tecplot 360 ex

User's Manual

Tecplot 360 2025 Release 1

Tecplot, Inc.

Table of Contents

Part 1: Introduction to Tecplot 360

Introduction

Tecplot 360 is a powerful tool for visualizing a wide range of technical data. It offers line plotting, 2D and 3D surface plots in a variety of formats, and 3D volumetric visualization.

The user documentation for Tecplot 360 includes these resources:

User's Manual (this document)

This manual provides a complete description of working with Tecplot 360 features.

Getting Started Manual

New users are encouraged to work through the tutorials provided in the Getting Started Manual to learn how to work with key features in Tecplot 360.

Scripting Guide

This guide provides Tecplot macro command syntax and information on working with macro files and commands.

Quick Reference Guide

This guide provides a handy reference for dynamic text, macro variables, keyboard shortcuts, special characters, and more.

Data Format Guide

This guide provides information on outputting simulator data to Tecplot 360 file format.

Installation Guide

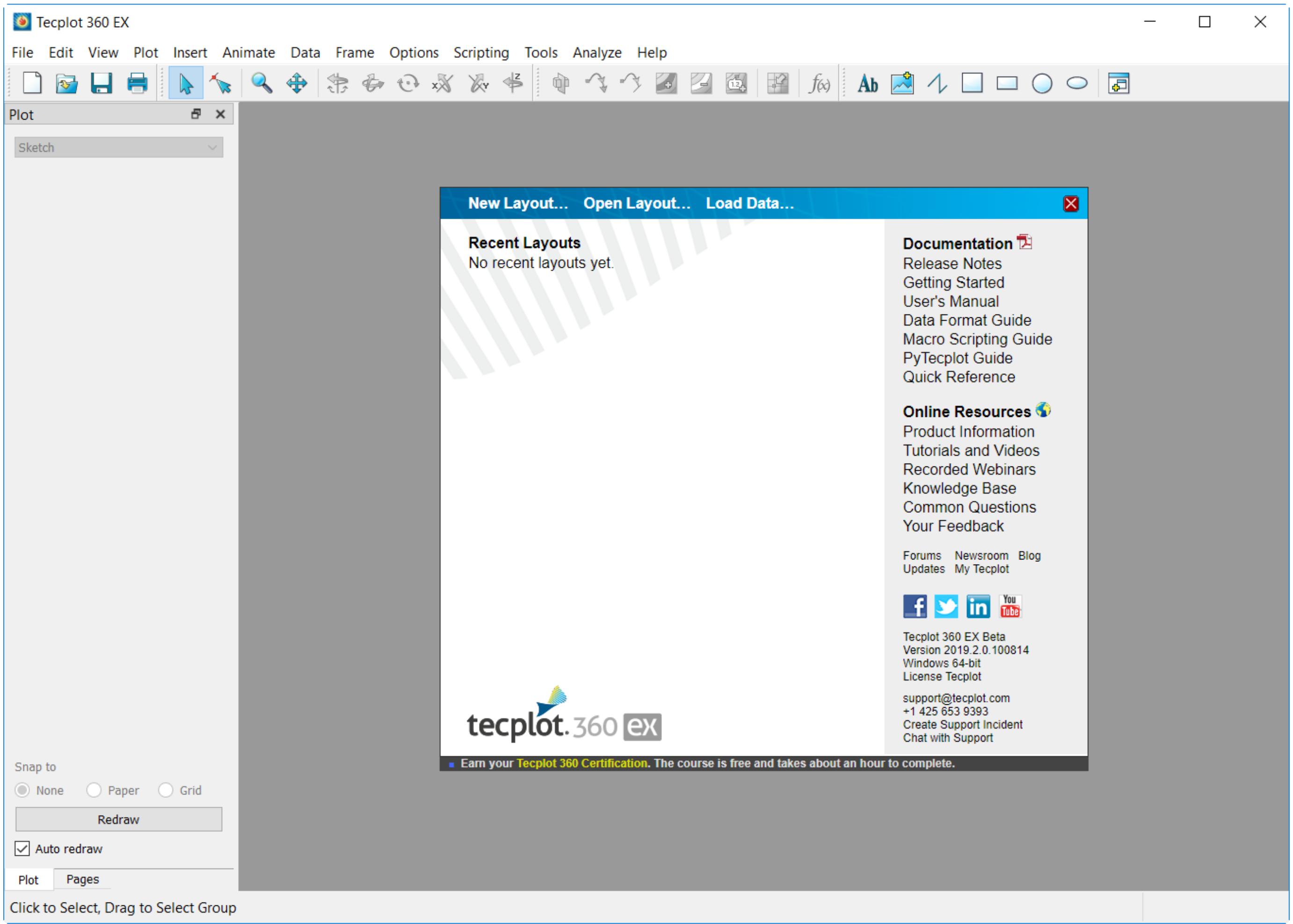
These instructions give a detailed description of how to install Tecplot 360 on your system.

Release Notes

These notes provide information about new and/or updated Tecplot 360 features.

Welcome Screen

When you start Tecplot 360, the Welcome Screen, shown below, appears. You can also open it at any time (even while working on a plot in Tecplot 360) by choosing **View→Welcome Screen**.



The Welcome Screen contains links that help you quickly get started working with your data and Tecplot 360.

- Across the top are links for creating a new layout and for opening existing layouts and data files.
- The left section provides a list of layouts you have recently worked with.
- The right column contains links to documentation and other resources to help you get the most out of Tecplot 360.
- At the very bottom of the right column is information for contacting Technical Support, including the version and platform information you will need for a support incident.

The Welcome Screen automatically disappears when you open a layout or create a new one. You may also close it manually by clicking the red X button in the upper right corner; this returns you to your existing layout if you have chosen **View**→**Welcome Screen** while you were working with a plot.

Input Devices

Tecplot 360 supports the following input devices:

Standard mouse

Used for pointing at, selecting, and manipulating objects in the workspace; changing the view;

choosing fields and manipulating controls in dialogs and sidebars; and performing actions or commands using the toolbar or menus. Most mice have multiple buttons and a scroll wheel, which provide quick access to specific functions.

Keyboard

Used for entering data and text and for activating commands using menu shortcuts. In conjunction with a mouse, can be used to switch modes when clicking or dragging.

3D mouse

A secondary input device that allows 3D Cartesian plots to be zoomed, translated, and rotated with a touch. Additionally, the always-active 3D mouse doesn't make you select mouse modes, commands, or keyboard shortcuts. A 3D mouse can therefore enable faster exploration of your data and improve the experience of working with your 3D plots.

> Tecplot 360 supports 3D mouse products from 3Dconnexion, such as the SpaceMouse, SpaceNavigator, and SpacePilot lines, and only on the Windows platform.

Actions available with a 3D mouse include:

Translation

Push the mouse cap left or right, pull it up, or push it down, to translate the plot left/right or up/down on the screen.

Zooming

Push the mouse cap toward you or away from you to zoom the plot. Hold down the Alt key while zooming to switch zoom methods.

The default zoom method depends on whether the current frame's 3D view mode is perspective or orthographic (see Three-dimensional View Details). The results of zooming using the 3D mouse and the Alt key are shown here.

Pushing or pulling mouse cap	In perspective mode	In orthographic mode
Normal (without Alt key)	Changes view distance	Changes view width
Alternate (pressing Alt key)	Changes view width	Changes view distance

Rotation

Rotation is around the defined center of rotation, by default the center of the data. However, the axis of rotation is relative to the screen rather than to the data.

X axis

Tilt the mouse cap forward or back to rotate around the screen's X axis, as if rolling your plot toward you or away from you.

Y axis

Twist the mouse cap clockwise or counterclockwise to rotate around the screen's Y axis, as if on a lazy susan.

Z axis

Tilt the mouse cap left or right to rotate around the screen's Z axis, like the hands of a clock.

With practice, it is possible to perform more than one of these operations at the same time. At first, though, it is best to be careful to invoke only one function at a time.

Most 3D mice also have one or more buttons, which can be mapped to commands that you use frequently using the software provided with the device.

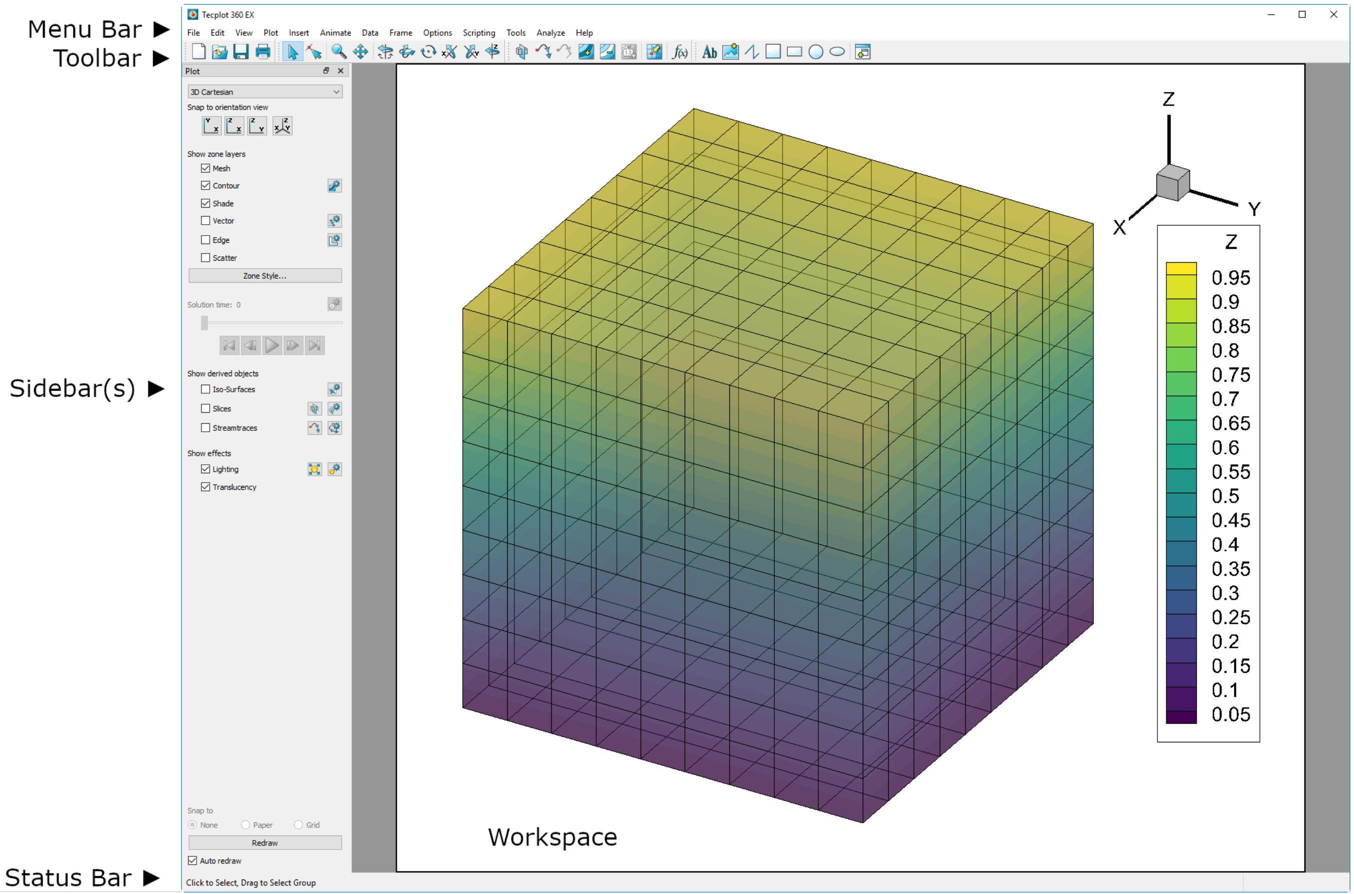
> An area of interest can be inspected in greater detail from any angle if the origin of rotation is assigned to it. Mapping the "o" key (orientation change) to a 3D mouse via the device GUI may facilitate a more natural workflow. Simply point the regular mouse cursor at the area of interest and press the orientation change button.

