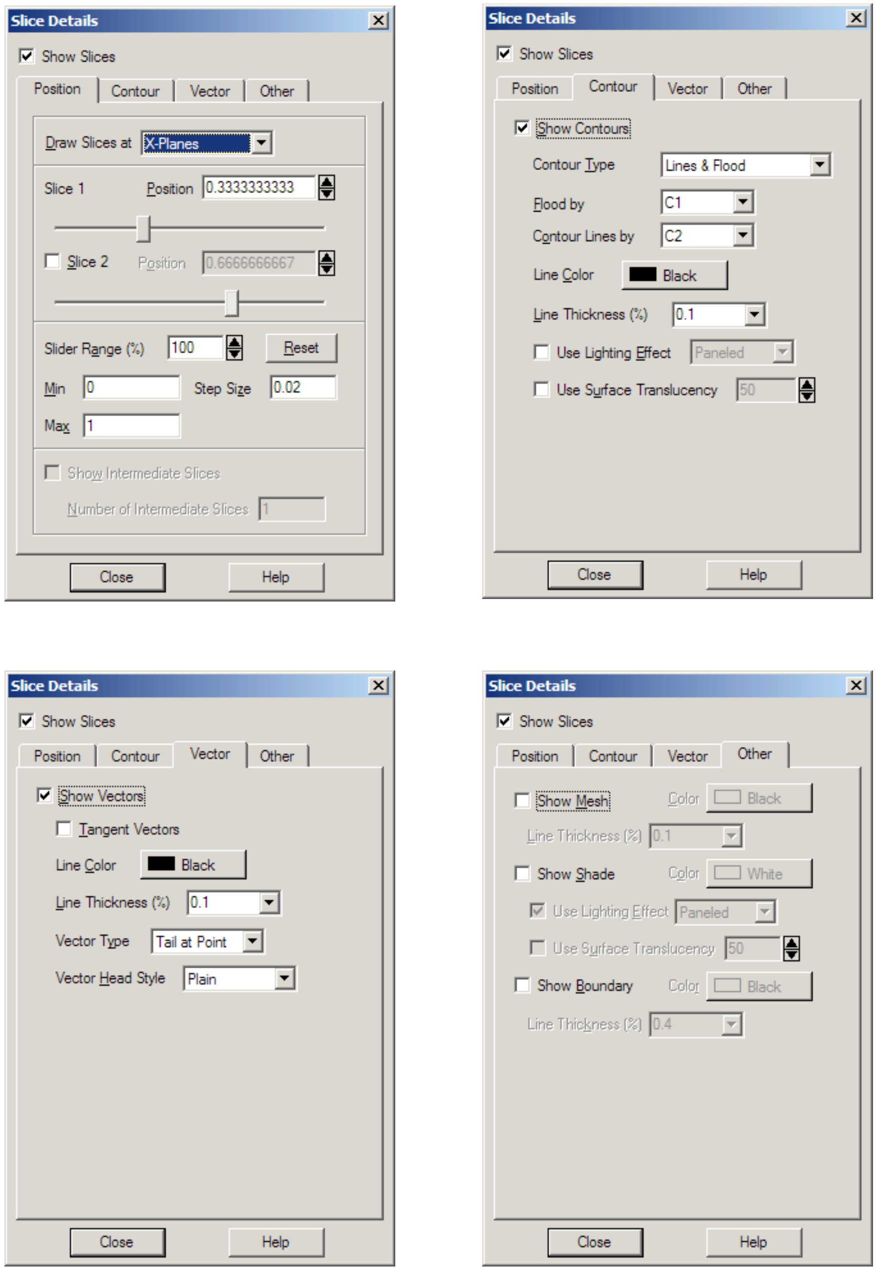


Figure 20-11. The pages of the Slice Details dialog, from upper left: the Position page, the Contour page, the Vector page, and the Other page.

- **Position:** Indicates the current location of your slice. Use the slider to select the position of your slice, or enter a value in the field.
- **Slider Range:** Limit the range for the slides. A value of 100 means the slider range is the same as the range of the axis variable currently being sliced.
- **Show Intermediate Slices:** Select this check box to show intermediate slices between the first and second slices.
- **Number of Intermediate Slices:** Enter the number of intermediate slicing planes in the text field. (Range 1-100.)

20.7.1.2. The Contour Page. Use the Contour page to control the contour attributes of your 3-D slices. The following options are available:

- **Show Contours:** Select this check box to show contours.
- **Contour Type:** Select the contour type of the flood from the drop-down. Lines, Flood, Lines and Flood, Average Cell Flood, and Primary Value Flood are available.
- **Flood by:** If you chose contour flooding, select the contour group by which to flood, or RGB flooding.
- **Contour Lines by:** If you chose contour lines, select the contour group by which to draw the lines.
- **Line Color:** Choose the line color from the Select Color dialog. Multi-Color will color the slice contour lines based on the contour group variable.
- **Line Thickness:** Specify the line thickness as a percentage of the frame width. You may enter a value in the text field, or choose one of the values in the drop-down.
- **Use Lighting Effect:** Select this check box to enable the lighting effect drop-down where you may choose Paneled or Gouraud shading.
- **Use Surface Translucency:** Select this check box to enable the surface translucency text field, where you may set the surface translucency from one (opaque) to 99 (translucent).

20.7.1.3. The Vector Page. Use the Vector page to control the vector attributes of your 3-D slices. The following options are available:

- **Show Vectors:** Select this check box to show vectors.
- **Tangent Vectors:** Select to use tangent vectors for your slices.
- **Line Color:** Choose the line color from the Select Color dialog. Multi-Color will color vectors based on the contour group variable. If no contour variable is set for the selected contour group, the Contour Details dialog will appear.
- **Line Thickness:** Specify line thickness as a percentage of the frame width. You may enter a value in the text field, or choose one of the values in the drop-down.
- **Vector Type:** Use this drop-down to set the vector type for your slices. Choose from Tail at Point, Head at Point, Anchor at Midpoint, and Head Only.

- **Vector Head Style:** Use this drop-down to set the vector head style for your slices. Choose from Plain, Filled, and Hollow.

20.7.1.4. The Other Page. Use this page to control the mesh, shade, and boundary attributes of your 3-D slices. The following options are available:

- **Show Mesh:** Select this check box to show mesh lines.
- **Color:** Choose the line color from the Select Color dialog. Multi-Color will color meshes based on the contour group variable. If no contour variable is set for the selected group when selecting Multi-Color, the Contour Details dialog will appear.
- **Line Thickness:** Specify the mesh line thickness as a percentage of the frame width. You may enter a value in the text field, or choose one of the values in the drop-down.
- **Show Shade:** Select this check box to show shading on the slice when Show Contour has not been selected or is set to Lines.
- **Color:** Choose the shade color from the Select Color dialog. Multi-Color and RGB coloring are not available—use flooded contours for multi-color or RGB flooding.
- **Use Lighting Effect:** Select this check box to enable the lighting effect drop-down where you may choose Paneled or Gouraud shading.
- **Use Surface Translucency:** Select this check box to enable the surface translucency text field, where you may set the surface translucency from one (opaque) to 99 (translucent).
- **Show Boundary:** Select this check box to show selected boundary lines on all slices.
- **Color:** Choose the boundary color from the drop-down of Tecplot's basic colors. Multi-Color and RGB coloring are not available.
- **Line Thickness:** Specify the boundary thickness as a percentage of the frame width. You may enter a value in the text field, or choose one of the values in the drop-down.

20.7.1.5. Slice Tool Use. The Slice tool allows you to position slice planes with your mouse. Select the tool from the sidebar, then click on a surface anywhere in your data. A slice will be positioned according to the XYZ-location of the nearest surface below where you clicked.

When adding a slice to volume data it is often a good idea to plot the original data using the Shade zone layer and set the translucency to a high level, such as 70 percent. This will allow you to see the outer bounds of your data while placing your slice. It is necessary to see the surface in order to be able to place your slice by mouse-click.

The Slice tool offers mouse and keyboard shortcuts which can greatly speed Tecplot use, especially when working with large amounts of data. These are:

Click: Place a start slice.

Drag: Move the start slice.

Shift-click: Place the end slice

Shift-drag: Move the end slice.

+: Turn on the start slice if no slices are active, or turns on the end slice if slices are already active.

- : Turn off the end slice if the end slice is active, or conversely, turns off the start slice if the end slice is not active.

I, J, K (ordered zones only): Switch to slicing constant I-, J-, or K-planes respectively.

X, Y, Z: Switch to slicing constant X-, Y, or Z-planes respectively.

1-9: Activate intermediate slices and set the number of intermediate slices to the number entered.

0: Turn off intermediate slices.

Alt-click/Alt-drag: Determine the XYZ-location by ignoring zones and looking only at derived volume objects (streamtraces, slices, iso-surfaces, slices).

20.7.2. Slice Extraction

In most cases it is not necessary to extract slices to zones. Most existing slice features allow you to set almost any style. There are cases where you may need to display multiple sets of slices in various directions, so it is necessary to extract at least some of the slices to zones.

20.7.2.1. Pre-Defined Slice Extraction. To extract slices that you have pre-defined with the Slice tool or the Slice Details dialog choose the Current 3D Slices option from the Extract sub-menu of the Data menu. This option will create a separate zone for each slice plane.

20.7.2.2. Arbitrary Slice Extraction. To extract a slice at an arbitrary orientation, or to slice a 3-D surface instead of a volume, use the Slice from Plane option from the Extract sub-menu of the Data menu.

Specify any of four different types of cutting planes, as follows:

- **Arbitrary:** An arbitrary cutting plane. You may specify the position and orientation of the cutting plane using three points or an origin and a normal vector, or you can interactively place and rotate the cutting plane using the controls in the Extract Slice dialog.
- **Constant X:** A cutting plane of constant X-value. You may specify the X-value either by entering a value, or using a position slider.
- **Constant Y:** As Constant X above, but for a cutting plane of constant Y-value.
- **Constant Z:** As Constant X above, but for a cutting plane of constant Z-value.

To slice a 3-D zone with a plane:

1. From the Data menu, choose Extract, then choose the Slice from Plane option. The Extract Slice from Plane dialog appears as in Figure 20-12.
2. Choose the option button corresponding to your desired slice plane (Arbitrary, Constant X, Constant Y, or Constant Z).
3. If you choose Arbitrary as your cutting plane, you can either use the Position sliders and Rotate About buttons to position the cutting plane, or choose one of the following buttons:

Figure 20-12. The Extract Slice from Plane dialog.

- **Three Points:** Calls up the Enter Three Points dialog, in which you specify the cutting plane by entering the X-, Y-, and Z-coordinates of three points on the cutting plane (nine numbers in all). These points must form a triangle; they cannot be coincident or collinear.
- **Origin and Normal:** Calls up the Enter Slice Origin and Normal dialog in which you specify the cutting plane by entering the coordinates of a point and the components of a normal vector. Using this option, you enter six numbers to specify the cutting plane: the X-, Y-, and Z-coordinates of a point on the cutting plane (called the slice origin), and the X-, Y-, and Z-components of a vector normal to the cutting plane (called the slice normal).

Use the X,Y, and Z Position sliders (or the associated text fields) to move the cutting plane's slice origin. A representation of the slice plane is shown in the workspace. Use Rotate About to rotate the slice plane about the slice origin.

4. To see a "trace" of the current slice, select the Show Trace check box. If Show Trace is selected, Tecplot draws an approximation of the intersection of the slicing plane with the active 3-D zones. For finite-element zones, the trace in fact draws all line segments of the intersections of the slicing plane with the cells in the zone. For IJK-ordered data, the trace is simply the line resulting from the intersection of the slicing plane and the outer surface of the zone. If Show Trace is not selected, Tecplot simply draws the intersection of the slicing plane with the axis box.
5. Choose to create slices from volume zones, surface zones, or surfaces of a volume zone. A slice from a volume zone will create a plane. A slice from a surface zone, or the surface of a volume zone, will be as a line or curve.
6. Click Extract to extract the slice as a finite-element surface zone.

Once you have created the slice zone, you may plot it, write it out to a data file, delete it, etc. It is the same as any zone that was read into Tecplot. If you slice volume zones the resulting slice zones created are finite-element surface, quadrilateral element-types. If you slice surface zones the resulting zones are finite-element surface, triangle element types.

See Figure 20-13 for an example of a zone created by a slice.

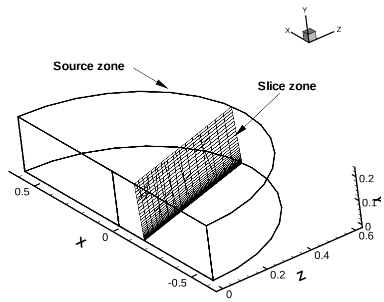


Figure 20-13. Zone extracted by slicing 3-D volume zone.

20.8. Special 3-D Volume Plots

Special 3-D volume plots include fence plots, so called because they look like a flat plane divided by fences, and analytic iso-surface plots, which plot iso-surfaces for analytic functions such as $F(x,y,z)=2x^2-3y^2-7z^2$, and cutaway plots, which are discussed in Chapter 26, "Blanking."

20.8.1. Fence Plots

A fence plot is a plot of planes of a 3-D data field. These planes may be IJ-ordered zones, or combinations of I-, J-, and K-planes of an IJK-ordered zone. In particular, the "bottom" plane of the plot is plotted, plus a few planes that are perpendicular to this plane. These perpendicular planes are the "fences." Typically, flooded contours are plotted on each plane. An example fence plot is shown in Figure 20-14.

Creating a fence plot with IJK-ordered data is simple; just perform the following steps:

1. Read in the IJK-ordered data set.
2. Select the Contour zone layer check box on the sidebar. (This will initially be an iso-surface plot.)
3. Deselect the Mesh zone layer.
4. From the sidebar click Zone Style. The Zone Style dialog appears.
5. From the Surfaces page set Surfaces to Plot to I&J&K Planes.
6. For each of I, J, and K, set the index range as appropriate. In Figure 20-14, the ranges chosen were as follows:
   - **I-planes:** *Start=1*, *End=1*, *Skip=1* (the bottom plane)

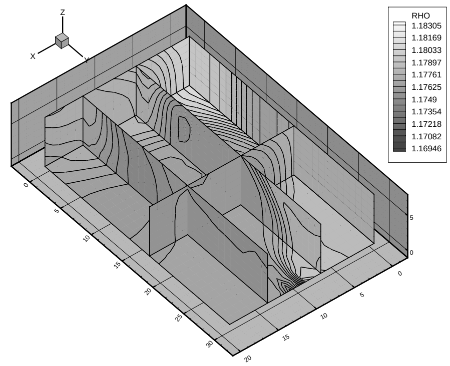


Figure 20-14. A fence plot.

- **J-planes:** *Start=1*, *End=Mx*, *Skip=7* (the three parallel planes)
- **K-planes:** *Start=1*, *End=Mx*, *Skip=17* (the two parallel planes)

7. From the Contour page of the Zone Style dialog, set the contour plot type to Flood or Both Lines and Flood. Set the Flood Translucency to Medium.
8. Select the Shade zone layer.
9. Redraw your plot. You should see a fence plot similar to that in Figure 20-14.

You can also create fence plots using IJ-ordered zones. For the best effect, the plotted zones should be perpendicular to each other when plotted in 3-D. For example, you can create a fence plot from the planes extracted using the procedure in Section 20.5, “I-, J-, and K-Plane Extraction.”

20.8.2. Analytic Iso-Surface Plots

Using Tecplot’s data manipulation tools, you can create iso-surface plots of 3-D volume analytic functions such as $F(x,y,z)=2x^2-3y^2-7z^2$. An iso-surface plot of this function in the range $x=0$ to $x=1$, $y=0$ to $y=1$, and $z=0$ to $z=1$ is shown in Figure 20-15.

To create an iso-surface plot of an analytic function, perform the following steps:

1. From the File menu, choose New Layout to clear the workspace.
2. From the Data menu, choose Create Zone, and then choose Rectangular. The Create Rectangular Zone dialog appears.
3. Enter the dimensions of I, J, and K. These will be, respectively, the number of points plotted in the X-, Y-, and Z-directions. Figure 20-15 was plotted using a dimension of 20 for each index.
4. Enter zero for each of XMin, YMin, and ZMin; enter one for each of XMax, YMax, and ZMax.

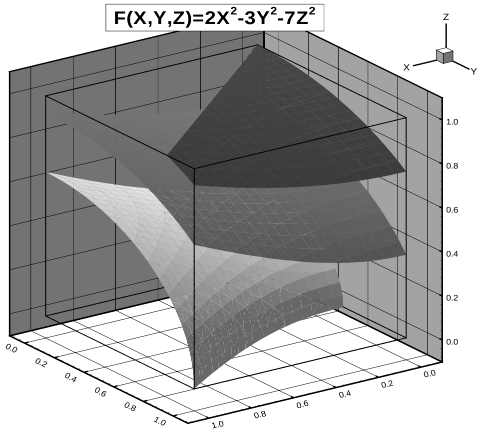


Figure 20-15. Iso-surface plot of an analytic function.

5. From the Data menu, choose Alter, then choose Specify Equations. The Specify Equations dialog appears.
6. In the Equation(s) text field, enter the following equation:

   ```
   V4 = 2*X*X - 3*Y*Y - 7*Z*Z
   ```
7. Click Compute to create the new variable.
8. Deselect the Mesh zone layer check box on the sidebar.
9. Bring up the Iso-Surface Details dialog from the Plot menu. When the Contour Details dialog appears, accept the default, v4, as the contour variable.
10. Turn on Iso-Surfaces.

CHAPTER 21 Printing

Printing your plot is the process of sending the plot image to an output device, print spooler, or a file. Typically the output devices is a printer, but it may be a plotter, film recorder, or typesetting machine. Instead printing it directly on the device, you can print any plot to a file and print that file later. If you are creating files for use in another program, you should use Tecplot's Export option to create your files—Export includes all the supported print file types, as well as several standard graphics formats such as TIFF, WMF, JPEG, and EPS. See Chapter 22, "Exporting," for complete details.

21.1. Plot Printing

To print a plot, select Print from the File menu. This brings up the Print dialog as shown in Figure 21-1.

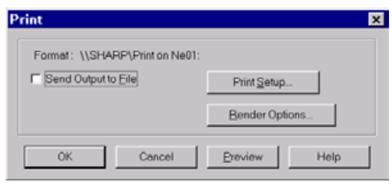


Figure 21-1. The Print dialog (under Windows).

From the Print dialog you can specify whether the output is sent directly to the printer (or print spooler) or to a file. If the Send Output to File check box is selected, the Print to a File dialog appears when you click OK, otherwise the output goes directly to the printer. If you do not select the Send Output to File check box, Tecplot prints the plot directly on the printer (or to the print spooler). The printed plot includes everything on the Tecplot paper. (The UNIX version also allows you to the number of output copies in the Print dialog.)

The Print dialog lets you specify options for your printer by accessing the Print Setup dialog as discussed in Section 21.3, "Printer Setup," and for setting additional controls for the plot via the Print Render Options dialog as discussed in Section 21.4, "Print Render Options." The Print dialog also provides access to Print Preview, which is discussed in Section 21.5, "Print Preview."

21.2. Paper Setup

You can set various parameters relating to the paper, including paper size and orientation, using the Paper Setup dialog or the Print Setup dialog. A change to your paper settings in either the Paper Setup dialog or the Print Setup dialog will automatically update the other.

21.2.1. Windows Print Setup Dialog

The Print Setup dialog is preferable under Windows for setting up your paper. It lists all the paper sizes your printer supports. Tecplot can produce output to fit virtually any paper size. You may select a paper size using the Print Setup dialog's Size drop-down.

The Print Setup dialog is called up by clicking Print Setup on the Print dialog. The Print dialog is accessed from the File menu's Print option.

Print Setup allows you to specify the paper source tray if your printer has multiple paper trays. Do this by choosing a tray from the Print Setup dialog's Source drop-down.

You can choose either Portrait or Landscape paper orientation from the Print Setup dialog. In Portrait orientation, the long axis of the paper is aligned with the vertical axis of the plot. In Landscape orientation, the long axis of the paper is aligned with the horizontal axis of the plot.

The Print Setup dialog is shown in Figure 21-2.

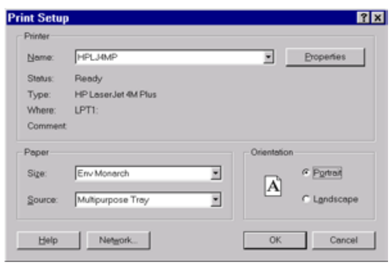


Figure 21-2. The Print Setup dialog under Windows.

21.2.2. Paper Setup Dialog

To adjust the paper size, orientation, and background color for your plots, select the Paper Setup option from the File menu. The current settings for these options are reflected in the representation of the paper in the workspace. (To view the paper, select the Show Paper on Screen check box in either the Paper Setup dialog or the Ruler/Grid dialog under the Workspace menu. This check box is selected by default.)

The Paper Setup dialog, in contrast with the Print Setup dialog under Windows, offers you only six paper sizes. These may not be compatible with the paper sizes your printer supports. You cannot select from multiple paper trays with the Paper Setup dialog. You may set screen display options and fill colors with the Paper Setup dialog. The Paper Setup dialog is shown in Figure 21-3.

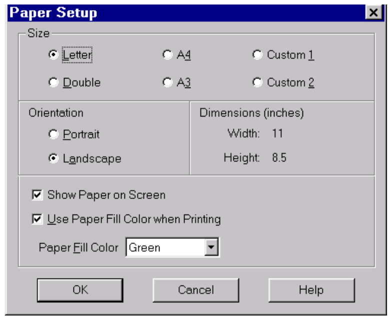


Figure 21-3. The Paper Setup dialog.

The following options are available in the Paper Setup dialog:

- **Size:** Choose the size of the paper from the following six selections:
  - Letter (8.5 x 11 inches).
  - Double (11 x 17 inches).
  - A4 (21x 29.7 cm).
  - A3 (29.7 x 42 cm).
  - Custom 1 (8.5 x 14 inches).
  - Custom 2 (8 x 10 inches).

  Under Windows, paper size Custom 2 is overwritten with the size selected in Print Setup if that size does not exist in Tecplot.

  You can customize all six paper sizes in the configuration file, as well as their hard-clip limits. The hard-clip limits are the lines on the edges of the paper that show where your printer cannot print. You can set the hard-clip limits to larger values for use as guides in placing your plots on the paper.
- **Orientation:** Choose the paper orientation. You have two options: Portrait and Landscape. In Portrait orientation, the long axis of the paper is aligned with the vertical axis of the plot. In Landscape orientation, the long axis of the paper is aligned with the horizontal axis of the plot.
- **Paper Fill Color:** Select a color to use for the paper background. This color is used to display the paper in the workspace. You can select the check box Use Paper Fill Color when Printing to have Tecplot print this background color on the hard-copy as well.

21.3. Printer Setup

You use the Print Setup dialog to set up Tecplot for printing on a particular printer. The available options are different on Windows and UNIX systems. The Print Setup dialog is called up by clicking Print Setup on the Print dialog. The Print dialog is accessed from the File menu's Print option.

21.3.1. Windows Printing

To set up for printing on Windows systems, select the Print option from the File menu. The Print dialog will appear, which was shown in Figure 21-1. Click Print Setup to launch the Print Setup dialog, shown in Figure 21-2.

You may choose to use the printer and specifications presented, or you may click Print Setup or (Print) Render Options to customize your printing.

You may choose to use the Windows default printer, or choose from any currently installed printers. To change to another installed printer, click on the Name drop-down and select another printer from the list.

21.3.2. UNIX Printing

Setting up to print under UNIX includes the following tasks:

- Specifying a spool command, if you are using a print spooler. This may include specifying a device-dependent startup string to condition the output device for the Tecplot output, or a mopup string to reset the output device upon completion of plotting.
- Specifying the precision of the output for those formats which support variable precision.
- Assigning pen colors to pens for pen plotters, if applicable.

You perform most of these tasks from the UNIX version of the Print Setup dialog, accessed from the Print option of the File menu, and shown in Figure 21-4. Some of the Print Setup dialog options launch additional dialogs, which are discussed at the appropriate places in the following sections.

21.3.2.1. Print Formats. In Tecplot, you can choose from any of the following print formats:

- **PostScript (color or monochrome):** PostScript is the recommended output format, since it supports all Tecplot fonts (including Greek and Math), color flooding (or gray-scale flooding), hidden surface (or line) removal, and overlaid frames (plots).
- **HP-GL (line color only):** The HP-GL format generates output for most HP pen plotters as well as other pen plotters that emulate the HP-GL language. Screen colors are mapped to pen numbers on the plotter. HP-GL output can be imported into some programs, such as WordPerfect, but it has some limitations. It does not support contour flooding, multi-coloring, or hidden-surface removal, and it is restricted to using the Tecplot stroke (screen) fonts.

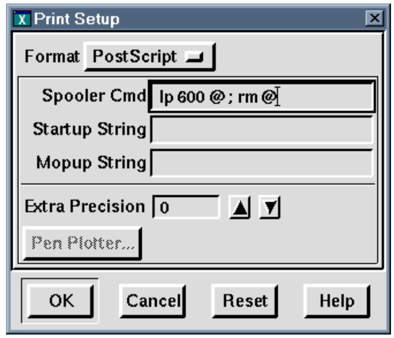


Figure 21-4. The UNIX version of the Print Setup dialog.

- **HP-GL/2 (color or monochrome):** The HP-GL/2 format can be used for HP-GL pen plotters as well as the HP LaserJet III, LaserJet 4, and PaintJet XL printers. When used with a supported printer (as opposed to a pen plotter), HP-GL/2 can show contour flooding, multicoloring and hidden-surface removal, but plots are always limited to using the Tecplot stroke fonts.

To choose the print format:

1. Call up the Print Setup dialog by clicking Print Setup on the Print dialog, accessed from the File menu.
2. Choose the desired format from the Format drop-down.

If your printer is incompatible with all of the above formats, you might consider a PostScript converter such as the freeware program Ghostscript. These programs interpret PostScript output and translate it to the native languages for dozens of supported devices.

21.3.2.2. Spool Commands. Printers on most UNIX systems are accessed via print spoolers that manage the print queue. Under UNIX, you typically use either the `lp` or `lpr` commands to send files to the print spooler. There may be command-line options that need to be set on your system, as well, such as a flag to specify a particular printer. You use the Print Setup dialog to specify the appropriate spool command for your system.

To specify the spool command for your system:

1. Call up the Print Setup dialog, and choose the desired format. (Spool commands will most likely be different for different print formats, and Tecplot stores one spool command for each print format.)
2. In the Spooler Cmd text field, enter the appropriate spool command for your system, using the `@` symbol to represent a file name.

For example, suppose you routinely use the following spool command to print a file named **`myfile.ps`**: “**`lpr -m -r myfile.ps`**.” The appropriate spooler command to enter in the Spooler Cmd field is then “**`lpr -m -r @`**.”

When printing to a spooler, Tecplot creates temporary files with names of the form **`tp??????`**, where the **`?`**s are randomly generated characters. Tecplot does not delete these temporary files automatically; commands to do so should be included in your spool command. In our example, the **`-r`** flag says to remove the file when done.

21.3.2.3. Startup and Mopup Strings. A startup string is an initialization string that sets up your output device to accept the plot created by Tecplot. A mopup string is a reset signal that tells your output device that the special output has ended. For most devices no startup or mopup strings are needed. However, some common devices, such as the HP LaserJet III when printing HP-GL/2, require both startup and mopup strings.

To specify a startup or mopup string:

1. Call up the Print Setup dialog, and choose the desired format. (Startup and mopup strings will be different for each format, and Tecplot stores one startup and mopup string for each print format.)
2. Enter the appropriate startup string or mopup string in the appropriate text field. Special characters are generated by using Macro Codes (such as “**`%E`**” for the escape character and “**`^`***nnn*” for any ASCII character with a decimal ordinal value of *nnn*). Check your printer documentation for the appropriate strings. For example, with some HP-GL implementations, the HP-GL startup string must be set to the following:
   `$E.J$E.N;19:$E.I81;;17:`

   The HP LaserJet III requires the HP-GL/2 startup and mopup strings shown below:
   - Startup String: **`$E%-1B`**
   - Mopup String: **`$E%0A^012`**

21.3.2.4. Printing Precision. For PostScript and HP-GL/2 output, you can control the numerical precision used in your print files. Print files contain numbers that define sizes and positions of pieces of the plot on the output paper. These numbers are defined as integers between zero and about 8,000. Usually, this provides sufficient resolution for most output devices. Occasionally, you may need more resolution. For example, printing to a high-resolution output device like a Linotronic typesetter may require more precision; making print output with very small cells or elements may also require more precision.

To increase the precision of the output, increase the value in the Extra Precision field of the Print Setup dialog. You specify one Extra Precision value for all formats that supports precision control. The precision is defined as the number of digits to the right of the decimal. Normally, precision is zero. The disadvantage of setting precision high is that the print files increase in size. The higher the Extra Precision setting, the larger your print files, but the more accurate the plot. Numbers above two are not normally required unless you need extremely fine resolution. The maximum setting for the precision is eight.

21.3.2.5. Pen Plotter Configuration. If you are using a pen plotter, you can use the Pen Plotter Device Configuration dialog shown in Figure 21-5 to specify plotter speed and pen assignments for particular colors and Tecplot object types. You access this dialog by clicking Pen Plotter on the HP-GL page of the Print Setup dialog. You can associate each of Tecplot's basic colors with any of the plotter's pens.

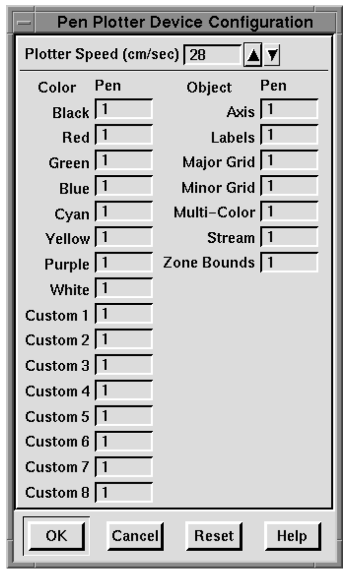


Figure 21-5. The Pen Plotter Device Configuration dialog.

You may also specify that the following object types be associated with a particular pen:

- Axes.
- Tick mark labels.
- Major grid lines.
- Minor grid lines.
- Objects colored with the multi-color option.
- Streamtraces.
- Zone boundaries.

To specify a pen assignment, simply enter the number of the desired pen in the appropriate color or object field.

To specify a plotter speed, either enter a value in the Plotter Speed field, or use the up and down arrow buttons to increment and decrement the current value.

Click OK to accept your changes, Cancel to quit with no changes, or Reset to reset the configuration to its last saved value.

The pen plotter configuration settings only affect the output only when the print format is HP-GL; they have no effect on PostScript or HP-GL/2. There are no settings for Flood Pens because flooding is not supported under HP-GL. The default is for all lines and text to use Pen 1.

21.4. Print Render Options

Clicking the Print dialog’s (Print) Render Options calls up the Print Render Options dialog, shown in Figure 21-6.

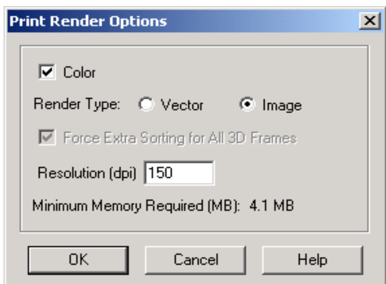


Figure 21-6. The Print Render Options dialog under Windows.

The Print Render Options dialog offers you the following choices:

- **Color:** Select this check box for color output; deselect the check box for monochrome output.
- **Vector:** Select this option to create print output using the drawing commands of the printer. The printer renders the plot, yielding higher resolution, but some plot options, such as translucency, are not available.
- **Image:** Select this option to create print output using an image. Rendering is done by Tecplot at the specified resolution, usually less than the printer’s resolution. However, all plot options are available.
- **Force Extra Sorting for all 3D Frames:** This option is available when the Vector option has been selected. Selecting this check box will cause Tecplot to use extra sorting in all 3-D frames. This overrides the setting in the Advanced 3D dialog. If this check box is not selected, Tecplot will choose sorting algorithms based on the Advanced 3D dialog options that were chosen for each frame. When printing 3-D plots in a vector graphics format, Tecplot must sort the objects so that it can draw those farthest from the screen first and those closest to the screen last. By default, Tecplot uses a quick sorting algorithm. This is not always accurate and does not detect problems, such as intersecting objects. If Extra Sorting is selected, Tecplot uses a slower, more accurate approach that detects problems.

- **Resolution (dpi):** Available when the Image option is selected. Enter the resolution in terms of dpi in the text field. Larger resolutions may result in an out-of-memory condition, or produce very large files. Smaller resolutions may yield less-attractive output images.

The Print Render Options dialog also indicates the amount of memory your final output will require when the selected Render Type is Image.

21.5. Print Preview

A preview of your screen image as it will be rendered for the printer may be generated by selecting Print Preview from the File menu. The preview image may be accessed by clicking Preview on the Print dialog, which is shown in Figure 21-1. There are several reasons for viewing the print preview image prior to sending the plot to the printer. They are associated with the image quality and reduced image content that can be supported for vector graphics printer formats such as PostScript.

As discussed in Section 21.4, “Print Render Options,” the default sorting algorithm used by Tecplot may have problems with intersecting objects. This will typically not show up in the OpenGL-rendered screen image. However, sorting errors may occur for vector print output. These will be visible in the preview. The Print Preview option provides access to the Print Render Options dialog, where you may improve sorting by selecting Force Extra Sorting for All 3D Frames. If extra sorting does not solve the problem, the only option available is to export the plot using an image format, discussed in Chapter 22, “Exporting.” By increasing the resolution for an image format you can obtain a quality comparable to PostScript without the sorting errors.

Vector graphics formats do not support translucency, contour flooding with Gouraud shading, or contour flooding using the continuous color distribution method (which is only available with OpenGL). Print Preview will not display translucency. Gouraud shading for contour flooding will be reduced to Paneled shading. Continuous color flooding will be reduced to color flooding with average-cell color. When you print, warning messages will be displayed to advise you about the unsupported plot styles. The resulting printed output will closely match the print preview image.

CHAPTER 22 *Exporting*

In Chapter 21, "Printing," we stated that all Tecplot plots could be printed to files, as well as directly to printers. Print files are useful only for printing at a later time. Sometimes, however, you want to create files for use in other applications, such as plots to be included in a word-processor document, or to be edited by a graphics program. Sometimes, you can use print files for these purposes. More often, however, you need plots in different formats. Use the Export dialog under the File menu to create files for export into other applications.

Tecplot generates types of export files—vector graphics and image. Vector export files have device-independent resolution and thus can be easily resized, but they have the same limitations as vector print output. Image output has the advantage of accurately representing translucency and smooth color gradations, but with the disadvantage of generally being larger than vector output, particularly when a high image resolution is specified. Image files are sometimes called raster or bit-mapped.

Tecplot exports the following formats:

- **AVI**: Image in Audio Visual Interleaved format (a common Windows movie file format). AVI files may contain multiple images for animations.
- **BMP:** Image in Windows Bitmap format.
- **Encapsulated PostScript (EPS):** Vector or image graphics in a special type of PostScript file designed for inclusion in other applications.
- **HP-GL:** Vector graphics mainly for pen plotters, although you can import these files into some applications.
- **HP-GL/2:** Vector graphics mainly for pen plotters, some HP LaserJets or PaintJets.
- **JPEG:** Image in JPEG format. JPEG files are very small for their resolution and quite common on the internet, but they do involve some loss of image quality that may affect certain plot images.
- **PNG:** Image in Portable Network Graphic format. Also common on the internet, but creating files with no loss in image quality but with larger file size than JPEG.
- **PostScript (PS):** Vector or image graphics suitable for direct printing, but usually unsuitable for import into other applications. It is recommended that you use the Encapsulated PostScript (EPS) format for importing into other applications.
- **Raster Metafile:** Image in NASA's Raster Metafile format. Raster Metafile files may contain multiple images for animations. Used for creating movies for Framer.

- **Sun Raster:** Image in Sun Microsystems' Sun Raster format.
- **TIFF:** Image in Tagged Image File Format.
- **WMF (Windows Metafiles):** Vector graphics to import into various Windows applications.
- **X-Windows:** Image in “xwd” (X-Window Raster) format.

On Windows and Macintosh systems, Tecplot can export directly to the clipboard instead of to a file. Windows systems export BMP and WMF directly to the clipboard. Macintosh systems export PICT files directly to the clipboard. PICT files are an image format unique to the Macintosh version of Tecplot, and are only available through when copying directly to the clipboard.

Certain images formats support anti-aliasing, a feature that removes “jaggies” from text, lines and edges. This feature is discussed at the end of this chapter.

22.1. Export File Creation

The basic procedure for creating an export file is the same regardless of the exported file format:

1. From the File menu, select Export. This calls up the Export dialog, shown in Figure 22-1.

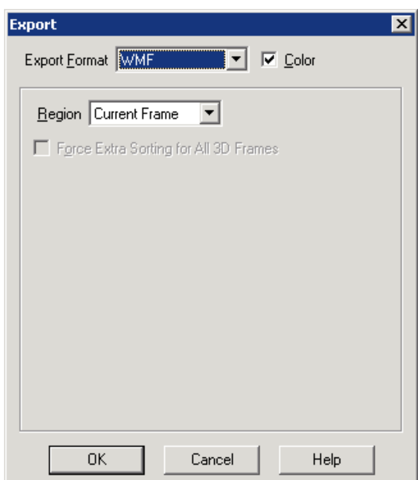


Figure 22-1. The Export dialog.

2. Select a format from the Export Format drop-down.
3. Set format-specific options.
4. Click OK in the Export dialog, and specify a file name for the export file.

22.2. Audio-Visual Interleaved (AVI) Export

The AVI format is used for viewing movies created in Tecplot. AVI is an image format, but Tecplot uses only 256-color images with AVI files. Thus, AVI can accurately represents some plots with translucency and smooth color gradations. The AVI export options are shown in Figure 22-2.

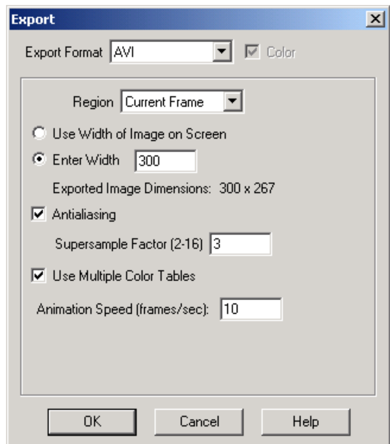


Figure 22-2. AVI options on the Export dialog.

When you select AVI in the Export dialog, you have the following options:

- **Region:** Choose to export only the current frame, or the smallest rectangle containing all frames, or everything shown in the workspace.
- **Use Width of Image on Screen:** Select this option to generate an image file the same size as the current plot on the screen. This option is forced on if you use on-screen image rendering in the Display Performance dialog. See Section 30.3, "Display Performance Dialog," for details.
- **Enter Width:** Select this option to specify a width (in pixels) for the generated image. The greater the width you specify, the longer it will take to export the image and the larger the exported file. However, a larger width increases the quality of your image. This option is not available if you have chosen to use on-screen image rendering.
- **Antialiasing:** Select this option to remove jaggies from the image. See Section 22.15, "Antialiasing Images," for details.
- **Supersample Factor:** Control the amount of antialiasing used in the image. See Section 22.15, "Antialiasing Images," for details.

- **Use Multiple Color Tables:** Select this check box to create an AVI file with a separate color table each step in the animation. If this check box is not selected, Tecplot scans all steps in the animation and creates one color table for the entire animation. Multiple color tables can provide better image quality for the animation, but many applications display only the first color table provided.
- **Animation Speed:** Set the speed of the animation in frames per second.

For more information on AVI files, see Chapter 29, “Animation.”

22.3. Bitmap (BMP) Export

BMP is an image format, and thus accurately represents plots with translucency and smooth color gradations. The BMP export options are shown in Figure 22-3.

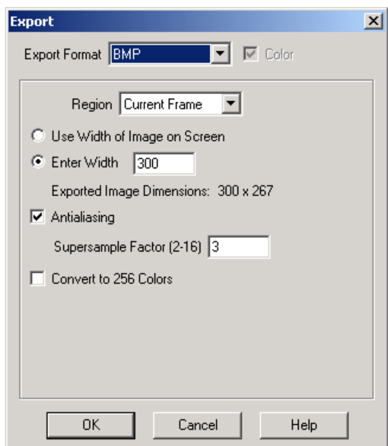


Figure 22-3. BMP options on the Export dialog.

When you select BMP in the Export dialog, you have the following options:

- **Region:** Choose to export only the current frame, or the smallest rectangle containing all frames, or everything shown in the workspace.
- **Use Width of Image on Screen:** Select this option to generate an image file the same size as the current plot on the screen. This option is forced on if you use on-screen image rendering in the Display Performance dialog. See Section 30.3, “Display Performance Dialog,” for details.
- **Enter Width:** Select this option to specify a width (in pixels) for the generated image. The greater the width you specify, the longer it will take to export the image and the larger the exported file. However, a larger width increases the quality of your image. This option is not available if you have chosen to use on-screen image rendering.

- **Antialiasing:** Select this option to remove “jaggies” from the image. See Section 22.15, “Antialiasing Images,” for details.
- **Supersample Factor:** Control the amount of antialiasing used in the image. See Section 22.15, “Antialiasing Images,” for details.
- **Convert to 256 Colors:** Select this check box generate an image with only 256 colors (down from a possible 16 million colors). Tecplot selects the best color match. The image will have a greatly reduced file size, but for plots with many colors, the results may be sub-optimal. Using this option with transparency, smooth color gradations, or antialiasing may result in poor image quality.

22.4. Encapsulated PostScript (EPS) Export

Encapsulated PostScript file (EPS) are Postscript files with additional commands that another program can use to determine the size of your plot. After you import your EPS file into another program, you can position it and usually resize it before printing. (If you try to send an EPS directly to a printer, it may not be positioned correctly on the paper. Use Tecplot’s PS export format to create files to send directly to a printer.)

EPS files may also contain a preview image, which allows you to see a rough picture of your plot in the program that loads the EPS file. There are two basic types of preview images:

- An EPS file with a TIFF preview image has a binary header giving some information about the TIFF image. The preview image is in binary at the bottom of the file. In order for a program to read in an EPS file with a TIFF preview image, it must be able to interpret these binary sections. This type of EPS file can not be sent directly to a printer.
- All of the other preview image types are included within the body of the EPS file as PostScript comments. An EPS file with these types of preview images can be sent directly to the printer. The printer ignores the preview commands, however, and may place the plot incorrectly.

For some applications, if you import an EPS file and print to a non-PostScript printer, only the preview image is printed. Under Windows you must tell Windows that the printer is a PS printer, or you also get the preview image.

The EPS export options are shown in Figure 22-4.

The following options are available:

- **Color:** Choose between color and gray-scale EPS output.
- **Vector/Image:** Choose the Render Type used—vector PS commands or a PS image in the file. Vector commands generally result in a smaller file, but a PS image is required to accurately represent translucency or smooth color gradations.

If you choose Vector Render Type, the following options are also available:

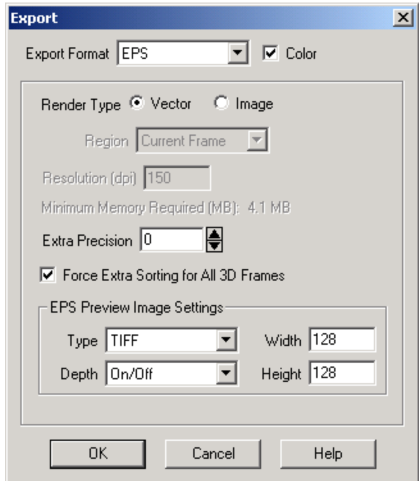


Figure 22-4. EPS options on the Export dialog.

- **Extra Precision:** Specify the number of decimal places to which size and position parameters are carried in the resulting vector-based EPS output. Use the increase or decrease arrows, or enter a value in the text field. Larger values create more accurate plots, but result in larger file sizes.
- **Force Extra Sorting for All 3D Frames:** Selecting this check box causes Tecplot to use extra sorting in all 3-D frames.This overrides the setting in the Advanced 3D dialog. If this check box is not selected, Tecplot chooses sorting algorithms based on the Advanced 3D dialog options for a given frame.

If you choose Image Render Type, the following options are available instead of the above:

- **Region:** Choose to export only the current frame, or the smallest rectangle containing all frames, or everything shown in the workspace.
- **Resolution:** Enter the resolution of the image in dots per inch. Larger values create more accurate plots, but result in larger file sizes.

Regardless of the Render Type, you may choose the type of preview image included in your EPS files. A preview image is a rough sketch of your print file used by importing programs. You may also specify the preview images size (in pixels) in the width and height fields. Low values make the preview image poor. High values can make the EPS file large. (Note: When using Render Type Image, these preview image width and height values are separate from the size of the actual EPS image. The actual EPS image size is determined by the Resolution setting.)

Tecplot provides the following options for the preview image:

- **None:** No preview image information is include. This is good for importing into applications that do not use preview image information.
- **TIFF:** Include a monochrome or gray-scale TIFF preview image. (Color preview images are not available.) This is the most common preview image format. You may specify an image depth for the preview image in the Depth drop-down. TIFF image depth options are described in Section 22.11, "Tagged Image File Format (TIFF) Export." (This preview image depth is separate from the depth of the actual image for EPS files generated with Render Type Image. The actual image depth is determined by Tecplot.)
- **EPSIV2:** Include a monochrome (one bit per pixel) Encapsulated PostScript Version 2 preview image. This is also a common preview image type in EPS files.
- **FrameMaker:** Include a monochrome preview image tuned to work with older versions of Adobe FrameMaker. This preview image type is rarely necessary.

22.5. HP-GL and HP-GL/2 Export

The Export dialog allows you to export plots in HP-GL and HP-GL/2, although these formats are usually used for printing directly to a plotter or LaserJet or PaintJet printer. Some applications import HP-GL format files, but the format strips much of the information out of the plot, and you are generally better off using another format.

If you want to export an HP-GL or HP-GL/2 file, the process for creating these export files is very similar to printing to a file on a UNIX system (even under Windows). See Chapter 21, "Printing," for details.

22.6. Joint Photographic Experts Group (JPEG) Export

JPEG is an image format, and thus accurately represent plots with translucency and smooth color gradations. However, JPEG is a highly compressible, "lossy" format, and can result in poor image quality for some types of images. The advantage of JPEG is very small file sizes and near universal acceptance on the internet. JPEG supports different qualities of compression, and Tecplot allows you to control the image quality (and thus, inversely, the file size).

When you select JPEG in the Export dialog, you have the following options:

- **Region:** Choose to export only the current frame, or the smallest rectangle containing all frames, or everything shown in the workspace.
- **Use Width of Image on Screen:** Select this option to generate an image file the same size as the current plot on the screen. This option is forced on if you use on-screen image rendering in the Display Performance dialog. See Section 30.3, "Display Performance Dialog," for details.

- **Enter Width:** Select this option to specify a width (in pixels) for the generated image. The greater the width you specify, the longer it will take to export the image and the larger the exported file. However, a larger width increases the quality of your image. This option is not available if you have chosen to use on-screen image rendering.
- **Antialiasing:** Select this option to remove “jaggies” from the image. See Section 22.15, “Antialiasing Images,” for details.
- **Supersample Factor:** Control the amount of antialiasing used in the image. See Section 22.15, “Antialiasing Images,” for details.
- **Encoding:** Choose an encoding method for the JPEG file. Standard encoding creates a JPEG which downloads one line at a time, starting at the top line. Progressive encoding creates a JPEG image that can be displayed with a “fade in” effect in a browser. This is sometimes useful when viewing the JPEG in a browser with a slow connection, since it allows an approximation of the JPEG to be drawn immediately, and the browser does not have to wait for the entire image to download. Given the same Quality level (see below), Standard encoded JPEG files look better, although they are larger than the equivalent Progressive encoded JPEG files.
- **Quality:** Select the quality of JPEG image. Higher quality settings produce larger files and better looking export images. Lower quality settings produce smaller files. For best results, use a quality setting of 75 or higher.

22.7. Portable Network Graphics (PNG) Export

PNG is an image format, and thus accurately represent plots with translucency and smooth color gradations. The PNG export options are shown in Figure 22-5.

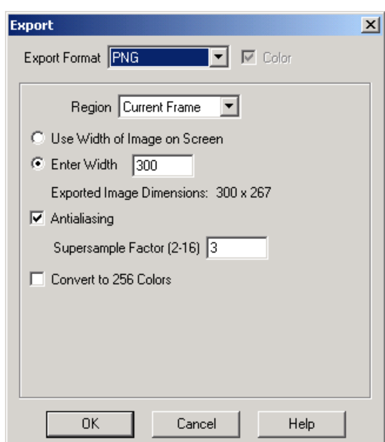


Figure 22-5. PNG options on the Export dialog.

When you select PNG in the Export dialog, you have the following options:

- **Region:** Choose to export only the current frame, or the smallest rectangle containing all frames, or everything shown in the workspace.
- **Use Width of Image on Screen:** Select this option to generate an image file the same size as the current plot on the screen. This option is forced on if you use on-screen image rendering in the Display Performance dialog. See Section 30.3, “Display Performance Dialog,” for details.
- **Enter Width:** Select this option to specify a width (in pixels) for the generated image. The greater the width you specify, the longer it will take to export the image and the larger the exported file. However, a larger width increases the quality of your image. This option is not available if you have chosen to use on-screen image rendering.
- **Antialiasing:** Select this option to remove jaggies from the image. See Section 22.15, “Antialiasing Images,” for details.
- **Supersample Factor:** Control the amount of antialiasing used in the image. See Section 22.15, “Antialiasing Images,” for details.
- **Convert to 256 Colors:** Select this check box generate an image with only 256 colors (down from a possible 16 million colors). Tecplot selects the best color match. The image will have a greatly reduced file size, but for plots with many colors, the results may be sub-optimal. Using this option with transparency, smooth color gradations, or antialiasing may result in poor image quality.

22.8. PostScript (PS) Export

The Export dialog allows you to export plots in PostScript, although this format is usually used for printing directly to a printer or print spooler. It is recommended that you use the Encapsulated PostScript (EPS) format for importing into other applications. See 22.4, “Encapsulated PostScript (EPS) Export,” for details.

If you want to export PostScript file (perhaps for later printing), the process for creating a PostScript export file is very similar to printing to a file on a UNIX system (even under Windows). See Chapter 21, “Printing,” for details.

22.9. Raster Metafile (RM) Export

Raster is an image format, but Tecplot create 256-color images with Raster Metafiles files. Thus, Raster Metafile can accurately represents some plots with translucency and smooth color gradations. The Raster Metafile format was defined by NASA, but is also the format used the Framer program for viewing movies created in Tecplot. The Raster Metafile export options are shown in Figure 22-6.

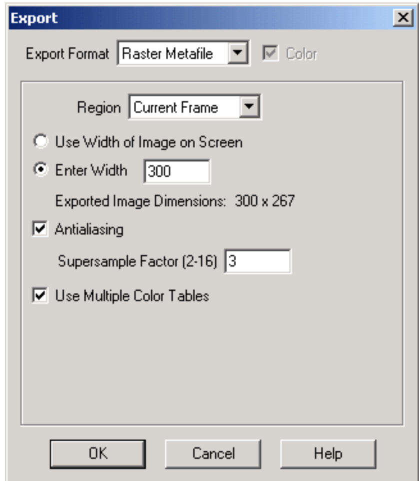


Figure 22-6. Raster Metafile options on the Export dialog.

When you select Raster Metafile in the Export dialog, you have the following options:

- **Region:** Choose to export only the current frame, or the smallest rectangle containing all frames, or everything shown in the workspace.
- **Use Width of Image on Screen:** Select this option to generate an image file the same size as the current plot on the screen. This option is forced on if you use on-screen image rendering in the Display Performance dialog. See Section 30.3, “Display Performance Dialog,” for details.
- **Enter Width:** Select this option to specify a width (in pixels) for the generated image. The greater the width you specify, the longer it will take to export the image and the larger the exported file. However, a larger width increases the quality of your image. This option is not available if you have chosen to use on-screen image rendering.
- **Antialiasing:** Select this option to remove jaggies from the image. See Section 22.15, “Antialiasing Images,” for details.
- **Supersample Factor:** Control the amount of antialiasing used in the image. See Section 22.15, “Antialiasing Images,” for details.
- **Use Multiple Color Tables:** Select this check box to create a Raster Metafile with a separate color table each step in the animation. If this check box is not selected, Tecplot scans all steps in the animation and creates one color table for the entire animation. Multiple color tables can provide better per-step image quality for the animation, but may result in flicker during playback.

For more information on Raster Metafiles and Framer, see Chapter 29, “Animation.”

22.10. Sun Raster (RAS) Export

Sun Raster is an image format, and thus accurately represent plots with translucency and smooth color gradations. Sun Raster files can be created in either of two formats—the standard format, which is not compressed, and a byte-encoded format, which is compressed. The Sun Raster export options are shown in Figure 22-7.

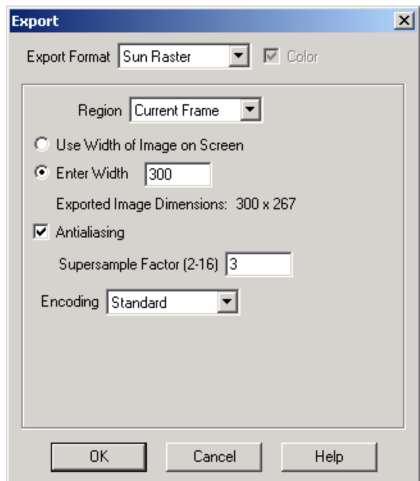


Figure 22-7. Sun Raster options on the Export dialog.

When you select Sun Raster in the Export dialog, you have the following options:

- **Region:** Choose to export only the current frame, or the smallest rectangle containing all frames, or everything shown in the workspace.
- **Use Width of Image on Screen:** Select this option to generate an image file the same size as the current plot on the screen. This option is forced on if you use on-screen image rendering in the Display Performance dialog. See Section 30.3, “Display Performance Dialog,” for details.
- **Enter Width:** Select this option to specify a width (in pixels) for the generated image. The greater the width you specify, the longer it will take to export the image and the larger the exported file. However, a larger width increases the quality of your image. This option is not available if you have chosen to use on-screen image rendering.
- **Antialiasing:** Select this option to remove jaggies from the image. See Section 22.15, “Antialiasing Images,” for details.
- **Supersample Factor:** Control the amount of antialiasing used in the image. See Section 22.15, “Antialiasing Images,” for details.

- **Format:** You may select Standard, which will create an uncompressed file, or Byte-Encoded, which will create a compressed file. You should select Byte-Encoded unless you have a compelling reason to do otherwise.

22.11. Tagged Image File Format (TIFF) Export

TIFF (Tagged Image File Format) is an image format, and thus accurately represent plots with translucency and smooth color gradations. Tecplot generates both color and gray-scale TIFF images. The Export TIFF dialog is shown in Figure 22-8.

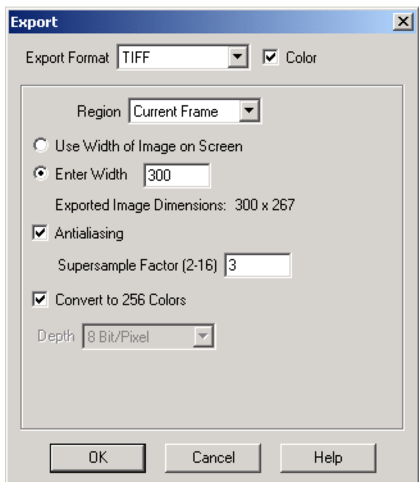


Figure 22-8. TIFF options on the Export dialog.

When you select TIFF in the Export dialog, you have the following options:

- **Color:** Choose between color and gray-scale TIFF output.
- **Region:** Choose to export only the current frame, or the smallest rectangle containing all frames, or everything shown in the workspace.
- **Use Width of Image on Screen:** Select this option to generate an image file the same size as the current plot on the screen. This option is forced on if you use on-screen image rendering in the Display Performance dialog. See Section 30.3, “Display Performance Dialog,” for details.
- **Enter Width:** Select this option to specify a width (in pixels) for the generated image. The greater the width you specify, the longer it will take to export the image and the larger the exported file. However, a larger width increases the quality of your image. This option is not available if you have chosen to use on-screen image rendering.

- **Antialiasing:** Select this option to remove jaggies from the image. See Section 22.15, “Antialiasing Images,” for details.
- **Supersample Factor:** Control the amount of antialiasing used in the image. See Section 22.15 “Antialiasing Images” for details.
- **Convert to 256 Colors:** Select this check box generate an image with only 256 colors (down from a possible 16 million colors). Tecplot selects the best color match. The image will have a greatly reduced file size, but for plots with many colors, the results may be sub-optimal. Using this option with transparency, smooth color gradations, or antialiasing may result in poor image quality.
- **Depth:** For gray-scale images, this specifies the number of shades of gray by how many bits of gray-scale information is used per pixel. The larger the number of bits per pixel, the larger the resulting file. Your options are:
  - **On/Off:** One bit per pixel using an on/off strategy. All background pixels are made white (on), and all foreground pixels, black (off). This setting creates small files and is good for images with lots of background, such as line plots and contour lines.
  - **1 Bit/Pixel:** One bit per pixel using gray scale values of pixels to determine black or white. Those pixels that are more than 50 percent gray are black; the rest are white. This setting creates small files that might be useful for a rough draft or a preview image.
  - **4 Bit/Pixel:** Four bits per pixel resulting in sixteen levels of gray scale. This setting generates fairly small image files with a fair number of gray levels. This setting works well for most preview image purposes.
  - **8 Bit/Pixel:** Eight bits per pixel resulting in 256 levels of gray. This setting is useful for full image representation, but the files generated by this setting can be large.

22.12. Windows Meta File (WMF) Export

WMF is a vector graphics format and thus can be easily resized by the importing application. WMF files can be imported into many applications. As a vector format, WMF cannot accurate represent plots with translucency or smooth color gradations. Selecting WMF from the Export Format drop-down displays WMF options, shown in Figure 22-9.

The following options are available:

- **Region:** Choose to export only the current frame, or the smallest rectangle containing all frames.
- **Color:** Choose between color and gray-scale WMF output.
- **Force Extra Sorting for All 3D Frames:** Selecting this check box causes Tecplot to use extra sorting in all 3-D frames.This overrides the setting in the Advanced 3D dialog. If this check box is not selected, Tecplot chooses sorting algorithms based on the Advanced 3D dialog options for a given frame.

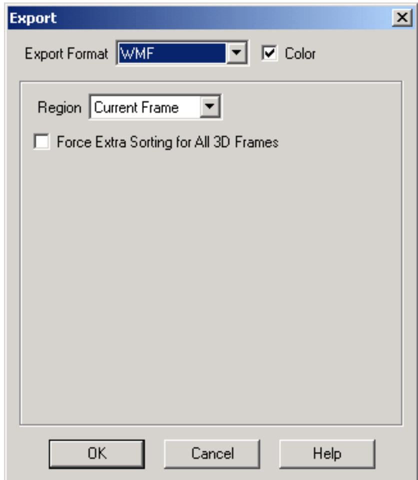


Figure 22-9. WMF options on the Export dialog.

22.13. X-Windows Format (XWD) Export

XWD (X-Windows format) is an image format, and thus accurately represent plots with translucency and smooth color gradations. XWD can be generated on Windows, Macintosh, and UNIX versions of Tecplot.

When you select XWD in the Export dialog, you have the same options as BMP or PNG. See Section 22.3, “Bitmap (BMP) Export,” or Section 22.7, “Portable Network Graphics (PNG) Export,” for details.

22.14. Clipboard Exporting to Other Applications

Tecplot’s Cut, Copy, and Paste commands work only within Tecplot. However, the Copy Plot to Clipboard command is available in the Windows and Macintosh versions of Tecplot. With this option, you can copy and paste Tecplot images directly into other applications such as Microsoft Word, Adobe FrameMaker, and many other applications. The Copy Plot to Clipboard dialog for Windows is shown in Figure 22-10.

The following options are available:

- **Region:** Choose to export only the current frame, or the smallest rectangle containing all frames, or everything shown in the workspace.

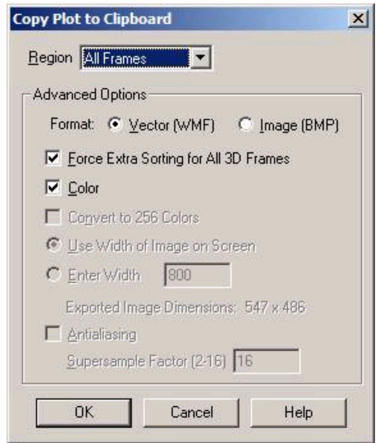


Figure 22-10. The Copy Plot to Clipboard dialog (Windows and Macintosh only).

- **Format:** On Windows, plots may be copied as a vector (WMF) or image (BMP) format. See Section 22.3, "Bitmap (BMP) Export," and Section 22.12, "Windows Meta File (WMF) Export," for a discussion of these formats. On the Macintosh, the plot is copied as a PICT image, and this is the only way to generate a PICT image from within Tecplot.
- **Force Extra Sorting for All 3D Frames:** Selecting this check box causes Tecplot to use extra sorting in all 3-D frames.This overrides the setting in the Advanced 3D dialog. If this check box is not selected, Tecplot chooses sorting algorithms based on the Advanced 3D dialog options for a given frame. This option is only available for WMF, and thus only on Windows.
- **Color:** Choose between color and gray-scale output.
- **Convert to 256 Colors:** On Windows, select this check box generate an image with only 256 colors (down from a possible 16 million colors). Tecplot selects the best color match. The image will take up less memory on your Windows clipboard, but for plots with many colors, the results may be suboptimal. Using this option with transparency, smooth color gradations, or antialiasing may result in poor image quality. On the Macintosh, the image is always generated with only 256 colors because that is all the PICT format supports.
- **Use Width of Image on Screen:** Select this option to generate an image file the same size as the current plot on the screen. This option is forced on if you use on-screen image rendering in the Display Performance dialog. See Section 30.3, "Display Performance Dialog," for details. This option is only available for BMP and PICT.

- **Enter Width:** Select this option to specify a width (in pixels) for the generated image. The greater the width you specify, the longer it will take to export the image and the larger the exported file. However, a larger width increases the quality of your image. This option is not available if you have chosen to use on-screen image rendering. This option is only available for BMP and PICT.
- **Antialiasing:** Select this option to remove jaggies from the image. See Section 22.15, “Antialiasing Images,” for details. This option is only available for BMP and PICT.
- **Supersample Factor:** Control the amount of antialiasing used in the image. See Section 22.15, “Antialiasing Images,” for details. This option is only available for BMP and PICT.

To copy and place a Tecplot image into a document or other art work:

1. In the Edit menu, click Copy Plot to Clipboard. Choose the appropriate options as discussed above, then click OK.
2. In your other software package, place your cursor or select the frame or site where you want to place the Tecplot plot.
3. From the menu of the other software package, execute the Paste command. The plot appears. Some packages push other content out of the way to create a spot for the plot, while others will draw the plot on top of existing content.
4. Resize and reposition the plot as needed within the other software package.

22.15. Antialiasing Images

Antialiasing removes jaggies from text, lines, and edges of image output formats by the process of supersampling. A large intermediate image is rendered and then reduced to the final image size. Each pixel on the final image is created from multiple rendered pixels. The width and height of the intermediate image are the width and height of the final image times some scale factor. This scale factor is the Supersample Factor. You can use values from 2 to 16. Factors greater than 3 are seldom necessary. Large scale factors take a lot more time and memory. Some graphics cards limit the dimensions of rendered images to a maximum of 2048x2048 or 4096x4096 pixels, and thus Tecplot cannot antialias if the intermediate image would be larger than this limit.

Antialiasing uses many colors. Certain image formats are limited to 256 colors, and cannot represent all antialiased images correctly. The image formats limited to 256 color include X-Windows, AVI, Raster Metafile, SunRaster, and any image format with the Convert to 256 Colors. With these formats, the antialiasing works fine for plots with a very limited selection of colors (like a red mesh on a black field). Otherwise, antialiasing with 256 colors is a waste of time and may result in a worse plot.

The antialiasing and 256-color problem can be amplified by animation formats. Both AVI and Raster Metafile support only 256-colors, and need to use them to display multiple frames. For these formats, try a test animation of a few steps with antialiasing on before creating the entire animation.

CHAPTER 23 *Data Spreadsheet*

All ordered and finite-element data can be viewed using Tecplot's data spreadsheet. The data may be modified within the spreadsheet in order to change the plots Tecplot produces.

The spreadsheet only allows the viewing and altering of data loaded into Tecplot. If you want to add zones, variables, or values, you can do so in your original data files before loading into Tecplot, or through the Create Zone or Alter options on the Data menu.

23.1. Data Set Viewing

The spreadsheet displays Tecplot's data differently depending on the type of zone being examined. I-ordered and finite element data sets are displayed with each zone's variable displayed in a column. IJ-ordered data sets are displayed in the spreadsheet with I along the rows and J along the columns. IJK-ordered data sets are displayed one plane at a time: selecting the K-plane displays I along the rows and J along the columns, selecting the J-plane displays I along the rows and K along the columns, and selecting the I-plane displays J along the rows and K along the columns. With IJK-ordered data the slice of interest can be selected by entering a specific index or using the up and down arrows provided.

You view data using the Spreadsheet option under the Data menu. To view your data set:

1. From the Data menu, choose Spreadsheet. The Data Spreadsheet dialog appears, as shown in Figure 23-1.

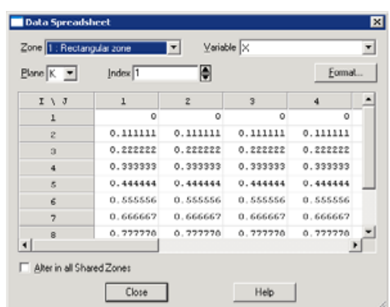


Figure 23-1. The Data Spreadsheet dialog.

2. From the Data Spreadsheet dialog select a desired zone and variable to examine.

3. Use the scroll bars to examine all of your data.
4. To see the values for a specific data point, you may click the mouse on that point in the plot. The spreadsheet will then be positioned to display that point.

You can change the format of data in a spreadsheet without changing the appearance of your plot. To change the data spreadsheet's display format:

1. On the Data Spreadsheet dialog click Format. The Data Format dialog appears.
2. Select a number format from the option menu that best represents the data of interest.
3. If available for the selected number format, specify precision (number of decimal places).
4. Enter a column width that best fits the data of interest.

23.2. Spreadsheet Data Editing

You can change your data set within Tecplot without changing your original data file. You do this by editing values in the cells of the spreadsheet. To modify data:

1. From the Data Spreadsheet dialog select a desired zone and variable to modify.
2. If the variable is shared with another zone or zones, the Alter in all Shared Zones toggle will be enabled. Select this toggle to keep the variable shared as you modify data, propagating changes to the other zones that share the variable.
3. Select the value of interest from the spreadsheet. This will highlight and expand the value to its full precision.
4. To replace the highlighted value simply enter the new value. Anything highlighted is instantly replaced with new digits entered.
5. To slightly modify a highlighted value select the value a second time. This will un-highlight the value and place the edit cursor at the desired position. Make desired modifications to the existing value.
6. To undo a modification of a given cell press Esc. To commit to a modification press the Enter, Tab, or Shift-Tab keys, or click on another cell.

CHAPTER 24 Data Operations

All plots in Tecplot, with the exception of sketches, rely on the data sets attached to each frame. You can modify, create, transform, interpolate, duplicate, and delete the data in the current data set using the Data menu. You can also use the data operation capabilities of Tecplot to create plots of analytical functions. By using Tecplot's linked layout files, or macro capabilities and equation files, you can create complex data operations that can be repeated on different data sets.

Changes to the data set within Tecplot do not affect the original data file(s). You can save the modified data to a data file by selecting Write Data File from the File menu. When you save a layout file, a journal of data operations is saved and those operations are repeated when the layout file is read at a later time. If the data in the file has changed, or the data file is overridden with a different file, the operations are applied to the new data. Alternatively, any data sets that have been modified are also saved to data files (see Section 5.3, "Layout Files, Layout Package Files, Stylesheets," for details).

24.1. Data Journaling

Data is often loaded into Tecplot from external sources, such as data files. On occasion you may modify this data prior to making a final plot. Some (but not all) of the data operations mentioned in this chapter modify data. Tecplot simultaneously "journals" the corresponding instructions. If you then save a layout file, it can reference the original data and include in the layout the instructions necessary to reconstruct the final data used for plotting.

If you perform some type of operation in Tecplot that it is unable to journal then you are prompted to save the data set to a new file when you go to save a layout file. This is necessary for the layout to reproduce exactly what you have in your plot.

24.1.1. Data Journaled Operations

Tecplot journals the following operations:

- Data alteration (except for derivatives).
- Creation of rectangular zones, circular zones, and 1-D zones from within Tecplot.
- Zone duplication.
- Zone deletion.

24.1.2. Data Journaling Example

Suppose you have axi-symmetric data for an airplane and you are only provided with one-half of the data for plotting. The final plot must show the full body of the airplane with flooded contours of pressure on one side of the plane and flooded contours of temperature on the other side.

To create the duplicate image in Tecplot perform the following steps:

1. Go to the Data menu. Select Duplicate from the Create Zone sub-menu to duplicate the original zones.
2. Also on the Data menu, select Specify Equations from the Alter sub-menu. The Specify Equations dialog appears. Set Y = -Y for all of the new zones.

The desired plot styles can now be assigned to the zones on both halves of the plane. The original data after loading and the plot with the duplicated zone are shown in Figure 24-1.

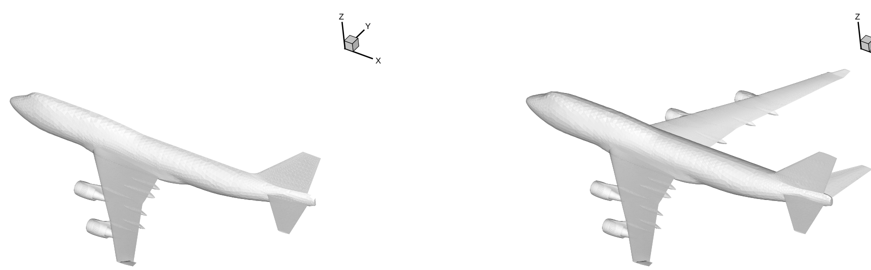


Figure 24-1. Axi-symmetric data before and after duplication.

A layout can now be saved that only needs to reference the original data file containing just the left half of the plane.

24.2. Data Alteration through Equations

Tecplot allows you to alter data in existing zones. You can alter the values of a variable or create new variables as functions of existing variables and index values. Data can be altered simultaneously in one or more zones (or in all zones). For ordered zones, you can also restrict the alteration to specified ranges of indices (I, J, and K).

To modify your data set, follow these steps.

1. From the Data menu, choose Alter.
2. From the Alter menu, choose Specify Equations. The Specify Equations dialog appears, as shown in Figure 24-2.
3. Using the Specify Equations dialog, perform the following steps:

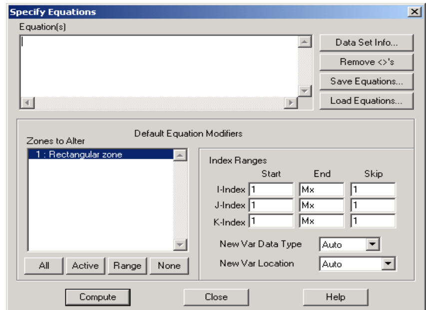


Figure 24-2. The Specify Equations dialog.

- Enter the equations.
- (Optional) Select the set of zones to alter using the appropriate region of the Specify Equations dialog. The default is to alter all zones. You may skip this step if you want to apply the equation to all zones.
- (Optional) Select the index ranges to alter in the selected zones. You may skip this step if you want to apply the equation to all points of the selected zones.
- (Optional) Select the data type of the new variable from the New Var Data Type drop-down. Options are Auto (the default, where Tecplot selects), Single, Double, Long Int, Short Int, Byte, Bit. If appropriate for the variable, memory usage can be reduced by using one of the smaller variable types.
- (Optional) Select the location of the new variable using the New Var Location drop-down. Options are Auto (the default), Node, Cell-Center. Auto is node unless all variables in the equation are located at the cell center.

4. Click Compute to alter the data. This is important. If you do not click Compute, the data values do not change.

These steps are discussed in more detail in the following sections.

24.2.1. Equation Syntax

Tecplot equations have the following form:

```
LValue = F(RValue1, RValue2, RValue3, ...)
```

Where *F()* is a mathematical expression, with some limitations and some extensions as discussed below. *LValue* is a reference to an existing or non-existing variable, and *RValueN* is a reference to a value (such as a constant, variable value, or index value).

There may be any number of spaces within the equation, between operators, and so forth. There cannot, however, be any spaces between the letters of intrinsic-function names nor for variables referred to by name. (See Section 24.2.1.1, “Equation Variables and Values.”)

If the variable on the left-hand side (*LValue*) already exists in the data set of the active frame, the equation is used to modify that variable. If the variable does not already exist, the equation is used to create a new variable as a function of existing variables.

If the equation contains a syntax error, the equation is displayed in an error dialog. The error dialog informs you of the approximate location of the syntax error.

Each equation occupies one line of the Equation(s) text field of the Specify Equations dialog. You can use multiple equations, all of which use as defaults, the zones and index ranges set up at the time you click Compute. Each equation is applied to all specified zones and data points before subsequent equations are computed.

24.2.1.1. Equation Variables and Values. A variable is specified in one of three ways: according to its order in the data file, by its name, or by a letter code.

A variable may be referenced according to its order in the data file. **V1** is the first variable in the data file, **V2** is the second, and so forth.

For example, to set the first variable in the data file equal to the sum of the values of the second and third variables, type:

```
V1 = V2 + V3
```

To create a new variable using this specification, you must specify the number of the next available variable. Assuming there are five variables in the data file, you can create a new variable that is equal to half of the fourth variable as follows:

```
V6 = V4 / 2
```

You must specify the number of the next available variable. If you try to assign the result to a higher numbered variable, Tecplot pops up an error dialog informing you that you have specified an invalid variable number.

A variable may also be referenced by its name. You refer to a variable by its name by enclosing the name within curly braces ("**{**" and "**}**"). For example, to set **V3** equal to the value of the variable named **R/RFR**, you can type:

```
V3 = {R/RFR}
```

Variable names are case insensitive, meaning that any combination of uppercase and lowercase letters matches the variable name. Leading and trailing spaces are also not considered. So the following equations are equivalent:

```
v3 = {R/rfr}
V3 = { r/rfr }
```

Spaces within the variable name are significant, so the following equation is not equivalent to the equations above:

```
V3 = {R / rfr}
```

If two or more variables have the same name, Tecplot uses the first variable. So, if both **V5** and **V9** are named **R/rfr**, **V5** is used.

The curly braces can also be used on the left-hand side of the equation. In this case, if a variable with that name does not exist, a new variable is created with that name. This is useful in equation files (see Section 24.2.1.6, "Equation Examples," for details). For example, the following equation sets a variable called **T/R** to the value of a variable called **T** divided by the value of a variable called **R**. If no variable called **T/R** exists, a new variable is created.

```
{T/R} = {T} / {R}
```

Variables may also be referenced by letter codes. Letter codes can also be used to reference index values. Valid letter codes are:

- **I**: The I-index value at the data point.
- **J**: The J-index value at the data point (1 for finite-element zones).
- **K**: The K-index value at the data point (1 for finite-element zones).
- **X**: The variable assigned to the X-axis (in XY-plots, all active mappings must have the same X variable in order for this variable name to be valid).
- **Y**: The variable assigned to the Y-axis (in XY-plots, all active mappings must have the same Y variable in order for this variable name to be valid).
- **Z**: The variable assigned to the Z-axis (if in 3-D).
- **A**: The variable assigned to the Theta-axis for Polar plots. For this variable to be valid, the plot type must be set to Polar Line. In addition, all active mappings must have the same Theta-variable.
- **R**: The variable assigned to the R-axis for Polar plots. The plot type must be Polar Line, and all active mappings must have the same R-variable for this variable name to be valid.
- **U**: The X-component of vectors (if set).
- **V**: The Y-component of vectors (if set).
- **W**: The Z-component of vectors (if set, and if in 3-D).
- **B**: The value-blanking variable (if set).
- **C**: The contour variable (if set).
- **S**: The scatter-sizing variable (if set).

Letter codes may be used anywhere on the right-hand side of the equation. Do not enclose them in curly braces. Some examples follow:

```
V3 = I + J
V4 = cos(X) * cos(Y) * cos(Z)
{Dist} = sqrt(U*U + V*V + W*W)
{temp} = min(B,1)
```

Those letter codes representing variables (all letter codes except **I**, **J**, and **K**) may be used on the left-hand side of the equation, as well. For example:

```
Z = X*X/(1+Y*Y)
W = 0
S = 1+ABS(S)
```

You receive an error message if the appropriate Tecplot feature is not available. For example, if you try to use **Z** and the current frame is not in the 3D Cartesian plot type, you get an error message. The variables referenced by the letter codes are for the current frame.

You may use auxiliary data containing numerical constants in equations. For example, using reference quantities read from the data file as auxiliary data to compute dimensional variables from non-dimensional variables. The syntax for using auxiliary data in equations is:

```
AUXZONE[nnn]:Name
AUXDATASET:Name
AUXFRAME:Name
```

For instance, a data set auxiliary data constant called **Pref** would be referenced using **AUX-DataSet:Pref**. Equations using this auxiliary data might appear as:

```
{P} = {P_NonDim} *  AUXDataSet:Pref
```

A frame auxiliary data constant called **Mach** would be referenced using **AUXFrame:Mach**. Zone auxiliary data constants may be referenced in one of two ways: **AUXZone:Name** uses the **Name** auxiliary data constant for the zone being operated on while **AUX-Zone[nnn]:Name** uses the **Name** auxiliary data constant for zone number **nnn**. For example, if you are operating on zones 1-3, **AUXZone:Reynolds** references the auxiliary data constant named **Reynolds** attached to zone 1 when operating on zone 1, the auxiliary data constant name **Reynolds** attached to zone 2 when operating on zone 2, and so on. For the same case, **AUXZone[2]:Reynolds** references the auxiliary data constant named **Reynolds** attached to zone 2 for all zones it is operating on.

24.2.1.2. Data Set Information. To view a host of information about the current data set, select Data Set Info on the Data menu. You may also select Specific Equations from the Alter sub-menu of the Data menu. The Data Set Information dialog appears, as shown in Figure 24-3. This dialog has five pages providing information on the zones and variables, general data set information including source files, variable sharing,the data journal, and auxiliary data. Zone/Variable Info is the default page; it provides a list of zones and variables in the data set.

To see more information about a zone, select one of the zone names listed under Zone(s) in the upper left-hand corner. The information about that zone is displayed in the lower left-hand corner. If it is an ordered zone, the I-, J-, and K-values are shown. If it is a finite-element zone, the number of points and elements are shown. While a zone is selected you may edit its name in the Zone Name text field.

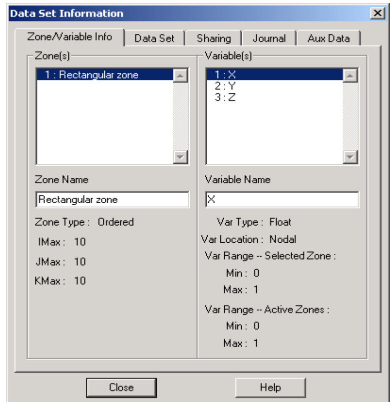


Figure 24-3. The Data Set Information dialog.

For viewing more information about a variable, select one of the variable names listed under Variables in the upper right-hand corner. Information about that variable, such as its type (double, float, long integer, and so forth), its location (nodal or cell-centered), its range in values, and the range in values for all active zones (field plots) or all referenced zones (line plots), is displayed in the lower right-hand corner. The set of all referenced zones is the set of all zones used by the current set of active line mappings. To see a variable's range for a particular zone, select the zone from Zone(s) while the variable is highlighted under Variables. While a variable is selected you may edit its name in the Variable Name text field.

Get additional information about the data set by clicking Data Set. If the data has not been changed since a data file was loaded, the name of that data file appears in the Data File(s) field. The Data Set Title field shows the current name of the data set. While a data set is selected you may edit its name in this field. The lower part of the page shows the Variable Load Mode, which affects your ability to load additional data files to append your current data set. See Chapter 5, "Working with Tecplot Files," for a discussion of loading variables by name or position. The Locked By field informs you if the current data set has been locked by an add-on. Add-ons can lock a data set, preventing the deletion zones or deleting the last frame associated with a data set.

Click Sharing for information about data sharing within a data set. Select the zone and variable of interest and a list of zones that variable is shared with appears. To the right is a list of zones the connectivity is shared with.

Information on data journaling for data sets is available on the Journal page. This shows the journaled macro commands. Select Expand Command to see details of the commands. For a complete description of data journaling, see Section 24.1, "Data Journaling."

Auxiliary data information is available on the Aux Data page. View auxiliary data information by zones, data sets, frames, or by name. When viewing zone information, all auxiliary data name/value pairs for selected zones are shown. Likewise, when viewing data set or frame information, all auxiliary data name/value pairs attached to the data set or selected frame are displayed. Viewing by name allows you to see the auxiliary data of a given name at all locations (frame, data set, all zones). This is useful because the same auxiliary data name may have different values at different locations.

24.2.1.3. Equation Operators and Functions. In an equation, the valid binary operators are as follows:

`+`	Addition.
`-`	Subtraction.
`*`	Multiplication.
`/`	Division.
`**`	Exponentiation.

Binary operators have the following precedence:

`**`	Highest precedence.
`*,/`	
`+,-`	Lowest precedence.

For example, the expression `V1+2*V2` is evaluated as `V1+(2*V2)`, not as `(V1+2)*V2`. Operators are evaluated from left to right within a precedence level (that is, "*" and "/" are evaluated left to right).

The available functions are as follows (except where noted, all take a single argument):

`SIN`	Sine (angle must be specified in radians).
`COS`	Cosine (angle must be specified in radians).
`TAN`	Tangent (angle must be specified in radians).
`ABS`	Absolute value.
`ASIN`	Arcsine (result is given in radians).
`ACOS`	Arccosine (result is given in radians).
`ATAN`	Arctangent (result is given in radians).
`ATAN2(A,B)`	Arctangent of `A/B` (result is given in radians).
`SQRT`	Positive square root.
`LOG, ALOG`	Natural logarithm (base *e*).
`LOG10, ALOG10`	Logarithm base 10.
`EXP`	Exponentiation (base *e*); `EXP(V1)=`*e*`**(V1)`.

MIN(A,B)	Minimum of **A** or **B**.
MAX(A,B)	Maximum of **A** or **B**.
SIGN	Returns -1 if argument is negative, +1 otherwise.
ROUND	Round off to the nearest integer.
TRUNC	Remove fraction part of a value.

LOG and **ALOG** are equivalent functions, as are **LOG10** and **ALOG10**.

First- and second-derivative and difference functions are also available. These are discussed later in this section.

To call an intrinsic function, place its argument within parentheses ("**(**" and "**)**"). For example, to set **V4** to the arctangent of **V1**, use:

```
V4 = ATAN(V1)
```

24.2.1.4. Zone Number Specification. By following a variable reference with square brackets ("**[**" and "**]**"), you can specify a specific zone from which to get the variable value. Some examples follow:

```
V3 = V3 - V3[1]
X = ( X[1] + X[2] + X[3] ) / 3
{TempAdj} = {Temp}[7] - {Adj}
V8 = V1[19] - 2*C[21] + {R/T}[18]
```

The zone number must be a positive integer constant less than or equal to the number of zones. The zone specified must have the same structure (I-, IJ-, or IJK-ordered or finite-element) and dimensions (*IMax*, number of nodes) as the current zone. If you do not specify a zone, the current zone is used.

Zone specification works only on the right-hand side of the equation. All values used on the right-hand side of the equation are the values before the alteration began. To specify zones for the left-hand side, use the Zones region of the Specify Equations dialog.

24.2.1.5. Index Specification. By following a variable reference with parentheses ("**(**" and "**)**"), you can specify indices. Indices can be absolute or an offset from the current index. For example:

```
V2 = ( V2(i+1) + V2(i-1) ) / 2
U = U(i+1,j) - U(i-1,j) + V(i+2,1) + 3*W(i-1)
{NTQ} = {TQ} + {TQ}(i-3,j+7,k-1) - {TQ}(3,j+1,k+8}
S = S(i+1,j) - V3(2) + {RFR/T}(J+2)
```

Index offsets are specified by using the appropriate index "**i**", "**j**" or "**k**" followed by a "**+**" or "**-**" and then an integer constant. Any integer offsets may be used. If the offset moves beyond the end of the zone, the boundary value is used. For example, **V3(i+2)** uses the value

`V3(IMAX)` when *I=IMax-1* and *I=IMax*. `V3(I-2)` uses the value of `V3(1)` when*I=1* or *I=2*.

Absolute indices are specified by using a positive integer constant only. For example, `V3(2)` references `V3` at index `2` regardless of the current `i` index.

Indices must be listed I-index, then J-index, then K-index. The J-index is omitted if the data set is I-ordered; the K-index is omitted if the data set is not IJK-ordered. Indices are not allowed for finite-element data.

Index specification works only on the right-hand side of the equation. If the indices are not specified, the current index values are used. To specify indices for the left-hand side, use the Index Ranges section of the Specify Equations dialog.

Indices may be combined with zone specifications. The zone is listed first, then the index offset. For example:

```
V3 = V3 - V3[1](i+1)
Y = Y[1] - Y[2](1) + Y(1,j+3) + Y
```

24.2.1.6. Equation Examples. In the following equation, `V1` (the first variable defined in the data file) is replaced by two and a half times the existing value of `V1`:

```
V1 = 2.5*V1
```

The following equation sets the value of a variable called `Density` to 205. A new variable is created if a variable called `Density` does not exist.

```
{Density} = 205
```

In the next equation, the values for `Y` (the variable assigned to the Y-axis) are replaced by the negative of the square of the values of `X` (the variable assigned to the X-axis):

```
Y = -X**2
```

The following equation replaces the values of `V3` with the values of `V2` rounded off to the nearest integer. A new variable is created if there are only two variables currently in the data set.

```
V3 = round(V2)
```

In the following equation, the values of the fourth variable in the data set are replaced by the log (base 10) of the values of the third variable.

```
V4 = ALOG10(V3)
```

Suppose that the third variable and fourth variable are the X- and Y-components of velocity and that there are currently a total of five variables. The following examples create a new variable (`V6`) that is the magnitude of the components of velocity.

```
V6 = (V3*V3+V4*V4)**0.5
```

or

```
V6 = sqrt(V3**2+V4**2)
```

The above operation can also be accomplished with the following equation (assuming you have already defined the vector components for the current frame):

```
{Mag} = sqrt(U*U + V*V)
```

The following equation sets the value of a variable named **diff** to the truncated value of a variable named **depth** subtracted from the existing value of **depth**:

```
{diff} = {depth} - trunc({depth})
```

In the next equation, **C** (the contour variable) is set to the absolute value of **S** (the scatter-sizing variable), assuming both **C** and **S** are defined:

```
C = abs(S)
```

In the following example, a new variable is created (assuming that only seven variables initially existed in the data file). The value for **V8** (the new variable) is calculated from a function of the existing variables:

```
V8 = SQRT((V1*V1+V2*V2+V3*V3)/(287.0*V4*V6))
```

The above operation could have been performed in two simpler steps as follows:

```
V8 = V1*V1+V2*V2+V3*V3
V8 = SQRT(V8/(287.0*V4*V6))
```

The following equation replaces any value of a variable called **TIME** that is below **5.0** with **5.0**. In other words, the values of **TIME** are replaced with the maximum of the current value of **TIME** and **5.0**:

```
{TIME} = max({TIME},5)
```

The following equation creates variable **V4** which has values of **X** at points where **X**<0; at other points, it has a value of zero (this does not affect any values of **X**):

```
V4 = min(X,0)
```

Another example using intrinsic functions is shown below:

```
V8 = 55.0*SIN(V3*3.14/180.0) + ALOG(V4**3/(v1+1.0))
```

You can also reference the I-, J-, and K-indices in an equation. For example, if you wanted to cut out a section of a zone using value-blanking, you could create a new variable that is a function of the I- and J-indices (for IJ-ordered data). Then, by using value-blanking, you could remove certain cells where the value of the value-blanking variable was less than or equal to the value-blanking cut-off value.

Here is an example for calculating a value-blanking variable that is zero in a block of cells from *I=10* to 30, and is equal to one in the other cells:

```
V3 = min((max(I,30)-min(I,10)-20),1)
```

The following equation replaces all values of **Y** with the difference between the current value of **Y** and the value of **Y** in zone 1. (If zone 1 is used for the data alteration, the new values of **Y** will be zero throughout that zone.)

```
Y = Y - Y[1]
```

The following equation replaces the values of **V3** (in an IJ-ordered zone) with the average of the values of **V3** at the four adjacent data points:

```
V3 = (V3(i+1,j)+V3(i-1,j)+V3(i,j+1)+V3(i,j-1))/4
```

The following equation sets the values of a variable called **TEMP** to the product of the values of a variable called **T** measured in four places: in zone 1 at two index values before the current data point, in the current zone at an absolute index of three, in zone 4 at the current data point, and in the current zone at the current data point.

```
{TEMP} = {T}[1](i-2) * {T}(3) * {T}[4] * {T}
```

24.2.1.7. Derivative and Difference Functions. Tecplot has a complete set of first- and second-derivative and difference functions. These functions are listed below:

```
ddx     ddy     ddz     dda     ddr
d2dx2   d2dy2   d2dz2
d2dxy   d2dyz   d2dxz
ddi     ddj     ddk     d2daz   d2dar     d2dr2z
d2di2   d2dj2   d2dk2
d2dij   d2djk   d2dik
```

The derivative function **ddx** is used to calculate $\partial/\partial x$; **d2dx2** calculates $\frac{\partial^2}{\partial x^2}$; **d2dxy** calculates $\frac{\partial^2}{\partial x \partial y}$, and so on. The derivatives **dda**, **ddr**, **d2da2**, and so forth, are with respect to the Theta angle (**a**) and radius (**r**) in Polar plots. They may only be used with Polar Line plot types. The difference functions **ddi**, **d2di2**, and so forth, calculate centered differences of their argument with respect to the indices I, J, and K based on the indices of the point. For example:

$$\mathbf{ddi}(V) = \frac{V_{i+1} - V_{i-1}}{2}$$

The difference functions may not be used with finite-element zones. The derivative and difference functions are used just like the intrinsic functions described above. For example:

```
V4 = ddx(V3)
V6 = d2dx2(v5)
```

```
{dC/dx} = ddx(C)
V8 = ddj(X)
{Vt12} = ddy({Vt11}(i+1)) + ddy({Vt11}(i-1))
Z = d2dj2(sin(v5*v6))
V9 = ddj(ddx({R/T}))
C = d2dij(C[1]-C)
{NEWVAR}=ddi(X)+ddj(Y)+ddk(Z)
```

The use of derivative and difference functions is restricted as follows:

- Derivatives and differences for IJK-ordered zones are calculated for the full 3-D volume. The IJK-mode for such zones is not considered.
- If the derivative cannot be defined at every data point in all the selected zones, the operation is not performed for any data point.
- Derivative functions are calculated using the current frame's axis assignments. Be careful if you have multiple frames with different variable assignments for the same data set.
- Derivatives at the boundary of two zones may differ since Tecplot operates on only one zone at a time while generating derivatives.