Tecplot 360 supports the following actions (built-in to the 3D mouse software) that may be mapped to buttons using the device software:

- Fit (same as **View**→**Fit Surfaces**)
- Front, Back, Top, Bottom, Left, and Right View
- Isometric View 1 (Tecplot 360 default view) and 2 (theta 180° from default)
- Roll CW and CCW (in 90° increments)

Interface

Five major sections make up the Tecplot 360 interface.



Menu Bar

The menu bar offers access to Tecplot 360 features.

File Edit View Plot Insert Animate Data Frame Options Scripting Tools Analyze Help

Tecplot 360's features are organized into the following menus:

File

Use the **File** menu to read or write data files and plot layouts, and print and export plots.

Edit

Use the **Edit** menu to select, undo, cut, copy, paste, and delete objects, and to change the draw order for selected items.

Generally, **Cut**, **Copy**, and **Paste** of items other than text work only within Tecplot 360. You can, however, copy an image of a selected frame or the text of a selected text object to the clipboard. See Clipboard Exporting to Other Applications.

View

Use the **View** menu to manipulate the way your data is shown or to open a sidebar or the Welcome Screen. See View Modification.

Plot

Use the **Plot** menu to control the style of your plots. The menu items available depend on the active plot type (chosen in the Plot sidebar).

Insert

Use the **Insert** menu to add text and geometries (polylines, squares, rectangles, circles, and ellipses). If you have a 3D zone, you may also use the **Insert** menu to insert a slice. If the plot type is set to 2D or 3D Cartesian, you may insert a streamtrace.

Animate

Use the **Animate** menu to create animations of your plot.

Data

Use the **Data** menu to create, manipulate, and examine data. Types of data manipulation available in Tecplot 360 include zone creation, interpolation, triangulation, and creation or alteration of variables.

Frame

Use the **Frame** menu to create, edit, and control frames.

Options

Use the **Options** menu to control your Tecplot 360 experience, including rulers, grids, and performance.

Scripting

Use the **Scripting** menu to play or record macros, and to access the **Quick Macros Panel**.

Tools

Use the **Tools** menu to launch an add-on.

Analyze

Use the **Analyze** menu to examine grid quality, perform integrations, generate particle paths, extract flow features, and estimate numerical errors.

Help

Get help for Tecplot 360 features, or view the About dialog, which contains version and platform information.

Context Menu and Toolbar

In many places within Tecplot 360, clicking with the right mouse button displays a *context menu* that gives you quick access to just the operations most relevant to what you're clicking. Often, these operations are also available in some other way, such as using a toolbar button, the pull-down menus, or a dialog.

In the Tecplot 360 workspace, the following objects have context menus with the specified functions:

Frames

Change frame order, load and save frame style, frame details

Axis

Axis settings

Slices

Hide, edit slice details, extract slices, slice style

Iso-surfaces

Hide, choose variable, edit iso-surface details, extract surfaces, change iso-surface style

Zones

Hide, deactivate, zone style

Streamtraces

Extract, delete, streamtrace settings

3D orientation axis

Hide axis, axis settings

Line maps

Deactivate, copy, change curve type, mapping style, Fourier transform (when applicable), show curve details on plot; write curve details or data to file

Legends

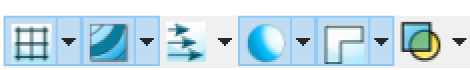
Hide, change box type, legend settings

Geometries (shapes)

Change line and fill color, delete, edit geometry settings. For polylines, you may also extract data along the line.

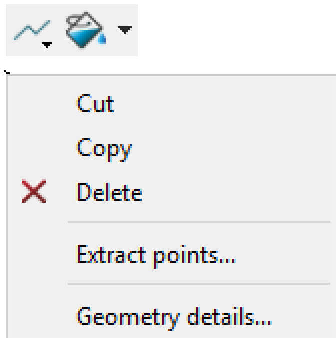
Text

Change color, cut, copy, delete, align(left, right, center, top, bottom), text settings.

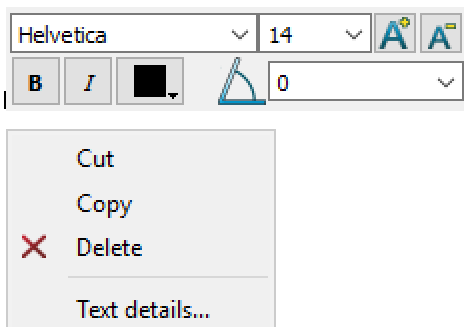


When working with zones, slices, and iso-surfaces, a context toolbar appears above the context menu. This toolbar allows you to turn on or off the grid, contour, vector, shade, edge, and translucency layers for the selected object(s) by clicking the icons. (Iso-surfaces do not have vector or edge layers.) Additionally, you may adjust frequently-used style settings for each layer using the drop-down menu to the right of each icon, for example selecting a color for the grid (or choosing a variable by which to color it).

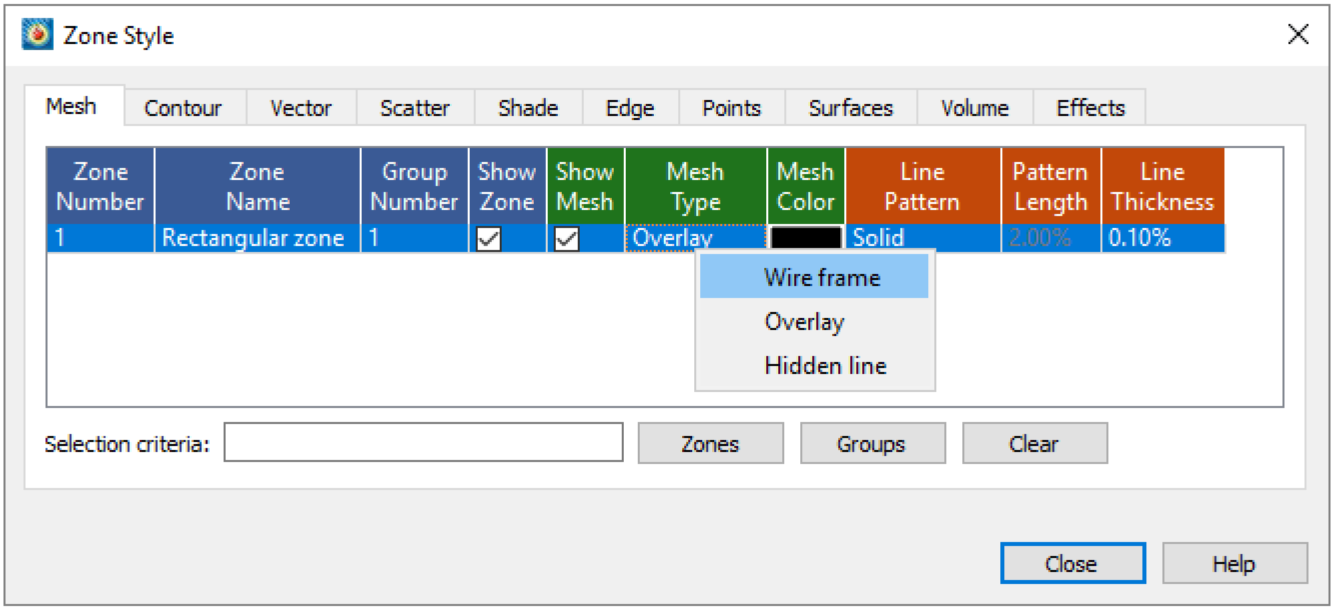
A different context toolbar appears when you right-click a line in XY or polar line plots. This toolbar allows you to turn on or off the line, symbol, and error bars for the selected line map by clicking the icons. You can also adjust the attributes of these (for example, line color, symbol shape, or error variable) using drop-down menus to the right of each icon.



When a geometry object is right-clicked, the Geometry context menu appears. This menu allows you to Cut, Copy, Delete the object as well as adjust the line and fill attributes. When multiple geometries are selected, the Alignment of the objects can be adjusted as well.



Right-clicking on a Text object brings up the Text context menu. This menu allows you to Cut, Copy, Delete the object as well as change the alignment or open the Text details dialog. You can also adjust the text attributes such as bold, italics, color, and text angle.



In some places, such as in the Mapping Style and Zone Style dialogs, right-clicking is the primary way to alter the displayed data.

The right mouse button is also used for translating (moving) the plot. This doesn't conflict with its use for context menus. The context menu appears when you right-click without moving the mouse; *dragging* with the right mouse button performs the translation operation.

Sidebars

Tecplot 360's three main sidebars provide easy access to frequently-used functionality:

Plot Sidebar

Includes controls for manipulating the appearance of your plot.

Pages Sidebar

A list of the pages currently open, allowing you to switch between them, and to create, rename, and delete pages.

Frames Sidebar

A list of the frames in the current page, allowing you to easily manage their order and other characteristics.

Initially, the Plot and Pages sidebars appear "docked" on the left side of the workspace—attached to the left side of the plot area. The Frames sidebar is initially hidden, but appears on the left side of the workspace when enabled using the **View** menu.

Any visible sidebar may be moved to the right side of the workspace, or even dragged out of the workspace entirely (for example, to move it to another display) by dragging its title bar.

When more than one sidebar is docked to the same side of the workspace, they can be combined in two ways:

Tabbed panels

Only one sidebar is visible at a time; tabs appear at the bottom of the sidebar are to choose which sidebar you want to use. This is the default mode; the screen image in Interface shows how this looks.

Sharing space vertically

Both sidebars are visible at the same time. You can drag the boundary between the two sidebars to adjust the proportion of the space used by each.

Which style is used depends on where you dock the second sidebar: drag to the top or bottom of the already-docked sidebar to split the area, or drag to the middle to combine them and use tabs to switch between them.

Additional sidebars are available for certain other features throughout Tecplot 360. For example, the Quick Macro Panel is a sidebar that initially appears docked to the right side of the workspace. Like the other sidebars, it may be "torn off" from the window or share space with another sidebar. The Probe Sidebar is another; it displays the results of probe operations.

Any sidebar may be closed if it is in your way. To open it again, choose the desired sidebar from the appropriate menu, or right-click any sidebar or the menu bar and choose the desired sidebar from the context menu.

Sidebar	Menu Command
Plot	View → Plot sidebar
Pages	View → Pages sidebar
Frames	View → Frames sidebar
Probe	Data → Probe sidebar
Quick Macro Panel	Scripting → Quick Macros

Plot Sidebar

The controls available in the Plot sidebar depend on the plot type of the active frame. For 2D or 3D Cartesian plot types, you can show or hide zone layers, and zone effects, and derived objects from your plot. For line plots (XY and polar) you can show or hide mapping layers.

You can open the Plot sidebar from the **View** menu if it is not currently visible. To customize your plot, simply:

- Select a plot type from the Plot Types drop-down menu.
- Use the toggle switches to add or subtract Map Layers/Zone Layers, or Zone Effects, or Derived Objects. Use the **Zone Style**/**Mapping Style** dialogs to further customize your plot by showing or

hiding zones in specific plot layers/mappings, changing the way a zone or group of zones is displayed, or changing various plot settings.

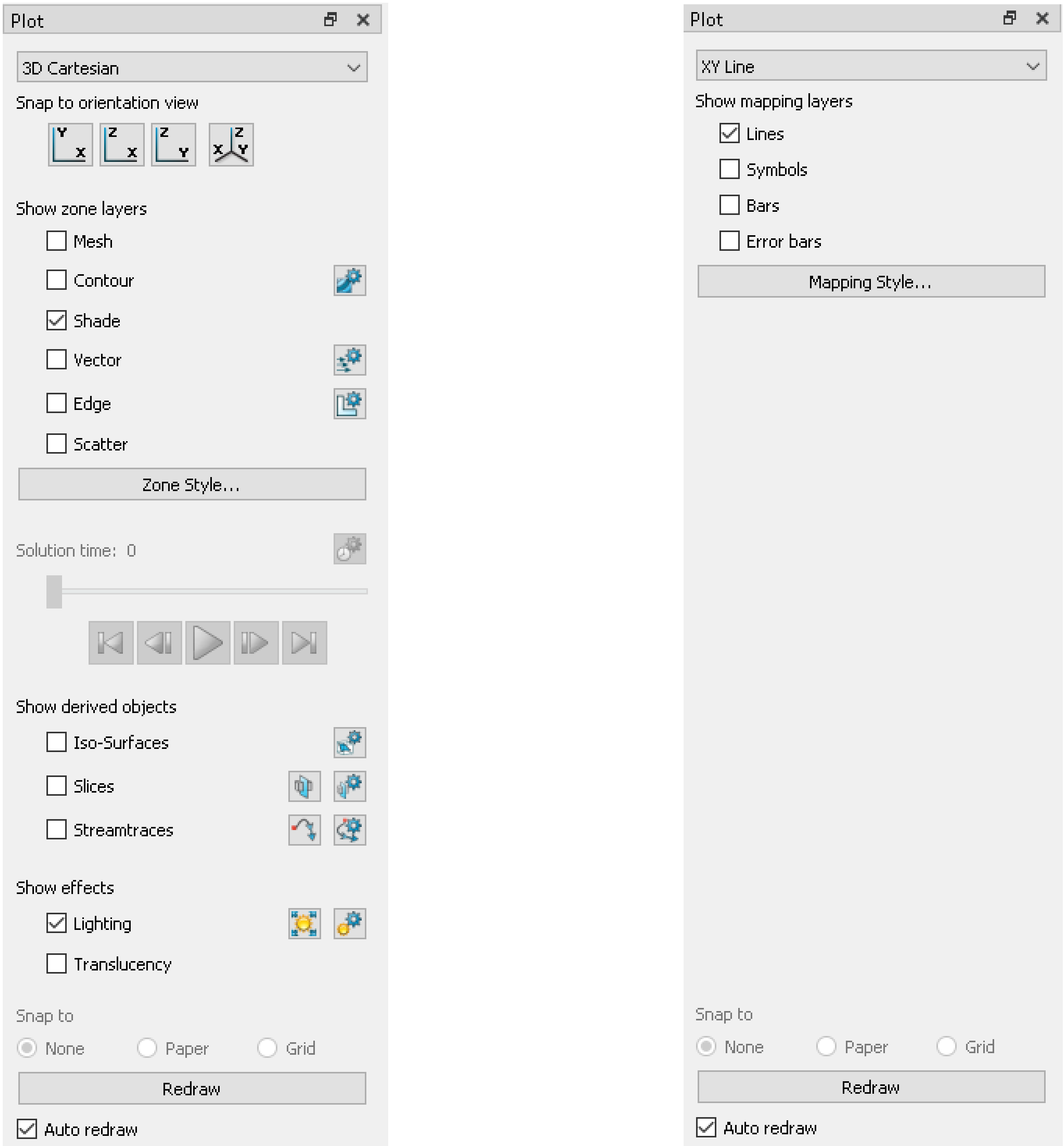


Figure 1. The Plot sidebar as it appears for a 3D Cartesian plot (left) and XY Line plot (right).

Plot Types

The Plot Type, combined with a frame's dataset, active layers, and their associated attributes, define a plot. Each plot type represents one view of the data. There are five plot types available:

3D Cartesian

3D plots of surfaces and volumes.

2D Cartesian

2D plots of surfaces, where the vertical and horizontal axis are both dependent variables (i.e. $x = f(A)$ and $y = f(A)$, where A is another variable).

XY Line

Line plots of independent and dependent variables on a Cartesian grid. Typically the horizontal axis (x) is the independent variable and the y-axis a dependent variable, $y = f(x)$.

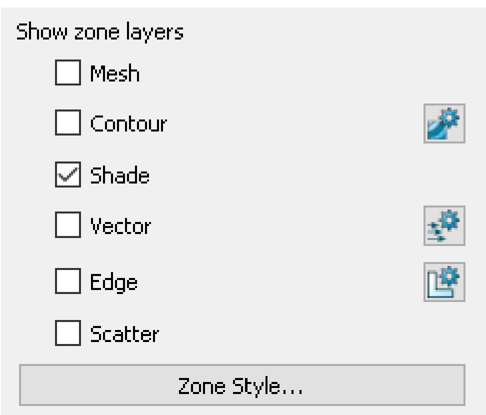
Polar Line

Line plots of independent and dependent variables on a polar grid.

Sketch

Create plots without data such as drawings, flow charts, and viewgraphs.

Zone Layers



A layer is a way of representing a frame's dataset. 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. The six zone layers for 2D and 3D Cartesian plot types are:

Mesh

A grid of lines connecting the data points within each zone.

Contour

Iso-valued lines, the region between these lines can be set to contour flooding.

Shade

Used to tint each zone with a solid color, or to add light-source shading to a 3D 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.

Vector

The direction and magnitude of vector quantities.

Edge

Zone edges and creases for ordered data and creases for finite element data.

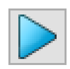
Scatter

Symbols at the location of each data point.

Zone Style

Select the **Zone Style** button to launch the **Zone Style** dialog. The **Zone Style** dialog is used to customize the zone layers that you have added to your plot. Refer to the chapter for each zone layer for details on working with the **Zone Style** dialog.

Transient Controls

When working with transient data, simply press the Play  button in the Plot sidebar to animate over time. The active frame will be animated from the Current Solution Time to the last time step. You may also drag the slider to change the Current Solution Time of your plot.

The Animation Controls have the following functions:

Jumps to the Starting Value. (Keyboard: Home)

Jumps toward the Starting Value by one step. (Keyboard: Left arrow)

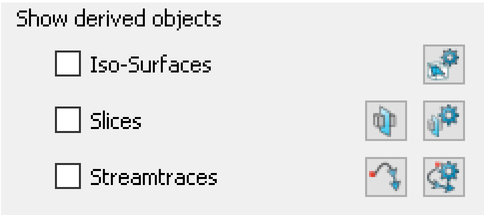
Runs the animation as specified by the 'Operation' field of the **Time Details** dialog. The Play button becomes a Pause button while the animation is playing. (Keyboard: Space bar)

Jumps toward the Ending Value by one step. (Keyboard: Right arrow)

Jumps to the Ending Value. (Keyboard: End)

Use the  button to launch the **Time Details** dialog. See Time Aware for more information on Time controls and the **Time Details** dialog.