Boundary values for first-derivative and difference functions (**ddx**, **ddy**, **ddz**, **ddi**, **ddj**, and **ddk**) of ordered zones are evaluated in one of two methods: simple or complex. The default is simple. The following parameter in the configuration file selects the method to use:

```
$!INTERFACE
  DATA {DERIVATIVEBOUNDARY=SIMPLE}
```

Change the parameter **SIMPLE** to **COMPLEX** to use the complex boundary condition.

For simple boundary conditions, the boundary derivative is determined by the one-sided first derivative at the boundary. This is the same as assuming that the first derivative is constant across the boundary (with the second derivative equal to zero).

For complex boundary conditions, the boundary derivative is extrapolated linearly from the derivatives at neighboring interior points. This is the same as assuming that the second derivative is constant across the boundary (with the first derivative varying linearly).

For second-derivatives and differences (**d2dx2**, **d2dy2**, **d2dz2**, **d2dxy**, **d2dyz**, **d2dxz**, **d2di2**, **d2dj2**, **d2dij**, **d2dk2**, **d2djk**, and **d2dik**), these boundary conditions are ignored. The boundary derivative is set equal to the derivative one index in from the boundary. This is the same as assuming that the second derivative is constant across the boundary.

You can create your own derivative boundary conditions by using the index range and the indices options discussed previously.

24.2.1.8. Variable Sharing Between Zones. For zones with the same structure and index ranges, you can set a variable to be shared by specifying that the variable for those zones have the values from one zone. For example, if zones 3 and 4 have the same structure and you compute **V3=V3[3]** for zones 3 and 4, **V3** will be shared. However, it is also possible for zones to lose the sharing of variables by equations altering those variables.

24.2.2. Zone Selection

You may select which zone(s) to alter using the Zones to Alter area of the Specify Equations dialog. By default, all zones are altered. If you are creating a new variable, all zones must be selected since all zones in a data set must have the same number of variables per data point.

24.2.3. Index Range and Skip Selections for Ordered Zones

For ordered zones, you can use the Index Ranges area of the Specify Equations dialog to select which data points to alter (based upon the index values). You can specify an I-Index range, a J-Index range, and/or a K-Index range.

For each index range, you enter the start index value, the end index value, and a skip factor. A skip factor of one applies your equation to every index value; two does every other; three, every third; and so forth. Use the special value **0** or **Mx** to specify the maximum index. You can also use the values **Mx-1** (to specify the index one less than the maximum index), **Mx-2**, and so forth. By default, these ranges are set to the entire range of points (that is, the start index defaults to one, the end index defaults to **Mx**, and the skip factor defaults to one).

The index ranges are applied to all ordered zones that are selected using the Zones to Alter area. Index ranges are ignored for finite-element zones; every data point of a finite-element zone is altered regardless of the settings in the Index Ranges area.

If you are creating a new variable, the new variable's value is set to zero at any index value that is skipped.

24.2.4. Data Type Specification for New Variables

If your equations are creating one or more new variables, you can specify a data type for these new variables using the New Var Data Type drop-down. By default, this is set to Auto, and Tecplot assigns the most appropriate data type to the variables. However, if you want to be sure that your new variables have a particular data type, choose the type from the New Var Data Type drop-down.

The following data types are available:

- **Single:** Four-byte floating point values.
- **Double:** Eight-byte floating point values.
- **Long Int:** Four-byte integer values.
- **Short Int:** Two-byte integer values.
- **Byte:** One-byte integer values (zero to 255).
- **Bit:** Either zero or one.

24.2.5. Data Location Specification for New Variables

If your equations are creating new variables, you can specify the location for these new variables using the New Var Location drop-down. By default this is set to Auto, and Tecplot presumes the variable is located at the node, unless all variables used in the equation are cell-centered. If you want to be sure of the location, specify node or cell-center from the New Var Location drop-down.

24.2.6. Equation Restriction Overriding

The zone and index restrictions specified in the equation dialog can be overridden on an equation by equation basis. To specify restrictions for a single equation add the colon character (**:**) at the end of the equation followed by one or more of the following:

Equation Restrictor	**Comments**
<***Z***=<*set*>>	Restrict the zones.
<***I***=*start*[,*end*[,*skip*]]>	Restrict the I-range.
<***J***=*start*[,*end*[,*skip*]]>	Restrict the J-range.
<***K***=*start*[,*end*[,*skip*]]>	Restrict the K-range.
<***D***=<*datatype*>>	Set the data type for the variable on the left hand side. This only applies if a new variable is being created.

For example, to add one to X in zones 1, 3, 4, and 5:

```
X=X+1:<Z=[1,3-5]>
```

The following example adds one to X for every other I-index. Note that zero represents the maximum index.

```
X=X+1:<I=1,0,2>
```

The next example creates a new variable of type Byte:

```
{NewV}=X-Y:<D=Byte>
```

24.2.7. Data Alteration

After you have set the index ranges and zones that are to be altered and have entered the equations, click Compute to apply the equations to your data. Each equation is applied to each data point in each zone independently of the other points. Each equation is applied to all specified zones and data points before subsequent equations are processed.

If an error occurs during the alteration (because of division by zero, overflow, underflow, and so forth), an error message is displayed, and all of the zones are restored to the state they were in before the bad equation was processed. Thus, if you have three equations A, B, and C, and B contains an error, the final state is the result of processing equation A.

24.2.8. Macros and Equations

Tecplot allows you to put your equations in macros. In fact, we sometimes refer to a macro with just equations as an equation file. An equation in a macro file is specified using the **$!ALTERDATA** macro command. Equation files may also include comment lines, and in fact must start with the comment **#!MC 1000**, like other macro files. If you are performing complex operations on your data, and/or the operations are repeated frequently, equation files can be very helpful.

You can create equation files from scratch using an ASCII text editor, or you can create your equations interactively using the Specify Equations dialog, and then save the resulting equations. The standard file name extension for equation files is **.eqn**.

For example, you might define an equation to compute the magnitude of a 3-D vector. In the Specify Equations dialog, it would have the following form:

```
{Mag} = sqrt(U*U + V*V + W*W)
```

In a macro file, it would have the following form:

```
#!MC 1000
$!ALTERDATA
  EQUATION = "{Mag} = sqrt(U*U + V*V + W*W)"
```

That is, the interactive form of the equation must be enclosed in double quotes and supplied as a value to the **EQUATION** parameter of the **$!ALTERDATA** macro command.

To read an equation file, click Load Equations on the Specify Equations dialog. The Load Equation File dialog appears. Select an equation file that contains a set of equations to apply to the selected zones of your data. The equations in the equation file are added to the list of equations in the dialog. All equations are applied to your data when you click Compute.

Equations in equation files may be calculated somewhat differently depending on whether the computation is done from within the Specify Equations dialog or by running the equation file as a macro. When loaded into the Specify Equations dialog, equations that do not contain zone or index restrictions use the current zone and index restrictions shown in the dialog. When processed as a macro file, such equations apply to all zones and data points. To include zone and index restrictions, you must include them in the equation file as part of the **$!ALTERDATA** command. See Section 24.2.8.2, “Zones To Operate On Specification,” and Section 24.2.8.4, “Range and Skip Factor Specification,” for details.

24.2.8.1. Equation File Comments. As with any macro file, any line with a pound sign (“**#**”) as the first character is considered a comment and ignored. For example:

```
#!MC 1000
#
# calculate magnitudes of vectors
#
```

```
$!ALTERDATA
  EQUATION = "{Mag} = sqrt(U*U + V*V + W*W)"
#
# normalize
#
$!ALTERDATA
  EQUATION = "U = U/{Mag}"
$!ALTERDATA
  EQUATION = "V = V/{Mag}"
$!ALTERDATA
  EQUATION = "W = W/{Mag}"
#
# done
#
```

24.2.8.2. Zones To Operate On Specification. To restrict the zones to operate on, specify the number of the zone or zones to the **$!ALTERDATA** command as a square-bracketed parameter:

```
$!ALTERDATA [zonelist]
  EQUATION = "equation"
```

where *zonelist* is a list of zones or zone ranges separated by a comma (“,”). Zone ranges are separated by a hyphen (“-”). For example, the following macro command restricts the equation to only the second zone:

```
$!ALTERDATA [2]
  EQUATION = "X=X+1"
```

If you do not specify the zones in your equation file, Tecplot defaults to the dialog settings if you read the equations into the Specify Equations dialog. (If you run the equations from a macro file, Tecplot defaults to all zones.) The default setting in the Specify Equations dialog is all zones. To create a new variable, you must have all zones available for data operation.

For example:

```
#!MC 1000
# do the following to default set of zones
$!ALTERDATA
  EQUATION = "Y = Y*2"
# now just do zone 1
$!ALTERDATA [1]
  EQUATION = "Y = 0"
# now just do zones 2 and 3
$!ALTERDATA [2-3]
  EQUATION = "Y = Y[4]"
# now just do zones 4, 5, 6, 7, and 19
$!ALTERDATA [4-7,19]
  EQUATION = "Y = Y(i-1) + Y(i+1)"
```

```
# now do all zones to create a new variable
$!ALTERDATA
  EQUATION = "{SomeNewVariable} = 0"
```

24.2.8.3. Zone Number Specification for Operands. By following a variable reference with brackets "`[`" and "`]`" you may designate a specific zone from which to get the variable value. For example:

```
V3 = V3 -V3[1]
X = ( X[1] + X[2] + X[3] ) / 3
{TempAdj} = {Temp}[7] - {Adj}
V7 = V1[19] - 2*C[21] + {R/T}[18]
```

The zone number must be a positive integer constant less than or equal to the number of zones. The zone designated must have the same structure (finite-element, I-, IJ-, or IJK-ordered) and dimensions (number of nodes and so forth). If you do not designate a zone, the current zone will be used.

Specifying a zone only works on the right-hand side of an equation. All values used on the right-hand side of the equation are the values from before the alteration began.

24.2.8.4. Range and Skip Factor Specification. You can restrict index ranges from within your Equation file by specifying the following parameters to the **`$!ALTERDATA`** macro command:

```
IRANGE = min, max, skip
JRANGE = min, max, skip
KRANGE = min, max, skip
```

where *min* is the minimum value for that index; *max*, the maximum; and *skip*, the skip factor. Index ranges are described in Section 24.2.3, "Index Range and Skip Selections for Ordered Zones." If these commands are not used, the default index ranges use the currently selected indices (if the equations are executed from a macro, all points are used). Index ranges are effective only for ordered zones.

For example:

```
#!MC 1000
#
# use default index range
#
$!ALTERDATA
  EQUATION = "{K/10} = round(K/10)"
#
# calculate d2dxy(C) everywhere
#
$!ALTERDATA
  IRange = 1,0,1
```

```
  JRange = 1,0,1
  KRange = 1,0,1
  EQUATION = "V5 = d2dxy(C)"
#
# set corner values to zero
#
$!ALTERDATA
 IRange = 1,0,0
 JRange = 1,0,0
 KRange = 1,0,0
 EQUATION = "V5 = 0"
```

24.2.8.5. New Variable Data Type Specification. You can specify the data type for new variables that are created in equations in the **$!ALTERDATA** command by including the following parameter:

DATATYPE = <datatype**>**

where *datatype* can be one of **SINGLE**, **DOUBLE**, **LONGINT**, **SHORTINT**, **BYTE**, or **BIT**.

For example, to create a new variable and set its data type to **BYTE**:

```
$!ALTERDATA
  EQUATION = "{NewV} = X-Y"
  DATATYPE = BYTE
```

24.2.8.6. New Variable Value Location Specification. You can specify the value location for new variables that are created in equations in the **$!ALTERDATA** command by including the following parameter:

VALUELOCATION = <valuelocation**>**

where *datatype* can be one of **SINGLE**, **DOUBLE**, **LONGINT**, **SHORTINT**, **BYTE**, or **BIT**.

For example, to create a new variable and set its data type to **BYTE**:

```
$!ALTERDATA
  EQUATION = "{NewCC} = X-Y"
  VALUELOCATION = CELLCENTERED
```

24.2.8.7. Variable Naming in an Equation File. To name a variable when creating it in an equation file, simply provide the name within curly braces (**{}**) on the left hand side of the equation, as in the example below:

```
#!MC 1000
$!ALTERDATA
  EQUATION = "{myvar} = V4 / V5"
```

To rename a variable using a macro, use the macro command **$!RENAMEDATASETVAR**, as in the example below:

```
#!MC 1000
$!RENAMEDATASETVAR
  VAR = 6
  NAME = "myvar"
```

The macro listed above may be played back using Play option of the Macro command under the File menu. It cannot, however, be loaded in as an equation in the Specify Equations dialog.

24.2.8.8. Equation File Examples. If you want to cut out a section of a zone using value-blanking, you can create a new variable that is a function of the I- and J-indices (for IJ-ordered data). Then, by using value-blanking, you can remove certain cells where the value of the value-blanking variable is less than or equal to the value-blanking cut-off value. Below is an example equation file for calculating a value-blanking variable that is zero in a block of cells from *I=10* to 28 and from *J=5* to 16 and is equal to one in the other cells.

```
#!MC 1000
## create a mask for the cells in the I-direction
$!ALTERDATA
  EQUATION =  "{MASK_I} = max(I,28) - min(I,10) - 18"
$!ALTERDATA
  EQUATION = "{MASK_I} = min({MASK_I},1)"
## create a mask for the J-direction
$!ALTERDATA
  EQUATION = "{MASK_J} = max(J,16) - min(J,5) - 11"
$!ALTERDATA
  EQUATION = "{MASK_J} = min({MASK_J},1)"
## create the value-blanking variable that is the
## intersection ("and") of the above conditions
$!ALTERDATA
  EQUATION = "{VBLANK} = {MASK_I} + {MASK_J}"
## create a second value-blanking variable that is
## the union ("or") of the above conditions
$!ALTERDATA
  EQUATION = "{OR_VBLANK} = {MASK_I} * {MASK_J}"
```

Another example of an equation file is shown below. In this example, **V8** is a new variable, so all zones must be selected when **V8** is created.

```
#!MC 1000
## The first three equations are applied to all zones
$!ALTERDATA
  EQUATION = "V4 = SQRT(V4) / 460.0"
$!ALTERDATA
  EQUATION = "V4 = (V4*V5 + V4*V6)/2"
$!ALTERDATA
  EQUATION = "V5 = SIN(V4/V3) * EXP(-0.53*V2/V1)"
## Creating New Variable, Operate on all zones
```

```
$!ALTERDATA [1,|NUMZONES|]
  EQUATION = "{CRITT} = 0.5396E-3 * V4**2"
## Operate on Zone 5 only
$!ALTERDATA [5]
  EQUATION = "V1 = V1 / 2"
$!ALTERDATA [5]
  EQUATION = "V2 = V2 / 2"
## Operate On Zones 6,7,8,9, and 23 Only
$!ALTERDATA [6-9,23]
  EQUATION = "V1 = V1 * 4.3"
$!ALTERDATA [6-9,23]
  EQUATION = "V2 = V2 * 4.3"
## Operate On All Zones, But Limit Index Ranges
## To Every Fifth Point in the I- or J-Direction
$!ALTERDATA
  IRange = 1,0,5
  JRange = 1,0,5
  EQUATION = "V5 = EXP(V3(i-2)+V5(i+4))"
##
##
```

24.3. Coordinate Transformation

All 2- and 3-D Tecplot plots use a rectangular coordinate system with axes X and Y. Some 2- and 3-D data may be represented in polar coordinate form, in which each point is represented by the radius *r* and by an angle θ (in radians). For line plots you should plot this data using the Polar Line plot type. For field plots, such a data set initially displays as a rectangle in Tecplot.

In 3-D, transformation, Tecplot assumes the current X-variable represents the radius ρ, the current Y-variable the angle θ (in radians), and the current Z-variable the angle ψ. Figure 24-4 shows ρ, θ, and ψ in the spherical coordinate system.

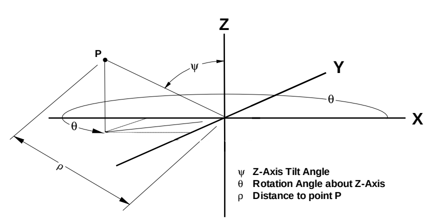


Figure 24-4. Three-dimensional angles of rotation.

Use the Transform Coordinates dialog to transform data sets in polar coordinates to a rectangular coordinate system. When you use this dialog, Tecplot assumes the current X-variable represents the radius **r**, and the current Y-variable represents the angle θ.These variables are listed in the upper portion of the Transform Coordinates dialog, shown in Figure 24-5.

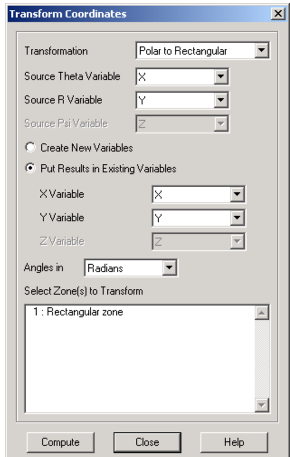


Figure 24-5. The Transform Coordinates dialog.

The following options are available:

- **Transformation:** Select the type of transformation for changing all points in one or more zones from one coordinate system to another. Options include Polar to Rectangular, Spherical to Rectangular, Rectangular to Polar, Rectangular to Spherical.
- **Theta/X Variable:** Selects variable to use for Theta/X in transformation equations.
- **R/Y Variable:** Selects variable to use for R/Y in transformation equations.
- **Psi/Z Variable:** Selects variable to use for Psi/Z in transformation equations.
- **Create New Variables:** When checked, new variables are created to hold the transformed values. Tecplot comes up with names so the data set integrity is maintained (no two variables with same names).
- **Put Results in Existing Variables:** When checked, results are put into variables in the current data set.
- **X/Theta Variable:** Selects variable to receive X/Theta values from transformation.
- **Y/R Variable:** Selects variable to receive Y/R values from transformation.
- **Z/Psi Variable:** Selects variable to receive Z/Psi values from transformation.
- **Angles in:** Selects whether to calculate using values in Theta and Psi variable as radians or degrees.
- **Select Zones to Transform:** Selects zones to alter (also displays that all are transformed).

To transform your data in polar coordinates to rectangular coordinates:

1. From the Data menu, choose Alter, then choose Transform Coordinates. The Transform Coordinates dialog appears as shown in
2. Select the type of transformation desired. Options include polar or spherical to rectangular, rectangular to polar or spherical. Verify that the X- and Y-variables shown represent *r* and θ, respectively, in your polar coordinate data. The data must be in radians.
3. Select the zones for the transformation, then click Compute.

24.4. Two-Dimensional Data Rotation

Use the 2D Rotate dialog to rotate 2-D field data about a user specified XY-origin. Unlike 3-D rotation, which does not alter the data but merely the user's view of the data, 2-D rotation actually modifies the data.

To rotate data in 2-D:

1. From the Data menu, choose Alter, then choose 2D Rotate. The 2D Rotate dialog appears, as shown in Figure 24-6.

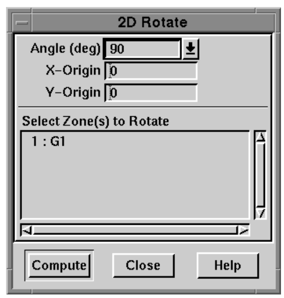


Figure 24-6. The 2D Rotate dialog.

2. Specify the angle of rotation, in degrees, using the Angle (deg) drop-down.
3. Enter the X- and Y-coordinates of the origin of rotation (that is, the point around which the data rotate).
4. Select the zones to rotate.
5. Click Compute.

24.5. Alter Shift Cell Centered

Use the Shift Cell-Centered Data dialog to recalculate the values of variables at your grid points under the assumption that the original data represented values observed at the centers of the grid cells.

For example, suppose you have grid points at X=1,2, ..., and your data is gathered at the cell centers, that is, at X=1.5, 2.5, ..., Y=1.5, 2.5, and so forth. When you create your Tecplot data file, simply map the cell-centered observations with the lowest indexed corner of the cell, so that, for example, the observations at (X,Y)=(1.5, 1.5) are identified with the grid point

(X,Y)=(1, 1). Then use the dialog Shift Cell-Centered Data to interpolate these cell-centered values onto the grid points. The final result is a node-centered data set with interpolated observations at each node.

The following options are available:

Zone(s): Select zones to be shifted. The default is no zones selected.

To select a single zone, click on it. To select a range of zones:

1. Click on the first or last zone in the range.
2. Shift-click on the last or first zone in the range.

To select an arbitrary set of zones:

1. Click on the first desired zone.
2. Ctrl-click on subsequent zones until all desired zones are selected.

Variable(s): Select variables for shifting the data. The default is no variables selected.

To select a single variable, click on it. To select a range of variables:

1. Click on the first or last variable in the range.
2. Shift-click on the last or first variable in the range.

To select an arbitrary set of variables:

1. Click on the first desired variable.
2. Ctrl-click on subsequent variables until all desired variables are selected.
3. To perform the data shift, click **Compute**.

24.6. Zone Creation

You create zones in Tecplot as part of any number of operations: zone duplication, iso-surface extraction, streamtrace zone generation, 3-D data slicing, finite-element boundary extraction, and triangulation. Most of these tasks are discussed in the chapter related to those topics—streamtrace zone generation, for example, is discussed in Chapter 13, “Streamtraces.” This section concentrates on zone creation that is also essentially data creation—creating new rectangular, circular, or linear zones. We also describe briefly the data duplication aspects of zone creation—zone duplication, mirror zones, and finite-element zone creation from polylines, and creation of 1-D zone by direct entry of values.

24.6.1. One-Dimensional Line Zone Creation

A 1D-line zone is an I-ordered set of points along a line. You can create a 1-D line zone as the first step in plotting an analytic function. Typically, you create the 1-D line zone, then use the Specify Equations dialog to modify the Y-variable, then plot the result.

To create the 1-D line zone:

1. From the Data menu, choose Create Zone, then choose 1-D Line. The Create 1-D Line Zone dialog appears.
2. In the text field labeled Number of Points, enter the number of points you want to have in the zone.
3. In the text fields labeled XMin and XMax, enter the range of X-values for the defined points.
4. Click Create. A dialog informs you that the zone was created. Tecplot uniformly distributes the points along the X-axis between XMin and XMax. Y, and any other variables, are set to zero.

24.6.2. Rectangular Zone Creation

Creating a rectangular zone is often the first step in interpolating irregular data into an ordered grid (see Section 20.4, “Three-Dimensional Volume Irregular Data Interpolation,”) or in plotting analytic functions as described in Section 20.8.2, “Analytic Iso-Surface Plots.”

Tecplot allows you to create a new ordered rectangular zone with the dimensions in the I-, J- and K-directions you specify. This is done either with the Create Rectangular Zone tool or the Create Rectangular Zone dialog. The zone that you create has the same number of variables as other zones in the data set, if any. Otherwise, it has just two or three variables, depending on the value specified for the K-dimension. For the 3D Cartesian plot type, you can specify three dimensions when creating a zone. For the 2D Cartesian plot type, you can specify two dimensions. If you have a data set attached to your plot, Create Rectangular Zone does not work in the XY Line or Sketch plot types.

If you have no current data set, Tecplot creates one with two or three variables, depending on the specified K-dimension. If you specify $K=1$ (the default), the data set is created as IJ-ordered, and has just two variables. If you specify $K>1$, the data set is created as IJK-ordered, and has three variables.

24.6.2.1. New Rectangular Zones in 2D Cartesian Plot Type. In 2D Cartesian plot type, you may create IJ-ordered zones. You may interactively draw the boundary of the new zone, or use the Create Rectangular Zone dialog to enter the X- and Y-coordinates of the lower-left and upper-right corners of the new zone.

To create a rectangular zone interactively:

1. From the sidebar, choose the Create Rectangular Zone tool. The tool is only available if a data set is attached to the current frame. Use the Create Rectangular Zone dialog to create a zone in an empty frame.
2. In the frame in which you want to create the zone, press down and hold the left mouse button to specify one corner of the zone.

3. Drag until the boundary of the zone is as desired, then release. The Create Rectangular Zone dialog appears (see Figure 24-7) with the X and Y extents for the rectangular zone already filled in. You can adjust the I- and J-dimensions and the X- and Y-coordinates.
4. Click Create to create the zone.

To create a rectangular zone using the Create Rectangular Zone dialog:

1. From the Data menu, choose Create Zone, then Rectangular. The Create Rectangular Zone dialog appears, as in Figure 24-7.

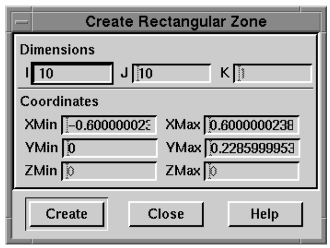


Figure 24-7. The Create Rectangular Zone dialog.

2. In the Create Rectangular Zone dialog, you enter the number of data points in the I-direction and the number of data points in the J-direction. Also enter the X- and Y-coordinates of the two corners.
3. Click Create.

Tecplot uniformly distributes the data points in the I- and J-directions. In this situation, the I-direction is along the X-axis, and the J-direction is along the Y-axis as shown in Figure 24-8. All other variables (those not assigned to the X- and Y-axes) are set to zero.

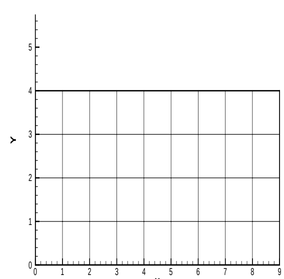


Figure 24-8. A rectangular zone.

After creating a rectangular zone, you can modify the X- and Y-coordinates and/or the other field variables using the tools under the Alter sub-menu of the Data menu.

24.6.2.2. New Rectangular Zones for the 3D Cartesian Plot Type. In 3D Cartesian plot type, you can create an I-ordered (linear) zone, an IJ-ordered (planar) zone or an IJK-ordered (3-D volume) zone. In 3D Cartesian, the Create Rectangular Zone mouse mode is disabled; you do not have the option to interactively draw the new zone.

To create a rectangular zone for the 3D Cartesian plot type:

1. From the Data menu, choose Create Zone, then choose Rectangular. The Create Rectangular Zone dialog appears.
2. Enter the dimensions (that is, number of data points) in the I-direction, the J-direction, and the K-direction in the text fields grouped under the label Dimensions.

   To create an I-ordered zone, enter one for both the J- and K-dimensions.

   To create an IJ-ordered zone, enter one for the K-dimension. *Z=ZMin* throughout the created zone.

   To create an IJK-ordered zone, enter a K-dimension greater than one.
3. Enter the minimum and maximum coordinate values for X, Y, and Z in the text fields grouped under the label Coordinates.
4. Click Create to create the new zone.

The values for X, Y, and Z are calculated at each data point in the new zone. Tecplot distributes the data points uniformly in the I-, J-, and K-directions. In this situation, the I-direction is along the X-axis, the J-direction along the Y-axis, and the K-direction along the Z-axis. All other variables are set to zero. If you plot the new zone, you will see a rectangular mesh with uniform spacing in the X-, Y-, and Z-directions. The mesh lines are straight and parallel to the axes.

Using Alter option under the Data menu, you can modify the X-, Y-, and Z-coordinates, and the values of the other variables as well, by using equations or Equation files. See Section 24.2, "Data Alteration through Equations."

24.6.3. Circular or Cylindrical Zone Creation

Tecplot allows you to create a new ordered circular or cylindrical zone with the dimensions in the I-, J-, and K-directions you specify. The I-dimension determines the number of points on each radius of the zones. The J-dimension determines the number of points around the circumference. The K-dimension determines the number of layers in the zone, creating a cylinder. You create a circular or cylindrical zone with the Create Circular Zone dialog, or, in 2D Cartesian, with the Create Circular Zone tool. The zone that you create has the same number of variables as other zones in the data set, if any. In the 3D Cartesian, you can specify three dimensions when creating a data set. In 2D Cartesian, you can specify two dimensions. In the Sketch and XY Line plot types, the create circular zone options are not available, except in Sketch when no data set is attached to the current frame.

If you have no current data set, Tecplot creates one with two or three variables, depending on the K-dimension. If you specify *K=1* (the default), the data set is created as IJ-ordered, and has just two variables. If you specify *K>1*, the data set is created as IJK-ordered, and has three variables.

24.6.3.1. New Circular Zones for the 2D Cartesian Plot Type. In the 2D Cartesian plot type, you may create circular IJ-ordered zones. You may interactively draw the boundary of the new zone, or use the Create Circular Zone dialog to enter the radius and X- and Y-coordinates of the center of the new zone.

To create a circular zone interactively:

1. From the sidebar, choose the Create Circular Zone tool. The tool is only available if a data set is attached to the current frame. Use the Create Circular Zone dialog to create a zone in an empty frame.
2. In the frame in which you want to create the zone, press down and hold the left mouse button to specify the center of the zone.
3. Drag the mouse until the boundary of the zone is as desired, then release. The Create Circular Zone dialog appears (see Figure 24-9) with the current origin and radius filled in for you. You may adjust the dimensions and coordinates as necessary.
4. Click Create to create the new zone.

To create a circular zone using the Create Circular Zone dialog:

1. From the Data menu, choose Create Zone, then Circular. The Create Circular Zone dialog appears, as in Figure 24-9.

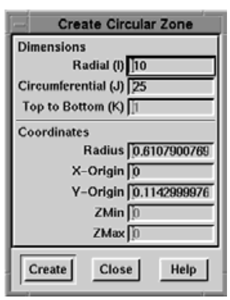


Figure 24-9. The Create Circular Zone dialog.

2. In the Create Circular Zone dialog, you enter the number of data points in the I-direction (radial) and the number of data points in the J-direction (circumferential). You must supply a J-dimension of at least 2.
3. Click Create to create the new zone.

Tecplot creates a zone in which I-circles are connected by J-radial lines, as shown in Figure 24-10. All other variables (those not assigned to the X- and Y-axes) are set to zero.

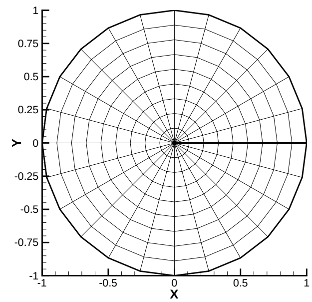


Figure 24-10. A circular zone.

After creating a circular zone, you can modify the X- and Y-coordinates and/or the other field variables using the tools under the Alter sub-menu of the Data menu.

24.6.3.2. New Cylindrical Zones for the 3D Cartesian Plot Type. In 3D Cartesian, you can create an IJ-ordered (planar) zone or an IJK-ordered (3-D volume) zone. In 3-D, the Create Circular Zone mouse mode is disabled, so that you do not have the option to interactively draw the new zone.

To create a cylindrical zone in 3-D Cartesian:

1. From the Data menu, choose Create Zone, then choose Circular. The Create Circular Zone dialog appears.
2. Enter the dimensions (that is, number of data points) in the I-direction (radial), the J-direction (circumferential), and the K-direction (layers) in the text fields grouped under the label Dimensions.

   To create an IJ-ordered zone, enter one for the K-dimension. *Z=ZMin* throughout the created zone.

   To create an IJK-ordered zone, enter a K-dimension greater than one.
3. Enter the radius, the X- and Y-coordinate values for the zone center, and the minimum and maximum Z-coordinates in the text fields grouped under the label Coordinates.
4. Click Create to create the new zone.

The values for X, Y, and Z are calculated at each data point in the new zone. If $K>1$, Tecplot creates a K-layered cylindrical zone having I-circles connected by J-radial planes as shown in Figure 24-11. All other variables are set to zero.

Using the Alter option from the Data menu, you can modify the X-, Y-, and Z-coordinates, and the values of the other variables as well, by using equations or equation files. See Section 24.2, “Data Alteration through Equations.”

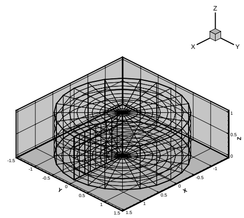


Figure 24-11. A 3-D circular zone.

24.6.4. XY Data Entry

If you have a fairly small number of XY-pairs, you can enter them directly into Tecplot to create a zone with XY-values.

To create an I-ordered zone for XY-plots:

1. From the Data menu, choose Create Zone, then choose Enter Values. The Enter XY-Values to Create a Zone dialog appears, as shown in Figure 24-12.

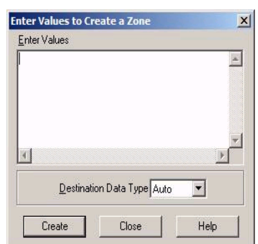


Figure 24-12. The Enter XY-Values to Create a Zone dialog.

2. In the text box labeled Enter XY Values, enter X- and Y-value pairs, one per line; first X, then one or more spaces, then Y.
3. If you would like to specify a data type for the data (long or short integer, float, double, byte, bit), select the desired data type from the drop-down labeled Destination Data Type.
4. Click Create to create the zone.

24.6.5. Data Extraction from an Existing Zone

You may create new zones by extracting (or interpolating) data from existing zones in a number of ways. Derived objects, such as contour lines, FE-boundaries, iso-surfaces, current slices, or streamtraces may be extracted to be independent zones. You may also extract data using a specified slice plane, discrete points, points from a polyline, or points from a geometry. Finally, you may extract a sub-zone of an ordered zone.

The procedures for extracting derived objects are discussed in the chapters related to those objects. For details see Chapter 10, “Contour Plots,” Section 19.4, “Boundary Extraction of Finite-Element Zones,” Section 20.6.3, “Iso-Surface Extraction,” and Section 12.7, “Streamtrace Extraction as Zones.” Extracting slices, both derived objects and arbitrarily defined, is described in Section 20.7.2, “Slice Extraction.”

24.6.5.1. Sub-Zone Extraction. You can only extract a sub-zone out of an existing I-, IJ-, or IJK-ordered zone. You cannot create a sub-zone out of a finite-element zone.

To create a sub-zone of an existing zone:

1. From the Data menu, choose Extract, then Sub-Zone. The Extract SubZone dialog appears, as shown in Figure 24-13.

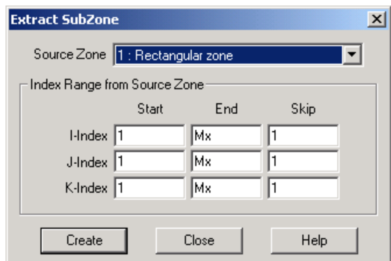


Figure 24-13. The Extract SubZone dialog.

2. Select the source zone from the Source Zone drop-down.
3. Specify the desired sub-zone as a range of I-, J-, and K-indices. For each of I-Index, J-Index, and K-Index (if applicable), specify a start index, an end index, and a skip. You may use the special value **`0`** or **`Mx`** to indicate the maximum of that index, and the values **`Mx-1`** to represent one index less than the maximum, **`Mx-2`** for two less than the maximum, and so forth.
4. Click Create to create the sub-zones. Each sub-zone is given the name “SubZone.”

24.6.6. Data Point Extraction

Another method for creating an I-ordered zone is to extract data points from the current data set using any of three methods:

- Choosing a discrete set of points with the mouse.
- Drawing a polyline with the mouse from which points are extracted at regular intervals.
- Selecting an existing polyline geometry from which points are extracted, either at regular intervals or at the points which define the geometry.

The difference between the second and third methods is that in the second method, the polyline used to extract the points is not a geometry and is not part of your plot—it is simply a transient structure used to extract the points. The points which define the polyline are not treated specially in any way. When you extract points from a polyline geometry, however, you can choose to use the points that define the polyline as the extracted points.

Note: To extract points from a geometry or polyline, it must lie within the boundaries of a zone with connectivity.

24.6.6.1. Discrete Point Extraction. To extract a discrete set of points with the mouse:

1. From the Data menu, choose Extract, then choose Discrete Points.
2. Click at each location from which you want to extract a point.
3. Double-click on the last data point or press Esc to end. The Extract Data Points dialog appears; use it to specify how to save the data.

24.6.6.2. Point Extraction from a Polyline. To extract points from a polyline:

1. From the Data menu, choose Extract, then choose Points from Polyline.
2. Click at the desired beginning of the line, and at all desired breakpoints.
3. Double-click on the last data point or press Esc to end. The Extract Data Points dialog appears; use it to specify how many points to extract and how to save the data.

24.6.6.3. Point Extraction from a Geometry. To extract points from a polyline geometry:

1. In the workspace, select the polyline geometry from which you want to extract data points.
2. From the Data menu, choose Extract, then choose Points from Geometry. The Extract Data Points dialog appears; use it to specify how many points to extract and how to save the data.

24.6.6.4. Data Point Extraction Controls. Use the Extract Data Points dialog to control how data points are extracted. Use the following controls:

- **Extract Data to:**
  - **File:** Select this check box if you want the data points extracted to an ASCII Tecplot data file. If this check box is selected, the Extract Data Points to File dialog appears when you click Extract. Use this dialog to specify a file name for the extracted data file.
  - **Zone:** Select this check box if you want the data points extracted to a zone in the current data set.
- **Include distance variable:** Select this check box if you want the extracted data file to contain an additional variable, `DISTANCE`, which contains the accumulated distance from the first point.

- **Number of points to extract:** Enter the number of points to extract. This field is sensitive only if you are extracting data points from a polyline or geometry. It is insensitive if you are extracting discrete points. If you are extracting from a geometry, you must also select the check box labeled “Extract regular points along a geometry.”
- **Extract regular points along geometry:** Select this check box if you want to extract the specified number of points distributed uniformly along the geometry. This check box is sensitive only if you are extracting points from a geometry.
- **Extract only points which define geometry:** Select this check box if you want to extract only the endpoints of the segments in the geometry. This check box is sensitive only if you are extracting points from a geometry.

After specifying any desired options, click Extract.

24.6.7. Zone Duplication

Tecplot can create a new zone by duplicating all or part of an existing zone. This is useful for creating projections, mirror images, and sub-zones of existing zones.

24.6.7.1. Full Zone Duplication. To create a full duplicate of one or more existing zones:

1. From the Data menu, choose Create Zone, then Duplicate. The Create Duplicate Zone dialog appears, as shown in Figure 24-14.

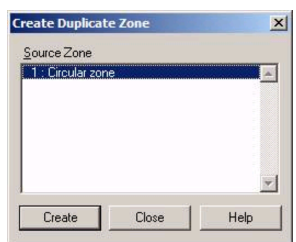


Figure 24-14. The Create Duplicate Zone dialog.

2. Select the source zone or zones from the scrolled list labeled Source Zone.
3. Click Create to create the duplicate zone or zones. Each duplicate zone has the same name as the zone of which it is a copy.

Duplicate zones can be used as the first step to create new zones which are projections of existing 3-D surface zones onto a plane. To do this, follow these steps:

1. Create a plot of a 3-D surface zone.
2. Create a duplicate of the surface zone using the procedures in Section 24.6.7.1, “Full Zone Duplication.”
3. From the Data menu, choose Alter, then choose Specify Equations.

4. Make the new zone a plane by setting the Z-coordinate to a constant value (-20,000 in this example) by entering the following equation in the Specify Equations dialog:

   `Z = -20000`

5. Select the new zone in the Zones to Alter region of the Specify Equations dialog, deselecting other zones as necessary.
6. Click Compute to complete modifying the new zone.

Figure 24-15 shows an example of a projection of a 3-D surface zone.

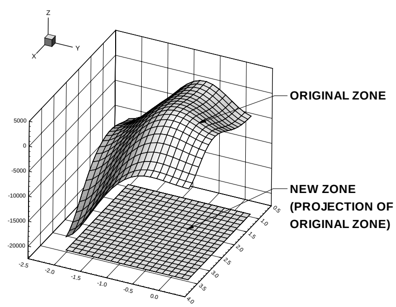


Figure 24-15. Projection of a 3-D surface.

24.6.7.2. Mirror Zone Creation. Tecplot makes it very simple to create a duplicate zone that is the mirror image of an existing zone if the desired mirror axis or mirror plane is one of the standard axes (2-D) or the plane determined by any two axes (3-D). To create a mirror image of one or more existing zones using one of these standard mirrors:

1. From the Data menu, choose Create Zone, then Mirror. The Create Mirror Zone dialog appears, as shown in Figure 24-16.

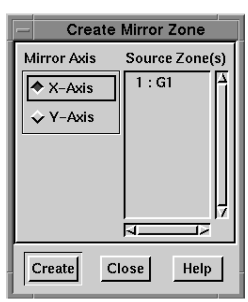


Figure 24-16. The Create Mirror Zone dialog for a 2-D image.

2. Select the source zone or zones from the scrolled list labeled Source Zone(s).

3. Specify the axis (2-D) or axis plane (3-D) to mirror about.
4. Click Create to create the mirror zone or zones. Each mirror zone has a name of the form “Mirror of zone sourcezone,” where sourcezone is the number of the zone from which the mirrored zone was created.

For example, consider the case where your input file describes a 90 degree wedge which occupies the positive X-positive Y-quadrant. You want to create the complete circle based on your inputs. To do this, perform the following steps:

1. Mirror the original zone about the X-axis.
2. Mirror both the original zone and the mirrored zone created in Step 1 about the Y-axis.

It is also possible to create a mirror zone that is mirrored about a different axis or plane. For example, suppose you wanted to mirror a 2-D zone about the line **`X=5`**. You can do this as follows:

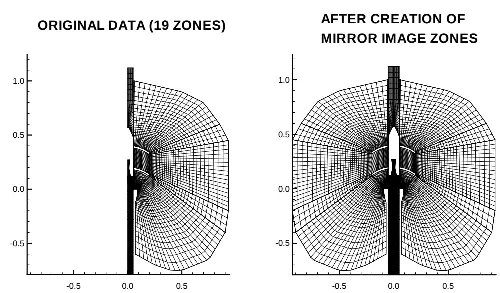


Figure 24-17. Creating a mirrored zone.

1. Create a mirror zone about the Y-axis as described above
2. From the Data menu, choose Alter, then choose Specify Equations.
3. Enter the following equation in the Equations text field:
   `X = 10 - X`
4. Specify the new mirror zone as the only zone to alter.
5. Click Compute to complete the process.

Figure 24-17 shows an example of creating mirror image zones.

24.6.8. Finite-Element Surface Zone Creation from I-Ordered Zones

To create a finite-element surface zone from two or more I-ordered zones go to the Data menu’s Create Zone sub-menu. Select the Polylines option. The Create Zone from Polylines dialog appears, as shown in Figure 24-18.

Only I-Ordered zones (the polylines) will appear in the list. Select two or more zones.

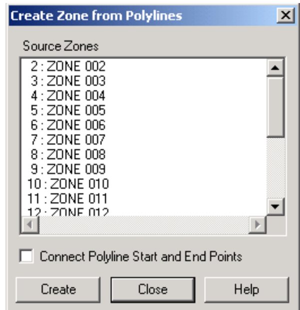


Figure 24-18. The Create Zone from Polylines dialog.

An option is also available to connect the start and end points for each supplied polyline. This is especially useful when creating 3-D surfaces. The supplied polylines and the resulting 3-D surface are shown in Figure 24-19.

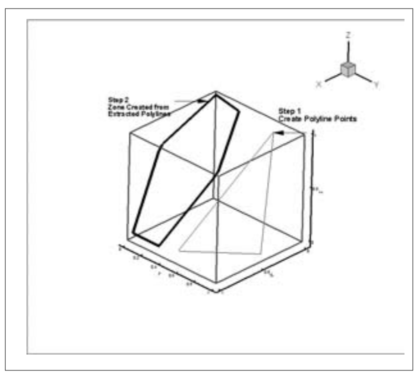


Figure 24-19. Supplied polylines and the resulting 3-D surface.

24.7. Zone or Variable Deletion

In any data set with more than one zone, you can use the Delete Zone dialog to delete any unwanted zones. You cannot delete all zones; if you attempt to delete all zones, the lowest numbered zone is not deleted.

To delete a zone:

1. From the Data menu, choose Delete. From the Delete sub-menu, select Zone. The Delete Zone dialog appears as shown in Figure 24-20.
2. Select the zone or zones you want to delete.
3. Click Delete to delete the zones.

Figure 24-20. The Delete Zone dialog.

To delete a variable:

1. From the Data menu, choose Delete. From the Delete sub-menu, select Variable. The Delete Variable dialog appears; it is similar to the Delete Zone dialog shown in Figure 24-20.
2. Select the variable or variables to delete.
3. Click Delete to delete the variables.

24.8. Irregular Data Point Triangulation

Triangulation is a process in which data points are connected to form triangles. You can use triangulation to convert irregular, I-ordered data sets into a finite-element surface zone. You can, in fact, use triangulation on any type of source zone, ordered or finite-element. But its use on irregular data is most common. Triangulation is one of two options for creating 2-D field plots from irregular data. The other is interpolation, discussed in Section 24.9, “Data Interpolation.” Triangulation preserves the accuracy of the data by creating an finite-element surface zone with the source data points as nodes and a set of Triangle elements. Triangulation is only available for the 2D Cartesian plot type.

Triangulation works best for 2-D data; you can, however, triangulate 3-D surface data as long as the Z-coordinate is single-valued (the surface does not wrap around on itself). When you triangulate 3-D surface data, the Z-coordinate of the data is ignored, causing a less-than-optimal triangulation in some cases.

To triangulate your data:

1. From the Data menu, choose Triangulate. The Triangulate dialog appears, as shown in Figure 24-21.
2. Select the zone or zones to triangulate from the scrolled list labeled Source Zone(s).
3. If you want to specify a boundary zone for the triangulation, select the Use Boundary Zone(s) check box and select the boundary zone or zones from the Boundary Zone(s) scrolled list. The boundary zones define the boundaries in the triangulation region; if you do not include boundary zones, Tecplot assumes the data points lie within a convex polygon and that all points in the interior can be connected.

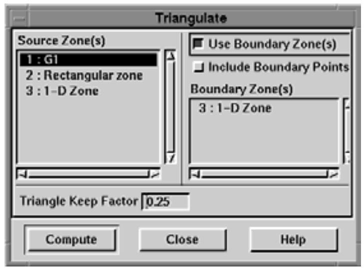


Figure 24-21. The Triangulate dialog.