Derived Objects

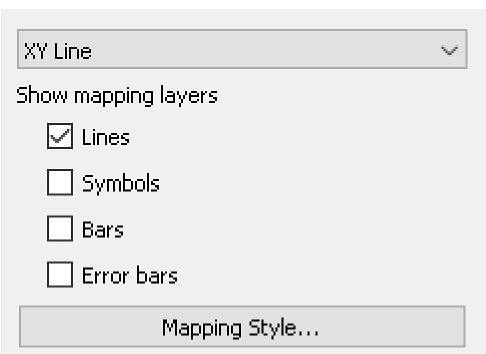


For Cartesian plot types (2D and 3D): Toggle-on Iso-surfaces, Slices, or Streamtraces from the Plot sidebar to add any or all of these elements to your plot. For convenience, the tool buttons for Slices and Streamtraces also appear here. The **Details** dialogs for each of the derived objects can be accessed via their respective buttons (, ,). Refer to Iso-Surfaces, Slices, or Streamtraces for details on working with these objects.

Zone Effects

For 3D Cartesian plot types, use the Plot sidebar to turn lighting and translucency on or off. Only shaded and flooded contour surface plot types are affected. Refer to Shade Layer and Translucency for additional information.

Map Layers



A layer is a way of representing a frame’s dataset. 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.

The four XY Line map layers are:

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.

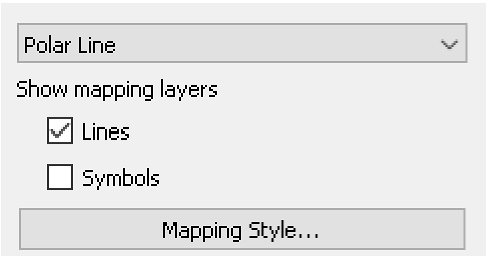
Bars

A pair of variables, X and Y, as a horizontal or vertical bar chart.

Error Bars

Allows you to add error bars to your plot.

The two map layers for Polar Line are:



Lines

A pair of variables, X and Y, as a set of line segments or a fitted curve.

Symbols

A pair of variables, e.g. X and Y, as individual data points represented by a symbol you specify.

Select the **Mapping Style** button to launch the **Mapping Style** dialog. The **Mapping Style** dialog allows you to customize the style settings for each of the plot layers and specify the points to plot. The pages of the dialog are discussed in detail in XY and Polar Line Plots.

Snap Modes

Snap modes allow you to easily place objects at convenient reference points, either on the axis grid or on the workspace paper. Most movable objects (including text, images, geometries, frames, axes, legends, and the orientation axis) can be snapped. Data points in zones and line maps are also subject to snap modes when being adjusted.

The snap controls in the Plot sidebar become available when such objects are selected using the Selector or the Adjustor or, in the case of text and geometries, when they are being placed initially using the appropriate tool.

None

No snapping is performed; objects may be positioned freely.

Snap to Grid

Constrain object movement to whole steps on the axis grid as defined in the Grid page of the Axis Details dialog (**Plot**→**Axis**); see Axis Grid Options. This can be useful for aligning objects with points of interest in a plot.

Snap to Paper

Constrain object movement to whole steps on the paper's grid as defined in the Ruler/Grid dialog (**Options**→**Ruler/Grid**); see Grid and Ruler Set-Up. This can be useful for positioning frames precisely for printing, or for absolute positioning of text, geometries, and other plot elements.

One or the other of the snap modes may be unavailable depending on the plot type or the kind of object being moved. For example, a frame cannot be aligned to the grid because it *contains* the grid, so the Grid snap mode is disabled when moving a frame.

Redraw Buttons

The redraw buttons allow you to keep your plot up to date: Clicking the middle mouse button redraws the current active frame.

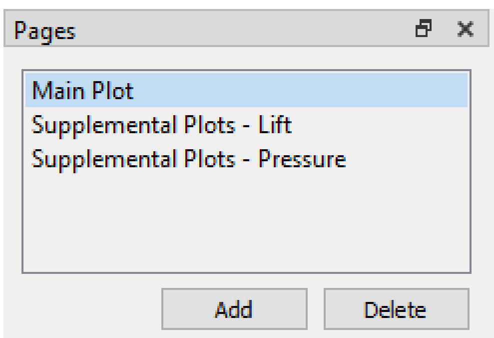
Auto Redraw

Use Auto Redraw

When selected, the plot will be automatically redrawn, whenever style or data changes. Some users prefer to turn this option off while setting multiple style settings and then manually press the **Redraw** or **Redraw All** button on the Plot sidebar to see a full plot.

> You can interrupt an auto-redraw at any time with a mouse click or key press.

Pages Sidebar



In Tecplot 360, a page is a container for any number of frames, each of which is a container for a plot. A layout can contain any number of pages (although many layouts contain only one). You can use the Pages sidebar to see the pages in the current layout, to choose which page is displayed in the workspace, and to create, rename, and delete pages.

The Pages sidebar may be opened from the View menu if it is not currently visible. In the Pages sidebar, you may:

- Click **Add** to add a new page. The new page is initially named "Untitled."
- Click a page to view it in the workspace. Only one page is visible at a time.
- Double-click a page to specify or change its name. The name becomes editable in the sidebar; simply type the new name, then press Enter.
- Click **Delete** to delete the active page.

> The **Cut**, **Copy**, and **Paste** commands on the Edit menu can be used to move or copy frames between pages. Click a frame border to select it, copy or cut it to the clipboard,

then switch to the destination page using the Pages sidebar and paste it.

Frames Sidebar

The Frames sidebar allows you to manage the frames on the current page, including activating and ordering them, which is particularly useful when you have many overlapping frames on a page. It is initially hidden, but may be opened from the **View** menu. See Frames for more information on working with frames.

Toolbar

Each of the tools represented in the Toolbar changes the mouse mode and allows you to interactively edit your plot.

The Toolbar is actually up to four separate toolbars, with buttons grouped by function.

File toolbar

Buttons for dealing with files (hidden by default)

View toolbar

Buttons for manipulating the view

Data toolbar

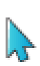
Buttons for working with data

Insert toolbar

Buttons that insert objects into the workspace.

In the default toolbar configuration, shown above, the toolbars appear next to each other. Each, however, can be moved independently to any edge of the workspace by grabbing the drag handle at the left edge of the toolbar. To hide or show a toolbar, right-click the menu bar and choose the desired toolbar.

Selector Tool

 Use the Selector tool to select objects in your workspace. To select multiple objects, hold down Shift while clicking the second and subsequent objects. The selected objects can then be moved (translated) using the Selector tool itself.

The following kinds of objects can be moved (translated) using the Selector tool:

- frames
- axis grid area
- text

- geometries
- contour labels
- streamtraces
- streamtrace termination line
- legends
- 3D frame axis

To select an object and open that object's attributes dialog, double-click any object

Adjustor Tool

Use the Adjustor tool to perform any of the following modifications to your plot and data:

- Location of individual or groups of data points in the grid.
- Values of the dataset variables at a particular point.
- Length or placement of individual axes (2D Cartesian and XY Line plot types only).
- Spacing between an axis label and its associated axis (2D Cartesian and XY Line plot types only).
- The points in a polyline (and therefore its shape).

The Adjustor tool behaves like the Selector tool when you click objects other than those listed above. For example, if you click a rectangular geometry, the entire object is selected, since the Adjustor tool has no special functionality with a rectangular geometry. When adjusting a polyline, click close to the points in the polyline to avoid the entire object being selected.

> The Adjustor tool can alter your data. Be sure you want to use the Adjustor tool before dragging points in the data region.

Click a point or object to select it. To select multiple points, you can hold Shift while clicking additional points after the first. In line plots, you can select points from only one mapping at a time.

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 first data point. The other selected points will move as a unit with respect to 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` key on your keyboard while adjusting the selected point. The `H` and `V` keys restrict the adjustment to the horizontal and vertical directions, respectively.

Zoom Tool

Zoom into or away from the plot. Geometries zoom with the plot; however, text does not.

When a mouse-click occurs (without dragging), the zooming is centered at the location of your click.

There are two zoom modes: plot zooming and paper zooming.

For plot zooming

drag the magnifying glass cursor to draw a box around the region that you want to fit into the frame. The box may be larger than the frame. Making the box larger than the frame zooms away from the plot. The region within the view box will be resized to fit into the frame.

> If **Snap to Grid** (located in the Sidebar) is selected, you cannot make the zoom box larger than the grid area.

To return to the previous view, choose the **View**→**Last** menu (`Control-L`). To restore the original 2D view, choose **View**→**Fit Everything** (`Ctrl-E`) or **View**→**Fit to Full Size** (`Ctrl-F`).

The results of plot zooming for the 2D plot type are dependent upon the axis mode selected in the **Axis Details** dialog (accessed via the **Plot** menu):

2D Independent Axis Mode

Allows the selected region to expand to exactly fit in the frame. The axes are rescaled independently to fit the zoom box.

2D Dependent Axis Mode

In dependent mode, the axes are not fit perfectly to the zoom box. The longest dimension from the zoom box is applied to an associated axis, and the other axis is resized according to the dependency relation.

For paper zooming

Shift-drag the magnifying glass cursor to draw a box about the region that you want to magnify. The plot is resized so that the longest dimension of the zoom box fits into the workspace. You can fit all frames to the workspace by using **Fit All Frames to Workspace** on the **View** menu.

> Clicking anywhere in your plot while the zoom tool is active will center the zoom around your click. Alternatively, Control-click centers the plot on the point that was clicked and zooms out.
>
> Use the center mouse button and drag (or hold down the scroll wheel and drag) to interactively zoom into or out of the plot.

3D mouse

Tecplot 360 also supports a 3D mouse, an input device that allows zooming (and translation and rotation) of 3D plots with a touch. You do not need to activate the Zoom tool to use the 3D mouse for zooming. Simply push the mouse's cap toward or away from you to zoom. Hold the Alt key while zooming with the 3D mouse to switch the zoom method (see Input Devices for details).

Avoid tilting the mouse cap, as this may induce unintended rotation.

Translate Tool

Use the Translate tool to translate data within a frame or the paper within the workspace.

While in Translate mode, drag the cursor to move the data with respect to the frame, or Shift-drag to move the paper with respect to the workspace.

> You may translate even when some other tool is selected by dragging with the right mouse button.