4. (If Use Boundary Zone(s) is selected) If you want to include the points in the boundary zones in the triangulated zone, select the Include Boundary Points check box.
5. (Optional) Modify the Triangle Keep Factor, if desired. This factor is used to define "bad" triangles on the outside of the triangulated zone. See below for complete details.
6. Click Compute to perform the triangulation.

At the completion of triangulation, Tecplot attempts to remove bad triangles from the outside of the triangulation. This does not include any triangles next to the boundary zone, only those along the edges where there is no boundary (or where the boundary zone points are excluded). The definition of a bad triangle is stored as the Triangle Keep Factor as a number between zero (three collinear points) and 1.0 (an equilateral triangle). Typical settings are values between 0.1 and 0.3; settings above 0.5 are not allowed.

After triangulating your data, you can use the resulting finite-element surface zone to create plots. Generally, you turn off the original zone(s) and plot the new zone only, but you can, for example, plot a scatter plot of the original zone(s) along with the contours of the new zone.

24.9. Data Interpolation

Interpolation, in Tecplot, means assigning new values for the variables at data points in a zone based on the data point values in another zone (or set of zones).

For example, you may have a set of data points in an I-ordered zone that are distributed in a random-like fashion in the XY-plane. This type of data is sometimes referred to as unordered, ungridded, or random data; in Tecplot, it is called irregular data. Using data in this form, you can create mesh plots and scatter plots, but you cannot create contour plots, light-source shading, or streamtraces. In Tecplot, you can interpolate the irregular I-ordered data onto an IJ-ordered mesh, and then create contour plots and other types of field plots with the interpolated data. You can also interpolate your 3-D, I-ordered irregular data into an IJK-ordered zone and create 3-D volume plots from the IJK-ordered zone. You can even interpolate to a finite-element zone.

There are three types of interpolation available:

- **Linear:** Interpolate using linear interpolation from a set of finite-element, IJ-ordered, or IJK-ordered zones to one zone.
- **Inverse Distance:** Interpolate using an inverse-distance weighting from a set of zones to one zone.
- **Kriging:** Interpolate using kriging from a set of zones to one zone.

Each of these options is described in the following sections.

24.9.1. Inverse-Distance Interpolation

Inverse-distance interpolation averages the values at the data points from one set of zones (the source zones) to the data points in another zone (the destination zone). The average is weighted by a function of the distance between each source data point to the destination data point. The closer a source data point is to the destination data point, the greater its value is weighted.

In many cases, the source zone is an irregular data set—an I-ordered set of data points without any mesh structure (a list of points). Inverse-distance interpolation may be used to create 2- or 3-D surface, or a 3-D volume field plots of irregular data. The destination zone can, for example, be a circular or rectangular zone created within Tecplot. See Section 24.6, “Zone Creation.”

To perform inverse-distance interpolation in Tecplot, use the following steps:

1. Read the data set to be interpolated into Tecplot (the source data).
2. Read in or create the zone onto which the data is to be interpolated (the destination zone).
3. From the Data menu, choose Interpolate, then choose Inverse Distance. The Inverse-Distance Interpolation dialog appears, as shown in Figure 24-22.

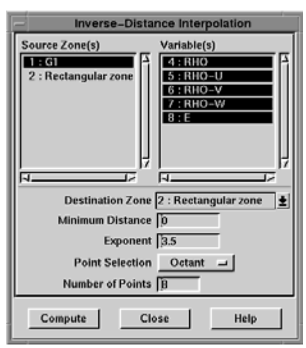


Figure 24-22. The Inverse-Distance Interpolation dialog.

4. Select the zones to be interpolated from those listed in the Source Zone(s) scrolled list.

5. Select which variables are to be interpolated from those listed in the Variable(s) scrolled list. By default, all variables are interpolated except those assigned to the X-, Y-, and Z-axes. If, after interpolating, you will be working with just one or two interpolated variables, you can speed up the calculations by interpolating only those needed variables.
6. Select the destination zone into which to interpolate. Existing values for the interpolated variables in the destination zone will be overwritten.
7. (Optional) Enter the minimum distance used for the inverse-distance weighting in the Minimum Distance text field. Source data points which are closer to a destination data point than this minimum distance are weighted as if they were at the minimum distance, thus reducing the weighting factor for such points. This tends to reduce the peaking and plateauing of the interpolated data near the source data points.
8. (Optional) Enter the exponent for the inverse-distance weighting in the Exponent text field.
9. (Optional) Select the method used for determining which source points to consider for each destination point from the Point Selection drop-down. There are three available methods, as follows:
   - **Nearest N:** For each point in the destination zone, consider only the closest *n* points to the destination point. These *n* points can come from any of the source zones. You specify *n* after selecting this option. This option may speed up processing if *n* is significantly smaller than the entire number of source points.
   - **Octant:** Like Nearest N above, except the *n* points are selected by coordinate-system octants. The *n* points are selected so they are distributed as evenly as possible throughout the eight octants. This reduces the chances of using source points which are all on one side of the destination point.
   - **All:** Consider all points in the source zone(s) for each point in the destination zone.
10. (Optional) If you specified Nearest N or Octant for the point selection method, enter the number of points.
11. Click Compute to perform the interpolation. While the interpolation is proceeding, a working dialog appears showing the progress of the interpolation. This dialog has a Cancel button allowing you to interrupt the interpolation.

    If you click Cancel during the interpolation process, the interpolation is terminated prematurely. The destination zone is left in an indeterminate state, and you should redo the interpolation.

Inverse-distance interpolation ignores the IJK-mode of IJK-ordered zones. All data points in both the source and destination zones are used in the interpolation.

Note: Tecplot uses the current frame's axis assignments to determine the variables to use for coordinates in interpolation, but it ignores any axis scaling that may be in effect.

24.9.1.1. The Inverse-Distance Algorithm. The algorithm used for inverse-distance interpolation is simple. The value of a variable at a data point in the destination zone is calculated as a function of the selected data points in the source zone (as defined in the Point Selection drop-down).

The value at each source zone data point is weighted by the inverse of the distance between the source data point and the destination data point raised to a power as shown below:

$$\varphi_d = \frac{\sum w_s \varphi_s}{\sum w_s}$$ (summed over the selected points in the source zone)

where φ_d and φ_s are the values of the variables at the destination point and the source point, respectively, and w_s is the weighting function defined as:

$$w_s = D^{-E}$$

D in the equation above is the distance between the source point and the destination point or the minimum distance specified in the dialog, whichever is greater. *E* is the exponent specified in the Exponent text field. The exponent should be set between 2 and 5. The algorithm is speed-optimized for an exponent of 4, although in many cases, the interpolation looks better with an exponent of 3.5.

Smoothing may improve the data created by inverse-distance interpolation. Smoothing adjusts the values at data points toward the average of the values at neighboring data points, removing peaks, plateaus, and noise from the data. See Section 24.10, “Data Smoothing,” for information on smoothing.

24.9.2. Kriging

Kriging is a more complex form of interpolation than inverse-distance. It works similar to inverse-distance interpolation discussed in Section 24.9.1, “Inverse-Distance Interpolation,” and is used for the same purposes. Kriging generally produces superior results to the inverse-distance algorithm but requires more computer memory and time.

To perform kriging in Tecplot, perform the following steps:

1. Read the data set to be interpolated into Tecplot (the source data).
2. Read in or create the zone onto which the data is to be interpolated (the destination zone).
3. From the Data menu, choose Interpolate, then choose Kriging. The Kriging dialog appears, as shown in Figure 24-23
4. Select the zones to be interpolated from those listed in the Source Zone(s) scrolled list.
5. Select which variables are to be interpolated from those listed in the Variable(s) scrolled list. By default, all variables are interpolated except those assigned to the X-, Y-, and Z-axes. If, after interpolating, you will be working with just one or two interpolated vari-

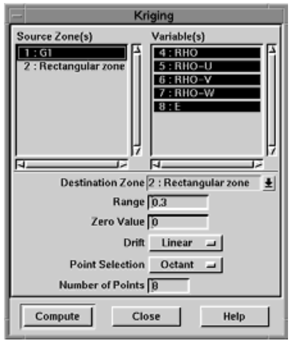


Figure 24-23. The Kriging dialog in Motif.

ables, you can speed up the calculations by interpolating only those needed variables. In kriging, interpolating fewer variables can have a significant effect on the speed of interpolation.

6. Select the destination zone into which to interpolate. Existing values for the interpolated variables in the destination zone will be overwritten.
7. (Optional) In the Range text field, enter the distance beyond which source points become insignificant for the kriging. The value is stated as the fraction of the length of the diagonal of the box which contains the data points. A range of zero means that any point not coincident with the destination point is statistically insignificant; a range of one means that every point in the data set is statistically significant for each point. In general, values between 0.2 and 0.5 should be used.
8. (Optional) In the Zero Value text field, enter the semi-variance at each source data point on a normalized scale from zero to one. Semi-variance is the certainty of the value at a data point. A value of zero means that the values at the source points are exact. Greater values mean the values at the source points have some uncertainty or noise. Zero is usually a good number for the zero value, and it causes the interpolated data to fit closely to all the source data points. Increasing the zero value results in smoother interpolated values that fit increasingly more to the average of the source data.
9. (Optional) Select the overall trend for the data in the Drift drop-down. This can be No Drift, Linear, or Quadratic.
10. (Optional) Select the method used for determining which source points to consider for each destination point from the Point Selection drop-down. There are three available methods, as follows:
    - **Nearest N:** For each point in the destination zone, consider only the closest *n* points to the destination point. These *n* points can come from any of the source zones.

- **Octant:** Like Nearest N above, except the *n* points are selected by coordinate-system octants. The *n* points are selected so they are distributed as evenly as possible throughout the eight octants. This reduces the chances of using source points which are all on one side of the destination point.
- **All:** Consider all points in the source zone(s) for each point in the destination zone.

This option is very important for kriging, since kriging involves the computationally expensive inversion and multiplication of matrices. The computational time and memory requirements increase rapidly as the number of selected source data points increases. In general, you should not use the All option unless you have very few source points.

11. If you specified Nearest N or Octant for the point selection method, enter the number of points.
12. Click Compute to perform the kriging. While the kriging is proceeding, a working dialog appears showing its progress. This dialog has a Cancel button allowing you to interrupt the kriging.

    If you click Cancel during the kriging process, the kriging is terminated prematurely. The destination zone is left in an indeterminate state, and you should redo the kriging.

Note: Tecplot uses the current frame's axis assignments to determine the variables to use for coordinates in kriging, but it ignores any axis scaling that may be in effect. Also, if the Drift is set to Linear or Quadratic, Tecplot requires that the points selected be non-collinear (non-coplanar in 3-D). To avoid this limitation, set the Drift to None. Tecplot requires that no points be coincident. You can eliminate coincident points by triangulation before you interpolate.

24.9.2.1. The Kriging Algorithm. For a detailed discussion of the kriging algorithm see:

> Davis, J. C., *Statistics and Data Analysis in Geology*, Second Edition, John Wiley & Sons, New York, 1973, 1986.

24.9.2.2. Kriging and Inverse Distance Interpolation Improvements. For better results with 3-D data, try changing the range of your Z-variable to one similar to the X-range the Y-range. Also, set Zero Value to 0.05.

24.9.3. Linear Interpolation

Linear interpolation differs from the two previous interpolation schemes previously discussed in that the source zone must have some 2- or 3-D structure. That is, the source zones must be IJ-ordered, IJK-ordered, or finite-element. Irregular I-ordered data cannot be used for the source zones. (For 2-D data, you may be able to first create a finite-element zone from an irregular, I-ordered zone by using triangulation. See Section 24.8, "Irregular Data Point Triangulation.")

Linear interpolation finds the values in the destination zone based on their location within the cells of the source zones. The value is linearly interpolated to the destination data points using only the data points at the vertices of the cell (or element) in the source zone(s).

To perform linear interpolation in Tecplot, perform the following steps:

1. Read the data set to be interpolated into Tecplot (the source data).
2. Read in or create the zone onto which the data is to be interpolated (the destination zone).
3. From the Data menu, choose Interpolate, then choose Linear. The Linear Interpolation dialog appears, as shown in Figure 24-24.

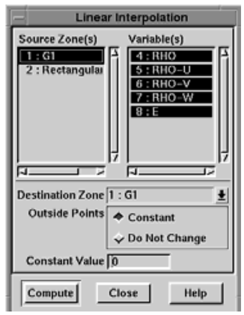


Figure 24-24. The Linear Interpolation dialog.

4. Select the zones to be interpolated from those listed in the Source Zone(s) scrolled list.
5. Select which variables are to be interpolated from those listed in the Variable(s) scrolled list. By default, all variables are interpolated except those assigned to the X-, Y-, and Z-axes. If, after interpolating, you will be working with just one or two interpolated variables, you can speed up the calculations by interpolating only those needed variables.
6. Select the destination zone into which to interpolate. Existing values for the interpolated variables in the destination zone will be overwritten.
7. (Optional) Select what to do with points that lie outside the source-zone data field. You have two options, represented by option buttons on the dialog: Constant, which sets all points outside the data field to a constant value that you specify; and Do Not Change, which preserves the values of points outside the data field. Do Not Change is appropriate in cases where you are using one interpolation algorithm inside the data field, and another outside.
8. (Optional) If you choose Constant as the Outside Points option, specify the constant value in the Constant Value text field.
9. Click Compute to perform the interpolation. While the interpolation is proceeding, a working dialog appears showing the progress of the interpolation. This dialog has a Cancel button allowing you to interrupt the interpolation.

   If you click Cancel during the interpolation process, the interpolation is terminated prematurely. The destination zone is left in an indeterminate state, and you should redo the interpolation.

24.9.4. Interpolation Alternatives

An alternative to 2-D interpolation is to triangulate your irregular data points. This creates a mesh of triangles (a Triangle element-type, finite-element zone) using the source data as node points. No interpolation is required. See Section 24.8, "Irregular Data Point Triangulation," for a description of triangulation.

24.10. Data Smoothing

You can smooth the values of a variable of any zone (in either 2- or 3-D) to reduce "noise" and lessen discontinuities in data. Smoothing can also be used after inverse-distance interpolation to reduce the artificial peaks and plateaus.

Smoothing is applied to XY-lines, in 2-D, across a 3-D surface, or in a 3-D volume, depending on the state of the current frame and the type of zone structure. Each pass of smoothing shifts the value of a variable at a data point towards an average of the values at its neighboring data points.

To smooth data in Tecplot, use the following steps:

1. From the Data menu, choose Alter, then choose Smooth. The Smooth dialog appears as shown in Figure 24-25.

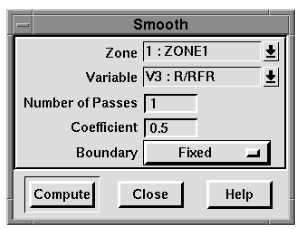


Figure 24-25. The Smooth dialog.

2. Select the zone to smooth from the Zone drop-down. The zone should not intersect itself.
3. Select the variable to smooth from the drop-down of variables in the zone. For the XY Line plot type, the variable must be a dependent variable for one active mapping for that zone.
4. (Optional) Specify the number of smoothing passes to perform. The default is 1. A greater number of passes will take more time but smooth the data more.
5. (Optional) Specify the relaxation factor for each pass of smoothing in the field labeled Coefficient. Enter a number between zero and one (exclusively). Large numbers flatten peaks and noise quickly. Small numbers smooth less each pass, rounding out peaks and valleys rather than eliminating them.
6. (Optional) Select the boundary conditions by which to smooth from the Boundary drop-down. The options are Fixed, First Order, and Second Order, as follows:

- **Fixed:** The points at the boundary are not changed in value. For finite-element data, only fixed boundary conditions may be used.
- **First Order:** The points at the boundary are smoothed based on the assumption that the first derivative normal to the boundary is constant. This will tend to cause contour lines of the smoothed variable to be perpendicular to the boundary.
- **Second Order:** The points at the boundary are smoothed based on the assumption that the second derivative normal to the boundary is constant. This option may overextrapolate derivatives at the boundary.

7. Click Compute to perform the smoothing. While the smoothing is underway, a working dialog appears showing the progress of the smoothing. This dialog has a Cancel button allowing you to interrupt the smoothing.

   If you click Cancel during the smoothing process, you will interrupt the smoothing, and Tecplot will report back the number of passes completed. For example, if you specified ten passes in Number of Passes but hit escape halfway through, Tecplot would report five passes complete.

Limitations to smoothing:

- Finite-element zones cannot be smoothed with anything other than Fixed boundary conditions.
- Tecplot uses the current frame's axis assignments to determine the variables to use for the coordinates in the smoothing, and also to determine whether the smoothing should be done with XY Line, 2-, or 3D Cartesian plot types. Be careful if you have multiple frames with different variable assignments for the same data set.
- Any axis scaling is ignored by Tecplot while smoothing.
- For I-ordered or finite-element line segment zones, the current frame can be in the XY Line, 2- or 3D Cartesian plot types. In XY Line, the variable must be the dependent variable of one active mapping for that zone.
- For IJ-ordered, finite-element triangle, or finite-element quadrilateral zones, the current frame can be a 2- or 3D Cartesian plot type, but you cannot smooth the variables assigned to the X- and Y-axes in 2D Cartesian.
- For IJK-ordered, finite-element tetrahedral, or finite-element brick zones, the plot type must be 3D Cartesian, and you cannot smooth the variables assigned to the X-, Y-, and Z-axes. The IJK-mode is ignored. The zone is smoothed with respect to the entire 3-D volume.
- Smoothing does not extend across zone boundaries. If you use a boundary condition option other than Fixed (such that values along the zone boundary change), contour lines can be discontinuous at the zone boundaries.
- Smoothing is performed on all nodes of a zone, and disregards value-blanking if it is active.

CHAPTER 25 Probing

In Tecplot probing is the ability to select a point and view the values of all variables at that point. You can also view information about the data set itself while probing. Similar to probing is probing-to-edit, a feature which allows you to modify your data interactively. To prevent you from inadvertently changing your data, probing-to-edit is disabled by default. You can enable the feature by toggling the option Allow Data Point Adjustment in the Edit menu. Use either the Probe At dialog or the Probe tool to obtain point information from a data field.

With the Probe At dialog, you can specify the location of the probe as set of spatial coordinates X, Y, and Z, one of the polar coordinates Theta and R, or as a set of I-, J-, and K-indices. You select one or more locations in the data field where information is to be collected, and the resulting information is displayed in the Probe dialog.

When you probe with the mouse, you can probe in either of two modes: Interpolate and Nearest Point. In Interpolate mode, accessed by a mouse click, the value returned is the linearly interpolated value for the specified locations. In Nearest Point mode, accessed by Ctrl-click, the value returned is the exact value at the closest data point in the field.

25.1. Field Plot Probing with the Mouse

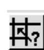
The most direct method of probing is to use the Probe tool. When you select the Probe tool from the sidebar and move the pointer into the workspace, where it becomes a cross-hair. Click at any point to probe in Interpolate mode, which calls up a dialog showing the probe information interpolated for that point. Ctrl-click at any location to probe in Nearest Point mode which will obtain probe information for the data point closest to the cross-hair.

The following table shows the information returned for each type of probe action for field plots. (All mouse click operations are using the left mouse button.)

Probe Action	Information Returned
Click	If the pointer is over a valid cell return the interpolated field values from all nodes in the cell. If multiple cells are candidates then for 2D Cartesian plots, the cell from the highest number zone is used, and for 3D Cartesian plots, the cell closest to the viewer is used.

Probe Action	Information Returned
Ctrl-Click	If the pointer is over a valid cell return the field values from the nearest node in the cell. If multiple cells are candidates then, for 2D Cartesian plots the cell from the highest number zone is used, and for 3D Cartesian plots the cell closest to the viewer is used. If the pointer is not over any cell then the field values from nearest data point as measured in distance on the screen are returned.
Shift-Ctrl-Click	Return the field values from the nearest point on the screen ignoring surfaces and regardless of zone number or depth of the point. This is useful in 3-D for probing on data points that are on the back side of a closed surface without having to rotate the object. In 2-D this is useful for probing on data points for zones that may be underneath other zones because of the order in which they were drawn.
Alt-Click (3D Cartesian plots only)	Same as Click except ignore zones while probing. (Probe only on streamtraces, iso-surfaces, or slices.)
Alt-Ctrl-Click	Same as Ctrl-Click except ignore zones while probing. (Probe only on streamtraces, iso-surfaces, or slices.)
Alt-Ctrl-Shift-Click	Same as Shift-Ctrl-Click except ignore zones while probing. (Probe only on streamtraces, iso-surfaces, or slices.)

To obtain interpolated variable values for the exact probed location:

1. Select the Probe tool, represented by , from the sidebar.
2. Move the pointer into the workspace. The pointer changes to a cross-hair.
3. Click at the desired location. A cross appears at the probed location, and the Probe dialog appears as shown in Figure 25-1, showing variable values. The variable values are interpolated linearly from the values of the data set. If you probe a 3-D volume zone in a 3D Cartesian plot, the probe cross-hairs point to locations on the surface of the plot, not to locations within the plot.
4. To view zone and cell information, click Zone/Cell Info to bring up the Zone/Cell Info page, as shown in Figure 25-1.
5. To view cell-centered values, click Cell Center. This shows values at the center of the cell you clicked on. For node-centered variables, this value will be interpolated from the corners of the cell.
6. Click at additional locations to view variable values or zone and cell information for other data points.

Note: Interpolate mode does not work for I-ordered data displayed in a 2D or 3D Cartesian plot; if you probe such data you will always get the error message “Point is outside of data

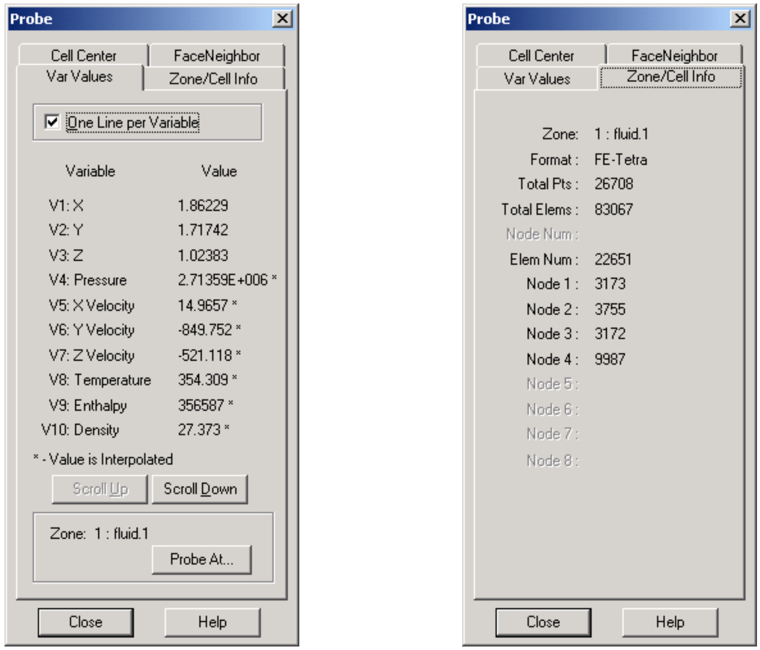
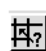


Figure 25-1. The Var Values and Zone/Cell Info pages of the Probe dialog.

field," because Tecplot cannot interpolate without a field mesh structure. You can, however, use the Nearest Point mode (described below) in such situations.

To obtain exact results for the data point nearest the probed location:

1. On the sidebar select the Probe tool, represented by .
2. Move the pointer into the workspace where it will change to a cross-hair.
3. Ctrl-click at the desired location. An XORed cross appears at the data point closest to the probed location, and the Probe dialog appears showing variable values, including the X-, Y-, and Z-coordinates of the nearest point.

   The variable values are the exact values for the nearest point. If the zone is not I-ordered and the cross-hairs are placed within the data field, the point reported is the closest data point in the cell pointed to by the pointer. (This may not be the closest point in the entire data field.) If you probe a 3-D volume zone in a 3D Cartesian plot, the probe cross-hairs point to locations on the surface of the plot, not to locations within the plot.
4. To view zone and cell information, click the Zone/Cell Info to call up the Zone/Cell Info page.
5. Ctrl-click at additional locations to view variable values or zone and cell information for other data points.

You may alternate between interpolated and exact values by clicking and Ctrl-clicking.

25.2. Advanced Field Plot Probing

By default a Tecplot probe first detects a zone cell face. It then finds the nearest point of that face if Ctrl-click was used while probing. However, advanced probe options let you probe points behind a cell face, or objects that are contained within zones.

25.2.1. Obscured Point Probing

Nearest point probing using Ctrl-click is limited in two ways. In a 2D Cartesian plot the Probe tool will select mesh intersection points only according to drawing order when using Ctrl-click. By default, when probing data in a 3D Cartesian plot using Ctrl-click with the Probe tool, the probe will only select the surface nearest you. Thus, if you are viewing a 3-D wire mesh, the probe will not select mesh intersection points shown on the far side of an enclosed surface, and you would need to rotate the view in order to select points on the far side.

To overcome this, you may use Shift-Ctrl-click. This option lets you select points to probe as if all visible points were projected onto the 2-D plane of the screen and a ruler was laid on the screen to measure the distance from the cross-hair to each point, allowing you to find the nearest point.

25.2.2. Streamtrace, Iso-Surface, and Slice Probing

In a 2D Cartesian plot Shift-Ctrl-click allows users to probe mesh intersection points regardless of drawing order. In a 3D Cartesian plot it will allow you to chose mesh intersection points independent of surface depth. Thus, when viewing a 3-D wire mesh for example, you may use Shift-Ctrl-click to select points on the far side of an enclosed surface.

You may probe for values on volume objects such as streamtraces, iso-surfaces and slices, with the Alt key. Using the Alt key alone when probing will give you interpolated values of the nearest volume object. Using Alt-Ctrl-click will probe a point where the mesh intersects the closest object. Using Alt-Shift-Ctrl-click will probe the closest point where the mesh intersects any object.

25.3. Field Plot Probing by Specifying Coordinates and Indices

If you want precise control over your probe location, or if you want to probe using I-, J-, and K-indices, or if you want to probe inside a 3-D volume, you need to use the Probe At dialog to specify the probe location. You can launch the Probe At dialog either from the Data menu (by choosing Probe At), from the Var Values page of the Probe dialog (using the Probe At button), or by clicking Details in Probe mode.

To probe at a specified location using spatial coordinates (in Interpolate mode):

1. Launch the Probe At dialog. The Probe At dialog appears, as in Figure 25-2, ready for you to enter X-, Y-, and Z-coordinates.
2. Enter the X-, Y-, and Z-coordinates of the desired probe location.
3. If the zone you are probing is a 3-D volume zone, select the check box labeled Probe Within Volume to ensure that the probe is performed at the indicated point. If you specify a position within a 3-D volume zone and the Probe Within Volume check box is not selected, Tecplot probes at the surface of the zone nearest the user’s eye along the ray defined by the specified point and the user’s eye.

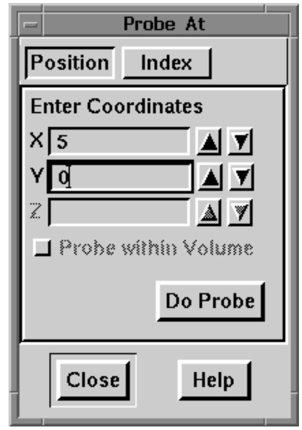


Figure 25-2. The Position page of the Probe At dialog.

4. Click Do Probe to perform the probe. The Probe dialog appears with interpolated values for the specified location.

To probe at a specified location using data set indices (in Nearest Point mode):

1. Launch the Probe At dialog. The Probe At dialog appears.
2. Click the Index button (Index tab in Windows) to bring up the Index page of the Probe At dialog, shown in Figure 25-3.
3. Select the desired zone from the Zone drop-down.
4. Enter the I-, J-, and K-indices of the desired probe location. (For finite-element and I-ordered data, you can enter only the I-index. For IJ-ordered data, you can enter both I- and J-indices. For IJK-ordered data, you can enter I-, J-, and K-indices.)
5. Click Do Probe to perform the probe. The Probe dialog appears.

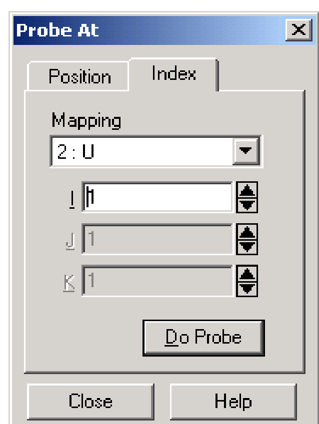


Figure 25-3. Index page of the Probe At dialog.

If you have already probed one point, you can specify new indices by increasing or decreasing the displayed values using the up and down arrows at the right of each index field. Doing this automatically performs the probe; you need not click Do Probe again.

25.4. Field Plot Probed Data Viewing

You view probed data in the Probe dialog. The Probe dialog has four pages:

- **Var Values:** Examine values of all variables at any selected location.
- **Zone/Cell Info:** Report characteristics of any location in a data field. The characteristics reported include the indices of the selected cell or point, the zone number, the dimensions of the zone, and the type of zone (ordered or finite-element).
- **Cell Center:** Examine values of all variables at the center of the clicked-on cell.
- **Face Neighbor:** Examine neighboring cells of the click-on cell.

25.4.1. Variable Value Viewing

The Var Values page of the Probe dialog, shown in Figure 25-1, lists every variable in the current data set, together with its value at the specified probe point. By default, each variable is shown on a single line, which allows display of about the first ten characters of the variable name and seven significant digits of the variable value.

To display longer variable names or see more digits of the value, deselect the check box labeled One Line per Variable. If there are more variables than will fit in one window, use the Scroll Up and Scroll Down buttons. The Var Values page also displays the zone name and number.

25.4.2. Zone and Cell Information Viewing

The Zone/Cell Info page of the Probe dialog, shown in Figure 25-1, lists the following information about any probed data point, regardless of the format of the data:

- The number and name of the probed zone.
- The format of the zone, either ordered or one of the finite-element formats:
  - **FE-Triangle.**
  - **FE-Quad.**
  - **FE-Tetra.**
  - **FE-Brick.**

For ordered zones, the following additional information is displayed:

- The maximum I-, J-, and K-indices of the zone. JMax is 1 for I-ordered data, and KMax is one for I-ordered and IJ-ordered data.
- The type of plane (I, J, or K) which was probed. (For I-ordered and IJ-ordered data, this is always K.)

- In Interpolate mode, the I-, J-, and K-indices of the principal data point of the cell containing the probed point.
- In Nearest Point mode, the I-, J-, and K-indices of the nearest point to the probed point.

For finite-element zones, the following additional information is displayed:

- The total number of points in the zone.
- The total number of elements (cells) in the zone.
- The number of the probed node. This field is filled in only if the point is probed in Nearest Point mode.
- The number of the probed element.
- The number of each node of the probed cell. There are three nodes for FE-surface triangle zones, four nodes for FE-surface quadrilateral and FE-volume tetrahedral zones, and eight nodes for FE-volume brick zones.

25.4.3. Cell Center Viewing

The Cell Center page of the Probe dialog, like the Var Values page, lists every variable in the current data set, but lists values at the center of the cell that was clicked on. By default, each variable is shown on a single line, which allows display of about the first ten characters of the variable name and seven significant digits of the variable value.

To display longer variable names or see more digits of the value, deselect the check box labeled One Line per Variable. If there are more variables than will fit in one window, use the Scroll Up and Scroll Down buttons. The Cell Center page also displays the zone name and number.

25.4.4. Face Neighbor Viewing

The Face Neighbor page of the Probe dialog displays cells that neighbor the selected cell. A cell is considered a neighbor if one of its faces shares all nodes in common with the selected cell, or if it is identified as a neighbor by face neighbor data in the data set. Refer to 4.1.2.9. “Face Neighbors” on page 47 for more information on face neighbor data.

25.5. Line Plot Probing with the Mouse

You may probe XY and Polar Line plots in much the same way you probe field plots. You can use the probe mouse mode to obtain interpolated variable values at any given location, or obtain exact values from a specified (X, Y) or (Theta, R) data point. When you probe an XY Line plot in the standard mode, Tecplot displays a vertical or horizontal line, depending on whether you are probing along an X- or a Y-axis.When you probe a Polar Line plot, a radial line or a circle is displayed depending on whether you are probing along the Theta- or R-axis. In either case, the probe is performed along the displayed line (or circle). Axis variable values of all active mappings that lie along the probe line are interpolated and displayed.

When you hold down the Ctrl key Tecplot enters Nearest Point probe mode, and the displayed line disappears. Now the exact location of the pointer is important. When you Ctrl-click, Tecplot displays the exact X- and Y- or Theta- and R-values of the data point closest to the location clicked.

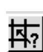
Note: When probing line plots, keep in mind whether you are in nearest point or interpolate mode. The presence or absence of the XORed line should indicate the current mode.The exact position of the mouse pointer, while relatively unimportant in interpolate mode, is significant in determining the nearest point.

25.5.1. Line Plot Probing in Interpolate Mode

Interpolate mode is the standard probe mouse mode in line plots just as for field plots. For XY Line plots, you can probe along any of Tecplot's five X-axes, or along any of Tecplot's five Y-axes. By default, probing is performed along the X1 axis. For Polar Line, probing is done along the Theta-axis by default.

Note: In Polar Line, many combinations of Theta- and R-values can result in the same point on the screen. When using the mouse in Interpolate mode to probe along the Theta-axis, Tecplot uses the Theta-value within the current Theta-axis range to determine the corresponding R-values reported in the Probe dialog. This behavior may even result in no probe information shown for a mapping that has Theta-values entirely outside the current Theta-axis range even though the mapping crosses the probe line on the screen. (For example, probing along the Theta-axis in interpolate mode misses a mapping representing only Theta-values several cycles outside the current Theta-axis range.) Similarly, when using the mouse in Interpolate mode to probe along the R-axis, Tecplot uses the R-value within the current R-axis range and may miss mappings that are shown on the plot but have R-values different from the R-axis range.

To enter the Probe Interpolate mode:

1. Choose the Probe tool, indicated by , from the sidebar.

2. Move the pointer into the workspace, where it becomes a cross-hair. When you move into the axis grid area, the cross-hair is augmented by a line indicating along which axis you are probing. Click at the desired location. A cross appears on each probed mapping at the probed location, and the Probe dialog appears with a title of Interpolated Values, as in Figure 25-4.
3. Read the desired information from the Probe dialog.

Repeat steps 2 and 3 as desired.

Figure 25-5 shows a workspace with an XY Line plot being probed along the X-axis.

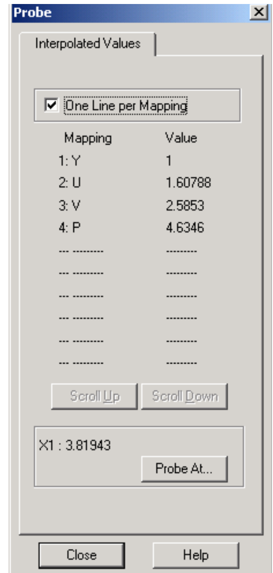


Figure 25-4. The Probe dialog for XY Line plots.

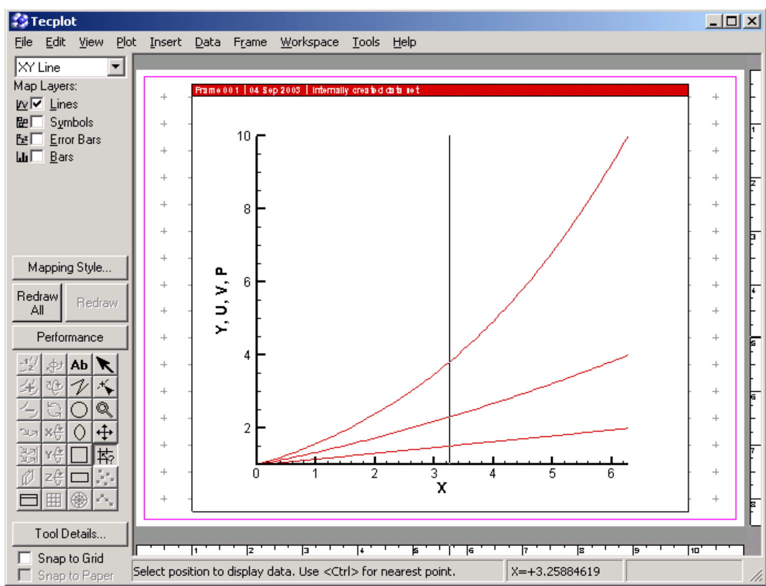


Figure 25-5. Probing an XY Line plot along the X-axis.

To specify which axis to probe along:

1. Choose the Probe tool from the sidebar.
2. Click Tool Details on the sidebar. The Probe At dialog appears. The Probe At dialogs for XY and Polar Line plots are shown in Figure 25-6

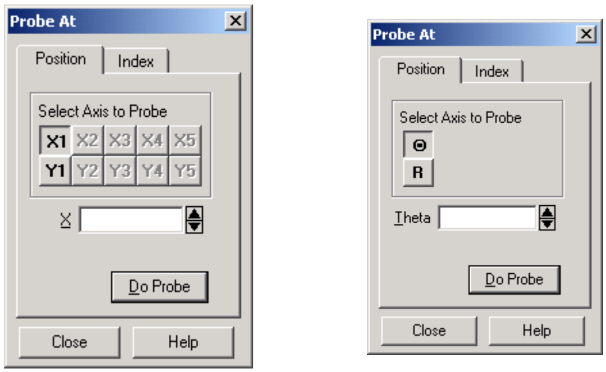


Figure 25-6. The Probe At dialog for XY and Polar Line plots.

3. Click the button labeled with the name of the axis you want to probe along.

Alternatively, you can press the X, Y, T, or R keys on the keyboard while moving the mouse over the plot to selects the axis to probe along. You can also press the 1, 2, 3, 4, or 5 keys to select different X- or Y-axes for multiple-axis XY Line plots.

25.5.2. Line Plot Probing in Nearest Point Mode

Nearest Point probe mode provides the exact X- and Y- or Theta- and R-values of the data point closest on the screen to the probed location, together with information on the mapping and the zone to which the probed point belongs. If a data point is common to multiple mappings, the probe returns information on the highest numbered mapping. For example, if a data point is plotted as part of two mappings, numbered 1 and 2, the probe results are displayed for mapping 2.

To enter the Probe Nearest Point mode:

1. Choose the Probe tool, indicated , from the sidebar.
2. Move the pointer into the workspace, where it becomes a cross-hair. When you move into the axis grid area, the cross-hair is augmented by a line indicating along which axis you are probing. Press and hold down the Ctrl key to see only the cross-hair as you move the mouse. Nearest Point probing is independent of the axis you were probing along.
3. Ctrl-click at the desired probe location. Remember, the nearest point is calculated from the actual location of the cross-hair.

A cross appears at the probed location, and the Probe dialog appears with the title Specific Values, as shown in Figure 25-7.

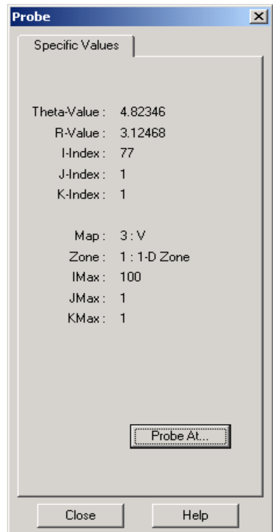


Figure 25-7. The Probe dialog for Polar Line plots (Nearest Point mode).

4. Read the desired data from the Probe dialog.
5. Repeat steps 3 and 4 as often as desired.

25.6. Line Data Probing by Specifying Coordinates and Indices

If you want precise control over your probe location, or if you want to probe using I-, J-, and K-indices, you need to use the Probe At dialog rather than the Probe mouse mode to specify the probe location. You can launch the Probe At dialog from the Data menu (by choosing Probe At), from the Probe dialog (using the Probe At button), or by clicking the Details button while in Probe mouse mode.

To probe at a specified location using position (in Interpolate mode):

1. Launch the Probe At dialog. The Probe At dialog appears, ready for you to enter a coordinate. By default the probe is done along the X1 axis for XY Line and Theta for Polar Line.
2. To probe along a different axis, click the button labeled with the name of the axis you want.
3. Enter the exact coordinate of the desired probe location.
4. Click Do Probe to perform the probe. The Probe dialog appears and a small cross appears on the plot to show the probed location.

To probe at a specified location using data set indices (in Nearest Point mode):

1. Launch the Probe At dialog. The Probe At dialog appears.
2. Click Index to bring up the Index page of the Probe At dialog, shown in Figure 25-8

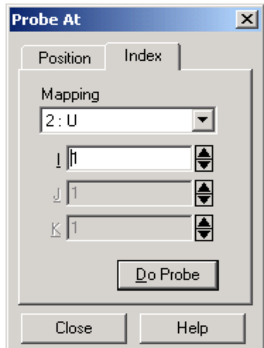


Figure 25-8. The Index page of the Probe At dialog for line plots.

3. Select the desired mapping from the Mapping drop-down.
4. Enter the I-, J-, and K-index of the desired probe location. (For finite-element and I-ordered data, you can enter only the I-index. For IJ-ordered data, you can enter both I- and J-indices. For IJK-ordered data, you can enter I-, J-, and K-indices.)
5. Click Do Probe to perform the probe. The Probe dialog appears.

If you have already probed one point, you can specify a new index by increasing or decreasing the displayed values using the up and down arrows at the right of the index fields. Doing this automatically performs the probe; you do not need to click Do Probe again. If you choose a combination of I-, J-, and K-indices that is valid for the data set but not included in the current mapping, the correct values are returned. No cross appears on the screen because that portion of the data is not being plotted.

25.7. Line Plot Data Viewing

The Probe dialog for line plots has two different forms, depending on whether you are probing for interpolated values or for information on the nearest data point.

25.7.1. Interpolated Line Plot Data Viewing

For interpolated values, the Probe dialog appears lists every active mapping and the interpolated value the opposing axis variable for that mapping. The value along the probed axis is listed at the bottom of the dialog. For example, Figure 25-4 shows a probe along the X1 axis and the corresponding Y-values.

In the Probe dialog, the probe value is dashed (---) if the probe is out of range for the mapping. The probe value is gray (inactive) if the mapping is not using the specific axis which you are probing. For example you probe the X1 axis and the mapping uses the X2 axis. This will only happen in XY Line plots with multiple X- or Y-axes.

By default, each mapping is shown on a single line, which allows display of about the first ten characters of the mapping name and seven significant digits of the variable value.

To display longer mapping names or see more digits of the value, deselect the check box labeled One Line per Mapping. If there are more mappings than will fit in one window, use the Scroll Up and Scroll Down buttons. Below the list of mappings, the position of the probe is listed.

25.7.2. Nearest Point Line Plot Data Viewing

In Nearest Point mode, the Probe dialog appears with the heading Specific Values (see Figure 25-7). You obtain this version of the Probe dialog if you use Probe At Index to specify a probe position, or if you use the Probe tool and Control-click to specify the probe position. This form displays the following information about the nearest data point to the probed position:

- X- or Theta-value.
- Y- or R-value.
- I-index.
- J-index.
- K-index.
- The number and name of the mapping associated with the data point.
- The number and name of the zone referenced in the mapping.
- The maximum I-index of the zone.
- The maximum J-index of the zone.
- The maximum K-index of the zone.
- For XY Line plots, the X-axis associated with the mapping.
- For XY Line plots, the Y-axis associated with the mapping.

25.8. Data Editing with Probe

Using the Adjustor tool, you can probe and edit specific data points. In Adjustor mode, you can actually modify the coordinates of your data with the mouse.

You can edit data points either by moving them with the mouse (in XY Line and 2D Cartesian plots only), or by using the Probe/Edit Data dialog to enter new values for any variable in the probed data point.

If you modify a shared variable with the Adjustor tool, the variable will be branched--a separate copy of the variable will be created for the edited zone. If you use the Probe/Edit Data dialog, you can inhibit branching by selecting the Alter in all Shared Zones toggle.

25.8.1. Data Editing with the Mouse

In XY Line and 2D Cartesian plots, you can select and move data points with the Adjustor mouse mode. You can select multiple data points and move them as a group. When you move data points with the mouse, you will not actually see the changes until you redraw the screen.

To edit your data with the mouse:

1. On the sidebar, choose the Adjustor tool, indicated by .
2. Move the pointer into the workspace, where it becomes the Adjustor.
3. Click on a single point to select it. In the Adjustor mode, you must be within one-half of the selection handle's width to select the data point. To select multiple points, you can either Shift-click after selecting your initial point to select additional points, or you can draw a group select band to select the points within the band. (In line plots, you can select points from only one mapping at a time.)
4. Once you have selected all desired points, move the Adjustor over the selection handles of one of the points, then click-and-drag to the desired location of the chosen data point. Other selected points will move as a unit with the chosen data point, maintaining their relative positions.

   For XY Line plots, if several mappings are using the same data for one of the variables, adjusting one of the mappings will result in simultaneous adjustments to the others. You can avoid this by pressing the H or V keys on your keyboard while adjusting the selected point. The H and V keys restrict the adjustment to the horizontal and vertical directions, respectively.

25.8.2. Data Editing with the Probe/Edit Data Dialog

To probe to edit using the Probe/Edit Data dialog:

1. On the sidebar, choose the Adjustor tool, indicated by the  button.
2. Move the pointer into the workspace, where it becomes the Adjustor.
3. Double-click on the point you want to edit, or click on the point and then click Object Details on the sidebar. The Probe/Edit Data dialog appears, as shown in Figure 25-9. All variables in the zone or mapping are listed, along with their values at the probed point. For 2D and 3D plot types, the Probe/Edit Data dialog has Scroll Up and Scroll Down buttons which are active if the data set has more variables than can be displayed on one page of the dialog.

   Note: If you attempt to double-click, but move the mouse between clicks, you may find that you have accidentally moved your data point.
4. If the variable you with to modify is shared by other zones and you want the modification to be used by all zones (and the variable to remain shared), select the Alter in all Shared Zones toggle.

5. Enter new values as desired.

The lower half of the Probe/Edit Data dialog is a copy of the Probe At dialog's Index page. You can use this area to specify a new zone or mapping to probe, along with the specific points to probe and edit. Thus, for example, you can edit one point, then increase or decrease the displayed indices to edit the next point along a mapping.

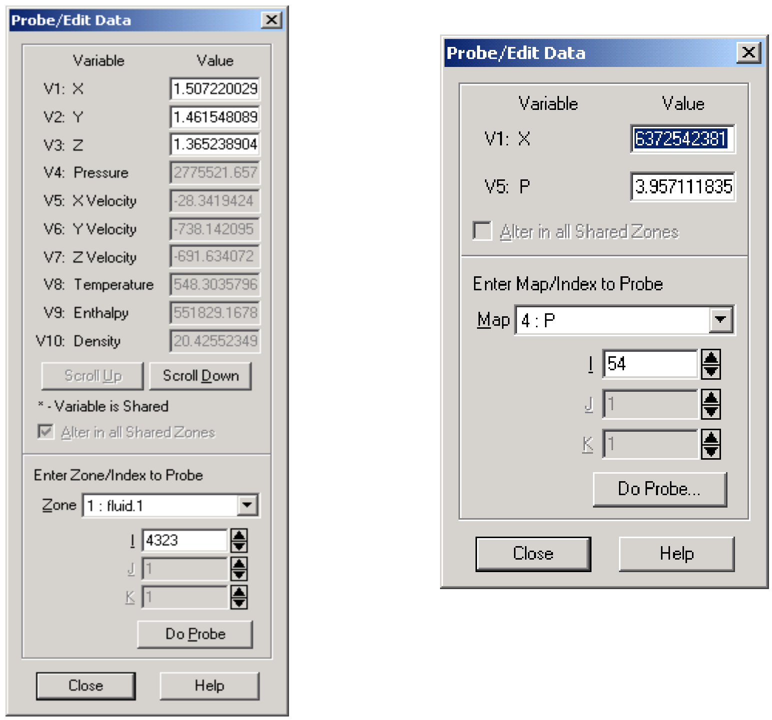


Figure 25-9. Probe/Edit Data dialog for field plots (left) and XY Line plots (right).

CHAPTER 26 Blanking

Blanking is the capability of Tecplot to exclude certain portions of zones from being plotted (in other words, selectively display certain cells or data points). In 3-D, the result is analogous to a cutaway view.

26.1. Two- and Three-Dimensional Blanking

In the following discussions, the term cell is used. In I-ordered data sets, a cell is the connection between two adjacent points. In IJ-ordered data sets, a cell is the quadrilateral area bounded by four neighboring data points. In IJK-ordered data sets, a cell is the six-faced (hexahedral) volume bounded by eight neighboring data points. For finite-element data sets, a cell is equivalent to an element.

There are three forms of blanking, as follows:

- **Value-Blanking:** Cells of all zones or line plot mappings are excluded based on the value of a variable (the value-blanking variable) at the data point of each cell or at the point where each cell intersects with a constraint boundary depending on the type of value blanking applied.
- **IJK-Blanking:** Cells of one IJK-ordered zone are included or excluded based on the index values.
- **Depth-Blanking:** Cells in a 3-D plot are visually excluded based on their distance from the viewer plane.

All types of blanking affect all field layers, except the Boundary zone layer. Value-blanking and IJK-blanking affect data operations such as streamtrace extraction, iso-surface extraction, slicing, and so on. Blanking affects linear interpolation, kriging and inverse-distance interpolation. Blanking is not performed in 3D plots when wire-frame sketches are drawn while rotating, translating, slicing, and so on.

For plots with more than one frame, changes in blanking settings are only applied to the current frame. Value blanking settings for multiple frames may be synchronized using frame linking. Refer to 18.1. “Attribute Linking Between Frames” on page 325 for more information on linking.

Blanking on volume zones may produce different results, depending upon the Surfaces to plot setting on the Surfaces page of the Zone Style dialog. See Section 20.1, “Surfaces to Plot,” for more details.

All types of blanking may be used in a single plot. They are cumulative: cells blanked from any of the options do not appear.

Value-blanking and depth-blanking affect all zones of all types of data, while IJK-blanking affects only one IJK-ordered zone. IJK-blanking is available for 2D or 3D Cartesian plot types; value-blanking is available for Line, 2D, or 3D plot types; depth-blanking is only available for the 3D plot type.

26.1.1. Two- and Three-Dimensional Value-Blanking

Value-blanking allows you to selectively eliminate or trim cells and elements from Line, 2-D, and 3-D field plots. The two forms of value-blanking are referred to as whole cell and precise blanking. The whole cell or precise blanking of cells is based on one or more user-defined constraints. For each active constraint you specify a value-blanking variable, a constant value or another variable, and a conditional statement telling Tecplot that region to blank in relation to the specified variable or constant. Whole cell blanking eliminates entire cells and therefore can result in a jagged blanking boundary, while precise blanking trims the display of data along a constraint boundary that you specify (precise blanking is only available for the 2D plot type).

26.1.1.1. Whole Cell Blanking. To use whole cell blanking:

1. From the Plot menu, choose Blanking/Value-Blanking. The Value-Blanking dialog appears as in Figure 26-1.

Figure 26-1. The Value-Blanking dialog when plot type is 2D or 3D Cartesian.

2. Select the Include Value-Blanking check box. Value-blanking has no effect until this check box is turned on and at least one constraint is activated (see step 4), regardless of the settings of the other value-blanking parameters.

Figure 26-2 demonstrates the various effects of whole cell and precise value-blanking modes.

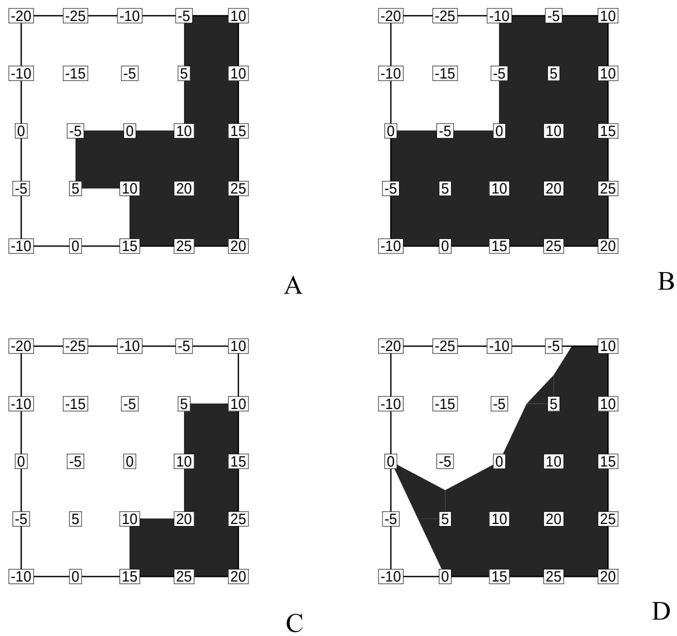


Figure 26-2. The effects of the different value-blanking options for a constraint where a variable is less than or equal to zero. The dark shading indicates the areas which are not blanked. A) Blank cell when primary corner is blanked. B) Blank cell when all corners are blanked. C) Blank cell when any corner is blanked. D) Trim cells along mathematical constraint boundary.

3. Choose the Blank entire cells option. Specify how the cell is blanked by selecting one of three selections from the option menu:
   - **All corners are blanked:** Cells are removed from the plot if all of their data points satisfy one or more of the active blanking constraints.
   - **Any corner is blanked:** Cells are removed from the plot if any of their data points satisfy one or more of the active blanking constraints.
   - **Primary corner is blanked:** Cells are removed from the plot based on the value at the primary corner of each cell. For ordered zones, the primary corner of a cell is the data point with the smallest indices of that cell. For finite-element zones, the primary corner is the first node listed in the data file's connectivity list for that element.

4. Activate a constraint by selecting the Active check box associated with the desired constraint number.
5. For any active constraint select the variable to use for value-blanking. This may be any variable, even one that is being used elsewhere in the plot. It is often convenient to create a new variable for use as the value-blanking variable. In this way you can manipulate its values without changing any other part of the plot. If no value-blanking variable is available, you can create one using the Specify Equations dialog, accessed from the Alter option of the Data drop-down (for instance, "{VBlank}=1"). See Section 24.2.1, "Equation Syntax."
6. Specify one of the following operations to describe how the blanking variable will be compared to the constant or variable following it:
   - **Is less than or equal to**: Cells for which the value-blanking variable has a value less than or equal to the specified constant or variable are removed from the plot.
   - **Is greater than or equal to**: Cells for which the value-blanking variable has a value greater than or equal to the specified constant or variable are removed from the plot.
   - **Is less than**: Cells for which the value-blanking variable has a value less than the specified constant or variable are removed from the plot.
   - **Is greater than**: Cells for which the value-blanking variable has a value greater than the specified constant or variable are removed from the plot.
   - **Is equal to**: Cells for which the value-blanking variable has a value equal to the specified constant or variable are removed from the plot.
   - **Is not equal to**: Cells for which the value-blanking variable has a value not equal to the specified constant or variable are removed from the plot.
7. Specify the value-blanking constant or variable used for comparison with the value-blanking variable:
   - **Variable**: This is the variable that is compared against the value-blanking variable to determine which cells are removed from the plot.
   - **Constant:** This is the number that is compared against the value-blanking variable to determine which cells are removed from the plot.

The Show Constraint Boundary check box will show you the line which separates the region of your data which is blanked from the region which is not blanked. Value-blanking has no effect on boundaries of an ordered zone. If the boundary is turned on, the boundary of the entire zone (without value-blanking) is plotted.

For finite-element data, value-blanking can affect the view of previously extracted boundaries, because each extracted boundary is a zone (see Section 19.4., "Boundary Extraction of Finite-Element Zones").

Figure 26-3 shows an IJ-ordered mesh with whole cell and precise value-blanking in use.

26.1.1.2. Precise Blanking. Precise blanking is only available for the 2D plot type. To use precise blanking:

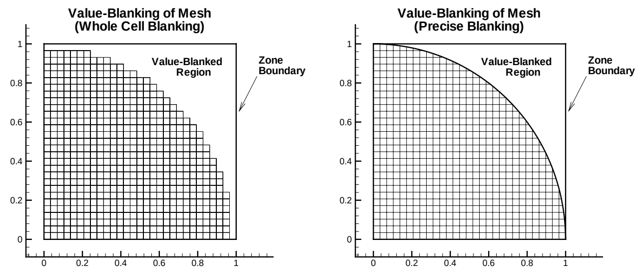


Figure 26-3. An IJ-ordered mesh with whole cell and precise value-blanking.

1. From the Plot menu, choose Blanking/Value-Blanking. The Value-Blanking dialog appears as in Figure 26-4.

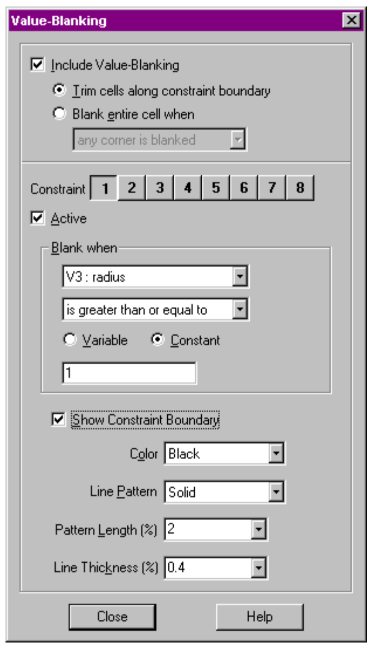


Figure 26-4. The Value-Blanking dialog for 2- and 3-D Cartesian plot types, set for precise blanking.

2. Select the Include Value-Blanking check box. Value-blanking has no effect until it is turned on and at least one constraint is activated (see step 4), regardless of the settings of the other value-blanking parameters.

Figure 26-2 demonstrates the various effects of whole cell and precise value blanking modes.

3. Choose the Trim Cells along Constraint Boundary option.
4. Activate a constraint by selecting the Active check box associated with the desired constraint number.
5. For any active constraint select the variable to use for value-blanking. This variable can be any variable, even one that is being used elsewhere in the plot. It is often convenient to create a new variable for use as the value-blanking variable. In this way you can manipulate its values without changing any other part of the plot. If no value-blanking variable is available, you can create one using the Specify Equations dialog (for example, "{VBlank}=1"). See Section 24.2.1, "Equation Syntax."
6. Specify one of the following operations to describe how the blanking variable will be compared to the constant or variable following it:
   - **Is less than or equal to:** Trim away all regions where the value-blanking variable is less than or equal to the specified constant.
   - **Is greater than or equal to:** Trim away all regions where the value-blanking variable has a value greater than or equal to the specified constant or variable.
7. Specify the value-blanking constant or variable used for comparison with the value-blanking variable:
   - **Variable:** This is the variable that is compared against the value-blanking variable.
   - **Constant:** This is the number that is compared against the value-blanking variable.
8. Select the Show Constraint Boundary check box to control the visibility of the defined constraint boundary line. The various attributes of the constraint line such as color, line pattern, pattern length, and line thickness can also be set to produce the desired effect.

Value-blanking has no effect on boundaries of an ordered zone. If the boundary is turned on, the boundary of the entire zone (without value-blanking) is plotted. For finite-element data, value-blanking can affect the boundary extraction. For more information see Section 19.4. "Boundary Extraction of Finite-Element Zones" on page 340.

Figure 26-3 shows a mesh with whole cell and precise value-blanking in use. Figure 26-5 demonstrates a more complex usage of precise value-blanking by overlaying multiple frames, each using some of the same constraints.

26.1.2. IJK-Blanking

IJK-blanking is available only for 3-D volume zones. IJK-blanking removes a selected portion of one IJK-ordered zone from the plot. This allows you to create cutaway plots, plots showing the exterior of some data set with a section "cut away" to show the interior, such as the plot shown in Figure 26-6. You define the blank region by specifying the following:

- The IJK-ordered zone in which the blanking is to be performed.

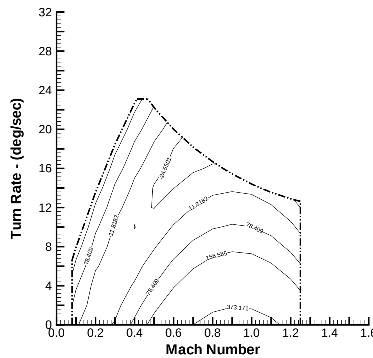


Figure 26-5. A single contour variable with precise blanking.

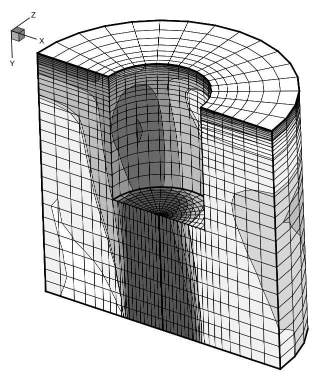


Figure 26-6. A cutaway plot created with IJK-blanking.

- I-, J-, and K-index ranges for the blank region, either using specific index values or percentages of the index range.
- Whether Tecplot should blank the interior or exterior of the defined region.

To use IJK-blanking, you must have an IJK-ordered zone, and the current plot type must be 2D or 3D Cartesian. Unlike value-blanking, which operates on all zones within a single frame, IJK-blanking can only be used on a single zone within a frame, and the zone must be IJK-ordered.

To use IJK-blanking:

1. From the Plot menu, choose Blanking/IJK-Blanking. The IJK-Blanking dialog appears as in Figure 26-7.

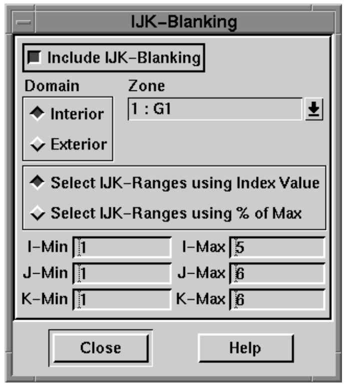


Figure 26-7. The IJK-Blanking dialog.

2. Select the Include IJK-Blanking check box to turn on IJK-blanking for the current frame. IJK-blanking does not take effect until this option is turned on, nor are any of the other controls sensitive.
3. Specify the domain of the IJK-blanking by choosing one of the following options:
   - **Interior:** Cells within the specified index ranges are blanked. Those outside are plotted. This creates a "hole" in the zone. The left side of Figure 26-8 shows an ordered zone with IJK-blanking with Interior domain.
   - **Exterior:** Cells outside the specified index ranges are blanked. Those inside are plotted. This plots a sub-zone of the zone. The right side of Figure 26-8 shows an ordered zone with IJK-blanking with Exterior domain.
4. Select the zone to which IJK-blanking is applied from the Zone drop-down. The zone must be IJK-ordered. You may select only one zone at a time.
5. Specify the format in which you will specify the index ranges by selecting one of the following option buttons:
   - **Select IJK-Ranges Using Index Values:** If you select this option, you specify the I-, J-, and K-index ranges using actual minimum and maximum indices.
   - **Select IJK-Ranges Using % of Max:** If you select this option, you specify the I-, J-, and K-index ranges as start and end percentages of the maximum index. For example, you could blank the middle third of a data set by setting the start percentage to 33.3 and the end percentage to 66.6.
6. Enter the I-, J-, and K-index ranges in the fields provided.

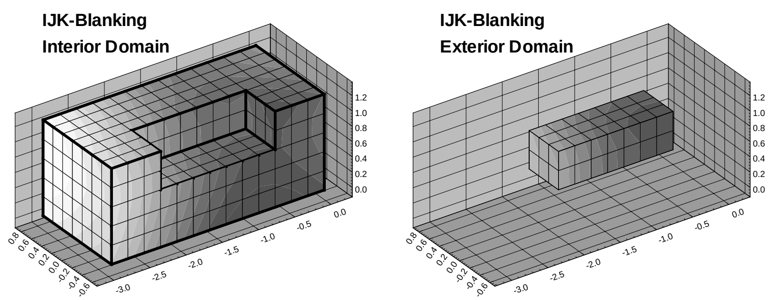


Figure 26-8. IJK-blanking with Interior domain (left) and Exterior domain (right).

When you save a layout, macro, or stylesheet, the IJK-blanking index ranges are stored as the percentage of the maximum index regardless of how you chose to enter them. This way, one file can be used for different zone sizes.

26.1.3. Cutaway Plots

Cutaway plots are plots of a 3-D volume zone in which a portion of the zone is blanked using IJK-blanking so that the interior of the zone can be seen. Create Figure 26-6 as follows:

1. Create an IJK-ordered zone, and create a 3-D contour plot of the zone. (Leave the Mesh zone layer turned on.)
2. Set the contour plot type to either Flood or Both Lines and Flood.
3. Use the IJK-Blanking dialog to blank out the appropriate region. Use the Interior blanking domain.

A more complex cutaway plot is shown in Figure 26-9. This plot contains iso-surfaces inside the cutout region.

To create the plot in Figure 26-9, continue with the following steps:

1. Set the contour plot type of the IJK-ordered zone to Both Lines and Flood.
2. Set the IJK-mode of the IJK-ordered zone to Volume.
3. Use the IJK-Blanking dialog to blank out the appropriate region, and choose the Exterior blanking domain. You should now have a plot of the region you want to blank out.
4. Call up the 3D Iso-Surface Details dialog from the Field menu. Select Show Iso-Surfaces and set the Draw Iso-Surfaces At drop-down to Each Contour Level.
5. From the Data menu, choose Extract, then choose Iso-Surfaces to extract the contour iso-surfaces of the region you want to blank out.
6. Change the contour plot type of the new iso-surface zones to Flood.

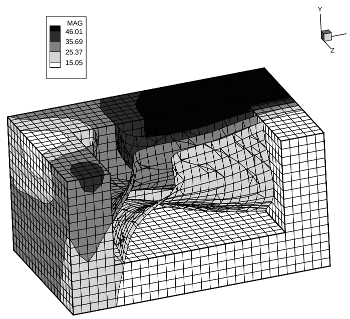


Figure 26-9. Cutaway plot with iso-surfaces inside cutout region.

7. Change the IJK-blanking domain to Interior. This changes the IJK-blanking to plot the entire zone minus the blanked region.
8. Set the IJK mode of the IJK-ordered zone to Face.

The iso-surfaces appear in the plot because they were extracted as separate zones.

26.1.4. Depth-Blanking

Depth-blanking removes cells in a 3-D plot based upon how close or far they appear from the screen. The options below are available on the Depth Blanking dialog, displayed by selecting Blanking/Depth Blanking from the Plot menu. This dialog is shown in Figure 26-10.

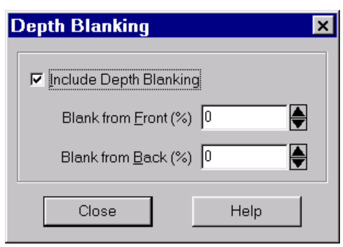


Figure 26-10. The Depth Blanking dialog.

- **Include Depth Blanking:** Select this check box to toggle depth-blanking on and off.
- **Blank from Front (%):** Blank cells appearing closer to the viewer than this plane. The value entered is the plane position in percentage of depth from the closest corner of the bounding box of the data to the furthest corner of the bounding box.

At the default of zero, the plane is at the depth of the closest corner of the bounding box. No cells on the front of the plot are blanked. At 50, the front half of the plot will be blanked. In particular, cells closer to the viewer than the front of the blanking plan, and cells further from the viewer than the blanking plane, may be blanked.

- **Blank from Back (%):** Blank cells appearing farther from the viewer than this plane. The value entered is the plane position in percentage of depth from the furthest corner of the bounding box of the data to the closest corner of the bounding box.

  At the default of zero, this plane is at the depth of the furthest corner of the bounding box. No cells on the back of the plot will be blanked. At 50, the back half of the plot will be blanked.

26.2. Line Plot Blanking

For line plots, blanking is the capability of Tecplot to exclude data points from consideration in the resulting plot. On a global scale, only value-blanking is available. To plot specific index ranges you can use the Indices page of the Mapping Style dialog to limit index ranges per mapping. The Curves page can provide another form of blanking, by allowing you to limit the range for the independent variable for individual mappings.

Figure 26-11 shows two plots. The original data for the plots contain some “bad” data points. The bad data points were identified as those with a Y-value greater than 0.6. The plot on the left uses all data points, including the bad data points, to draw a curve. The plot on the right has filtered out the bad data points by using value-blanking where all points are removed if Y > 0.6. Blanking does not necessarily have to be on the independent or dependent variable.

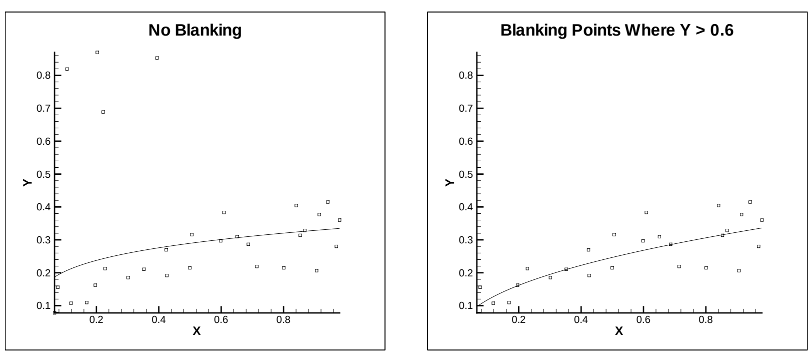


Figure 26-11. XY Line plots showing the effect of value-blanking.

CHAPTER 27 *Macro Commands*

Macro files allow you to automate Tecplot. Macro files contain a sequence of macro commands, and may contain macro function definitions. Macro functions act like macros-within-macros: they allow you to combine macro commands that you frequently use into a single unit callable from within another macro. This chapter focuses on the Tecplot menu options for recording and playing back macros. The *Tecplot Reference Manual* describes the Tecplot macro language in detail.

Macros are very useful for performing repetitive operations such as setting up frames, reading in data files and layout files, manipulating data, and creating plots. They are also necessary for running Tecplot in batch mode. See Chapter 28, “Batch Processing.”

The Macro sub-menu, found under the File menu, provides the following control options:

- **Play:** Calls up the Load/Play Macro File dialog, to select a macro file to load and play.
- **View:** Select this option to call up the Macro Viewer dialog, which provides several command buttons for stepping through and debugging a macro file.
- **Record:** Calls up the Macro Recorder dialog, which provides several command buttons for recording a series of actions to a macro file for playing back at a later time.

27.1. Macro Creation

The simplest way to create a macro is to have Tecplot record it for you. You can then use any ASCII text editor to edit the macro file. While editing your file, you can, for example, add macro function definitions, or add loops and other control commands.

Tecplot’s Macro Recorder lets you record a macro as you perform a sequence of actions interactively to obtain precisely the results you want. After recording your macro, you can edit it with an ASCII text editor to remove redundant operations, compress repetitive actions into loops, and otherwise modify the macro. Using the Macro Recorder is the quickest and surest way to become familiar with the Tecplot macro language.

To record a macro with the Macro Recorder:

1. From the File menu, choose Macro, then choose Record. The Write Macro File dialog is displayed.
2. Specify a macro file name.

3. Click OK to initiate the recording of the macro file. The Macro Recorder dialog now appears, as shown in Figure 27-1. This dialog must remain up during the recording session.

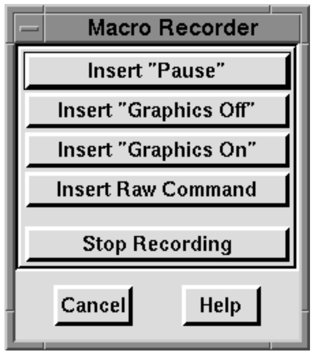


Figure 27-1. The Macro Recorder dialog.

4. Perform the actions you want recorded using the Tecplot interface.
5. Click Stop Recording on the Macro Recorder dialog when you are finished with the sequence of actions you want recorded.

While recording macros, you can use any of the following four buttons on the Macro Recorder dialog to add specific macro commands to your macro:

- **Insert "Pause":** Adds a "pause" command to the macro. When you play a macro including a pause command, Tecplot displays a message box when it reaches the pause command, and waits for you to click OK before continuing to process the macro.
- **Insert "Graphics Off":** Adds a "graphics off" command to the macro. When you play a macro containing a "graphics off" command, Tecplot stops displaying graphics in the workspace from the "graphics off" command until a "graphics on" command is encountered.
- **Insert "Graphics On":** Adds a "graphics on" command to the macro.
- **Insert Raw Command:** Brings up a dialog in which you can enter any valid Tecplot macro command. For example, you can add "**`$!LOOP 10`**" at the start of a section you want to repeat 10 times, then "**`$!ENDLOOP`**" at the end. See the *Tecplot Reference Manual* for information on the Tecplot macro language.

27.1.1. Macro Functions

When editing your macros, you can add macro function definitions and macro function calls. Macro functions have the following form:

```
$!MACROFUNCTION
  NAME = functionname
 .
 .
 .
$!ENDMACROFUNCTION
```

Between **$!MACROFUNCTION** and **$!ENDMACROFUNCTION**, you can include any legal macro command except **$!MACROFUNCTION**. These included macro commands are associated with the *functionname* specified as the value of the **NAME** parameter, but are not executed until the macro function is called with the **$!RUNMACROFUNCTION** macro command.

For example, the following macro function turns on the Contour zone layer, turns off the Mesh zone layer, sets the contour plot type to Both Lines and Flood for zones 1, 2 and 3, then chooses gray scale color mapping:

```
$!MACROFUNCTION
  NAME = "graycontour"
  RETAIN = Yes
$!FIELDLAYERS SHOWCONTOUR = YES
$!FIELDLAYERS SHOWMESH = NO
$!FIELD [1-3]  CONTOUR{CONTOURTYPE = BOTHLINESANDFLOOD}
$!COLORMAP CONTOURCOLORMAP = GRAYSCALE
$!REDRAW
$!ENDMACROFUNCTION
```

The **RETAIN** parameter tells Tecplot to retain the macro function definition for use in subsequent macro calls; this allows you to define a macro function once in some macro you load every time you run Tecplot, and continue to use it throughout your Tecplot session. See Section 27.4, “Doing More with Macros,” for a more sophisticated example of a macro function.

Use the **$!RUNMACROFUNCTION** macro command to call your macro function. For example, to call the “graycontour” macro function defined above, use the following macro command:

```
$!RUNMACROFUNCTION "graycontour"
```

If the macro function requires any parameters, you combine them into a parenthesized list which you give as a second argument to **$!RUNMACROFUNCTION**, as in the following example:

```
$!RUNMACROFUNCTION "Process Plane" (K,|LOOP|)
```

You can use the **$!RUNMACROFUNCTION** command within other macro functions; calls may be nested up to ten deep.

27.2. Macro Play Back

Once you have created a macro file, you have four methods in Tecplot for playing it back.

The four different methods are:

- From the command line.
- From the File menu, under the Macro Play option in the interface.
- Stepping through commands using the Macro Viewer.
- From the Quick Macro Panel.

The following sections explain each of these methods.

27.2.1. Macro Play Back Preparation

Often, the commands in a macro file rely on Tecplot being in a particular state. It is usually a good practice to have commands at the start of a macro that force Tecplot into a known state, **`$!NEWLAYOUT`** is a good command to do this since it deletes all data sets and frames and creates a single empty frame with a default size and position.

If you will always run your macro from the command line, then you can be sure Tecplot will be in its initial state when the macro begins processing. Including layout files, data files or stylesheet files on the command line along with the macro file is fine as long as the macro expects them.

If your macro performs some intermediate task, it is up to you to make sure Tecplot is in the same (or a similar) state when you run the macro as the state the macro was designed to start in.

27.2.2. Macros from the Command Line

You can immediately play a macro when you first start Tecplot by simply including the name of the macro file on the command line. In Windows, this can be accomplished by either dragging and dropping a macro file onto the Tecplot icon, or by using the command line from an MS-DOS dialog.

For example, to run the macro file **`mymacro.mcr`** from a UNIX or DOS command line prompt, type:

```
tecplot mymacro.mcr
```

If you name your macro file without the **`.mcr`** extension you can still run Tecplot with the macro file. However, you must include the **`-p`** flag on the command line. To run the macro file called **`mymacro.mmm`** you would type:

```
tecplot -p mymacro.mmm
```

In Windows you cannot drag and drop a macro file onto the Tecplot icon if it does not have the **`.mcr`** extension. Tecplot will think it is an ASCII data file and attempt to read it in as such.

If you want the macro viewer to automatically appear so you can see the macro commands prior to their execution, you can include the **`-z`** flag on the command line.

Macros can also be played back in batch mode (i.e., no graphics are displayed). See Chapter 28, “Batch Processing,” for details.

27.2.3. Macros from the Interface

You can play a macro from within Tecplot by using the Play option under the File main menu. This plays back the macro file without stopping until it reaches the end of the file.

To play back a macro file from the Tecplot interface:

1. From the File menu, choose Macro, then choose Play. The Load/Play Macro File dialog is displayed.
2. Specify a macro file name.
3. Click OK.

Tecplot immediately starts playing the specified macro file.

27.2.4. Macros from the Quick Macro Panel

The Quick Macro Panel (Figure 27-2) is Tecplot’s quick access mechanism for storing and retrieving your favorite, commonly used macro functions. This panel allows you to quickly play a macro function by clicking on the button in the panel that is linked to that macro function.

The Quick Macro Panel is linked to a special macro file that contains only macro function definitions. When Tecplot first starts up, it looks for this file under one of the following names, in the following order:

1. The file **tecplot.mcr** in the current directory.
2. The file **.tecplot.mcr** in your home directory (UNIX), or **tecplot.mcr** in the your home directory (Windows). Under Windows, your home directory is determined by the two environment variables **HOMEDRIVE** and **HOMEPATH**. If they are not set, Tecplot skips your home directory.
3. The file **tecplot.mcr** in the Tecplot home directory.

If Tecplot finds the file, it loads it and associates each button on the Quick Macro Panel with a specific macro function.

You can specify a different Quick Macro file by adding the **-qm** option flag in front of the macro file name to the command line.

The following command starts Tecplot and installs the macro functions defined in the file **myteccmd.mcr** into the Quick Macro Panel:

```
tecplot -qm myteccmd.mcr
```

If you want Tecplot to call up the Quick Macro Panel immediately after start up, include the **-showpanel** option flag at the end of the command as well.

Figure 27-2. The Quick Macro Panel.

For example, the following command starts Tecplot and immediately calls up the Quick Macro Panel:

```
tecplot -qm myteccmd.mcr -showpanel
```

To see an example of a macro function file, look at the Quick Macro file **`qmp.mcr`** located in the examples/mcr sub-directory below the Tecplot home directory.

Once the Quick Macro Panel has been installed you can run a macro by pressing its associated button on the Quick Macro Panel.

27.2.5. Macro Linking to Text and Geometries

Each text or geometry you create can be linked to a macro function. This macro function is called whenever the user holds down the control key and clicks the right mouse button on the text or geometry.

For example, if you have pieces of text, each representing a different well, Ctrl-right click on any piece could run a macro that brings up an XY-plot of that well's data.

Macro functions are specified with the "Link to Macro function" field in the Geometry dialog or in the Text Options dialog. If desired, the macro function may be listed with one or more parameters.

27.3. Macro Debugging

Use the Macro Viewer to step through and debug your macro file. This dialog allows you to add and delete breakpoints, view and set watch variables, and view state variables local to the macro currently loaded into the Macro Viewer. Selecting View from the Macro sub-menu displays the Macro Viewer dialog, shown in Figure 27-3. Using the Macro Viewer to look at a macro file built with the Macro Recorder is a quick and easy way to explore the Tecplot macro language.

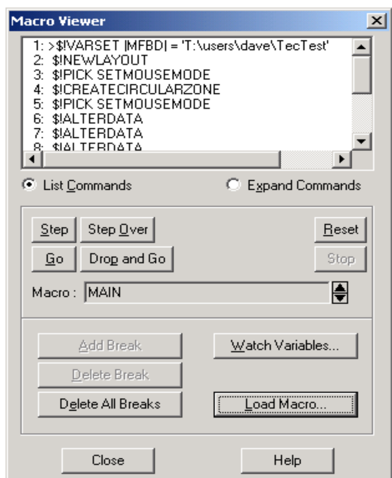


Figure 27-3. The Macro Viewer dialog.

To load a macro file into the Macro Viewer, click Load Macro. This calls up the Load/Play Macro File dialog for you to specify which macro file to load. Macro files typically have the extension **`.mcr`**.

The specified macro is loaded into the macro viewer for you. If you already had a macro loaded, it is discarded and the new macro is loaded in its place.

The Macro Viewer dialog displays the text of the currently loaded macro file at the top of the dialog. A **>** (greater than) marks the currently active line, that is, the line that Tecplot is about to evaluate. Click Step to evaluate the currently active line. The **>** sign then moves to the next line.

27.3.1. Macro Context

In the Macro Viewer dialog, the field labeled Macro displays the name of the macro or macro function you are currently evaluating. In most cases, this field displays the name **MAIN**, which means that the macro commands currently shown in the macro text display come from within the main macro body, that is, not from inside a macro function. If the macro you are viewing contains a call to a macro function, then when you evaluate (or step into) that macro function call (when you evaluate a **$!RUNMACROFUNCTION** command) the name displayed in the Macro field changes. The new name displayed is the name of the macro function just called. At the same time, the display in the top of the Macro Viewer dialog changes to show the macro function text for the called macro function.

Pressing on the up and down arrows located at the right hand side of the Macro display field shifts the macro context, that is, it lets you move between the text of the called macro function, and the text of the calling macro or macro function. If you switch context to the calling macro function, the **$!RUNMACROFUNCTION** command that called the macro is displayed with a **^** (caret) in front of it. This helps you quickly determine which command line called the macro function currently under evaluation. The down arrow then moves you back down a level to the called macro you were just viewing.

27.3.2. Macro Command Display Format Changes

Tecplot displays the macro in the viewer in one of two formats: a short format that lists the macro commands, one command per line, and a long format which expands a single, simple macro command to show all of its sub-commands and parameters. The short format is the default for the Macro Viewer.

- To choose the short form, select the List Commands option.
- To choose the long form, select the Expand Commands option.

27.3.3. Macro File Evaluation with the Macro Viewer

The Macro Viewer dialog's main purpose is to allow you to step through a macro's commands in a variety of ways so you can view and debug a macro file. With this dialog you can evaluate each line, including the commands within nested macro function calls, or just have Tecplot run the macro automatically while you watch.

27.3.3.1. Macro Step-Through Line by Line. The main activity you do in the Macro Viewer dialog is evaluate macro commands line by line. The **>** (greater than) marks the currently active command, that is, the command that Tecplot is about to evaluate. It moves to the next command after the currently active command is evaluated.

To evaluate a macro command, click Step. When a **$!RUNMACROFUNCTION** command is encountered, the macro viewer steps into the called function.

Step Over also processes each macro command, line by line however, when a **$!RUNMACRO-FUNCTION** command is encountered the entire function is processed.

You can also view or play a macro all the way through without stopping. To play the macro without stopping after each step, click Go. Tecplot continues until it either receives a stop signal from the Stop button, or it finishes playing the macro, or it encounters a breakpoint. See Section 27.3.4, "Breakpoint Addition and Deletion," on using breakpoints.

To stop the playing of a macro that the Go control started, click Stop.

You can restart a macro from the beginning, so you can evaluate it again from within the Macro Viewer. To restart the evaluation of a macro, click Reset.

Note: If your macro assumes Tecplot is in a particular state when it starts processing then you must make sure Tecplot is in this state before you click Reset and start the macro processing.

27.3.4. Breakpoint Addition and Deletion

An important debugging feature that the Macro Viewer provides is the ability to add breakpoints within a macro's command stream. A breakpoint is a flag you can insert anywhere in a macro that tells Tecplot to immediately suspend evaluation. Tecplot stops the action of a playing macro at the breakpoint to allow you to explore what is happening at that point in the macro file.

To add a breakpoint to a macro command:

1. Highlight the command in front of which you want to place the breakpoint by clicking on the command with the mouse cursor.
2. Click Add Break to add a breakpoint at the selected macro command. A **B** displayed at the beginning of the highlighted macro command indicates the breakpoint's placement.

To delete a breakpoint from a macro command:

1. Highlight the command for which you want to delete the breakpoint by clicking on the command with the mouse cursor.
2. Click Delete Break to delete the breakpoint.

To delete all the breakpoints set in a macro, click Delete All Breaks. This removes all the breakpoints within a macro.

27.3.5. Variable Value Viewing while Debugging

Another debugging feature that the Macro Viewer provides is the ability to specify and view specific user defined, or system defined internal variables—that is, to specify watch variables. Use the Macro Variables dialog, shown in Figure 27-4, that is displayed after clicking Watch Variable to specify and view watch variables.

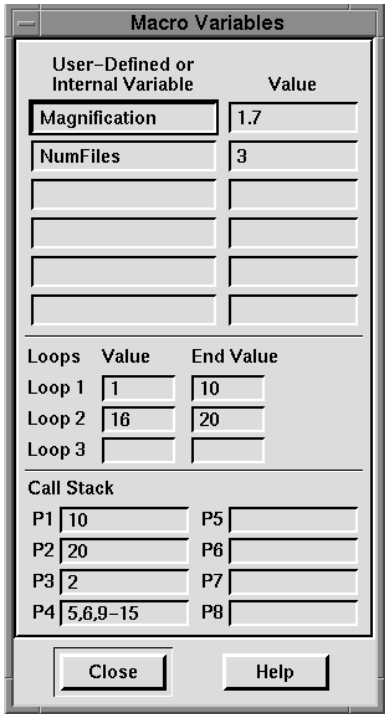


Figure 27-4. The Macro Variables dialog.

To specify a watch variable in a macro:

1. Click Watch Variables in the Macro Viewer dialog to bring up the Macro Variables dialog.
2. Type the name of the variable you want to watch in one of the User-Defined or Internal Variable text fields.

Leave this dialog open, off to one side, to watch the changing values in the Value column to the right of the variable name as the macro is playing.

The Macro Variables dialog also automatically displays any loop iteration values and command parameter stack calls that occur as a macro is played. In the Loops display area, the Value column for loops displays the loop iteration counter and changes as the system cycles through the loop sequence. The End Value displays the total number of iterations set for that loop. Tecplot lists the iteration values for up to three levels of nested loops. The Call Stack area displays the parameter values used in calling the currently active macro function. Each field displays one parameter value up to the limit of eight parameters per macro function call. The display changes to show the state of the currently called macro function as Tecplot evaluates a **$!RUNMACROFUNCTION** command.

27.3.6. Macro Variable Modification

After a breakpoint stops macro evaluation, you can change the value of a variable, a loop, or a stack call parameter. The ability to change these values in the middle of evaluating a macro allows you to either check a certain condition or to get out of a problem situation.

To change the value of a variable, a loop or a call stack parameter:

1. Highlight the variable value you want to change by selecting the value in the text field.
2. Type the new value.
3. Click Go or Step to continue the macro evaluation with the new value.

27.4. Doing More with Macros

There is much more to macro files than just recording and playing them. Tecplot provides you a whole macro programming language that allows you to develop tools for performing various functions, from developing your own interactive demonstration to transforming your data with a series of algorithms. See the *Tecplot Reference Manual* for details of Tecplot's macro language and macro files.

27.4.1. Multiple File Processing

Suppose you need to create hardcopy plots with a specific style for a large number of input data files. You can do this with a macro that does the following:

- Reads in data files named `t`*nnn*`.plt` where *nnn* counts from 1 to 50.
- Applies a predefined layout to the data set.
- Generates a PostScript file named `t`*nnn*`.ps` (*nnn* counting from 1 to 50), one for each data file.

There are a number of ways to create a macro like the one described. The easiest way involves using the **$!OPENLAYOUT** command feature that allows you to replace data files referenced in the layout file itself with other data files:

1. Read in a representative example data file and create the layout you want to plot. If you are creating a contour plot make sure the contour levels you choose will be good ones for all files that are processed.
2. Create a macro file that references the layout file created in Step 1.

If the layout file generated in Step 1 is called **cont.lay** then the final macro to process the data files is as follows:

```
#!MC 1000
#       Use a variable to store the number of
#       files to process.
```

```
$!VarSet |NumFiles| = 50
#       Make sure the output is PostScript.
$!ExportSetup
  ExportFormat    = PS
  Palette         = MONOCHROME
 #       Begin the loop
$!Loop |NumFiles|
#       Here is where we make use of the special feature
#       (i.e. the AltDataLoadInstructions option) of the $!OPENLAYOUT
#       command that allows us to override the named
#       data files within the layout file. Also make use of
#       the intrinsic LOOP macro variable.
$!OpenLayout "cont.lay"
  AltDataLoadInstructions = "t|LOOP|.plt"
#
#       Set the name of the file to be printed.
#
$!ExportSetup
  ExportFName = "t|LOOP|.ps"
#
#       Create the PostScript file.
#
$!Export
$!EndLoop
$!Quit
```

27.5. Macros, Layouts or Stylesheet Use

Tecplot layout files are simply macro files. Why not just create the plot you want, then save a layout to preserve it, rather than recording a macro to recreate it? Layout files are typically very simple macro files; they do not include loops, data alterations performed by many add-ons, or any such “programming.” Macros are generally used to do more complex tasks than layout files.

Tecplot processes layout and stylesheet files in a slightly different manner than it does macro files. In general, commands processed from a macro file undergo rigorous error checking and adjustments are sometimes made when values are outside certain limits. Layout and stylesheet files are treated more as a single unit by Tecplot. Individual commands are given more latitude because Tecplot assumes that the layout or stylesheet at one time represented a valid state. This is also why, while stepping through a macro file, the **$!OPENLAYOUT** and **$!READSTYLESHEET** commands are processed in a single step and you are not able to step “into” either of these commands. For layout and stylesheet files, error checking is not done until after the file is processed. By contrast, macro files are checked and adjusted line by line during processing. So, results from a layout file or stylesheet that you modify by hand may not be the same as those from a macro file.

CHAPTER 28 Batch Processing

You can run Tecplot in batch mode to create plots without displaying any graphics to the screen. This saves a lot of time when processing multiple files for printing or export. In batch mode, Tecplot can be executed locally on your workstation computer or, under UNIX, remotely using an ASCII terminal. The only limitation for batch mode operation is that under UNIX, you must use the Mesa version of Tecplot to create export files in bitmap formats. (The OpenGL version requires screen resources not available in batch mode.) Under Windows, all export formats are available.

28.1. Batch Processing Setup

To prepare for batch processing, you generally perform the following steps:

1. Create a macro file to control the batch processing. You may do this either interactively, by recording a Tecplot session, or using an ASCII text editor, or both. See Chapter 27, "Macro Commands."
2. Create layout and stylesheet files as necessary.
3. Prepare data files.
4. Debug the macro file by running Tecplot while not in batch mode.

Macros are necessary to do batch processing. When Tecplot is launched in batch mode it requires that you provide the name of a macro file to execute. The minimal command to launch Tecplot in batch mode is as follows:

`tecplot -b -p` *macrofile*

The **`-b`** flag instructs Tecplot to run in batch mode and the **`-p`** *macrofile* tells Tecplot the name of the macro file to execute. See Appendix A, "Tecplot Command Line Options," for more command line options.

How the macro file interacts with layout and/or stylesheet files is the subject of the following sections. Different strategies work with different situations depending on your data and what you want to do with it.

28.2. Batch Processing Using a Layout File

Combining layout files with batch processing is both powerful and flexible. With layout files you can organize a plot using one or more frames in a single file. The layout file manages data sets and can be altered on the fly, either on the command line or within a macro that loads the layout file.

For example, suppose you want to do the following sequence of tasks in batch mode:

- Load a data file from a user supplied file name.
- Create a specific style of plot.
- Create a PostScript file of the plot.

You can set up the batch job as follows:

1. Obtain a representative data file to be plotted.
2. Interactively, create a layout of the style of plot you want. (For this example, name the file **`batch.lay`**).
3. Create (using a text editor) the following macro (for this example call this macro **`batch.mcr`**):

```
#!MC 1000
$!ExportSetup
  ExportFormat = PS
  Palette = MONOCHROME
$!Export
$!Quit
```

The above macro can be modified to choose a different driver or palette depending on your situation.