While the Translate tool is active, you can zoom your image by pressing + to magnify, - to shrink. If you are Shift-dragging to move the paper, the rescale buttons + and - will magnify or shrink the paper, as long as you have the mouse button depressed.

3D mouse

Tecplot 360 also supports a 3D mouse, an input device that can be used to translate 3D plots at any time with just a touch, regardless of the tool or mode selected.

- Pull the mouse's cap up or push it down to translate up or down on the screen.
- Move the mouse's cap left or right to translate left or right on the screen.

  Avoid tilting or rotating the mouse when using these motions to avoid rotating the plot.

Three-dimensional Rotation

There are six 3D rotation mouse modes:

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 a 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 selected a rotation mouse mode, you can quickly switch to any of the others using the following keyboard shortcuts:

Drag	Rotate about the defined rotation origin with your current Rotate tool.
Alt-drag	Rotate about the viewer position using your current Rotate tool.
Middle-click-and-drag/Alt-right click-and-drag	Smooth zoom in and out of the data.
Right-click-and-drag	Translate the data.
Control-right-click-and-drag or Command-right-click-and-drag (Mac)	This option can be used without first selecting a rotation mouse mode. Simply hover over your intended point of origin, and then Control-right-click-and-drag to translate the image.

3D mouse

Tecplot 360 also supports a 3D mouse, an input device that can be used to rotate 3D plots at any time regardless of the tool or mode selected. The axis of rotation is relative to the screen rather than to the data.

- Tilt the mouse's cap forward or backward to rotate around the screen's X axis, as if rolling toward or away from you.
- Twist the mouse's cap clockwise or counterclockwise to rotate around the screen's Y axis, as if on a lazy susan.
- Tilt the mouse's cap left or right to rotate around the screen's Z axis, like the hands of a clock.

Slice Tool

 Use the Slicing tool to control your slice(s) interactively.

The following keyboard/mouse options are available when the Slice tool is active:

+	Primary Slices, Start End Slices Active - Turn on intermediate slices (if not already active) and adds a slice. Primary Slices active [ONLY] - Turns on Start/End Slices and adds a slice. Start/End Slices active [ONLY] - Turns on Start/End Slices and adds a slice.

-	Primary Slices, Start End Slices Active - Removes start and end slices. Primary Slices active [ONLY] - Removes the primary slice. Start/End Slices active [ONLY] - Removes the Start and End Slices.
Click/Drag	Updates the position of the primary slice (if active). If only start and end slices are visible, click updates the position of the starting slice.
Alt-click/Alt-drag	Determine the XYZ-location by ignoring zones and looking only at derived volume objects (streamtraces, slices, iso-surfaces).
Shift-click	Switches from one Primary slice to Start/End Slices by adding a slice.
Shift-drag	Move the start or end slice (whichever is closest to the initial click location). Show Start/End Slices is activated, if necessary.
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.
1-8	Numbers one through eight switch to the corresponding slice group.

Add Streamtrace

 Select the Add Streamtrace tool to add a streamtrace interactively by clicking anywhere in your plot. Select the number of streamtraces to include with each click (rake) using 1-9 on the keyboard.

Keyboard Shortcuts

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

Shift
: Draws a rake on concave 3D volume surfaces. These rakes are normally not drawn, as they occur outside of the data

Refer to Streamtraces for more information.

Streamtrace Termination Line

 Select the Add Streamtrace Termination Line tool to add a streamtrace termination line interactively.

To draw a Streamtrace Termination Line:

- Move the cursor into the data region.
- Click once at the desired starting point for the line.
- Click again at each desired break point.
- When the polyline is complete, double-click on the last point of the polyline, or press ESC on your keyboard.
- The drawn polyline ends any streamtraces that pass through it.

Add Contour Level

 Select the Add Contour Level tool to add a contour level by clicking anywhere in the active data region. A new contour level, passing through the specified location, is calculated and drawn.

Alt-click	Place a contour line by probing on a streamtrace, slice, or iso-surface.
Click	Place a contour line.
Control-Click	Replace the nearest contour line with a new line.
Drag	Move the new contour line.
-	Switch to the **Delete Contour Level** tool.

The following keyboard and mouse shortcuts are related to the **Add Contour Level** tool.

Delete Contour Level

 Select the **Delete Contour Level** tool to delete a contour level by clicking anywhere in the active data region. The contour line nearest the specified location is deleted.

> Use the + key to switch to the **Add Contour Level** tool and the - key to switch back to the **Delete Contour Level** tool.

Add Contour Labels

 Select the **Add Contour Label** tool to switch to the Contour Label mode, enabling you to add a contour label by clicking anywhere in the active data region.

A contour label is added to the plot at the specified location; its level or value information is taken from the nearest contour line. This allows you to place labels at a slight offset from the lines they label.

The Contour type must be lines or lines and flood in order for this tool to be active. You can set the contour type on the Contour page of the **Zone Style** dialog.

Probe Tool

Select the **Probe At** tool to probe for values of the dataset's variables at a particular point.

To obtain interpolated values of the dataset variables at the specified location, click at any point in the data region.

To obtain exact values for the data point nearest the specified location, Control-click at the desired location.

For XY plots, when you move into the axis grid area, the cursor cross hair is augmented by a vertical or horizontal line, depending on whether you are probing along the X-axis or the Y-axis. You can change the axis to probe simply by pressing X to probe the X-axis or Y to probe the Y-axis.

Specify Equations

Opens the Specify Equations dialog. See Data Alteration through Equations.

Insert Text

Select the Add Text tool to add text to any frame.

When this tool is in use, the Snap To buttons in the Plot sidebar become available to allow you to easily align your text with features of your plot.

Insert Image/Georeferenced Image

Select the Insert Image tool to insert an image or Georeferenced image to the plot.

See Images and Georeferenced Images for more information.

Insert Geometries

Use the geometry buttons to insert geometries into your plot. When these tools are in use, the Snap To buttons in the Plot sidebar become available to allow you to easily align your geometry with features of your plot.

- Polylines
- Squares
- Rectangles

- Circle
- Ellipse

Create New Frame

Select the Create Frame tool to create a new frame.

To add a frame:

- Click once in the workspace to anchor one corner of the frame.
- Drag the diagonal corner until the frame is the desired size and shape.

> If you have data loaded before you create a new frame, you can attach the existing dataset to the new frame by changing the plot type.

Status Line

The status line appears along the bottom of the Tecplot 360 window to provide a progress bar and other information when Tecplot 360 computes lengthy calculations. Other "hints," such as coordinates and tips for using the currently-selected tool, also appear here.

Tecplot 360 Workspace

In the Tecplot 360 *workspace*, you can create sketches and plots. You create each sketch or plot within a *frame*. Each visible measure of the workspace is called a *page*. The current state of the workspace, including the sizing and positioning of frames, the number and contents of pages, the location of the data files used by each frame, and all current attributes for all frames, makes up a *layout*. By default, the workspace displays a representation of where the paper plots are drawn, as well as a reference grid and rulers. The frame most recently selected is called the *active frame*.

You can include multiple pages in your layout, each of which can contain multiple frames. Use the Pages Sidebar (open it from the View menu if it’s not visible) to create and manage pages.

Getting Help

Tecplot 360 features a fully integrated Help system. Detailed help is accessible by:

- Selecting **Tecplot 360 Help** from the **Help** menu.
- Clicking the **Help** button in any dialog.

The **Help** dialog supports text search, has hypertext links, and provides detailed information on all menus and dialogs.

You may also send an e-mail to support@tecplot.com with your questions.

Using the Workspace

This chapter discusses the structures and features of Tecplot 360 that act the same regardless of the data type or plot layers you are using. These include:

Data Hierarchy

How Tecplot 360 manages data.

Interface Coordinate Systems

Tecplot 360's use of different coordinate systems, and when and where they occur.

Frames

Areas in the workspace in which you can create plots and control formatting.

Workspace Management Options

Factors that determine the color and orientation of your paper, as well as the ruler and grid, in order to precisely size and position objects. For in-depth information on Display Performance, refer to Performance Dialog.

View Modification

The commands to zoom, translate, and fit plots within frames.

Edit Menu

The commands to cut, copy, and paste plot elements in the workspace.

Data Hierarchy

Tecplot 360 structures data in two levels: datasets, which display in frames, and zones, which make up a dataset. Each dataset is composed of one or more zones, and each zone contains one or more variables. All zones within a dataset contain the same set of variables.