Use the following command to run the job in batch mode:

`tecplot -b -p batch.mcr -y tecplot.out batch.lay` *mydatafile*

Layout files are self-contained. They contain all the information necessary to create a plot including the name(s) of the data file(s) to load. When you supply the names of data files on the command line (*mydatafile* in the above example) along with the layout file, Tecplot replaces the data files referenced in the layout file with the ones from the command line. If two or more data files are to be combined to form a single data set, use the "**+**" symbol to join the data file names.

The final result is a file called **`tecplot.out`** which contains the PostScript commands.

28.3. Multiple Data File Processing

In Section 28.2, "Batch Processing Using a Layout File," we set up Tecplot to process a user-supplied data file (or data files) and create a single output file. If the above procedure is to be repeated for a large number of input files (one at a time), you can do this by using a loop: either outside Tecplot in the operating system or within Tecplot using the flow-of-control commands in the Tecplot macro language.

28.3.1. Looping Outside Tecplot

The following examples show the command files for launching Tecplot in an operating system loop on two different operating systems. Tecplot processes five data files named **d***nn***.plt** and creates ten output files named **d***nn***.out** where *nn* goes from 1 to 10.

28.3.1.1. Looping Outside Tecplot (UNIX). Create a shell script with the following commands:

```
#!/bin/sh
n=1
while test $n -le 10
do
    tecplot -b -p batch.mcr -y d$n.out batch.lay d$n.plt
    n=`expr $n+1`
done
```

28.3.1.2. Looping Outside Tecplot (Windows). Create a batch file with the following commands:

```
for %%f in (d1 d2 d3 d4 d5 d6 d7 d8 d9 d10)
do tecplot -b -p batch.mcr -y %%f.out batch.lay %%f.plt
```

28.3.2. Looping Inside Tecplot

In Section 28.3.1, "Looping Outside Tecplot," we set up Tecplot to process multiple data files using the operating system language to do the looping. There are two drawbacks to this procedure:

- The operating system languages are not portable between different operating systems.
- Tecplot must be continuously started and stopped each time a new data set is processed.

A more efficient approach is to loop through the data files inside Tecplot. Here, the layout file and the data files are all named within the Tecplot macro. The command line in this example is simple, as follows:

```
tecplot -b -p batch.mcr
```

The Tecplot macro is set up as follows:

```
#!MC 1000
$!EXPORTSETUP
  EXPORTFORMAT = PS
  PALETTE = MONOCHROME
$!LOOP 10
$!OPENLAYOUT "batch.lay"
  ALTDATALOADINSTRUCTIONS = "d|LOOP|.plt"
$!EXPORTSETUP
  EXPORTFNAME = "d|LOOP|.out"
```

```
$!EXPORT
$!ENDLOOP
$!QUIT
```

The **$!OPENLAYOUT** command loads in **batch.lay** but replaces the data file referenced in the layout with the file names in the **ALTDATALOADINSTRUCTIONS** sub-command. The **$!EXPORTSETUP** command is used in two places. Initially it is used to set the export format. Later it is used just to change the name of the file to export to. The **$!EXPORT** command does the actual exporting.

28.4. Batch Processing Using Stylesheet Files

Instead of using layout files, you can use stylesheet files when batch processing. In general, batch processing with stylesheets works just like the batch processing described above in “Processing Multiple Data Files,” except a stylesheet file is used instead of a layout file.

If you want to make many different plots using the same data set, stylesheets will be more efficient than layout files.

28.5. Batch Processing Diagnostics

Each time Tecplot is run in batch mode it creates a file defined by the name in the **BATCHLOGFILE** environment variable, or, if the environment variable is not defined, by a file named **batch.log** in the directory where Tecplot was started. If the name given in the **BATCHLOGFILE** environment variable is a relative path, the directory name where Tecplot was started is prefixed. A running commentary on actions performed in Tecplot, as well as warning and error messages, are sent to the **batch.log** file.

28.6. Macros Moved to Different Computers or Directories

The file **tecplot.phy** is created each time you run Tecplot interactively. It contains information about the physical characteristics of your computer system as well as information about the size of the Tecplot process window used during the last Tecplot session. It also contains the name of the last layout file used by Tecplot. If you are developing macros on one computer, but using them for batch processing on a different computer, you must transfer the **tecplot.phy** file from the development computer to the computer where you will run Tecplot in batch mode. Under UNIX, the same is true if you are developing macros in one directory, but will be processing them in batch mode in a different directory. See Section 30.6, “Tecplot.phy File Location Configuration,” for information on the location of your **tecplot.phy** file.

CHAPTER 29 *Animation*

Animation is the process of showing a succession of still images to give the impression of motion. Tecplot provides a variety of methods for creating animated plots, and exporting them to movie files for playback at a later time. There are three basic animation methods available:

- **Animation Tools:** Perform simple animations using the dialogs in the Animate sub-menu under the Tools menu. The Animate sub-menu allows you to animate zones, mappings, contour levels, IJK-planes, IJK-blanking, and streamtraces. The animation is viewed within Tecplot, or exported to a movie file which can be played back outside of Tecplot.
- **Manually:** Interactively create movies by creating an initial plot, exporting the image as either a AVI or Raster Metafile movie, then repeatedly changing and appending new images to the same movie file.
- **Macros:** Use a macro to perform multiple, repetitive changes, and write each image to a movie file.

This chapter discusses these methods, as well as ways to play back movies that have been created in Tecplot.

29.1. Animation Tools

Use the Animate sub-menu under the Tools menu to have Tecplot cycle through your data, automatically displaying zones, IJK-planes, or any of several other plot elements, one after the other, until your entire data set has been displayed. The following plot elements may be animated using the dialogs in the Animate submenu:

- Contour Levels.
- IJK-Planes.
- IJK-Blanking.
- Mappings.
- Slices.
- Streamtraces.

- Zones.

29.1.1. Contour Level Animation

Use the Animate Contour Levels dialog to display all or a specified subset of the contour levels defined in the current frame, one at a time.

To animate contour levels:

1. From the Tools menu, choose Animate, then choose Contour Levels. The Animate Contour Levels dialog appears, as shown in Figure 29-1.

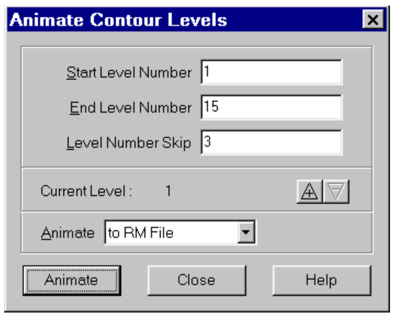


Figure 29-1. The Animate Contour Levels dialog.

2. Specify a start contour level (the first contour level you want displayed), an end contour level, and a contour level skip in the fields provided. If you specify a start level having a higher number than the end level, Tecplot cycles backward from the start to the end.
3. If you want Tecplot to create a movie file containing the animation, select “to AVI file” or “to RM file” from the Animate drop-down.
4. Click Animate to run the animation automatically, or use the + and - in the Current Level area to “step through” the animation one contour level at a time. (These cycle through the range of levels specified by Start Level Number and End Level Number; if your range is reversed, so are their actions.)

29.1.2. IJK-Plane Animation

Use the Animate IJK-Planes dialog to display all or a specified sub-set of the IJK-planes in the current data set, one at a time. You can choose to animate either the I-, J-, or K-planes.

To animate IJK-planes:

1. From the Tools menu, choose Animate, then choose IJK-Planes. The Animate IJK-Planes dialog appears, as shown in Figure 29-2.
2. Specify the set of planes to animate: I-Planes, J-Planes, or K-Planes.

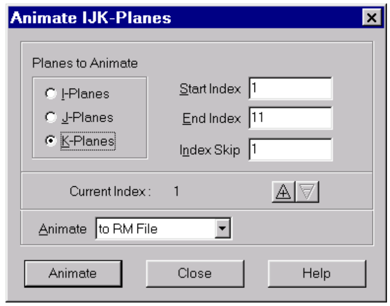


Figure 29-2. The Animate IJK-Planes dialog.

3. Specify a start index (the first plane you want displayed), an end index, and an index skip in the fields provided. If you specify a start index having a higher number than the end index, Tecplot cycles backward from the start to the end.
4. If you want Tecplot to create a movie file containing the animation, select “to AVI file” or “to RM file” from the Animate drop-down.
5. Click Animate to run the animation automatically, or use + and - in the Current Index area to “step through” the animation one plane at a time. (These cycle through the range of planes specified by Start Index and End Index; if your range is reversed, so are their actions.)

Figure 29-3 shows an example of animating I-planes in an IJK-ordered zone.

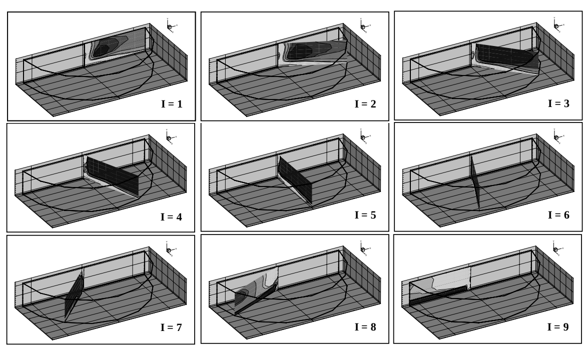


Figure 29-3. An animated sequence of I-planes.

29.1.3. IJK-Blanking Animation

Use the Animate IJK-Blanking dialog to animate a sequence of Tecplot renderings starting with an initial set of blanked IJK indices and proceeding in a series of interpolated steps to a final set of blanked IJK indices. Before you can animate IJK-blanking, you must first specify which zone you want to use in the IJK-Blanking dialog.

To animate a sequence of IJK-blankings, you must first turn on IJK-blanking, then use the Animate IJK-Blanking dialog:

1. From the Style menu, choose IJK-Blanking. The IJK-Blanking dialog appears.
2. Select the Include IJK-Blanking check box. The remaining controls in the dialog become active.
3. From the Zone drop-down, select the desired zone and specify whether the interior or exterior of the zone should be blanked.
4. Click Close.
5. From the Tools menu, choose Animate, then choose IJK-Blanking. The Animate IJK-Blanking dialog appears, as shown in Figure 29-4.

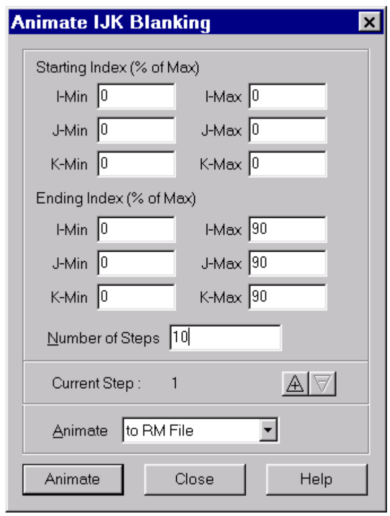


Figure 29-4. The Animate IJK-Blanking dialog.

6. Specify an initial set of blanked IJK-indices in the text fields grouped under the title Starting Index (% of Max). Enter a range of indices for each of I, J, and K. Index values are entered as percentages of the maximum index.
7. Specify a final set of blanked IJK-indices in the text fields grouped under the title Ending Index (% of Max). Enter a range of indices for each of I, J, and K.
8. Specify the number of steps—that is, the number of renderings required to move from the initial IJK-blanking to the final IJK-blanking. The minimum number of steps is two.
9. If you want Tecplot to create a movie file containing the animation, select “to AVI file” or “to RM file” from the Animate drop-down.
10. Click Animate to run the animation automatically, or use + and - in the Current Step area to “step through” the animation one step at a time.

29.1.4. Mapping Animation

Use the Animate Mappings dialog to display all or a specified subset of the XY or Polar Line mappings defined in the current frame, one at a time.

To animate mappings:

1. From the Tools menu, choose Animate, then choose Mappings. The Animate Mappings dialog appears, as shown in Figure 29-5.

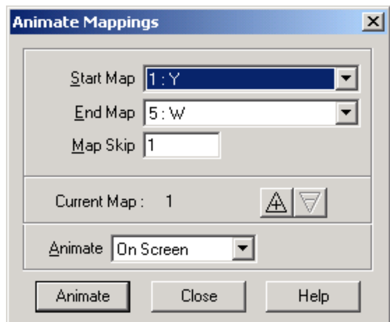


Figure 29-5. The Animate Mappings dialog.

2. Specify a Start Map (the first line mapping you want displayed), an End Map, and a Map Skip in the fields provided. If you specify a Start Map having a higher number than the End Map, Tecplot cycles backward from the start to the end.
3. If you want Tecplot to create a movie file containing the animation, select “to AVI file” or “to RM file” from the Animate drop-down.
4. Click Animate to run the animation automatically, or use + and - in the Current Map area to “step through” the animation one mapping at a time. (Theses cycle through the range of mappings specified by Start Map and End Map; if your range is reversed, so are their actions.)

You can try this with the demo data file, **`demo/xy/rainfall.plt`**.

29.1.5. Slice Animation

Use the Animate Slices dialog to animate a sequence of slices through your data. Use the Slice tool or the 3D Slice Details dialog to configure the start and end slices for your animation. The Animate Slices dialog is shown in Figure 29-6.

To animate a sequence of slices perform the following steps:

1. Use the Slice tool or the 3D Slice Details dialog to define a start and end slice.
2. From the Tools menu, choose Animate, then select Slices. The Animate Slices dialog appears, as shown in Figure 29-6.
3. Enter the number of slices to animate.

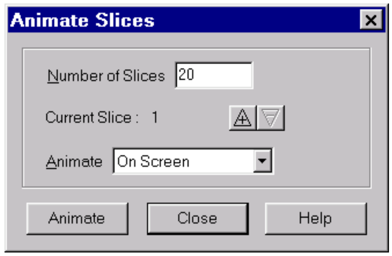


Figure 29-6. The Animate Slices dialog.

4. If you want to create a movie file containing the animation select "to AVI file" or "to RM file" from the Animate drop-down.
5. Click Animate.

If the slices are currently assigned to I-, J-, or K-planes, the total number of slices you can animate is limited to the total number of I-, J-, or K-planes possible between the start and end slice planes. If the slice planes are X-, Y-, or Z-planes you may specify any number of slices larger than or equal to two.

29.1.6. Streamtrace Animation

Use the Animate Streamtraces dialog to create animated images of streamtraces. You specify the number of images shown for each streamtrace cycle, and the number of cycles to show. The resulting animation shows streamtrace markers and/or streamtrace timing dashes at each step of the animation, "moving" down the streamtrace. Before you can animate streamtraces, you must turn on either the timing dashes or timing markers or both, using the Streamtrace Details dialog under the Field menu. See Section 12.6, "Streamtrace Timing," for details.

To animate your streamtraces:

1. Create a number of streamtraces. (For information on how to do this, see Chapter 12, "Streamtraces.")
2. If you have not already done so, select Show Dashes or Show Markers from the Timing page of the Streamtrace Details dialog. (Choose Streamtrace Details from the Field menu to call up this dialog.)
3. From the Tool menu, choose Animate, then choose Streamtraces. The Animate Streamtraces dialog appears, as shown in Figure 29-7.
4. If you want Tecplot to create a movie file containing the animation, select "to AVI file" or "to RM file" from the Animate drop-down.
5. Specify the number of steps per cycle and the number of cycles in the fields provided.
6. Click Animate to begin the animation.

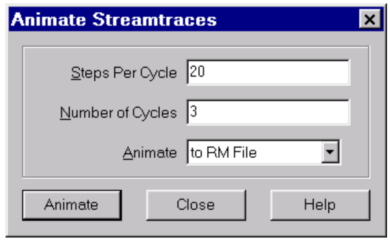


Figure 29-7. The Animate Streamtraces dialog.

29.1.7. Zone Animation

Use the Animate Zones dialog to display all or a specified subset of the zones in the current data set, one at a time.

To animate zones:

1. From the Tools menu, choose Animate, then choose Zones. The Animate Zones dialog appears as shown in Figure 29-8.

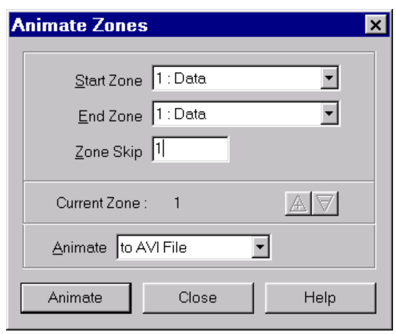


Figure 29-8. The Animate Zones dialog.

2. Specify a start zone (the first zone you want displayed), an end zone, and a zone skip in the fields provided. If you specify a start zone having a higher number than the end zone, Tecplot cycles backward from the start to the end.
3. If you want Tecplot to create a movie file containing the animation, select “to AVI file” or “to RM file” from the Animate drop-down.
4. Click Animate to run the animation automatically, or use + and - in the Current Zone area to “step through” the animation one zone at a time. (These cycle through the range of zones specified by Start Zone and End Zone; if your range is reversed, so are their actions.)

For example, suppose you have several sets of 2-D data defined at different times. At each time value, the data point positions are the same, but the variables defined at the data points are different. You could organize this body of data into one data file where each time value is allocated to a separate zone. In Tecplot, you could set up the plot style the way you want for all zones, then use Animate Zones to view the data for all time values by activating each zone, one at a time.

As an example, you might try loading the data file **`demo/2d/cylinder.plt`** and animating its zones. The cylinder data set has three zones.

29.1.8. Movie File Creation

Each of the animation dialogs offers you the option of saving the current animation as an AVI or Raster Metafile (Framer Movie File).

To save the current animation as a movie file:

1. In the appropriate Animation dialog, select “to AVI file” or “to RM file” from the Animate drop-down. When you click Animate, the appropriate Export dialog will appear.
2. Choose the image width and whether to use multiple color tables.
3. If you choose AVI as the export file type, you may specify animation speed in frames per second. Animation speed is only available for AVI files.
4. When you click OK, the Select Move File dialog will appear. Enter a file name.
5. Click OK to create the movie file. The images are written to the movie file while the Working dialog is displayed on your screen. (The screen image will not change.)

See Section 29.5.2, “Raster Metafiles Viewing in Framer,” for information on running Framer on your movie files.

29.2. Movie File Creation Manually

You can create a sequence of Raster Metafile images interactively as follows:

1. Create the first plot.
2. Select Export from the File menu and select Raster Metafile from the list.
3. Setup the desired characteristics on the Raster Metafile dialog, and click OK.
4. Select a file name when the Select Export File dialog is presented, then click Save.
5. The first image will be saved after you click Save on the Select Export File dialog, then the Record Animation File dialog will appear.
6. Create the next plot interactively.
7. When satisfied with the next image, click Record Next Image on the Record Animation File dialog.
8. Repeat steps 6 and 7 to continue adding images to the animation. When done, click Finish Animation on the Record Next Image dialog.

29.3. Movie Creation with Macros

The Tecplot macro language expands the capabilities of Tecplot's standard animation features. The macro commands allow you to do almost anything you can do interactively, and export images to movie files. You can also use loops to repeatedly rotate 3-D objects, cycle from one active zone to another, and so on, to create your movie. See Chapter 27, "Macro Commands," for detailed information regarding the Tecplot macro language.

A typical macro file for making movies has the following form:

```
#!MC 1000
 ... optional commands to set up the first image
$!EXPORTSETUP
  EXPORTFORMAT = AVI
  EXPORTFNAME = "mymovie.avi"
$!EXPORTSTART
$!LOOP 50
 ... commands to set up next image
  $!REDRAWALL
  $!EXPORTNEXTFRAME
$!ENDLOOP
$!EXPORTFINISH
```

For example, the following macro file duplicates the actions performed by the Animate Zones dialog:

```
#!MC 1000
##  Set up Export file type and file name.
$!EXPORTSETUP
  EXPORTFORMAT = AVI
  EXPORTFNAME = "C:\temp\timeseries.avi"
##  Begin Animating
$!LOOP |NUMZONES|
## Change the active zone.
## The |Loop| variable is equal to the current
## loop cycle number.
  $!ACTIVEFIELDZONES = [|Loop|]
  $!REDRAWALL
##  This series of $!IF statements ensures
##  that a new AVI file will be created when
##  the macro is started.
  $!IF |Loop| == 1
    $!EXPORTSTART
  $!ENDIF
  $!IF |Loop| != 1
    $!EXPORTNEXTFRAME
  $!ENDIF
```

```
$!ENDLOOP
$!EXPORTFINISH
```

To run this macro, do the following:

1. Create a macro file like the example above. Edit the macro file to specify a new export file name if necessary.
2. Load in a multi-zone data file.
3. Switch to 2D Cartesian plot type.
4. From the File menu, choose the Play option from the Macro sub-menu, then choose the macro file you specified in step 1.

Note: Version 8.0 macros for creating animation files (Raster Metafiles or AVI) must be revised to work with Version 9.0 or later, as shown in the example above.

29.4. Advanced Animation Techniques

There may be times when you want to include information in your animation which tells viewers about the time step, current zones, or a mapping. There are several ways this can be done.

29.4.1. Image Size

When you need a particular size for your animation image, such as 300 by 250 pixels, first edit your frame to the correct width and height. Then export only the current frame.

29.4.2. Text Changes by Attaching Text to Zones

This method works best if you are using the Zone Animation function under the Tools menu. First, create several text strings in your data file, and use the **ZN=** parameter to attach each text string to a zone or mapping. (See section 4.1.3, “Text Record,” for details on attaching text to zones.) You should have a separate text string for each zone in your data set that will be used in your animation. An example of this is:

```
ZONE T="Temp. distribution, Time = 0.5 seconds" I=51, J=51 F=POINT
.
.
.
list of variable values
.
.
.
TEXT X=70, Y=90, T="Time = 0.5 seconds", F=COURIER, CS=FRAME, H=2, ZN=1

ZONE T="Temp. distribution, Time = 1.0 seconds" I=51, J=51 F=POINT
```

```
.
.
.
```
list of variable values
```
.
.
.
TEXT X=70, Y=90, T="Time = 1.0 seconds", F=COURIER, CS=FRAME, H=2, ZN=2
```

Next, use the Animate Zones or Mappings tool to create your animation. Only the text string attached to the current zone in your animation will be visible. You can also use Tecplot's dynamic text feature (see Section 16.1.11. “Dynamic Text”) to insert a zone name into your text strings. For example:

```
ZONE T="Time = 1.0 seconds" I=51, J=51 F=POINT
.
.
.
```
list of variable values
```
.
.
.
TEXT X=70, Y=90, T="&(ZONENAME:2)", F=COURIER, CS=FRAME, H=2, ZN=2
```

29.4.3. Text Changes Using the Scatter Symbol Legend

You also can show the name of the current zone in your animation with the use of the scatter legend. Although you may not be using Scatter zone layer in your plot, the scatter legend will show the name of the current zone during the animation. See Section 13.7, “Scatter Legends,” for more information about using the scatter legend in Tecplot.

Turn on the scatter legend by selecting the Scatter Legend option on the Field menu, then make the scatter symbol invisible in the legend by changing the scatter color to white for all of your zones in the Scatter page of the Zone Style dialog (or changing to the same color as your frame background if it is not white).

29.4.4. Text Changes Using Macros

If you are using a macro to generate your animation, you can include a command to attach a text string that contains the current time step:

```
$!Loop 20
  $!Pick AddAll
    SelectText = TRUE
  $!Pick Clear
```

```
.
.
.
Commands to change the existing plot
.
.
.
$!AttachText
  Text = "Time = |Elapsed_Time| seconds."
  XYPos
  {
   X = 70
   Y = 90
  }
$!Redraw
$!If |Loop| == 1
$!ExportStart                                                    $!EndIf
$!If |Loop| == 1
$!ExportNextFrame
$!EndIf
     $!Varset |Elapsed_Time| += 0.5
$!EndLoop
```

The above loop is an example of how you could use a user-defined macro variable (discussed in the *Tecplot Reference Manual*) to insert an increasing time value into a text string. This example uses a variable called **|Elapsed_Time|**, which we consider to be the current time. One could alternatively set **|Elapsed_Time|** to be a text string that uses dynamic text:

```
$!Varset |Elapsed_Time| = "%(ZONENAME:|LOOP|)"
```

With each loop iteration, the **|LOOP|** macro variable will increment and the name for each zone will be inserted into **|Elapsed_Time|** each time through the loop. (Zone 1, 2, 3, 4, and so on.)

29.4.5. Multiple Frames Animation Simultaneously

Animation of plots in multiple frames requires the use of a macro. The **$!FRAMECONTROL PUSHTOP** command is used to switch between each frame. The following template demonstrates how this is done with a layout where each frame contains a similar plot:

```
#!MC 1000

##Set the number of images (movie frames) in the animation.
$!VARSET |NumCycles| = 10
```

```
$!EXPORTSETUP
  EXPORTFNAME   = "2frames.rm"
  EXPORTFORMAT  = RASTERMETAFILE
  BITDUMPREGION = ALLFRAMES
.
.
Insert commands to set up first frame, if necessary.
.
.
## Outer loop.
$!LOOP |NumCycles|
  ##  Inner loop cycles through each frame in the current layout.
  $!LOOP |NumFrames|
    .
    .
    Insert commands to change the plot in the current frame.
    .
    .
    ##This command pushes the topmost (active) frame to the back,
    ##making the next frame active.
    $!FrameControl PushTop
  $!EndLoop
  ##  This series of $!IF statements ensures
  ##  that a new AVI file will be created when
  ##  the macro is started.
  $!IF |Loop| == 1
    $!EXPORTSTART
  $!ENDIF
  $!IF |Loop| != 1
    $!EXPORTNEXTFRAME
  $!ENDIF
$!ENDLOOP
$!EXPORTFINISH
```

29.5. Movie File Viewing

The following tools allow you to view movie files you have created with Tecplot.

29.5.1. AVI Files

AVI format is the standard video format for Windows platforms. Below are some applications that can be used to view and/or edit AVI files:

- **Media Player:** A standard movie viewer included with Windows.
- **Xanim:** A program for playing a wide variety of video formats on UNIX X11 machines. More information is available at xanim.polter.net.

- **Premier:** A powerful tool for professional digital video editing. More information is available at www.adobe.com.

29.5.2. Raster Metafiles Viewing in Framer

Raster Metafile is a NASA-defined standard format for storing bit images and may contain one or more images. You can create a Raster Metafile in Tecplot either interactively, or using a Tecplot macro. For many types of repetitive plots (such as rotations, where each image is a slightly rotated version of the previous image), macros provide a very convenient means of simplifying Raster Metafile creation.

The Raster Metafile format is defined in the following reference:

> Taylor, N., Everton, E., Randall, D., Gates, R., and Skeens, K., NASA TM 102588, *Raster Metafile and Raster Metafile Translator.* Central Scientific Computing Complex Document G-14, NASA Langley Research Center, Hampton, VA. September, 1989.

Once you have created your Raster Metafile, you can view the resulting file with Framer. Framer is a utility program that is included with Tecplot. It allows you to view files stored in Raster Metafile format and runs independently of Tecplot. You may freely distribute Framer so that others may view your movies.

The Motif version of Framer is run from your shell prompt; the Windows version can be launched from the Tecplot program folder under the Start button. You may freely distribute the Framer executable to allow others to view your animation.

To launch Framer at a command line (shell prompt, Run command, and so forth), use the following command:

`framer` *[options] [rmfile]*

where *[rmfile]* is the name of a file containing Raster Metafile bitmaps created by Tecplot, and *[options]* is one or more of the options listed in Section 1 of Appendix B, "Utility Command Line Options."

To run Framer on UNIX type:

`framer` *[filename]*

If you do not specify a file name, Framer prompts you for one. (Under Windows you get the file dialog shown in Figure 29-9. In this dialog, you can choose to set buffering [equivalent to the **`-b`** flag] and/or multiple color maps [equivalent to the **`-m`** flag].) For a list of Framer command lines, see Appendix B.1, "Framer."

Figure 29-10 shows the main Framer window under Windows.

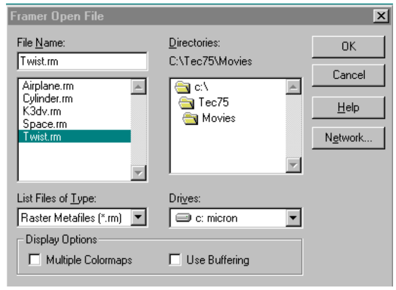


Figure 29-9. The Framer Open File dialog under Windows.

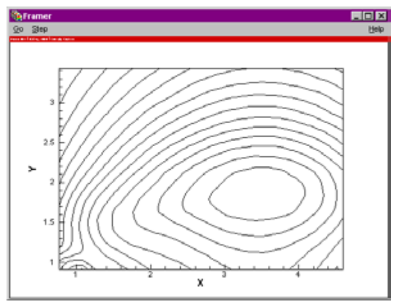


Figure 29-10. The Framer application window under Windows.

CHAPTER 30 *Customization*

Tecplot comes with a complete set of factory defaults for creating plots of all kinds: frame attributes, such as the initial frame size, background color; axis attributes, such as axis line color and range; plot attributes, such as boundary line thickness and contour plot type; and so on. You can modify virtually all of these defaults through the use of a configuration file. A Tecplot configuration file is a special type of Tecplot macro file that Tecplot reads on start up; the settings in the configuration file override Tecplot's factory defaults.

You can create a configuration file from scratch, using any ASCII text editor, or you can have Tecplot create one for you using the Save Configuration option in the File menu's Preferences sub-menu. You can also use the Preferences sub-menu and the Display Performance dialog to modify many settings interactively.

This chapter discusses creating and editing configuration files. The names of the files used will vary from platform to platform; this chapter concentrates on UNIX and Windows files.

30.1. Configuration Files

Tecplot looks for configuration files in one of three places: the current working directory, the user's home directory, and the Tecplot home directory. In the current working directory, Tecplot searches for a file named **`tecplot.cfg`**. If it finds it, it uses the settings in that file to override the factory defaults. If not, Tecplot searches your home directory for a file named **`.tecplot.cfg`** (**`tecplot.cfg`** in Windows). If Tecplot finds this file, it uses the settings in that file to override factory defaults. If there is no **`.tecplot.cfg`** file in your home directory, Tecplot looks in the Tecplot home directory for a file named **`tecplot.cfg`**, and uses the settings in that file to override the factory defaults. (Under Windows, your home directory is determined by the two environment variables **`HOMEDRIVE`** and **`HOMEPATH`**. If they are not set, Tecplot skips your home directory.)

You can save configuration files with any valid file name. To have Tecplot use them, you must either rename them to have one of the names Tecplot searches for by default, or, more commonly, use the "**`-c`**" command line option when starting Tecplot. For example, the following command starts Tecplot using the configuration settings in the file **`mydefs.cfg`**:

```
tecplot -c mydefs.cfg
```

System administrators can use the **`tecplot.cfg`** file in the Tecplot home directory to set system-wide defaults, then others on the system can copy the system configuration file to their own home directories and make any desired changes. The settings in your local configuration file are used instead of the settings in the system configuration file. A configuration file needs to include only those options for which you want to override defaults.

Tecplot under Motif has a second type of configuration file, an X11 resource file (app-defaults file) that controls the appearance of the Tecplot application and its dialogs. Most users do not need to concern themselves with this file; nothing in the resource file has any affect on the plots you create with Tecplot, either on screen or on paper. However, if you are an experienced Motif and X11 user, you may want to modify some of the resources to improve the appearance of Tecplot's windows and dialogs on your display. Section 30.4, "Interface Configuration Under UNIX," explains how to do this.

30.1.1. Configuration File Creation

The simplest way to create a configuration file is to change the appropriate settings using the Tecplot interface, then save the configuration. For example, suppose you want to have your paper orientation default to portrait and have your default export format be Encapsulated PostScript (EPS). You can modify the settings using the appropriate Tecplot dialogs, then save the configuration file.

To save a Tecplot configuration file:

1. Change settings as desired using Tecplot dialogs.
2. From the File menu, choose Preferences, then choose Save Configuration. The Save Configuration dialog appears as shown in Figure 30-1.

Figure 30-1. The Save Configuration dialog.

3. Check the file name listed. If the file name is correct, click OK. If you want to save to a different file, click Change File Name and specify a new file name.

Here is the configuration file resulting from the changes described above:

```
#!MC 1000
$!PAPER
  ORIENTPORTRAIT = YES
$!EXPORTSETUP
  EXPORTFORMAT = EPS
$!FRAMELAYOUT
   XYPOS
    {
    X = 1
    Y = 0.25
    }
  WIDTH = 9
  HEIGHT = 8
```

This configuration file specifies portrait orientation, and sets the Export format to EPS, just as desired. The only other settings saved are the default frame layout settings.

You can, however, obtain a configuration file that includes most of the factory defaults as follows:

1. From the File menu, choose Preferences, then choose Save Configuration. The Save Configuration dialog appears.
2. Select the check box labeled Include Factory Defaults.
3. Check the file name listed. If the file name is correct, click OK. If you want to save to a different file, click Change File Name and specify a new file name.

The created file contains factory defaults for the following types of Tecplot settings:

- Interface details.
- RGB color assignments for Tecplot's basic colors.
- Default paper layout.
- Print and export setup information.

If you modify any setting from these four types interactively and then save your configuration, the modifications are saved. However, modifications to other types of settings will not be saved.

You are not limited, however, to changing merely those settings which appear in the saved configuration file. Most settings which can be modified by one of Tecplot's SetValue macro commands can be changed in the configuration file. These other settings must be changed, however, by editing the configuration file.The simplest way to do this is to create a layout or macro with the settings you want, then copy and paste the appropriate SetValue commands into your configuration file. (Once you become more familiar with the macro language, it may be simpler to type in the appropriate SetValue command directly.) See the *Tecplot Reference Manual* for complete details on the SetValue and other macro commands.

For example, suppose you want your 2-D axes to appear cyan. You can add this preference to your configuration file as follows:

1. Using the Tecplot interface, create a 2-D plot with cyan axes, either recording your steps as a macro, or saving the result as a Tecplot layout.
2. Edit the resulting macro or layout, scanning for the lines that set the 2-D axis colors. The following example shows the commands that specify the X- and Y-axis details in a layout of a 2-D plot with cyan axes:

```
$!TWODAXIS
  XDETAIL
    {
    RANGEMIN = -2.99985003471
    RANGEMAX = 15.001799985
    GRSPACING = 5
    AXISCOLOR = CYAN
    }
  YDETAIL
    {
    RANGEMIN = -2.99985003471
    RANGEMAX = 13.4283224277
    GRSPACING = 2
    AXISCOLOR = CYAN
    }
```

3. Discard everything but the lines that actually set the color:

```
$!TWODAXIS
  XDETAIL
    {
    AXISCOLOR = CYAN
    }
  YDETAIL
    {
    AXISCOLOR = CYAN
    }
```

4. Paste the resulting lines into your configuration file.

30.1.2. Plot Default Setting

A single **$!FIELD** command can be included to set plot defaults. The command cannot specify a zone, and is not effective for values set dynamically by Tecplot, such as Mesh Color. In the example below, the default contour type is Flood, scatter symbol shape is Delta, and scatter size is 1.8.

```
$!FIELD
```

```
    CONTOUR
      {
      CONTOURTYPE = FLOOD
      }
    SCATTER
      {
      FRAMESIZE = 1.8
      SYMBOLSHAPE
        {
        GEOMSHAPE = DEL
        }
      }
```

In the same way, a single **$!LINEMAP** command can be added for line mapping defaults. In the example below, XY and Polar Line mappings will have a dashed line pattern, and symbols will be filled circles.

```
$!LINEMAP
   LINES
     {
     LINEPATTERN = DASHED
     }
   SYMBOLS
       {
       SYMBOLSHAPE
          {
          GEOMSHAPE = CIRCLE
          }
       FILLMODE = USELINECOLOR
       }
```

30.1.3. Interface Configuration

The many members of the **$!INTERFACE** macro help you configure Tecplot's user interface and graphics drawing capabilities. Although some of these commands can be executed in any Tecplot macro the best place to put these is in the Tecplot configuration file, **tecplot.cfg**. Below are a few examples. Refer to the *Tecplot Reference Manual* for a complete listing.

30.1.3.1. Interface Configuration Options.

```
$!INTERFACE MOUSEACTIONS = {MIDDLEBUTTON {SIMPLEDRAG=ZOOMDATA}}
```

Specify the action of the middle mouse button click and drag. Several other options for the middle and right mouse buttons are listed in the *Tecplot Reference Manual*. These commands can only be executed from the Tecplot configuration file.

$!INTERFACE UNIXHELPBROWSERCMD = *string*

Specify the command to execute to launch the browser on UNIX systems for viewing the Help files. This command can only be executed from the Tecplot configuration file.

$!INTERFACE SHOWWAITDIALOGS = (YES, NO)

You can disable the launch and display of all Wait dialogs by setting this to **NO**. (Wait dialogs are launched during long operations and give you the ability to cancel the operation.) This is useful on some Linux systems where transient dialogs do not drop properly, leaving a gray box that obscures part of Tecplot's drawing area.

$!INTERFACE USESTROKEFONTSONSCREEN = (YES, NO)

If set to **YES** all text drawn in the work area will be drawn using Tecplot's internal stroke fonts. If set to **NO** the native True Type fonts will be used instead. This option has no effect under UNIX.

$!INTERFACE USESTROKEFONTSFOR3DTEXT = (YES, NO)

If set to **YES** all 3-D text drawn in the work area will be drawn using Tecplot's internal stroke fonts. 3-D text consists of ASCII scatter symbols, and node and cell labels when the current plot type is 3D Cartesian. For 3-D text, this setting overrides the setting of **USESTROKE-FONTSONSCREEN**. If set to **NO** the native True Type fonts will be used instead. This option has no effect under UNIX.

30.1.3.2. OpenGL-Specific Configuration Options. Several options are available to further tune Tecplot to operate with the OpenGL capabilities of your platform. To assign values to these parameters you must use the **$!INTERFACE OPENGLCONFIG**. A complete list of these options is given in the *Tecplot Reference Manual*.

```
$!INTERFACE OPENGLCONFIG
  { SCREENRENDERING { DOEXTRADRAWFORLASTPIXEL = (YES, NO)}}
```

Some OpenGL implementations use an optimization for line drawing that omits the last pixel in the line. Set this to **YES** to change all line drawing to force the last pixel to be drawn. This setting applies only to drawing on the screen.

```
$!INTERFACE OPENGLCONFIG
  { SCREENRENDERING { STIPPLEALLLINES = (ALL, CRITICAL, NONE)}}
```

Set to **ALL** to make all lines drawn using stippling. Set to **CRITICAL** to use stippling for stroke and user-defined fonts. Set to **NONE** to disable stippling. This setting applies only to drawing on the screen.

$!INTERFACE OPENGLCONFIG

```
{ IMAGERENDERING { DOEXTRADRAWFORLASTPIXEL = (YES, NO)}}
```

Some OpenGL implementations use an optimization for line drawing that omits the last pixel in the line. Set this to **YES** to change all line drawing to force the last pixel to be drawn. This setting applies only to exporting images from Tecplot.

```
$!INTERFACE OPENGLCONFIG
  { IMAGERENDERING { STIPPLEALLLINES = (ALL, CRITICAL, NONE)}}
```

Set to **ALL** to make all lines drawn using stippling. Set to **CRITICAL** to use stippling for stroke and user-defined fonts. Set to **NONE** to disable stippling. This setting applies exporting images from Tecplot.

30.1.4. Default File Name Extensions

The default extensions for file names in file input-output dialogs can also be changed in the configuration file. These settings are changed via the **FNAMEFILTER** sub-command in the **$!FILECONFIG** macro command.

- **COLORMAPFILE:** Specifies the default extension for color map files.
- **INPUTDATAFILE:** Specifies the default extension for input data files.
- **OUTPUTASCIIDATAFILE:** Specifies the default extension for ASCII output files.
- **OUTPUTBINARYDATAFILE:** Specifies the default extension for binary output files.
- **INPUTLAYOUTFILE:** Specifies the default extension for input layout and layout package files.
- **OUTPUTLAYOUTFILE:** Specifies the default extension for output layout files.
- **OUTPUTLAYOUTPACKAGEFILE:** Specifies the default extension for output layout package files.
- **STYLEFILE:** Specifies the default extension for stylesheet files.
- **MACROFILE:** Specifies the default extension for macro files.
- **EQUATIONFILE:** Specifies the default extension for equation files.

For example, to change the default extension for input data files to be **.tbl** use:

```
$!FILECONFIG
  FNAMEFILTER
  {
    INPUTDATAFILE = "*.tbl"
  }
```

30.1.5. Default Temporary Directory

Tecplot writes out a number of temporary files. To tell Tecplot where to place these files, put the following macro command in the **tecplot.cfg** file:

```
$!FILECONFIG
```

```
TEMPFILEPATH = "tempfilepath"
```

where *tempfilepath* is the new path. The default path is system dependent.

30.2. Interactive Customization

Using the Preferences sub-menu under the File menu, you can interactively control the colors used throughout Tecplot, the size options available in most Tecplot dialogs, and several miscellaneous parameters.

30.2.1. Color Preferences Dialog

To change the RGB values of Tecplot's basic colors, use the Color Preferences dialog, shown in Figure 30-2. This dialog is displayed by selecting Preferences/Colors... from Tecplot's File menu.

To change a color, click on it in the palette and alter its RGB values with the sliders. As you move the sliders, the box in the upper right corner of the dialog shows the color as currently specified. You may alter multiple colors by selecting those colors and changing their RGB values. Choosing Reset Selected Color or Reset All Colors will restore the default RGB values. All color changes take effect when you click OK.

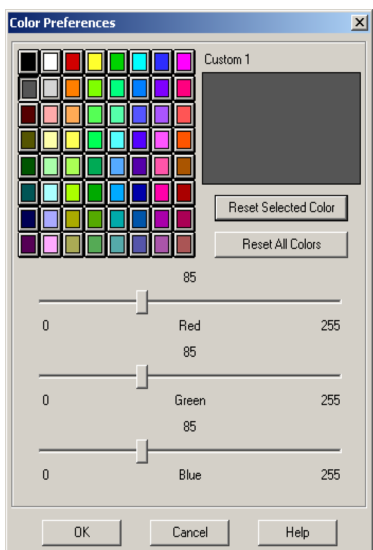


Figure 30-2. The Color Preferences dialog.

30.2.2. Size Preferences Dialog

To set size options, use the Size Preferences dialog, shown in Figure 30-2. This dialog is displayed by selecting Preferences/Sizes... from Tecplot's File menu.

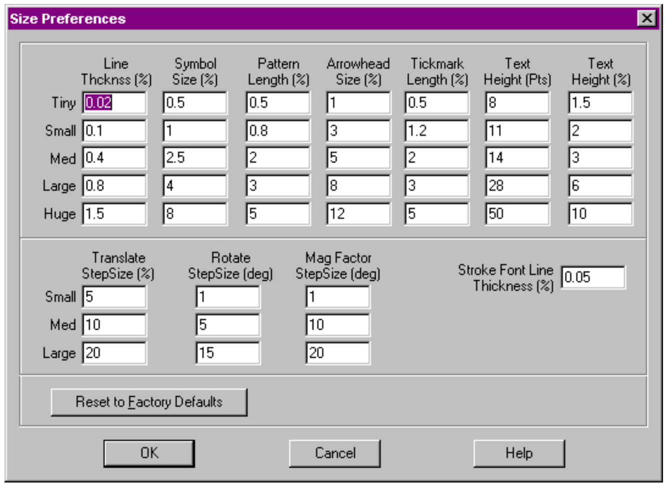


Figure 30-3. The Size Preferences dialog.

These options determine the choices available in drop-down such as Line Thickness that occur throughout the interface.

You can control the following sets of sizes:

- Line thickness.
- Symbol size.
- Pattern length.
- Arrowhead size.
- Tick mark length.
- Text height (in both points and frame units).
- Translate step size.
- Rotate step size.
- Magnification step size.
- Stroke font line thickness.

30.3. Performance Dialog

30.3.1. Best Practices For Rendering Performance

The factory settings in the Performance dialog are designed for moderately sized data and may need adjusting to optimize Tecplot's rendering performance. There are many combinations of “Plot Approximation” and “Graphics Cache” modes however two combinations meet most user's needs.

1. For moderate to large size data turn on approximation plots and set the “Plot Approximation” mode to “Automatic” when dealing with one frame or “Non-Current Frames Always Approximated” for multiple frames. Also set the “Graphics Cache” mode to “Cache All Graphics”.
2. For large to very large size data (usually determined when option #1 is not providing responsive interactive performance) set the “Plot Approximation” mode to “All Frames Always Approximated” and the “Graphics Cache” mode to “Cache only Lightweight Graphics Objects”.

With either case, adjust the “Approximate Plot as % of Full Plot” value to give an acceptable balance between good interactive performance and sufficient detail.

30.3.2. Rendering

Plot Approximation

Use Auto Redraw: When selected, Tecplot will automatically redraw the plot whenever style or data changes. Some users prefer to turn this option off while setting multiple style settings and then manually pressing Tecplot's Redraw or Redraw All button on the sidebar to see a full plot. Please note that even if Tecplot is in the middle of an auto-redraw it can be interrupted with a mouse click or key press. Note that this toggle is directly available on the Tecplot sidebar in the Performance menu.

Approximate Plots for Better Speed: When selected, Tecplot builds an approximate representations of the plot. The degree of detail of the approximation is controlled by the other Plot Approximation settings. Note that this toggle is directly available on the Tecplot sidebar in the Performance menu.

Tecplot has three approximate modes: automatic, non-current frames always approximated, and all frames always approximated.

Automatic: When selected, and if the number of data points is above the point threshold, (see below) Tecplot will render the approximate plot for style, data, and interactive view changes followed immediately by the full plot. This option provides for good interactive performance with the final plot always displayed in the full representation.

Non-Current Frames Always Approximated: When only one frame exists this option is exactly like automatic mode. If more than one frame exists the current frame is set to automatic mode while the other frames are always approximated.