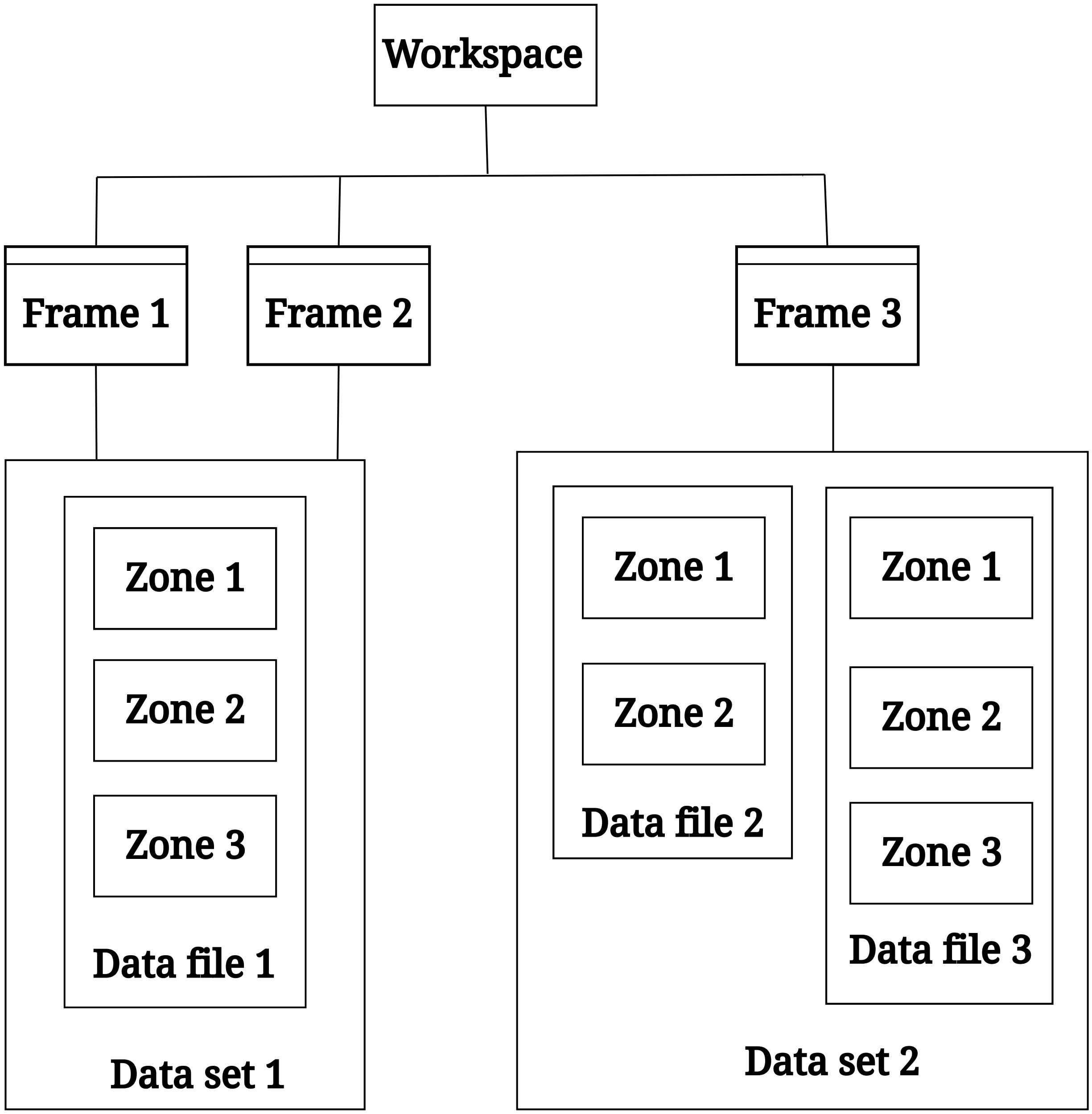


Figure 2. This chart displays a visual representation of Tecplot 360's data hierarchy. Frames 1 & 2 display Dataset 1, and Dataset 1 contains three zones from one data file. Frame 3 contains Dataset 2, which is composed of five zones (two from data file 2 and three from data file 3).

Data Displays in Frames

Tecplot 360 always displays data in a window called a "frame". By default, Tecplot 360 opens one frame when you launch Tecplot 360 or open a new layout, but you can display multiple plots concurrently in Tecplot 360 by adding additional frames. You can add frames to the workspace using the **Frame** menu. You can display one dataset in one frame, or you can share one dataset between multiple frames. Using the same dataset in multiple frames allows you to generate different plots of the same data. To learn more about working with frames, see Frames.

Frames appear on pages. You can manage multiple pages in Tecplot 360 using the Pages Sidebar and macro commands in the default Quick Macro Panel. Each page has its own collection of frames.

A Dataset Includes All Data in a Frame

A dataset is defined as "all of the data in a frame". To create a dataset, load one or more data files into an empty frame in Tecplot 360, or create a zone in a frame.

Datasets Are Made of Zones

A zone represents a subset of a dataset. A dataset can be composed of a single zone or several zones. Zones are either defined in a data file or created directly in Tecplot 360. The number of zones in a dataset is the sum of the number of zones in each of the data files included in the dataset, plus any zones created within Tecplot 360.

Typically, a data file is divided into zones based on its physical coordinates. For example, a dataset of an airplane may consist of a zone for each wing, each wheel, the nose, etc. Alternatively, zones may be defined based on the material. For example, a dataset of a fluid tank may have a zone for the tank itself and additional zones for each fluid inside the tank.

> All zones in a given dataset must have the same variables defined for each data point.

Interface Coordinate Systems

Tecplot 360 incorporates a number of coordinate systems, including the paper, frame, and the physical coordinate systems for the plot (2D, 3D, XY, or Polar). The origins of each coordinate system and their relationship to one another is shown in Figure 3.

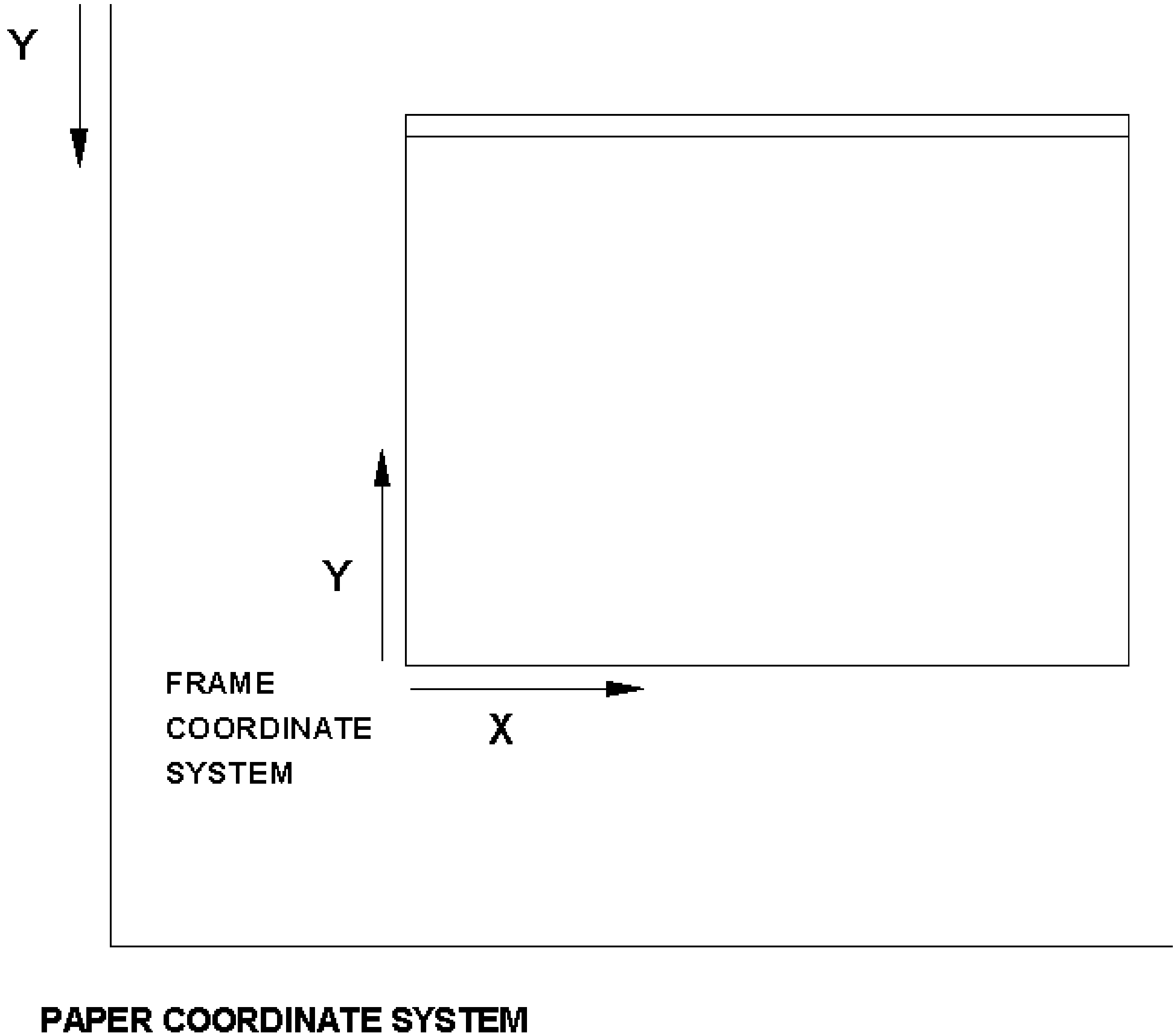


Figure 3. Tecplot 360 Coordinate System. The physical coordinate system(s) of the dataset (for example, 3D Cartesian or 2D Cartesian) are encompassed in the Frame Coordinate System.

The physical coordinate system (2D or 3D) depends on the plot type of the active frame. Two-dimensional physical coordinates are often referred to as *grid coordinates*. The Grid coordinate system is aligned with the coordinate system used by the plot axes; the Frame coordinate system is fixed to the frame and does not change when the plot is zoomed, translated, or rotated.

In 2D Cartesian plots, objects such as text labels and geometries are drawn in either the Frame or the Grid coordinate system. In 3D Cartesian plots, these objects are drawn in either the Frame coordinate system, or in what is known as the *Eye coordinate system*. The eye coordinate system is aligned with the Grid coordinate system; so objects drawn in the Eye coordinate system move with the data as you zoom and translate, but remain fixed when you rotate the plot.

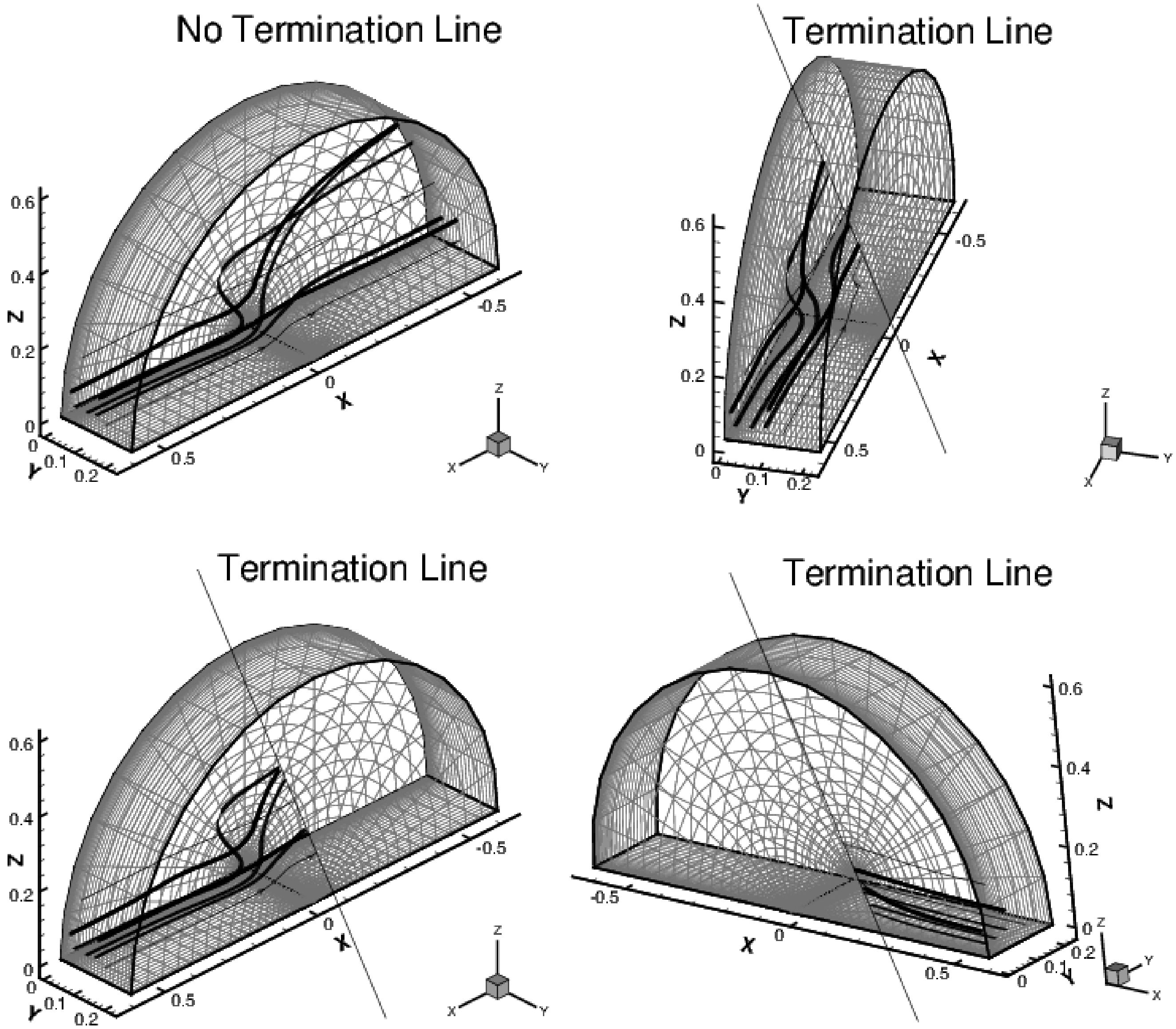


Figure 4. A 3D volume plot with streamribbons and a streamtrace termination line. This figure illustrates how the termination points vary as the plot is rotated. Notice that the termination line itself remains in place on the screen as the plot is rotated.

Frames

All plots and sketches are displayed within frames. Initially, the Tecplot 360 workspace contains one frame. You may add additional frames, resize and reposition frames, modify background color, and specify border and header appearance. Tecplot 360 acts upon only one frame, The Active Frame, at any given time (however, frames may be linked, so that changing one may also change others). The active frame displays with a darker, thicker border than the inactive frames so that you can easily see which frame is active.

Tecplot 360 uses the height of the frame for objects scaled by frame units, such as font size. When you enter a frame unit value into a dialog or when you set frame size and position on the paper, you may specify a different unit system (inches, points, centimeters, or pixels). Tecplot 360 automatically converts the values into frame units.

The Active Frame

The active frame, indicated by default with a darker and thicker border, is the frame in your workspace affected by all operations you do in Tecplot 360. To make a frame the active frame, click anywhere inside that frame with the Selector tool (or any other tool except the **Create New Frame** tool). Reference the following sections for additional actions you can perform on frames.

Frame Creation

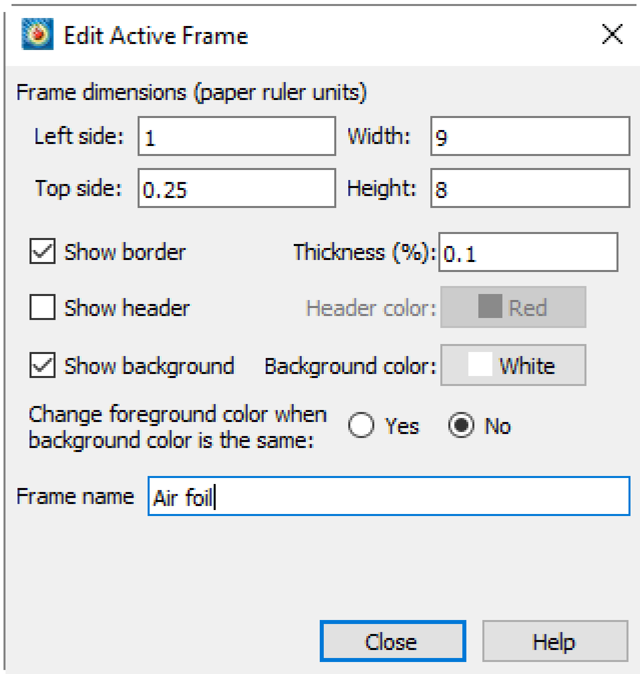
To create a new frame, which you can use to display a new dataset or another view of an already-loaded data set, select the New Frame button in the toolbar. Or select **Create New Frame** from the **Frame** menu. Selecting either the **New Frame tool** or **Create New Frame** in the **Frame** menu will change your selector tool into a small + symbol. Click with this tool in your workspace where you want one corner of your new frame, and drag across the workspace to where you would like the opposite corner of your new frame.

> Tecplot 360 automatically makes the new frame the active frame.
>
> For optimal printing of your plot(s), draw frames that sit completely within the paper displayed in the workspace. See Create Multiple Frames for information on simultaneously creating multiple frames.

> **Data Set Inheritance**
>
> After creating a new frame, you can assign the frame to share the dataset of another frame by changing the frame order and the plot type of the new frame. When you switch a frame without data out of Sketch plot type, Tecplot 360 searches for another frame with which to share data. It performs this search in the order that the frames are drawn on-screen, starting from the top and moving down. The empty frame will inherit data from the first frame in the draw order that contains data.

Edit Active Frame

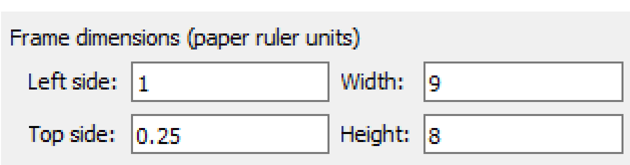
The **Edit Active Frame** dialog (accessed from the **Frame** menu) allows you to adjust the dimensions and style of your active frame. When you are working with multiple frames, select a frame with your selector tool to make that frame active. Then select **Edit Active Frame** from the **Frame** menu to adjust the position and properties of that frame.



Frame Dimensions

You can size and position a frame using your mouse, the keyboard, or a dialog. To use only your mouse, select the frame to activate the resizing handles (black boxes on the edges and corners of the frame). Click and drag on a handle to resize a frame, or click and drag on an edge in a location without a handle to change the frame location. To specify size or position with a dialog, select **Edit Active Frame** from the **Frame** menu.

In the **Edit Active Frame** dialog, you may specify the exact location for the frame's left and top sides, along with width and height.



Left Side

Left edge of the frame, relative to the workspace.

Top Side

Top edge of the frame, relative to the workspace.

Width

Width of the frame (coordinates are: *left side to left side + width*).

Height

Height of the frame (coordinates are: *top side to top side - height*).

The units in the Frame Dimensions region of the dialog are based on the units set for the **Ruler**

Spacing in **Options**→**Ruler/Grid**.

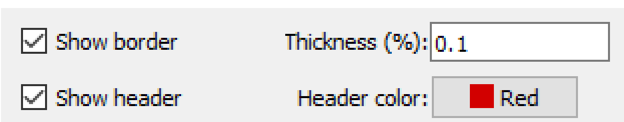
You may also use the mouse or the arrow keys to resize and position frames. Click anywhere on a frame’s header or border to activate resizing handles for the frame. To scale frames proportionally (maintaining the vertical to horizontal aspect ratio) select the frames, then press + on your keyboard to enlarge or - to reduce.

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.

To fit the active frame to the entire printable region of US letter-size paper (landscape orientation), set Left Side=0.127, Top Side=0.125, Width=10.75, and Height=8.25. Or, set Width=8.25 and Height=10.75 for portrait orientation.

Frame Border and Header Controls

Use the **Edit Active Frame** dialog (accessed from the **Frame** menu) to adjust the frame border or header.



Toggling-off **Show Border** makes the frame border invisible. Use the Thickness box in the **Edit Active Frame** dialog to adjust the line thickness of the border.

Remember that Tecplot 360 uses a thicker border to indicate The Active Frame, so making borders thicker can make it become more difficult to determine which frame is active.

The frame header displays when both **Show Border** and **Show Header** are toggled-on. If you turn off the border by toggling-off **Show Border**, the header turns off as well.

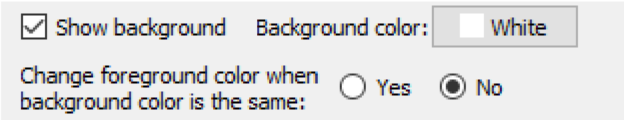
The frame header contains user-configurable information which defaults to:

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

where `FrameName` is the frame’s name, `date` is the date the frame was created or revised, and `DataSetTitle` is the title of the active dataset. You can change these defaults in your configuration file using the `$!GLOBALFRAME` command (see the Scripting Guide).

Frame Background Color Modification

Select the Color box in the **Edit Active Frame** dialog (accessed from the **Frame** menu) to adjust the frame background color. Toggle-off **Show Background** to set the frame background to transparent.