All Frames Always Approximated: When selected, and if the number of data points is above the point threshold, Tecplot will only render the approximate plot in any frame. To see the full representation press the Redraw or Redraw All button on the sidebar.

Point Threshold for Automatic Approximation: This value controls when Tecplot will consider using approximate plots (if approximate plots are turned on). The value to use is highly dependent on the computer's hardware capabilities. A computer with lots of RAM and a high performance graphics card with good OpenGL hardware acceleration can have a much larger number than the factory default.

Approximate Plot as % of Full Plot: This value controls the percentage of geometric detail represented by the approximate plot. The larger the percentage the more closely the approximation represents the original plot but the interactive performance is reduced. This number should be adjusted until there is a balance between good interactive performance and sufficient detail. Usually the percentage should be set to be less than or equal to 50. If values larger than 50% are needed to provide sufficient detail consider not using approximate plots at all.

Graphics Cache

Tecplot uses OpenGL to render plots. OpenGL provides for the ability to cache graphic instructions for rendering and can re-render the cached graphics much faster than having Tecplot send the instructions again. This is particularly true for interactive manipulation of a plot. However this performance potential comes at the cost of using more memory. If the memory need is too high the overall performance could be less. Tecplot has three graphics cache modes: cache all graphics, cache only lightweight graphics objects, and do not cache graphics.

Cache All Graphics: When selected, Tecplot assumes there is enough memory to generate the graphics cache. Assuming this is true Tecplot's rendering performance will be optimal for interactive manipulation of plots.

Cache Only Lightweight Graphics Objects: Lightweight objects include approximate plots and some other minor items but do not include full plots. For memory constrained problems this is a good setting. Consider using this option in conjunction with the “Plot Approximation” mode set to “All Frames Always Approximated”.

Do Not Cache Graphics: When memory constraints are very limited consider using this option. If you intend on interacting with the plot also consider setting the “Plot Approximation” mode set to “All Frames Always Approximated”.

High Quality Font Usage

The Windows and Linux 32 bit versions of Tecplot support high quality TrueType font usage. Windows platforms are shipped with the TrueType fonts used by Tecplot. On the Linux platforms they have to be obtained and installed (see the Release Notes). Tecplot has three high quality font modes: when possible, for large characters only, and never.

When Possible: When selected, Tecplot uses any of its TrueType fonts that are available for any size text. This produces the best rendering quality however it is somewhat slower for large amounts of text.

For Large Characters Only: When selected, Tecplot uses the TrueType fonts only for larger characters. Small characters will use Tecplot's built-in stroke fonts. This is a good blend of quality and performance.

Never: When selected, Tecplot never uses TrueType fonts. This is the default mode for platforms other than Windows and Linux 32 bit.

Image Export Options

Some graphics card hardware does not support off-screen rendering needed for exporting images. In addition, most graphic hardware is slower at producing images off-screen than on-screen. To accommodate a variety of graphic hardware Tecplot provides two image export modes: safe, and fast.

Safe (Render Image Off-Screen): When selected, Tecplot will render all exported images off-screen. This allows images to be created that are not bound by the physical size and state of the Tecplot drawing area.

Fast (Use On-Screen Image): When selected, Tecplot will grab the pixels from the physical Tecplot drawing area. Any rendering damage, such as occluding windows or partially drawn images will become part of the exported image as it simply takes the pixels directly from the physical Tecplot drawing area. In addition the image size is bound by the physical size of the Tecplot drawing area.

30.3.3. Miscellaneous

Variable Derivation

When Tecplot needs to create a nodal variable from a cell centered one it uses a prescribed derivation method. Tecplot provides two such derivation methods: fast and accurate.

Fast (Linear): When selected, Tecplot uses linear interpolation to derive a nodal variable from a cell centered one.

Accurate (Laplacian): When selected, Tecplot uses Laplacian interpolation to derive a nodal variable from a cell centered one.

Data I/O

Use Memory Mapped I/O: When selected, Tecplot will use system level memory mapping functions to map Tecplot variables directly over block data in a binary data file or layout package file. The advantage of mapping variable data is that Tecplot will only load the variable when it is first used. In addition the mapped variable data can be shared between other Tecplot sessions running on the same machine. Memory mapped I/O is most useful when there is a large number of data points to load from a file and they are not all being used by Tecplot at the same time. Only variable data that is in a binary block format (the default for plt files generated by Tecplot) can be memory mapped.

Status Information

Sometimes the updating the status line slows down processing (mainly when remotely displaying Tecplot on X terminals) or is just annoying so Tecplot provides several toggles to turn on or off the status line and tool tips. In addition you can control what kind of information is shown on the status line.

30.4. Interface Configuration Under UNIX

In UNIX, the style of the graphical user interface for Tecplot is configured for the most part by a resource file called **`Tecplot100`** which resides in the **`app-defaults`** sub-directory below the Tecplot home directory. If you edit this file the changes will affect all users. Alternatively, you can add entries to a file called **`.Xdefaults`** which resides in your own **`$HOME`** directory if you want the changes to apply only to your own execution of Tecplot. If the file **`.Xdefaults`** does not already exist in your home directory, you can create one.

30.4.1. Default Size of Tecplot

The resource lines that affect the default Tecplot process window size are:

```
*Tecplot.main_dialog.width: 900
*Tecplot.main_dialog.height: 720
```

Changing either the value 900 or the value 720 will change the default size of the Tecplot process window.

30.4.2. Look and Feel

Tecplot now ships with two options, and may default to either depending on the platform used. In order to change from option to the other, refer to the Tecplot100 file as mentioned above.

30.5. Custom Character and Symbol Definition

When Tecplot starts up, it reads the font file (“**`tecplot.fnt`**”). This file contains information that defines the appearance of text characters on the screen. Tecplot defines and draws characters on the screen as a set of straight lines called strokes. These stroked characters approximate the appearance of characters for the screen.

The font file is an ASCII file that can be edited using an ASCII text editor. You can modify the shape, size, and resolution of existing stroke-font characters or add completely new ones. In PostScript print files, text characters are generated using PostScript defined fonts, not the stroked fonts. If you are using the Windows version of Tecplot and the Windows print drivers are active, then all text except text using the User-Defined fonts is serviced by the Windows printer driver. However, HP-GL and HP-GL/2 print files use the stroked fonts, and the text characters in bitmap export files are also in stroked fonts (since they are generated from the screen). The inter-character spacing in all output files is determined by the character-width definitions in the font file. When using PostScript print files or the Windows print drivers, changing the font commands affects only the character shape for User-Defined fonts and the character spacing for all fonts.

The Font File is structured as follows:

```
#!FF 4
 CharCellHeight
 Stroke command set for Helvetica Font
 Stroke command set for Greek Font
 Stroke command set for Math Font
 Stroke command set for User-Defined Font
 Stroke command set for Times Font
 Stroke command set for Times Italic Font
 Stroke command set for Courier Font
```

The file type and version are on the first line ("FF" means Font File). *CharCellHeight* is the interline spacing (that is, the height of a capital M plus some vertical space) in the units of a two-dimensional coordinate system used to define the stroke-font characters. The baseline of the characters is at zero. Before Tecplot uses the character definitions, they are normalized by the character cell height.

Following the character cell height, there are seven sets of stroke commands, one set for each font as shown above. Each stroke command set consists of definitions for the characters in the font. Each font has a base set of 96 characters (character indices 32 to 127). Some fonts also include an extended set of characters (character indices 160 to 255). The extended characters are needed to complete the character sets for most of the common European languages.

All seven stroke command sets must be present, and each must have at least one character defined. Each stroke command set begins with the definition for a space (character index 32). After that, characters within a stroke command set may be defined in any order. If a character is not defined in the Font File, it is drawn as a blank.

Each character in a stroke command set is defined as follows:

CharIndex NumCommands CharWidth

```
 Command1
 Command2
 Command3
 .
 .
 .
 CommandNumCommands
```

CharIndex is the character index which ranges from 32 to 127 and 160 to 255 for each font (see Figure 16-5 for the matching of the character index to the English, Greek, Math, and standard User-Defined font characters), *NumCommands* is the number of stroke commands defining the character that follows, and *CharWidth* is the character width, which determines the spacing of the characters.

A command may be in one of the following forms:

- **m** *x y*.
- **d** *x y*.
- **mr** *dx dy*.
- **dr** *dx dy*.

A command that begins with an **`m`** is a move command. A command that begins with a **`d`** is a draw command. Commands **`mr`** and **`dr`** are relative move and relative draw commands. The *x* and *y* are the absolute coordinates within the character cell. The *dx* and *dy* are the relative coordinates with respect to the previous location (increments from the position attained by the previous command). All coordinates are specified as integers. Figure 30-4 shows an example of a character cell and the commands used to define the lowercase letter "y." The height of the character cell is 48.

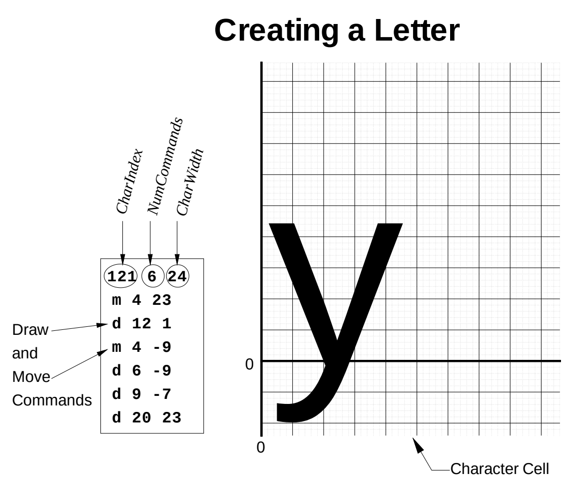


Figure 30-4. Defining a user-defined character.

Figure 30-5 shows a symbol being defined. Symbols should be centered about (0, 0) so that they are centered about the point they mark. The font file included with Tecplot contains many User-Defined font stroke commands. Most of these are for creating extra plotting symbols, accessible when you use the Symbol Type "Other," enter an ASCII character, and specify the User-Defined font.

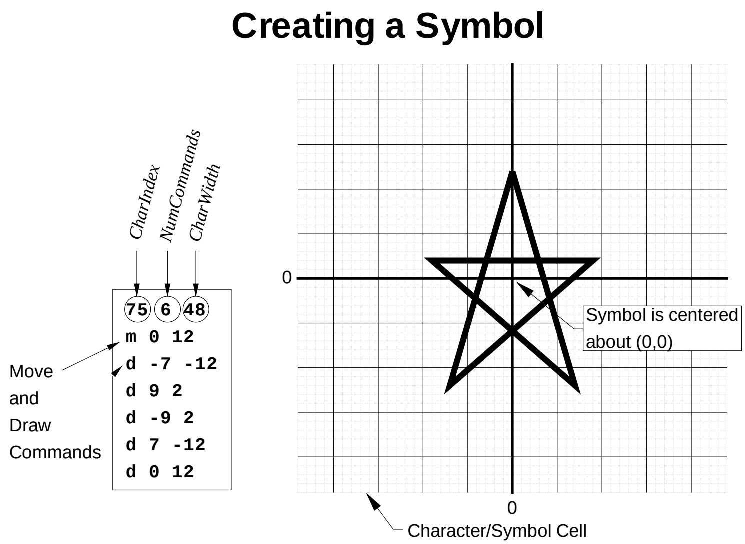


Figure 30-5. Defining a user-defined plotting symbol.

30.6. Tecplot.phy File Location Configuration

Whenever Tecplot starts, it tries to load a **`tecplot.phy`** file. This file contains information useful for running macros in batch mode (see Chapter 28, "Batch Processing," for more information) and also the name of the last layout file used in Tecplot. Whenever Tecplot exits, it writes out a new **`tecplot.phy`** file.

The place Tecplot looks for the **`tecplot.phy`** file is based on the following search:

1. Tecplot checks the environment variable **`TECPHYFILE`**. If this variable is set, Tecplot uses the value of this variable as the name of the **`tecplot.phy`** file. By default, this variable is not set. You can set this environment variable to control the location and name of the **`tecplot.phy`** file on a user-by-user basis.
2. (Windows Only) Tecplot checks the Windows registry for the key **`HKEY_LOCAL_MACHINE\SOFTWARE\Amtec Engineering, Inc.\Tecplot 10.0`**. If the value **`PhyFile`** is set under this key, then it is used as the name of the **`tecplot.phy`** file. This value is set by the installation program. You can use the command **`regedit`** from the Start Menu's Run option to edit the registry if you want to change or delete this key.

3. Tecplot uses the file called **`tecplot.phy`** in the directory where Tecplot is started. Note that this is the default behavior under UNIX.

Thus, using the default installation, Windows versions of Tecplot will write a **`tecplot.phy`** to one specific location (usually the Tecplot home directory), and UNIX versions will always use a **`tecplot.phy`** file in the directory where Tecplot is started.

The Windows version can be made to act like the UNIX version by deleting the value **`PhyFile`** from **`HKEY_LOCAL_MACHINE\SOFTWARE\Amtec Engineering, Inc.\Tecplot 10.0`** in the Windows registry with **`regedit`**.

Under both Windows and UNIX, the environment variable **`TECPHYFILE`** can be set to override this behavior.

CHAPTER 31 Add-Ons

Add-ons are a way to extend the basic functionality of Tecplot. They are executable modules designed to perform specific tasks. Amtec has produced a number of add-ons that load data in a variety of formats, allow advanced editing, or extend Tecplot's capabilities. By using the Tecplot *Add-on Developer's Kit* (ADK), users can create their own add-ons to generate plots, transform or analyze data, or perform a broad range of specialized tasks.

31.1. Tecplot Add-Ons

Add-ons are external programs that attach themselves to Tecplot and are accessed through the Tecplot interface. When Tecplot is started, it goes through various initialization phases, including the processing of the **`tecplot.cfg`** file, the loading of the Tecplot stroke font file (**`tecplot.fnt`**) and the initialization of the graphics. After all of this has been completed, Tecplot begins to look for add-ons.

A number of the add-ons currently used by Tecplot are data file loaders or converters, which allow users to read non-Tecplot data files. These are:

- **loadplot3d:** A PLOT3D data loader.
- **loadxls:** An Excel file loader (Window).
- **loadss:** A spreadsheet file data loader.
- **gridgen:** A GridGen file data loader.
- **loaddxf:** A Data eXchange Format (DFX) data loader.
- **loadhdf:** A Hierarchical Data Format (HDF) data loader.
- **loaddem:** A Digital Elevation Map (DEM) data loader.
- **loadimg:** An add-on that loads bitmaps as a group of geometries.
- **loadcgns:** A CFD General Notation System (CGNS) data loader.
- **loadfluent:** A Fluent data loader for **`.cas`** and **`.dat`** files (versions 5 to 6.1).

These show up under the Import option of the File menu. The primary difference between loaders and converters are that loaders bring up more complex dialogs than do converters, which only bring up dialogs based on Tecplot's standard Load Data File(s) option.

Tecplot also uses add-ons for extended curve-fits with XY Line plots. They may be accessed by selecting the Curve Type's Extended option, located on the Mapping Style dialog.

Curve-fit add-ons include:

- **crvstineinterp:** A curve-fit using Stineman interpolation.
- **crvgen:** A curve fit where users define the equation.

Other add-ons may be accessed through the Tools drop-down on Tecplot's menu bar. They include:

- **advqet:** This calls up the Advanced Quick Edit dialog.
- **crsfez:** Allows extractions from finite-element sub-zones.
- **cstream:** Circle stream (this allows users to place a rake of streamtraces in a circular pattern).
- **statechange:** View all Tecplot state change information (used primarily for add-on development).
- **viewbin:** Binary data file viewer.

31.1.1. Advanced Quick Edit

The Advanced Quick Edit dialog is shown in Figure 31-1.

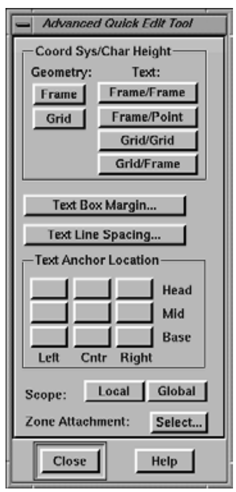


Figure 31-1. The Advanced Quick Edit dialog.

Selecting the Advanced Quick Edit Tool option from Tecplot's Tools menu allows you to make rapid changes to text and geometries selected in the current frame. This tool allows operations that cannot be performed with the standard Quick Edit Tool accessible via the Quick Edit option.

Controls on the Advanced Quick Edit Tool dialog are sensitive to user input only when one or more text and/or geometries are selected. Some controls are specific to either text or geometries, while others apply to both. If the selected objects are a mix of text and geometries, the controls that apply only to geometries will only affect the geometries you have selected. Similarly, controls that apply specifically to text will only affect text, even if the selected objects are a mix of text and geometries.

For more information, access Tecplot's Help.

31.1.2. Circle Stream

The Circle Stream add-on is used to place a "rake" of streamtraces starting from a selected circle geometry. The Circle Stream dialog is shown in Figure 31-2.

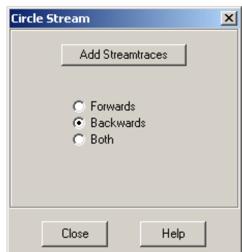


Figure 31-2. The Circle Stream dialog.

For more information, and to see an example using Circle Stream, access Tecplot's Help.

31.1.3. Create Multiple Frames

Use the Create Multiple Frames add-on to make a set of new frames with uniform size and spacing within the current frame. The total number of new frames will be the product of the frames across and the frames down.

The Create Multiple Frames dialog is shown in Figure 31-3.

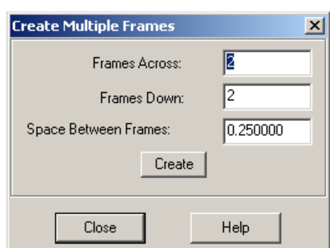


Figure 31-3. The Create Multiple Frames dialog.

For more information, and to see an example using Create Multiple Frames, access Tecplot's Help.

31.1.4. Create Finite-Element Sub-Zone

Selecting the Create SubFEZone option from Tecplot's Tools menu allows you to create a finite-element zone containing all elements that are completely visible in the current frame. This option is only available for 2D Cartesian plot types, and all elements must be of the same type, either triangular or quadrilateral. The Create FE Sub-Zone dialog is shown in Figure 31-4.

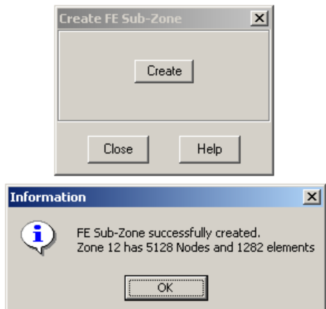


Figure 31-4. The Create FE Sub-Zone dialog and its resulting Information dialog.

For more information, and to see an example using Create SubFEZone, access Tecplot's Help.

31.1.5. Export DXF

The Export DXF File dialog is shown in Figure 31-5.The DXF Export add-on exports data in

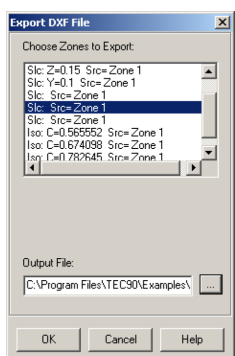


Figure 31-5. The Export DXF File dialog.

DXF (drawing interchange) format. If the data type is finite-element and the element type is triangular or quadrilateral:

- If the plot type is 3D Cartesian, a DXF 3DFACE entity is created for each triangle or quadrilateral.
- If the plot type is not 3D Cartesian, a DXF POLYLINE entity is created for each triangle or quadrilateral.
- If the element type is not triangular or quadrilateral, then no DXF entities are exported.

If the data type is I-, IJ-, or IJK-ordered:

- If the plot type is 3D Cartesian and JMax = 1 and KMax = 1, then a DXF POLYLINE entity is exported for all points in the selected zone(s).
- If the plot type is 3D Cartesian and JMax or KMax is greater than 1, then DXF 3DFACE entities are exported for all points in the selected zone(s).
- If the plot type is not 3D Cartesian, then a DXF POLYLINE entity is exported connecting all points in the selected zone(s).

For more information, access Tecplot's Help.

31.1.6. Extend Macro

The Extend Macro add-on extends Tecplot's macro language with macro commands. The commands include:

```
$!ADDONCOMMAND ADDONID='extendmcr' COMMAND='QUERY.ZONENAMEBYNUM nnn VVV'
```

Get the string for zone *nnn* and assign to variable *VVV*.

```
$!ADDONCOMMAND ADDONID='extendmcr' COMMAND='QUERY.VARNAMEBYNUM nnn VVV'
```

Get the string for variable *nnn* and assign to variable *VVV*.

```
$!ADDONCOMMAND ADDONID='extendmcr' COMMAND='QUERY.ZONENUMBYNAME
"zonename" VVV'
```

Get the number of zone named *zonename* and assign to variable *VVV*.

```
$!ADDONCOMMAND ADDONID='extendmcr' COMMAND='QUERY.VARNUMBYASSIGNMENT
assignment VVV'
```

Get the number of variable by assignment and assign to variable *VVV*.

These are the assignment options:

- **X:** Variable assigned to the X-axis.
- **Y:** Variable assigned to the Y-axis.
- **Z:** Variable assigned to the Z-axis.
- **U:** Variable assigned to be the U-vector component.
- **V:** Variable assigned to be the V-vector component.
- **W:** Variable assigned to be the W-vector component.
- **C:** Variable assigned to contours.
- **S:** Variable assigned to scatter sizing.
- **B:** Variable assigned to the first constraint for value-blanking.

$!ADDONCOMMAND ADDONID='extendmcr' COMMAND='QUERY.DATASETTITLE VVV'

Get the string for the data set title and assign to variable *VVV*.

$!ADDONCOMMAND ADDONID='extendmcr' COMMAND='STRING.LENGTH StrSource VVV'

Get the length of string **StrSource** and assign to variable *VVV*.

$!ADDONCOMMAND ADDONID='extendmcr' COMMAND='STRING.FINDPATTERN StrSource StrPattern VVV'

Get the sub-string from **StrSource** starting at pattern **StrPattern** and going to the end of **StrSource**. Returns "**NOTFOUND**" if not found.

$!ADDONCOMMAND ADDONID='extendmcr' COMMAND='STRING.SUBSTRING StrSource start end VVV'

Get the sub-string from **StrSource** starting at position start and ending at position end. Put the result in *VVV*.

$!ADDONCOMMAND ADDONID='extendmcr' COMMAND='QUERY.ACTIVEZONES VVV'

Get the set of active zones and put the result in *VVV*.

Note: The set string does not include any blank spaces. If zones 2, 4, 6, 7 and 8 are active, *VVV* would have the string "2, 4, 6-8."

$!ADDONCOMMAND ADDONID='extendmcr' COMMAND='QUERY.ISZONEACTIVE ZZZ VVV'

Test to see if zone *ZZZ* is currently active. If so, *VVV* is set to "**YES**," otherwise it is set to "**NO**."

More information on the Extend Macro add-on is available via Tecplot's Help.

31.1.7. Extrude

The Extrude Options dialog is shown in Figure 31-6.The Extrude add-on creates a 3-D volume

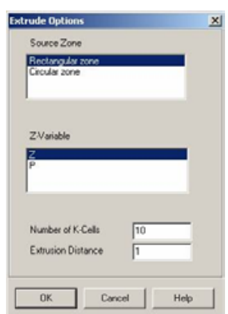


Figure 31-6. The Extrude Options dialog.

or surface zone by duplicating the source zone and translating it in the Z-direction until the specified number of K-cells are created. If the source zone is a surface, a volume zone will be created. If the source zone is a line, a surface zone will be created.

An example of using Extrude would be to create a cylindrical (open ended) surface by extruding a circular line. For simplicity, the circular line will be created as a sub-zone of a 2-D, Tecplot-generated, circular zone.

To do this, perform the following steps:

1. Generate a circular zone by selecting Circular from the Data menu's Create Zone submenu. The Create Circular Zone dialog appears. Set K to 1 and click Create.
2. Select the Tool's menu Extrude option. The Extrude dialog appears. Set the Extrusion Distance to 5, use the defaults for the other fields, then click OK.
3. Answer Yes when asked if you want to create the Z-variable. The result is shown in Figure 31-7.

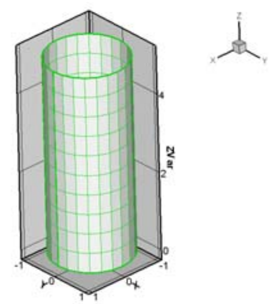


Figure 31-7. An example of using the Extrude Options dialog.

For more information on Extrude, including another example of using Extrude, access Tecplot's Help.

31.1.8. General Text Loader

The General Text Loader add-on allows you to read ASCII text data files in a variety of formats. You can specify variable and data set title information or indicate specific places in your data file to read them from. Instruction settings for reading a type of file can be saved and restored so they do not have to be entered again each time a new file of the same type is loaded.

The General Text Loader dialog is shown in Figure 31-8. For more information, and to see an example using General Text Loader, access Tecplot's Help.

31.1.9. Prism-Grid

The Prism-Grid dialog is shown in Figure 31-9. The Prism-Grid add-on creates a 3-D volume grid from a surface grid defining the bottom of a body of water. (For example, measured points defining the bed and banks of a river.) The volume grid, composed of layers of prisms, extends from the bottom to the surface. Points in the original surface zone above the specified water lever are blanked so they are not used in the definition of the volume grid.

Figure 31-8. The General Text Loader.

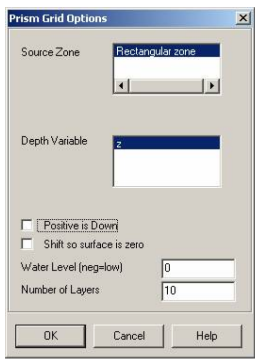


Figure 31-9. The Prism-Grid dialog.

An example of using Prism-Grid would be to define the bottom of a body of water. Normally the data defining the bottom (depth) of the body of water would be read from a file. In this example, however, we generate a rectangular zone with a simple parabolic variation of depth.

To do this, perform the following steps:

1. Generate a rectangular zone by selecting Rectangular from the Data menu's Create Zone sub-menu. The Create Rectangular Zone dialog appears. Set XMin to -1, YMin to -1, and the I- and J-dimensions to 50. Accept the defaults for the rest of the fields.
2. Create the depth variable with the Specify Equations option from the Data menu's Alter sub-menu. The Specify Equations dialog appears. Use the equation **`{Depth} = x**2 + y**2 - 0.5`**.
3. Select Prism-Grid from the Tool's menu. Accept the defaults and click OK.

Sections of the surface which are above the water level (zero) are removed, the rest of the surface is triangulated, and the volume between the bottom and the water level is filled with ten layers of triangular prisms. The result is shown in Figure 31-10.

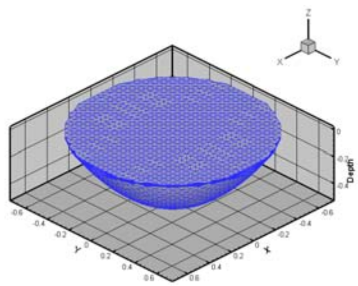


Figure 31-10. An example of using Prism-Grid.

At this point, experimental data, such as water temperatures or velocities, could be interpolated to the volume data and iso-surface, slices, or streamtraces could be generated. For more information, including macro commands for Prism-Grid, access Tecplot's Help.

31.1.10. Sort

The Sort dialog is shown in Figure 31-11.The Sort add-on sorts the values of a data set using

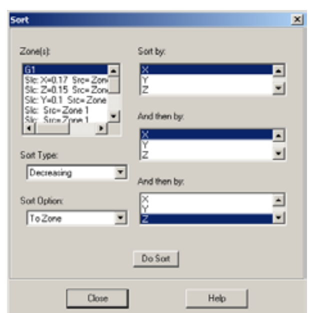


Figure 31-11. The Sort dialog.

one variable as a key. Further variables can be selected in order to further define how the data is sorted. Sort will only work with ordered data. Data may be sorted in either ascending or descending order. Also, data may be sorted In Place or To Zone. For more information, including macro commands for Sort, access Tecplot's Help.

31.1.11. Statistics Calculator

The Statistics dialog is shown in Figure 31-12. The Statistics Calculator extends Tecplot's capability to compute simple descriptive statistics. It computes mean, median, variance, stan-

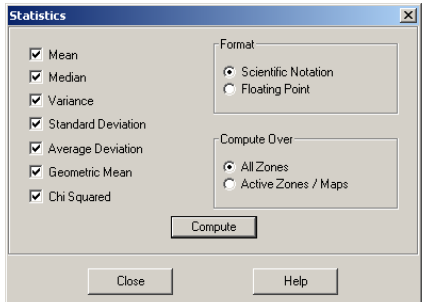


Figure 31-12. The Statistics dialog.

dard deviation, average deviation, geometric mean, and chi square. For more information, including macro commands and formulas for Statistics Calculator, access Tecplot's Help.

31.1.12. Tecplot GUI Builder

Tecplot GUI Builder's dialog is shown in Figure 31-13.The Tecplot GUI Builder is used to

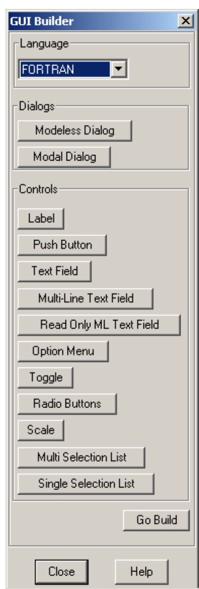


Figure 31-13. The Tecplot GUI Builder dialog.

generate graphical user interfaces for Tecplot add-ons. You will commonly start with the file, **`gui.lay`**, which was created by default if you used the Add-On Wizard or **`CreateNewAd-`**

dOn shell scripts to create your add-on. To build an interface, open this layout file in Tecplot and add an assortment of controls to modal or modeless dialogs. For more information, access Tecplot's Help.

31.1.13. Tetra-Grid

The Tetra-Grid dialog is shown in Figure 31-14.The Tetra-Grid add-on takes well data and

Figure 31-14. The Tetra-Grid dialog.

generates a tetrahedral mesh. Value-blanking may be used to eliminate wells and/or data points within wells.

The following requirements must be met for Tetra-Grid to work:

- The wells must be I-ordered zones.
- There must be at least three wells.
- Each well must contain at least two data points that are not blanked.

The Tetra-Grid dialog contains a list of I-ordered zones in the current data set. Choose the zones you want to use and click OK. The tetrahedral zone will be created and added to the end of the list of zones. You must activate this zone yourself.

As an example of using Tetra-Grid, say data for five different wells has been collected. Some wells have three data points, others have four. The data for each well is assigned to a separate I-ordered zone in Tecplot.

The input data is:

```
VARIABLES = "Easting (m)" "Northing (m)" "Elevation (ft)"
ZONE T="41-14-08" I=3, J=1, K=1,F=POINT
 3.437500000E+00 9.375000000E-02 2.819946289E+00
 3.375000000E+00 9.375000000E-02 1.811889648E+00
 3.437500000E+00 9.375000000E-02 8.199462891E-01
ZONE T="41-14-09" I=4, J=1, K=1,F=POINT
 2.687500000E+00 1.796875000E+00 2.212158203E+00
 2.687500000E+00 1.796875000E+00 1.500000000E+00
```

```
 2.437500000E+00 1.796875000E+00 1.179992676E+00
 2.375000000E+00 1.796875000E+00 1.799926758E-01
ZONE T="41-14-11" I=3, J=1, K=1,F=POINT
 1.875000000E+00 4.000000000E+00 2.509948730E+00
 1.875000000E+00 4.000000000E+00 1.509948730E+00
 1.812500000E+00 4.000000000E+00 5.018920898E-01
ZONE T="41-15-02" I=4, J=1, K=1,F=POINT
 0.000000000E+00 2.375000000E+00 2.089965820E+00
 0.000000000E+00 2.375000000E+00 1.089965820E+00
 0.000000000E+00 2.375000000E+00 5.089965820E-01
 0.000000000E+00 2.375000000E+00 8.996582031E-02
ZONE T="41-15-03" I=3, J=1, K=1,F=POINT
 1.250000000E+00 0.000000000E+00 2.000000000E+00
 1.500000000E+00 0.000000000E+00 1.016113281E+00
 1.250000000E+00 0.000000000E+00 4.687308319E-10
```

The wells do not have to be vertical or even straight. The resulting plot is shown in Figure 31-15. The figure shows the wells before and after running Tetra-Grid. A slice is added

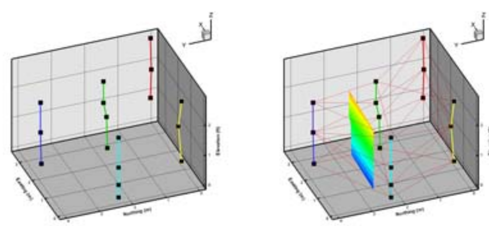


Figure 31-15. The original well data (left), tetrahedral zone from the well data (right).

to the plot with the new tetrahedral mesh to show how you can demonstrate volume properties with the new zone. For more information, including macro commands and another example using Tetra-Grid, access Tecplot's Help.

31.1.14. View Binary

The View Binary dialog is shown in Figure 31-16. The ViewBin add-on allows you to view the information in a Tecplot binary data (`.plt`) file.

The ViewBin dialog has the following option:

- **Show Raw Data:** Select this option to view zone data.

On some machines the font used to display the header information may not be a mono-pitched font and consequently some of the results may not line up directly below the table header. For more information, access Tecplot's Help.

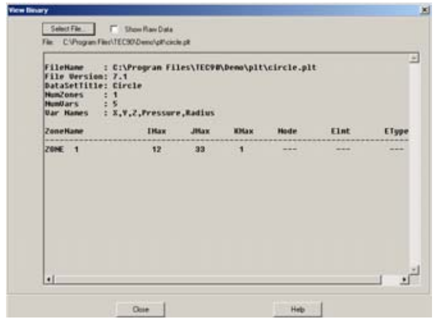


Figure 31-16. The View Binary dialog.

31.2. Tecplot Utilities

Several utilities are included with the Tecplot distribution. They are discussed in the following sections.

31.2.1. Excel Macro

The Excel Macro provides a convenient way to load data directly from your Excel spreadsheet into Tecplot. When loaded it adds an option to Excel's Tools menu called Tecplot, and a toolbar containing a button marked Tecplot. Both launch Tecplot and load the data in the highlighted region of the spreadsheet. The Excel macro offers many advantages over the Excel loader in Tecplot (accessed from the Import option from the File menu).

These include:

- **Highlight and Plot:** The Excel macro is easier to use than the conventional Excel loader. Click in the upper left cell of the region or highlight the entire region, and then click on the Tecplot button in the tool bar or on the Tecplot option in Excel's Tools menu.
- **Multiple Zones:** The Excel Macro makes loading multiple zones much easier. Highlight the entire region and then click on the Tecplot button in the tool bar or on the Tecplot option in Excel's Tools menu. If your zones are separated by blank rows or columns, then the macro will load them to Tecplot.
- **Formulas:** The highlighted region of the spreadsheet can contain formulas, or can be created entirely with formulas. The current Excel loader (using the Import option from the File menu) does not work where formulas are present.

A Read Me file, located in the **`Util/Excel`** directory, further describes installation and use of this macro.

As an example, let's say you have 3-D data obtained by drilling a number of wells and measuring contaminant concentrations of various chemicals at different depths. Your data is in Excel, and you want to load the data into Tecplot to get a visual representation of the contamination. The data has nine variables and twenty-seven zones, as shown in Figure 31-17.

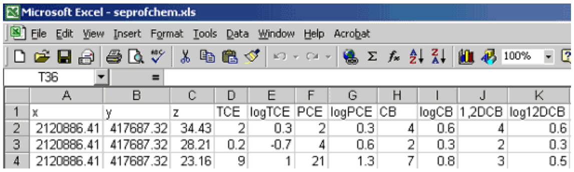


Figure 31-17. The beginning well data in Excel.

Perform the following steps to import your data and visualize the contaminant plumes:

1. Load your Excel data using the new Excel macro. (Accessing this dialog is shown in Figure 31-18.) Make sure you have a blank row separating the zones in Excel.

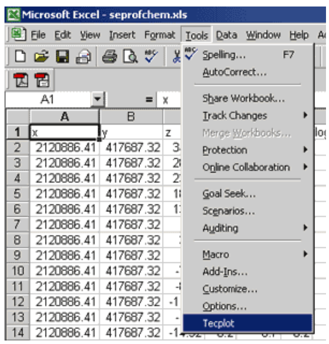


Figure 31-18. Accessing the Excel Loader macro via Excel's menu bar.

2. Starting with the top left-hand cell, highlight all twenty-seven zones and nine variables, as shown in Figure 31-19.

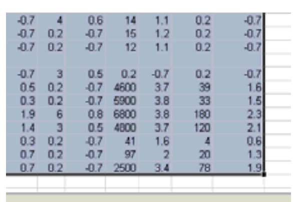


Figure 31-19. Highlighting your data in Excel.

3. Click on Tecplot in Excel's Tools menu. The menu option launches Tecplot with the selected data loaded.

4. Switch to 3D Cartesian plot mode to see the location and measurement depths of the well samples. The resulting plot is shown in Figure 31-20. Your wells have different depths, so

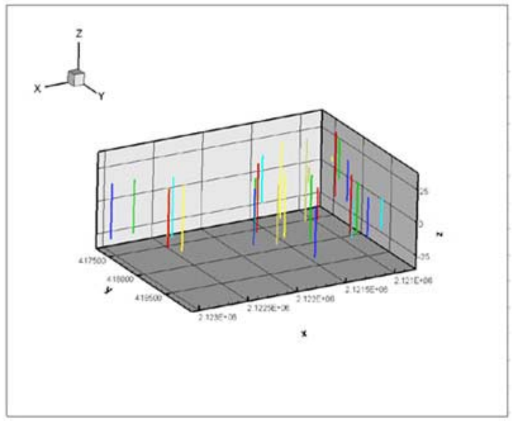


Figure 31-20. The Excel well data plotted in Tecplot.

the number of measurements are not the same for each well (there are only three measurements at well five).

31.2.2. Framer

A shareware utility for viewing RasterMetafile animations created by Tecplot. It is described in Section 29.5.2, "Raster Metafiles Viewing in Framer," and Appendix B.1, "Framer." Framer is shown in Figure 31-21.

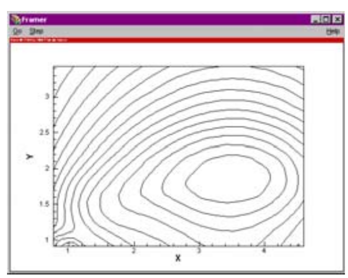


Figure 31-21. The Framer interface.

31.2.3. Lpkview

A utility to catalog, preview or unpack a layout package file into its component data and layout files. It is described Section 5.3.3.3, "Layout Package Utility," and Appendix B.2, "LPKView."

31.2.4. Preplot

A utility to convert an ASCII data file into a Tecplot binary file. It is described Section 4.5, “ASCII Data File Conversion to Binary,” and Appendix B.3, “Preplot.”

31.2.5. Rmtoavi

A utility to convert a RasterMetafile animation into an AVI animation. It is described in Appendix B.4, “Raster Metafile to AVI (rmtoavi).”

31.2.6. Pltview

A utility to view the header information for a Tecplot binary file. The **Pltview** utility is shown in Figure 31-22.

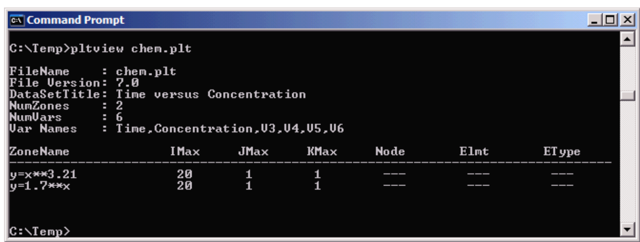


Figure 31-22. The **Pltview** utility.

31.2.7. Poly-Grid

This utility converts almost-random data into finite-element data. The random data must be arranged in non-intersecting polylines where each polyline can have any number of points. It is documented in detail in **TEC100\util\polygrid\readme.txt**.

31.3. Add-on Use

To use these add-ons, you may edit the **tecplot.add** file (located in the **TEC100HOME** directory), uncommenting the appropriate lines. For example, to use the **advqet** add-on, find the line which reads **#$!LoadAddOn "advqet"** in the **tecplot.add** file, remove the **#** sign, then save your changes. When you start Tecplot again, Advanced Quick Edit Tool will be an option under the Tools menu.

Finally, there are a number of add-ons related to Amtec’s ADK (*Add-on Developer’s Kit*). This consists of one add-on, **GuiBuild**, or Tecplot GUI Builder, along with several samples. These samples are add-ons which do not load automatically when you first install Tecplot.

If you want to build your own add-ons, you should refer to ADK documentation in the **<tecplot-home-dir>/adk/doc**. If this is not present, you may install the ADK by running the Tecplot installation program again, selecting to include the ADK during the installation.

31.3.1. Add-On Loading

You can customize lists of add-ons to be loaded by different Tecplot users in your network, or by a single user starting Tecplot with different commands.

31.3.1.1. Add-Ons Loaded by All Users. In a normal installation of Tecplot, the add-ons you want loaded by all users of Tecplot are named in an add-on load file called **`tecplot.add`**, located in the Tecplot home directory. The only command allowed in a **`tecplot.add`** file is the **`$!LoadAddOn`** command. The following is an example of a typical **`tecplot.add`** file:

```
#!MC 1000
$!LoadAddOn "cfdtool"
$!LoadAddOn "streamtool"
```

31.3.1.2. Secondary Add-On Load File Specification. You may also instruct Tecplot to load a different list of add-ons by naming a second add-on load file using one of the following methods:

- Include **`-addonfile`** *addonfilename* on the command line.

or

- Set the environment variable **`TECADDONFILE`**.

Both of these methods tell Tecplot the name of another add-on load file to process.

31.3.1.3. Add-On on the Command Line Specification. You can also instruct Tecplot to load a particular add-on via the command line. The following flags are available:

`-loadaddon` *libname*

or

`-loadaxaddon` *activeXname*

where

libname

The full name (including path and extension) of a **`V7Standard`** add-on (the only choice in UNIX).

activeXname

The name of an ActiveX style add-on. (The supplier of the add-on will tell you what type it is.)

You may specify the **`-loadaddon`** or **`-loadaxaddon`** flag as many times as you want on the command line.

If your add-on is named with the proper suffix for your platform (**`.dll`** for Windows, **`.sl`** for HP UNIX, and **`.so`** for all other UNIX platforms) you can simply name the add-on on the command line without using the **`-loadaddon`** flag.

After add-ons are loaded, Tecplot re-processes all command line arguments not processed earlier (for graphics and add-on initialization). This ordering allows for a data reader add-on (discussed later) to be used to load data specified on the command line.

31.3.2. $!LoadAddOn Command Use

The **`tecplot.add`** file is a special macro file that is executed at startup time and contains one or more **`$!LoadAddOn`** commands to load add-ons into Tecplot. **`$!LoadAddOn`** is, in fact, the only macro command allowed in a **`tecplot.add`** file. The syntax for the **`$!LoadAddOn`** command is:

`$!LoadAddOn` "*libname*"
  `AddOnStyle =` *addonstyle*

where

libname

The name of the shared object library file (see below). This must be in quotes.

addonstyle

The add-on style. This can be either **`V7Standard`** or **`V7ActiveX`**. **`V7Standard`** is the default.

Special rules govern how *libname* name is specified. In all cases the filename extension is omitted. If you assign *libname* to just the base name of the shared object library, then Tecplot will do the following:

- **UNIX:** The shared library to load will come from the file specified by:
  - *Tecplot-Home-Directory*`/lib/lib`+*basename*+*platform-specific-extension*

where *platform-specific-extension* is **`.sl`** for HP platforms and **`.so`** for all others.

- **Windows:** If the add-on is of type **`V7Standard`** and just the base name is supplied, the add-on *basename*`.dll` will be searched for in the following directories (in this order):
  - The directory where the Tecplot executable resides.
  - The Windows system directories.
  - The directories in your **`PATH`** environment variable.

If an absolute path name is used in *libname*, then in Windows, **`.dll`** is appended and in UNIX **`.so`** or **`.sl`** is appended.

On Windows using **`V7ActiveX`** style add-on libraries, Tecplot connects to the add-on via the *libname* entry in the registry.

APPENDIX A Command Line Options

A.1. Tecplot Command Line

The general form of the Tecplot command line is:

`tecplot` *[options] [layoutfile] [datafiles] [macrofile]*

where *options* is one or more of the following:

`-addonfile` *filename*	Load add-ons listed in *filename*.
`-b`	Run Tecplot in batch mode (**`-p`** option is also required).
`-c` *cfgfile*	Use *cfgfile* for the configuration set up instead of the default configuration file.
`-d` or **`-display`** *computername*	Displays Tecplot on computer *computername* (UNIX only). The computer, *computername*, must have X-server capability with the GLX extension.
`-datasetreader` *readername*	Instruct Tecplot to use the data set reader *readername* when loading data files specified on the command line. See Section A.7, "Specifying Data Set Readers on the Command Line," for details.
`-debug` *dbugfile*	Send debug information to the file *dbugfile*. Information is displayed to aid in debugging a new Tecplot configuration file, macro file, or binary data file. You may specify the minus sign ("**`-`**") for *dbugfile* to send the debug output to the "standard output."
`-demo`	Run Tecplot in demo mode (only reads demo files).
`-develop`	Launch Tecplot in a mode used to develop add-ons (UNIX only).
`-f` *fontfile*	Use *fontfile* for the font file instead of the default font file **`tecplot.fnt`**.
`-h` *homedir*	Use *homedir* for the Tecplot home directory instead of the default home directory or the directory stored in the operating system environment variable **`TEC100HOME`**. (See the *Tecplot Installation Notes*.)
`-loadaddon` `"`*addonname*`"`	Load add-on *addonname*.
`-loadaxaddon` `"`*axaddonname*`"`	Load Active-X add-on *axaddonname* (Windows only).

-m *cmapfile*	Select initial color map file to load.
-n	List node information (UNIX only).
-nobatchlog	Suppress creation of the file **batch.log** during batch mode operation.
-nostdaddons	Do not load add-ons in **tecplot.add**.
-p *macfile*	Play the macro in the file *macfile*. Note that if your macro file has an **.mcr** extension you do not need to use **-p**.
-q	Use quick playback mode. Ignores delay and pause commands.
-qm *quickpanelfile*	Load macro functions for the Quick Macro Panel from *quickpanelfile* instead of the default file **tecplot.mcr**.
-r *prtfile*	Set the default file name for routing Print Files to *prtfile*. This name can be reassigned interactively while running Tecplot.
-s *stylfile*	Use *stylfile* as a stylesheet for the first Tecplot frame.
-showpanel	Show the Quick Macro Panel immediately when Tecplot starts up.
-v	Print version number of Tecplot.
-x	Run Tecplot full screen.
-y *exportfile*	Same as **-r** except for exported files.

In the command line, data files is one or more data files. These files are assigned to the first data set. You can also give the name of a layout file (typically having a "**.lay**" extension). Tecplot processes the layout immediately upon starting up. If both a layout file and data files appear on the command line, Tecplot substitutes the data files from the command line for the data files referenced in the layout file. When you read in a layout package file ("**.lpk**") you will not get this behavior.

A.2. Using the Command Line in Windows

Most of the Tecplot command line options are available in Windows. To use them, you should start Tecplot from the Run command. In Windows the Run command is launched from the Start button. Under Windows you may also use the command line from the DOS prompt (otherwise known as the command prompt).

A.3. Using Command Line Options in Windows Shortcuts

All of the command line options that can be entered at the DOS or Command prompt by using the Run command can also be used in a Windows shortcut.

A.3.1. Creating Shortcuts

If you frequently run Tecplot using the same command line flags, it may be useful to create a shortcut on your Windows desktop that launches Tecplot with the desired command line flags. Here's how this can be done:

1. Right click in any blank space on your Windows desktop. A drop-down appears.
2. Select New.
3. Select Shortcut from the next drop-down that appears.
4. The “Create Shortcut” dialog will appear (Figure A-1).

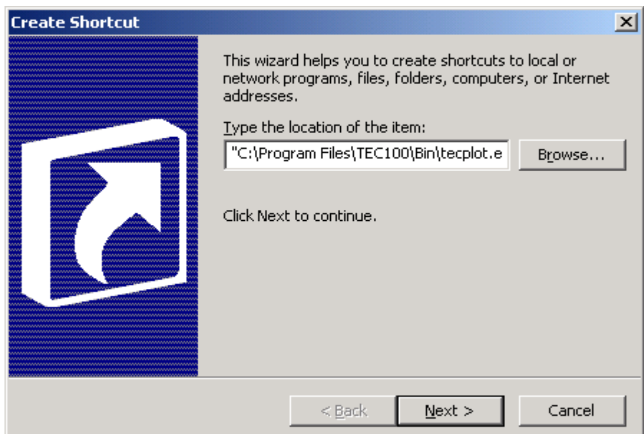


Figure A-1. The Create Shortcut dialog in Windows.

Type the location of the Tecplot executable, along with any command flags you want to specify. You can also click Browse if you are not sure where Tecplot is located. An example command line is:

```
"C:\Program Files\TEC100\BIN\Tecplot.exe" -p C:\Me\mymacro.mcr
```

5. Click Next.
6. Select a name for your shortcut, then click on Finish. An example name would be:

```
Run my Macro
```

A new shortcut icon will be placed on your Windows desktop. To run Tecplot using the command line options you specified, simply double-click on the new shortcut icon.

A.3.2. Changing Shortcuts

You can alter an existing shortcut by doing the following:

1. Right-click on the shortcut icon you want to change.
2. Select Properties from the drop-down.

3. On the Shortcut page (Figure A-2), modify the command line by changing the setting for Target. To change the working directory that Tecplot runs under, change the Start in location.

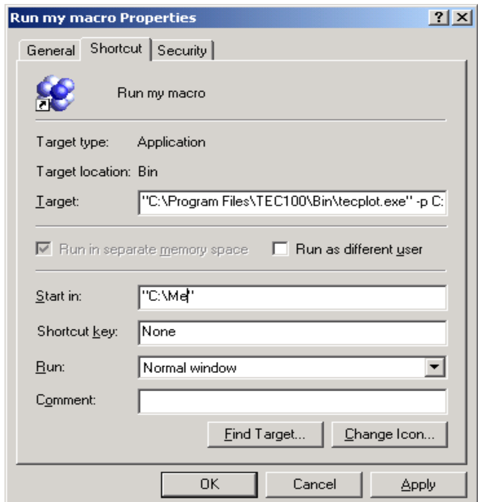


Figure A-2. The Shortcut page in Windows.

A.4. Additional Command Line Options in Motif

Under UNIX, you can use additional command line flags which are passed to the window manager to control how the application window is displayed. These include **`-geometry`** (for specifying the location and position of the application window), **`-fg`** and **`-bg`** (for specifying foreground and background window colors), and others. See the X11 reference for your system for complete details on these options.

A.5. Overriding the Data Sets in Layouts by Using "+" on the Command Line

This section describes how to load an alternate data set into a layout using the command line. A method for overriding the layout data set interactively, in the Open Layout dialog, is described in Section 5.1.2, “Loading Data from Other Software Packages.”

When loading a layout from the command line, you may override the data files used in that layout by specifying them on the command line after the layout file name. For example:

```
tecplot amt.lay t4.plt
```

This loads the **amt.lay** layout with the **t4.plt** data file instead of the data file specified in **amt.lay**.

If **amt.lay** had more than one data set associated with it, **t4.plt** would replace the first data set. If you wanted to replace multiple data sets, specify each file on the command line like so:

```
tecplot amt.lay t4-1.plt t4-2.plt t4-3.plt
```

This will use **t4-1.plt** as the first data set, **t4-2.plt** as the second, etc.

If **amt.lay**'s first data set had more than one data file associated with it, **t4-1.plt** would replace all data files in the first data set. If you wanted to specify more than one data file for the data set, you can use the **+** to specify they are all part of the first data set like so:

```
tecplot amt.lay t4-1.plt+t4-2.plt+t4-3.plt
```

In this case, **t4-1.plt**, **t4-2.plt**, and **t4-3.plt** are all combined into one data set that replaces the first data set of **amt.lay**.

You can combine both multiple data sets and multiple files per data set like so:

```
tecplot amt.lay ds1a.plt+ds1b.plt ds2.plt ds3a.plt+ds3b.plt
```

In this case, the files **ds1a.plt** and **ds1b.plt** are combined and replace the first data set, **ds2.plt** replaces the second data set, and **ds3a.plt** and **ds3b.plt** are combined to replace the third data set in **amt.lay**.

If you do not know which data set to substitute in your layout, look at the top of the layout file. It will look something like this:

```
#!MC 1000
$!VarSet |LFDSFN1| = 'temp.plt'
$!VarSet |LFDSFN2| = 'chem.plt'
$!VarSet |LFDSFN3| = 'pos.plt'
```

So you can, for example, replace **chem.plt** with **chem1.plt** and **chem2.plt** using the following command line:

```
tecplot amt.lay temp.plt chem1.plt+chem2.plt pos.plt
```

A.6. Tecplot Command Line Examples

To run Tecplot without pre-loading any data files, use:

```
tecplot
```

To run Tecplot loading the data file **ex1.plt** as the first data set, use:

```
tecplot ex1.plt
```

To run Tecplot loading the data files **ex1.plt**, **ex2.plt**, and **ex3.plt** as the first data set, use:

```
tecplot ex1.plt ex2.plt ex3.plt
```

To run Tecplot using **/usr/myhome** as the Tecplot home directory and loading the Tecplot configuration file **/usr/myhome/myset.cfg**, use:

```
tecplot -h /usr/myhome -c /usr/myhome/myset.cfg
```

To read a Tecplot layout file **sumtr1.lay**, you would use:

```
tecplot sumtr1.lay
```

To read a Tecplot layout file **calc.lay** and replace the first data set referenced in the layout file with the data file **temp.plt**, you would use:

```
tecplot calc.lay temp.plt
```

For example, suppose the layout file **t.lay** has two frames. The two frames reference different data sets. Suppose you want to start Tecplot, load this layout file, and have frame one use the data set defined in **a.plt** and have frame two use the data set defined by loading in **b.plt** and **c.plt** together. You can do this with the following command:

```
tecplot t.lay a.plt b.plt+c.plt
```

In UNIX, to determine the path or alias that the **tecplot** command calls, you would use:

```
which tecplot
```

A.7. Specifying Data Set Readers on the Command Line

Special care should be taken when using the **-datasetreader** option on the command line. The following rules apply if **-datasetreader** is used:

1. The **-datasetreader** flag must be followed by the data set reader name and then immediately followed by a space separated list of commands to be passed on to the data set reader. No further Tecplot options are allowed after this point.
2. The data set reader name must be placed in quotes if it contains spaces.
3. Only one data set reader can be specified on the command line.
4. If a layout file is also specified (prior to **-datasetreader**) then you can only override the first data set load instructions referenced in the layout file.

Following is an example:

Suppose you have a layout file (**mylayout.lay**) that uses the PLOT3D loader. To launch Tecplot via the command line and override the PLOT3D load instructions use:

```
tecplot mylayout.lay -datasetreader "plot3d loader" -ISET 1,,5 -b -
3DW -GF blunt.g
```

Everything from the **-ISET** parameter and following are instructions to be sent to the PLOT3D loader. Note that the instructions themselves are not entirely contained within any quotes. If your data reader requires instructions that themselves contain spaces then you must surround those instructions with quotes.

APPENDIX B Utility Command Line Options

B.1. Framer

To launch Framer at a command line (shell prompt, Run command, and so forth), use the following command:

framer *[options] [rmfile]*

where *[rmfile]* is the name of a file containing Raster Metafile bitmaps created by Tecplot, and *[options]* is one or more of the following:

-b [*nf*]	Use buffered mode. Framer reads *nf* frames into memory and displays only those frames. Frames not read are not displayed. This mode displays images much faster, but requires extra memory. If *nf* is not specified, Framer reads as many frames as possible up to the total limit on frames (see **-max** parameter).
-c *nc*	Use no more than *nc* colors (X-Windows only). On some machines, you may need to use "**-c 128**" to allow two copies of Framer to run at the same time.
-cycle *nn*	Start Framer in "cycle" mode (as if **C** were pressed), and continue for *nn* complete cycles (unless interrupted by user input), and then exit.
-d *dfile*	Send debug information to *dfile*. Use "**-d2**," "**-d3**," "**-d4**,", etc., for more detailed debug information.
-f *start,end,skip*	Display frames starting with frame number *start* and ending with frame number *end*, skipping by *skip* frames.
-g	Use gray scale for image instead of color.
-help	Print help information.
-loop *nn*	Start Framer in "loop" mode (as if **L** were pressed), and continue for *nn* complete loops (unless interrupted by user input), and then exit.
-m	Allow for multiple color maps. Without this flag, Framer assumes the first color map in the Raster Metafile is valid for all images in that file.

`-max` *nn*	Specify upper limit on total number of images in the Raster Metafile. The default value is 512.
`-noinfo`	Do not print initial copyright notice, help info, or count of buffered frames.
`-p` *ms*	Pause at least ms milliseconds between each frame. This does not affect the rate of the single frame keys (**`+`** and **`-`**).
`-w` *wc*	Width correction. (Use "**`-w -1`**" for Tecplot Version 4 images.)
`-x`	Run full screen.

If you do not specify a file name, Framer prompts you for one. You can choose to set buffering (equivalent to the **`-b`** flag) and/or multiple color maps (equivalent to the **`-m`** flag).

While Framer is running, you can press the following keys to control it:

`B`	Move backward through frames (or left mouse button).
`C`	Cycle forward and backward through frames.
`F`	Move forward through frames (or middle mouse button on a three-button mouse or right mouse button on a two button mouse).
`L`	Loop repeatedly forward through frames.
`Q`	Quit Framer (or right mouse button on three button mouse) or Escape key.
`S`	Stop cycling or looping (or spacebar).
`R`	Redraw the current frame.
`1`	Move to the first frame.
`+`	Move forward one frame.
`-`	Move backward one frame.
`<`	Increase the minimum delay between frames by 50 milliseconds. This decreases the speed at which frames are displayed.
`>`	Decrease the minimum delay between frames by 50 milliseconds. This increases the speed at which frames are displayed.

Under Windows, these Framer commands are also available from the Go and Step menus.

B.2. LPKView

Following is a description of the utility's syntax. Brackets (**`[]`**) surround optional parameters and the vertical bar (**`|`**) separates one mutually exclusive set of options from another:

lpkview [[-t] | [-ild] | [[-c <preview command>] -p]] filename where the options are described as follows:

-t	Show table of contents.
-i	Extract image (for example, a Portable Network Graphics or **.png** format).
-l	Extract layout.
-d	Extract data.
-c	(UNIX only) Specify preview command.
-p	(UNIX only) Preview image.

Option **-t** may not be used with any other options and options **-i**, **-l**, and **-d** may not be used with options **-c** and **-p**. If no command line options are specified **-i**, **-l**, and **-d** are assumed by default.

Note: Under UNIX, if the **-p** option is specified without specifying a preview command, **-c**, the following default preview command is used:

$MOZILLA_HOME/netscape -remote "OpenURL(%s)"

where **%s** is substituted by **lpkview** with the file name of the temporarily extracted preview image. The default command assumes that the environment variable **$MOZILLA_HOME** is set, Netscape is installed under **$MOZILLA_HOME**, and that **lpkview** has been added to Netscape as a helper application.

To add **lpkview** as a helper application bring up Netscape's Preferences dialog. This is usually accomplished by selecting Preferences from Netscape's Edit menu. Within the Preferences dialog locate and select the Applications page. Within the Applications page select New and add **lpkview** as a new helper application by entering the following information:

Description:<*optionally leave this blank*>

MIMEType: application/x-tecplot-lpk

Suffixes: lpk

Handled By: <*select "Application"*>

Application: $TEC90HOME/bin/lpkview -p %s

Assuming that you have correctly set the **$TEC90HOME** environment variable, if you browse with Netscape and click on a layout package file, it will run **lpkview** as a helper application and display the preview image in your browser.

If you choose to specify your own preview command, there are several requirements:

- The path to the preview command must be fully specified.
- If relative, it must be located in one of the directories specified in your **$PATH** environment variable

- The command must contain a **%s** that can be substituted by **lpkview** with the file name of the temporarily extracted preview image.

For example, if you wanted the preview command to be the UNIX **file** utility. Then, running the following command:

    lpkview -c "file %s" -p myplot.lpk

might produce the following output:

    /var/tmp/aaaa005L7:   data

Where file **/var/tmp/aaaa005L7** is the temporarily extracted preview image. The temporary file is removed as soon as the preview command completes.

B.3. Preplot

The following options are used with standard Tecplot data files:

-d	Turn on debug echo. Use **-d2**, **-d3**, **-d4** for more detailed debug information.
-r	Reverse the bytes of the output binary data file (generally not required).
-iset *[zone]*, *[start]*, *[end]*, *[skip]*	
	Create the binary data file using only the specified range and skipping for the I-index. The arguments are optional, but the commas are not. The *zone* parameter specifies which zone this option affects; if not specified, all zones are affected. The *start* parameter is the starting I-index; the default is one. The *end* parameter is the ending I-index; the default is the last index value. The *skip* parameter specifies the I-interval, that is, the distance between indices; one means every index is used, two means every other index, and so on. For example, **-iset 1, 3, 7, 2** indicates that for zone 1 only I-index values of **3**, **5**, and **7** are used. Only one **-iset** option is allowed per zone.
-jset *[zone]*, *[start]*, *[end]*, *[skip]*	
	Same as **-iset** above, except with respect to the J-index.
-kset *[zone]*, *[start]*, *[end]*, *[skip]*	
	Same as **-iset** above, except with respect to the K-index.
-zonelist *start[*:*end[*:*skip]]*, ...	
	Specify the zones to process. You may supply more than one specification. By default Preplot processes all zones.

The following options are used with PLOT3D data files:

-d	Turn on debug echo. Use **-d2**, **-d3**, **-d4** for more detailed debug information.
-r	Reverse the bytes of the output binary data file (generally not required).
-plot3d	Input file is in PLOT3D format. This flag is required for PLOT3D data.
-b	Input file is binary.
-f	Input file is binary-FORTRAN, that is, there are record markers.
-foreign	Reverse bytes of input file.
-function	The **.q** file is a **.f** file.
-functionandq	There are both **.f** and **.q** files present.
-gridonly	Read grid variables only.
-i	Input file includes PLOT3D IBLANK variable.
-m	Input file is multi-grid (usually more than one grid block).
-ip *ilist*	Extract planes of constant *i* for all *i* in *ilist*. (Requires 3-D whole data.)
-jp *jlist*	Extract planes of constant *j* for all *j* in *jlist*. (Requires 3-D whole data.)
-kp *klist*	Extract planes of constant *k* for all *k* in *klist*. (Requires 3-D whole data.)
-1d	Input PLOT3D file is 1-D.
-2d	Input PLOT3D file is 2-D.
-3dp	Input PLOT3D file is 3-D planar.
-3dw	Input PLOT3D file is 3-D whole.

B.4. Raster Metafile to AVI (rmtoavi)

The **rmtoavi** utility will convert a Raster Metafile animation to an AVI animation. The following is a description of the utility's syntax. Brackets (**[]**) surround optional parameters. Options must be specified separately.

rmtoavi *[options]* **filename[.rm]** *[outputfilename]*

Filename is the name of the Raster Metafile to convert. Only one file name may be specified. The input file must end with the **.rm** extension.

The *[outputfilename]* is the name of the converted output AVI file. If the output file name is not specified, the input file name is used with an **.avi** extension. If any of the file names contain spaces, they must be enclosed in quotes.

For example, the command **rmtoavi test.rm** will create the file **test.avi**. If the output file exists, **rmtoavi** will prompt to overwrite it unless the **-y** option is used (see below).

The *[options]* are described as follows

-help	Prints help information.
-q	Suppress startup banner and information message.
-y	Suppress query to overwrite an existing AVI file.
-d [nn]	Progress indicator. This prints a dot (**.**) every **[nn]** frames processed. If **[nn]** is not specified, it defaults to ten.
-m	Use multiple color palettes in the converted AVI file. Each frame of an AVI or Raster Metafile animation is limited to 256 colors. AVI animations can use either one set of 256 colors for the entire animation or a separate set of 256 colors for each frame. If you use the **-m** option, then each frame of the output AVI file will use a separate set of 256 colors. Since color information is read from the input Raster Metafile, this option only affects the output AVI animation if the Raster Metafile was originally exported using multiple color palettes.
-speed nn	Sets the speed of the output AVI file to *nn* frames per second. The default is ten.

APPENDIX C *Mouse and Keyboard Operations*

C.1. Extended Mouse Operations

The middle and right mouse buttons are powerful tools you may use to immediately zoom and translate your data without having to switch to the Zoom or Translate tools on the sidebar. This advanced mouse/keyboard functionality is available when using any 3D rotate, Contour, Geometry (except Polyline), Probe, Slice, Streamtrace Placement, Translate, Zoom, or Zone Creation tools. If you have a two button mouse use the Ctrl key in conjunction with the right mouse button to achieve middle mouse button capabilities.

The following table lists all of the capabilities of the middle and right mouse buttons.

Action	Middle Button/Ctrl-Right Button	Right Button
`Click`	Redraw. If the pointer is in the current frame then the current frame is redrawn. Otherwise, redraw all frames.[a]	Switch from the current tool to the Selector.[b]
`Drag`	Smoothly zoom in or out. An upward motion zooms out. A downward motion zooms in.	Translate.
`Alt-Drag`	In 3D Cartesian plots, move the viewer further from (upward motion) or closer to (downward motion) the object. In all other plot types, this behaves like the Drag action	Same as the Drag action.

a. This is the default action for a click. It may be configured with the **`$!INTERFACE MOUSEACTIONS {MIDDLEBUTTON...`** command.

b. This is the default action for a click. It may be configured with the **`$!INTERFACE MOUSEACTIONS {RIGHTBUTTON...`** command.

C.2. Mouse Tool Operations

The following tables contain all mouse/keyboard operations you may use with the various sidebar tools. All mouse button operations utilize the left button.

3D Rotate tools:

`Drag`	Rotate about the defined rotation origin with your current Rotate tool.
`Alt-Drag`	Rotate about the viewer position using your current Rotate tool.
`C`	Move rotation origin to probed point, ignoring zones.
`O`	Move rotation origin to probed point of data.
`R`	Rollerball rotation.
`S`	Spherical rotation.
`T`	Twist rotation.
`X`	X-axis rotation.
`Y`	Y-axis rotation.
`Z`	Z-axis rotation.

Contour Add tool:

`Alt-Click`	Place a contour line by probing on a streamtrace, slice, or iso-surface.
`Click`	Place a contour line.
`Ctrl-Click`	Replace the nearest contour line with a new line.
`Drag`	Move the new contour line.
`-`	Switch to the Contour Remove tool.

Contour Remove tool:

`Click`	Removes the contour line nearest to the probed location.
`+`	Switch to Contour Add tool if you are using Contour Remove.

Geometry Polyline tool:

`A`	Allow translation of polyline segments in all directions.
`H`	Restrict translation of current polyline segment to horizontal.
`U`	Pen up, while drawing polyline.
`V`	Restrict translation of current polyline segment to vertical.

Probe tools.

`Click`	If the pointer is over a valid cell return the interpolated field values from all nodes in the cell. If multiple cells are candidates then, for 2D Cartesian plots the cell from the highest number zone is used and for 3D Cartesian plots the cell closest to the viewer is used.
`Ctrl-Click`	If the pointer is over a valid cell return the field values from the nearest node in the cell. If multiple cells are candidates then, for 2D Cartesian plots the cell from the highest number zone is used and for 3D Cartesian plots the cell closest to the viewer is used. If the pointer is not over any cell then the field values from nearest data point as measured in distance on the screen are returned.
`Shift-Ctrl-Click`	Return the field values from the nearest point on the screen ignoring surfaces and regardless of zone number or depth of the point. This is useful in 3-D for probing on data points that are on the back side of a closed surface without having to rotate the object. In 2-D this is useful for probing on data points for zones that may be underneath other zones because of the order in which they were drawn.
`Alt-Click`	Same as Click except ignore zones while probing. (Probe only on streamtraces, iso-surfaces, or slices.)
`Alt-Ctrl-Click`	Same as Ctrl-Click except ignore zones while probing. (Probe only on streamtraces, iso-surfaces, or slices.)
`Alt-Ctrl-Shift-Click`	Same as Shift-Ctrl-Click except ignore zones while probing. (Probe only on streamtraces, iso-surfaces, or slices.)
`T, R` `X, Y`	When probing, press R or T on your keyboard to switch dependencies in Polar Line or X or Y in XY Line.

Slice tools:

`+`	Turn on the start slice if no slices are active, or turn on the end slice if slices are already active.
`-`	Turn off the end slice if the end slice is active, or turn off the start slice if the end slice is not active.
`Click`	Place a start slice.
`Drag`	Move the start slice.

`Alt-click/Alt-drag`	Determine the XYZ-location by ignoring zones and looking only at derived volume objects (streamtraces, slices, iso-surfaces, slices).
`Shift-click`	Place the end slice.
`Shift-drag`	Move the end slice.
`I, J, K (ordered zones only)`	Switch to slicing constant I-, J-, or K-planes respectively.
`X, Y, Z`	Switch to slicing constant X-, Y-, or Z-planes respectively.
`0-9`	Numbers one through nine activate intermediate slices and set the number of intermediate slices to the number entered; zero turns off intermediate slices.

Streamtrace Placement tools (3D Cartesian plots only):

`D`	Switch to streamrods.
`R`	Switch to streamribbons.
`S`	Switch to surface lines.
`V`	Switch to volume lines.
`1-9`	Change the number of streamtraces to be added when placing a rake of streamtraces.

Translate/Magnify tool:

`Drag`	Translate the data.
`Shift-Drag`	Translate the paper.
`-`	If the drag was started with Shift, this will reduce the magnification of the paper. Otherwise, this will reduce the magnification of the data.
`+`	If the drag was started with Shift, this will increase the magnification of the paper. Otherwise, this will increase the magnification of the data.
`- drag`	Decrease magnification on the paper.
`+ drag`	Increase magnification on the paper.

Zoom tool:

`Click`	Center the zoom around the location of your click.

C.3. Picked Object Options

`-`	Reduce the size of the object. If multiple objects are selected, all object positions will be shifted towards the first object selected.
`-`	Increase the size of the object. If multiple objects are selected, all object positions will be shifted away from the first object selected.
`Del`	Delete picked object(s).
`Ctrl-C`	Copy picked object(s) to the clipboard.
`Ctrl-V`	Paste picked object(s) from the clipboard.
`Ctrl-X`	Cut picked object(s).

C.4. Other Keyboard Operations

`Ctrl-A`	Paste stored frame view to current frame.
`Ctrl-C`	Copy selected objects to paste buffer.
`Ctrl-D`	Redraw all frames.
`Ctrl-F`	Fit current image to full size.
`Ctrl-L`	Restore last frame view.
`Ctrl-O`	Open layout.
`Ctrl-P`	Print.
`Ctrl-Q`	Exit Tecplot.
`Ctrl-R`	Redraw the current frame.
`Ctrl-S`	Save current layout.
`Ctrl-U`	Call up the Publish dialog to control Web publishing.
`Ctrl-W`	Save current layout as a specified file.

APPENDIX D Glossary

The following terms are used throughout the *Tecplot User's Manual* and are included here for your information.

2-D

Plotting in two dimensions. Line plots of one or more variables (XY and Polar Line plots) are not considered 2-D.

2-D Cartesian Plot

A plot of some variable by location on a single plane using two axes.

3-D

Plotting in three dimensions. Three-dimensional plotting can be subdivided into 3-D surface and 3-D volume.

3-D Cartesian Plot

A plot displaying a 3-D scattering of points, surfaces, or volumes using three axes.

3-D Sorting

The process by which Tecplot determines which surface to plot first. The various cells are sorted relative to the viewer and then plotted from farthest away to closest.

3-D Surface

Three-dimensional plotting confined to a surface. For example, the surface of a wing.

3-D Volume

Three-dimensional plotting of data that includes interior data points of a volume, as well as those on the surface. For example, the vector field around a wing.

Active Zone

A zone that is activated in the Zone Style dialog.

Antialiasing

In computer graphics, the process of removing or reducing the jagged distortions in curves and diagonal lines so that the lines appear smooth or smoother.

ASCII Data File

A data file composed of human-readable statements and numbers using ASCII characters.

Aspect Ratio

The ratio of lengths of the sides of an object. In the 3D Cartesian plot type, the ratio is that of the longest side to the shortest side.

Auxiliary Data

Metadata attached to zones, data sets, and frames.

Banded Contour Flooding

A field plot where the surface between contour lines is filled with a constant color.

Binary Data File

A data file composed of machine-readable data. This type of file is created by converting ASCII data files with Preplot, or by directly creating them from an application.

Blanking

A feature of Tecplot that excludes certain cells and points from a plot. There are three types of blanking: value-blanked, IJK-blanking, and depth-blanking.

Block

A data file format in which the data is listed by variable. All the point values of the first variable are listed first, then all the point values of the second variable, and so forth.

Boundary

A 2- or 3-D field plot option. Plotting the boundary of a zone plots the connection of all outer lines (IJ-ordered zones), finite-element surface zones, or planes (IJK-ordered zones).

Boundary Cell Faces

A set of un-blanked cell faces in a 3-D volume zone which have only one neighboring volume cell. In contrast, interior cell faces have two neighboring volume cells, one on either side, which share the face. For an IJK-ordered zone the boundary cell faces are on the exterior of the zone. That is, the first and last I-planes, the first and last J-plans, and the first and last K-planes. For a finite-element 3-D volume zone, boundary cell faces are on the exterior of the zone and the surface of any voids within the zone.

Bounding Box of Data

The smallest rectangular box, aligned with the coordinate axes, which completely encloses all data points.

Brick

An element type of finite-element volume data composed of eight node points arranged in a hexahedron-like format. This element type is used in 3-D volume plotting.

Carpet Plot

A 3-D surface plot formed by a 3-D plot where the variable is plotted in the third dimension and is singular-valued with respect to the independent variables.

Case Insensitive

Text that may be in upper- or lowercase letters.

Cell

Either an element of finite-element data, or the space contained by one increment of each index of IJ- or IJK-ordered data.

Cell-Centered Values

Values located at the center of the cell (assumed to be the centroid).

Color Map

A color spectrum used to plot contour flooding and multi-colored objects.

Color Map File

A file that contains a description of a color map.

Connectivity List

The second portion of a finite-element data file where the relationships between points are given to define elements. Cells of the appropriate element type are defined by listing the node point indices. The number of node points per cell is determined by the element type.

Continuous Contour Flooding

A field plot where a color is assigned to each point in a mesh, based upon the contour variable and the color map. Each face is filled with colors interpolated between the corner nodes. This results in a smooth variation of color over the surface.

Contour

A field plot type that plots iso-valued lines, or color flooding based on the values of a specified variable.

Curve Type

The function used to fit the data points in an XY-plot.

Custom Labels

Text strings contained within a data file or text geometry file which define labels for your axes or contour table. You may select Custom Labels anywhere you can choose a number format, the result is the text strings in place of numbers.

Cutaway Plot

A 3-D volume plot where a portion of a 3-D volume zone is cut-away by blanking to reveal the interior.

Cutting Plane

A planar surface used to slice 3-D volume or surface zones.

Data File

A file that contains data used for plotting in Tecplot.

Data Format

The type of zone data as specified by the format parameter in a Tecplot data file, such as: BLOCK, POINT, FEBLOCK, or FEPOINT.

Data Loader

A Tecplot add-on which allows you to read non-Tecplot data files.

Data Point

An XYZ-point at which field variables are defined.

Data Set

A set of one or more zones. A data set may be plotted in one or more frames, however, a single frame may only plot one data set. A data set may be created by loading one or more data files.

Dependent

An axis mode requiring the axes to maintain a fixed ratio to one another.

Depth

For image export, the number of bits stored per pixel. For depth-blanking, the component of distance from the viewer position in a screen normal coordinate system.

Depth-Blanking

A blanking option which excludes cells in a 3-D plot, based upon their depth into the image. Cells closer than a plane of a certain depth, as well as cells further than a plane of another depth, may be blanked.

Derived Volume Objects

Graphic objects which are visible in the plot and created from zone data, but are not zones. Examples include iso-surfaces, 3-D slices, and streamtraces.

Display List

A group of OpenGL commands that have been stored for subsequent execution. Using display lists can, depending upon the hardware involved, dramatically speed up graphics rendering. Using display lists also requires more memory.

Draw Level

A draw behavior setting for modifying the image quality and rendering speed during various operations, such as rotation. Options vary from Trace, a simplified wire-frame mesh which is rendered quickly, to Full.

Element Type

The form of individual elements in a finite-element zone. There are four types: Triangle and Quadrilateral (finite-element surface types), and Tetrahedron and Brick (finite-element volume types). The element type of a zone determines the number of nodes per element and their orientation within an element.

Exposed Cell Faces

The set of those cell faces in 3-D volume zones that have only one un-blanked neighboring volume cell. By comparison, interior cell faces have two neighboring cells, one on either side, which share the face. The exposed cell faces include boundary cell faces and interior cell faces exposed by blanking. (One of the neighboring cells has been blanked.)

Extended Curve-Fit

A Tecplot add-on which extends Tecplot's XY-plot curve-fitting capabilities.

Extra 3D Sorting

Perform extra work to resolve hidden surface problems encountered during 3-D sorting.

FE

An abbreviation for finite-element, a common means of arranging data for calculations. (Often referred to as "unstructured.")

FEBLOCK

A data file format for finite-element zones in which the node data is listed by variable. All the node values of the first variables are listed first, then the node values of the second variable, and so forth. This section is followed by a connectivity list.

Fence Plot

A plot of planes of a 3-D data field.

FEPOINT

A data file format for finite-element zones in which the node data is listed by point-by-point. All the variable values of the first point are listed first, then the variable values of the second point, and so forth. This section is followed by a connectivity list.

FE Surface

A finite-element zone of the element type Triangle or Quadrilateral. These zones are used for 2- and 3-D surface plots.

FE Volume

A finite-element zone of the element type Tetrahedron or Brick. These zones are used for 3-D volume plots.

Field Plot

Includes 2D Cartesian and 3D Cartesian plot types. Generally used to display the spacial relationship of data. Mesh, Contour, Vector, Scatter and Shade are all considered field plots. XY and Polar Line plots and the Sketch plot type are not field plots.

File Path

An option which specifies the directory for Tecplot to search for a given type of file. For instance, a linked layout saved with absolute file path contains the complete directory structure to load the associated file.

Finite-Element

A type of data point ordering. Data is arranged by listing the data points (called nodes), and then listing their relationships (called elements). The element type of the zone determines the number of nodes which are contained in each element, as well as the exact relationship of nodes within an element. There are four different element types supported by Tecplot: triangle, quadrilateral, tetrahedron and brick.

Font Modifier

The modifier used to embed Greek, Math, or User-Defined characters in a text string.

Frame

Boxed areas within the workspace where sketches and plots are created.

Geometry

An arrangement of objects or parts that suggests geometric figures.

Grid Area

One or more rectangular regions defined and bounded by the grid axes.

Grid Axes

An axis option which displays the coordinates of the grid along the various spatial dimensions.

Gridline

A set of lines drawn from one or more axes that extend from the tick marks on an axis across the grid area.

Grid Point

In 2-D, the intersection of gridlines.

I-Ordered

A type of data point ordering where each point is listed one at a time (that is, by one index). Used mainly in XY-plots. In 2- or 3-D, this type of data point ordering is sometimes called irregular, and is only useful for scatter plots, or for interpolating or triangulating into 2-D, 3-D surface, or 3-D volume zones. (This type of data can also be used for 2- or 3-D vector plots if streamtraces are not required.)

IJ-Ordered

A type of data point ordering where the points are arranged in a 2-D array. used for 2-D and 3-D surface plotting.

IJK-Blanking

A feature to include or exclude portions of an IJK-ordered zone based on index ranges.

IJK-Ordered

A type of data ordering where the points are arranged in a 3-D array. Used for 3-D volume plotting as well as 2-D and 3-D surface plotting.

Image Format

Any of the raster or bit-mapped graphic formats supported by Tecplot.

Inactive Zone

A zone loaded into Tecplot which does not appear in the plot. A zone can be deactivated using the Zone Show option on any page of the Zone Style dialog.

Independent

Axis mode allowing each axis to have a range that is not affected by the ranges of other axis or axes.

Interpolate

To assign new values for the variables at data points in one zone based on the data point values in another zone (or set of zones).

Internal Macro Variable

A read-only macro variable which allows you to access certain key values in Tecplot. For example, **`$NUMVARS`** gives the number of variables.

I-Plane

In an ordered zone, the connected surface of all points with a constant I-index. In reality, I-planes may be cylinders, spheres, or any other shape.

Irregular Data

Points which have no order, or at least no order which can be easily converted to IJ- or IJK-ordering.

Iso-Surface

A surface within a 3-D zone where the contour variable has a constant value at all locations.

Journal

Place where data manipulation/creation/deletion instructions are logged.

JPEG

Joint Photographic Experts Group (development group for lossy compressed 24 bit color image storage format; also a file extension).

J-Plane

In an ordered zone, the connected surface of all points with a constant J-index. In reality, J-planes may be cylinders, spheres, or any other shape.

K-Plane

In an IJK-ordered zone, the connected surface of all points with a constant K-index. In reality, K-planes may be cylinders, spheres, or any other shape.

Layout File

A specialized macro file which preserves a plot created within Tecplot. When the layout is opened, it restores Tecplot to the state it was in when the layout file was saved.

Layout Package File

A binary layout file with the data embedded.

Line Map

A set of points from a single zone where one variable is assigned to an X-axis and another is assigned to a Y-axis. You can define many XY-maps for an XY-plot.

Macro

A file containing a list of instructions, called macro commands, which can duplicate virtually any action performed in Tecplot.

Macro Command

An instruction given to Tecplot in a macro file. Macro commands always start with a dollar sign and then an exclamation mark. For example, $!Redraw refreshes a plot view.

Macro File

A file which contains a series of macro commands. Macro files are run from the command line, or through the Run option of the Macro sub-menu of the File menu.

Macro Function

A self-contained macro sub-routine that can be called.

Macro Variable

A holding place for numeric values in a macro file. There are two types of macro variables: user-defined (you set and retrieve the value), or internal (Tecplot sets the value and you may retrieve it).

Map Layer

One way of displaying a line mapping, such as with line, bars, symbols, and so forth. One mapping may be displayed with one or more layers.

Median Axis

In 3-D, the grid axis which when scaled is not the shortest nor the longest axis.

Menu Bar

The top bar of the Tecplot screen used to select menu options.

Mesh

A 2- or 3-D field plot type which plots connections between data points.

Multi-Colored

Any Tecplot object which is colored by the value of the contouring variable. Multi-colored objects may include mesh, scatter symbols, vectors, contour lines, and streamtraces.

Multi-Line Text

Text which spans two or more lines.

Node

A point in finite-element data.

Number Format

The style of numbers to display for a data or axis label; exponent, integer, float, and so forth.

OpenGL

A graphics library for high-end 3-D graphics. It usually takes advantage of hardware acceleration for 3-D rendering.

Ordered Data

A type of data point organization which consists of a parameterized series of points. There are seven types of ordered data: I-, J-, K-, IJ-, JK-, IK-, and IJK-ordered. I-, IJ-, and IJK-ordered are the most common.

Plot Type

Determines the type of plot which is displayed in a frame. For example, 2D Cartesian plot, 3D Cartesian plot, XY Line plot, Polar Line plot, or Sketch plot.

PLOT3D

A plotting package developed by NASA. Useful because the file format can be converted to a Tecplot binary data file by Preplot.

Point

A data file format for an I-, IJ-, or IJK-ordered zone in which the data is listed by point. All of the variable values for the first data point are listed first, then all the variable values for the second data point, and so forth.

Polar Line Plot

A plot of radius versus angle, or visa versa. The polar axes are the radial axis (by default zero at the origin) and theta axis (by default zero for any data on the right running horizontal line).

Precise Dot Grid

In 2-D, the points of intersection of the imaginary lines extending from the X- and Y-axes' tick marks.

Preview Image

A display of your plot as it will appear when printed.

Primary Corner

The point in an ordered zone's cell that has the minimum index values for that cell, or the first listed node of a finite-element cell.

Print File

An output file which contains a description of the plot. (Used for making hard copies.)

Print Format

The type of print output. For example. PostScript, HP-GL/2, and so forth.

Quadrilateral

An element type of finite-element surface data which is composed of four node points arranged in a quadrilateral. Used in 2- and 3-D surface plotting.

Quick Macro Panel

A user-defined panel accessed from the Tool menu which allows quick access to your macro functions.

Rake

A specified line from which two or more streamtraces are generated.

RGB Color Flooding

The assignment of color based on Red, Green, and Blue components defined at field data locations.

Ribbon

(See *Streamribbon*.)

Rod

(See *Streamrod*.)

Scatter

A 2- or 3-D field plot type which plots a symbol at each data point.

Shade Plot

A 2- or 3-D field plot type which plots solid color or colors with lighting effect over the cells of the data.

Sharing

Variable sharing allows a single storage location to be used by more than one party. For example, if the X-variable is shared between zones five and seven only one storage location is created. The storage is not freed by tecplot until the number of parties accessing the data is reduced to zero. Variables and connectivity information may be shared.

Sidebar

The area to the left of the Tecplot workspace.

Sketch Plot

A plot which displays only text and geometries. These plots are in the Sketch plot type.

Slice

A set of data created by the intersection of a plane with 3-D zones.

Snap-to-Grid

Lock any object on the screen to the closest grid point. The position and size of the object will be affected by changes to the grid.

Snap-to-Paper

Lock any object on the screen to the underlying paper. The position and size of the object will not be affected by changes to the grid.

Sort

A measurement from one to two of the amount of work Tecplot should do to resolve hidden-surface problems during 3-D sorting. Selecting two will increase the time required for each redraw and will generate messages about the number of cells with a potential conflict.

Specular Highlights

Rendering a surface such that it displays qualities similar to those of a smooth reflecting surface such as metal.

Step Size

The fraction of a cell over which Tecplot streamtraces are integrated. Step size in Tecplot is variable, changing with the vector field and the size and aspect ratio of the cells.

Stream

An option of vector plots to plot particle traces through the vector field.

Stream Format

The current type of streamtraces being placed in Tecplot. For example, Surface Line, Volume Line, Volume Ribbon, or Volume Rod.

Streamline

A 2- or 3-D line which is parallel to the vector filed along its entire length. For a steady state vector field, this is the same as a simple particle trace which marks the path of a massless particle in the vector field.

Streamribbon

A particle trace with a width which not only follows the flow field (its center being a regular streamline), but which also twists with the vorticity of the vector field.

Streamrod

A particle trace with a polygonal cross-section and a width which not only follows the flow field (its center being a regular streamline), but which also rotates with the vorticity of the vector field.

Streamtrace

Any type of particle trace: streamlines, streamribbons, or streamrods.

Streamtrace Zone

Any streamtrace which has been extracted to form a new zone.

Stylesheet

A type of file which contains the definition of how the plot in a single frame is to be plotted. The stylesheet does not contain any zone data but does contain information about views, axes positions, zone attributes, and so forth.

Supersampling Factor

When antialiasing an image for export, the factor Tecplot uses when creating an intermediate image that is then resized down to the final image size. The larger the value, the smoother the resulting image at the cost of performance. Values of more than 3 are seldom necessary.

Surface Line

A type of 3-D streamline which is confined to remain on a 3-D surface. Also used to refer to 2-D streamlines.

Tetrahedron

An element type of finite-element surface data which is composed of four node points arranged in a tetrahedron. (Used in 3-D volume plotting.)

Translucency

A property allowing you to see through an object to areas within or beyond it. In Tecplot you may vary the amount of translucency, controlling the extent that an object closer to you obscures one it overlays.

Triangle

An element type of finite-element surface data which is composed of three node points arranged in a triangle. (Used in 2- and 3-D surface plotting.)

Unordered or Unorganized Data

(See *Irregular Data*.)

Value-Blanking

A feature of Tecplot used to trim or eliminate cells based on one or more user-defined constraints for variable values.

Variable

One of the values defined at every data point in a Tecplot data set or data file.

Vector

A short line or arrow showing the direction and or the magnitude of vector qualities.

Volume Line

A type of 3-D streamline which is not confined to remain on a surface and may travel through 3-D volume data.

Volume Zone

Any zone that is IJK-ordered, finite-element tetrahedron, or finite-element brick.

Vorticity

The measurement of the tendency of a vector field to rotate about a point. (Also called "curl.")

Workspace

The portion of your screen where you can create Tecplot frames. This includes but is not limited to the region covered by the displayed paper.

XY-Dependent

A 3-D axis mode where X and Y are fixed (dependent), but Z is free to vary in ratio (independent).

XY Line Plot

Plots one variable assigned to one axis versus another variable assigned to another axis. Log plots, bar charts, curve fitted lines are all examples of XY Line plots.

Zone

A subset of a data set which is assigned certain plot types. Zones may be activated (plotted) or deactivated (not plotted). Each zone has one type of data ordering: I-, IJ-, IJK-, or finite-element. Zones are typically used to distinguish different portions of the data. For example, different calculations, experimental versus theoretical results, different time steps, or different types of objects, such as a wing surface versus a vector field around a wing.

Zone Layers

One way of displaying a 2- or 3-D plot's data set. The plot is the sum of the active zone layers, which may include mesh, contour, vector, shade, scatter and boundary.

APPENDIX E Limits of Tecplot Version 10

The following hard limits apply to Tecplot Version 10.0.

Item	Limit
Maximum number of data points per variable	Over 2 billion
Maximum number of zones per data set	32,700
Maximum number of variables per data set	32,700
Maximum number of mappings	32,700
Largest floating point absolute value	10^{150}
Smallest non-zero floating point absolute value	10^{-150}
Maximum number of picked objects	1500
Maximum number of data sets	128 (Limited by max. number of frames)
Maximum number of frames	128
Maximum number of value blank constraints	8
Maximum number of contour groups	4
Maximum number of geometries	limited by memory
Maximum number of polylines per line geometry[a]	50
Maximum number of points per circle or ellipse	720
Maximum number of custom label sets	10
Maximum number of custom labels per set	5000
Minimum frame width or height	0.1 inches
Maximum frame width	500 inches
Maximum streamtraces per frame	5000
Maximum number of color map overrides	16

Item	Limit
Maximum preview width for EPS files	1024
Maximum preview height for EPS files	1024
Maximum number of user-defined color map control points	50
Maximum number of raw user-defined color map entries	800
Maximum number of characters in variable name	128
Maximum number of characters in zone title	128
Maximum number of characters in data set title	256
Maximum number of views per view stack	16
Maximum number of characters in an auxiliary data string	32000

a. A polyline is a continuous series of line segments, and can be a subset of a line geometry.

The following soft limits may be changed via the Tecplot configuration file:

Number of:	Windows	UNIX	Hard Limit
Points per line[a]	3000	5000	500,000
Contour levels	150	400	5000
Characters per text label	1023	1023	10,000

a. Points per line is the limit on the number of points allowed in the following: line segment geometries, stream termination lines, and contour lines. For line segment geometries, this is the total number of points used in all polylines contained in the geometry.

The following hard limits apply to plot style:

Item	Limit
Printing Gouraud shaded plots with continuous flooding	On screen or exported bitmap image only

Item	Limit
Printing plots with translucency	On screen or exported bitmap image only

Index

H

P

Q

X

Y

Z