Frame Name Modification

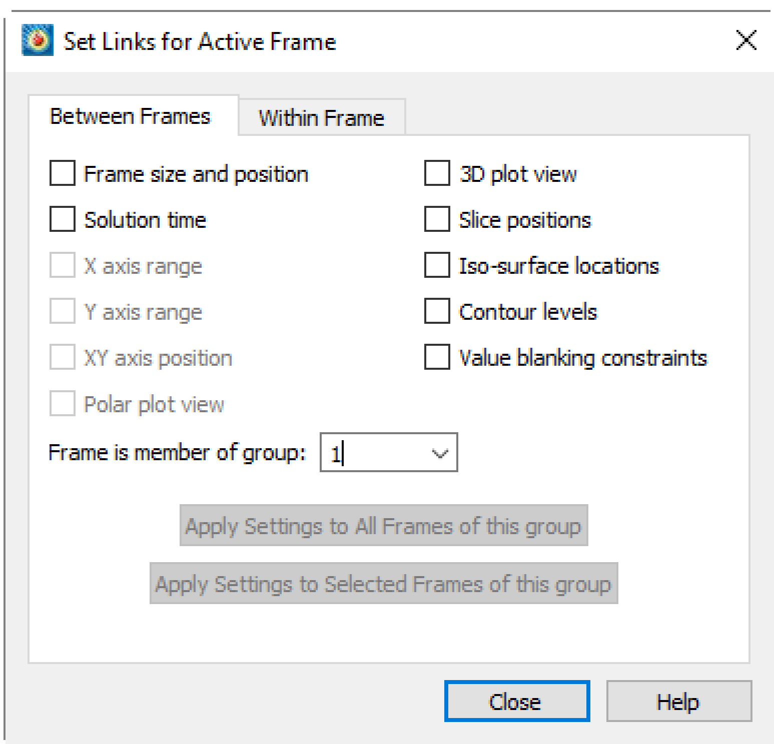
Enter text in the Frame Name region of the **Edit Active Frame** dialog (accessed from the **Frame** menu) to change the name of the active frame. The text entered here is displayed as the name of the frame.

Frame Linking

The **Set Links for Active Frame** dialog, accessible through **Frame**→**Frame Linking**, allows you to link specific style attributes either *between* frames or *within* a frame. Linking between frames, on the Between Frames page of the dialog, allows you to quickly make changes in one frame and propagate them to a number of other frames. Linking within frames, on the Within Frame page, links attributes between similar objects within a frame.

Attribute Linking Between Frames

Use the Between Frames page of the **Set Links for Active Frame** dialog (accessed from **Frame**→**Frame Linking**) to link the following attributes (shown below):



Frame Size and Position

Use this option to overlay transparent frames. (See Frame Background Color Modification.)

Solution Time

All frames display the same solution time.

X Axis Range, Y Axis Range (For XY Line and 2D plots)

Links the X-Axis or Y-Axis range and the positioning of the left and right sides of the viewport.

> For XY Line plots, axis range linking applies only to the first axes, X1 and/or Y1.

XY Axis Position (For XY Line and 2D plots)

Links the positioning of the X and Y-Axes between frames, including the method used for positioning the axes, such as aligning with an opposing axis value.

Polar Plot View

Link views for frames using the Polar Line plot type.

3D Plot View

Link the 3D axes and 3D view.

Slice Positions

Link slice positions and slice planes for active slices (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 2D and 3D plots.

Value Blanking Constraints

Link all value-blanking attributes.

> It is not necessary to close and reopen the dialog between frames. Simply select another frame with the dialog open to edit linking for the newly active frame.

Frame Linking Groups

You can segregate frames into groups so that only frames in that group receive changes made in the linked attributes. By default, all frames are added to Group 1. Add a frame to a group by selecting the appropriate group number from the **Frame is a Member of Group** drop-down menu on the Between Frames page of the **Set Links for Active Frame** dialog. You can assign frames to groups 1-20. New frames added to a group take on the characteristics of previous members of the group.

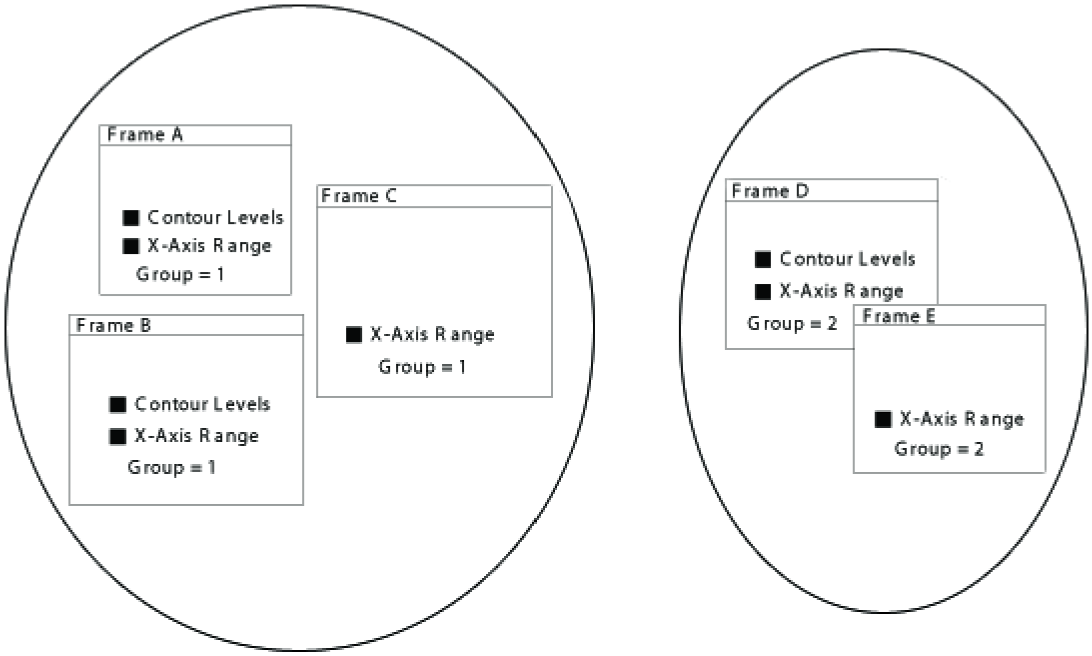


Figure 5. Five frames in two groups with different linking options.

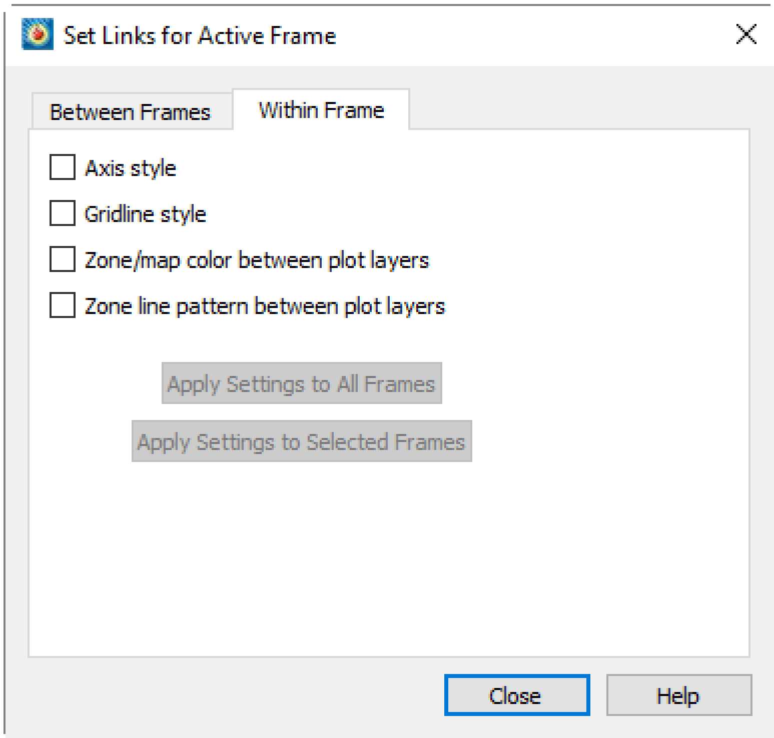
Propagating Between-Frame Link Attributes to Other Frames

Once link attributes are set in a frame group, 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 selected group of frames to have the same link attributes, select the appropriate **Apply Settings to...** button to quickly propagate the link settings. Alternatively, you can select each frame individually, making the same selections on the **Set Links for Active Frame** dialog for each chosen frame.

When 2D or XY Line frames have dependent axes and these axis ranges are linked, a "best-fit" attempt is made 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.

Attribute Linking Within A Frame

The Within Frame page of the **Set Links for Active Frame** dialog (shown below) allows you to link the following attributes:



Axis Style

Link activation, colors, line styles, and font styles for objects associated with axes.

Gridline Style

Link activation, colors, and 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.

Settings changed on this dialog take effect immediately. You may also apply the settings to all frames or just to selected frames using the buttons in the 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 turn on the same Within-Frame Linking properties in other frames.

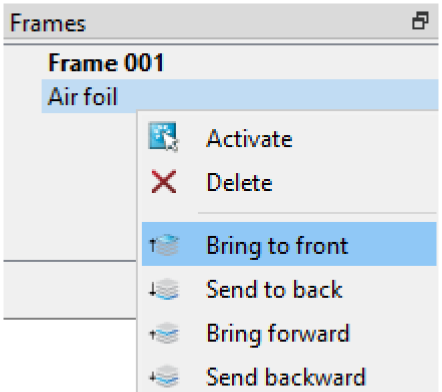
Frame Deletion

Delete a frame by selecting the frame and then choosing **Edit**→**Delete** or pressing the Delete key. This action is not undoable.

Managing Frame Order

Frames can overlap, and when they do, one is drawn in front of the other. You can think of the frames as a stack of papers pinned to a bulletin board. The order of frames can be managed by right-clicking a frame name in the Frames sidebar and choosing one of the following options.

> The Frames sidebar is initially hidden. To display it, choose **View → Frames Sidebar**.



Bring Forward

To move a frame one step closer to the front of the frame stack, right-click its name and choose **Bring Forward** from the context menu.

Send Backward

To move a frame to one step closer to the back of the frame stack, right-click its name and choose **Send Backward** from the context menu.

Bring to Front

To move a frame to the front of the frame drawing order, right-click its name and choose **Bring to Front** from the context menu. That frame will then appear in front of all other frames.

Send to Back

To move a frame to the bottom of the frame drawing order, right-click its name and choose **Send to Back** from the context menu. That frame will then appear behind all other frames.

The context menu in the Frames sidebar also contains an option to **Activate** the selected frame. This does not change the drawing order of the frame, but does make it The Active Frame.

Save Frame Style

You can save a frame's style for later use or to share with another frame by selecting **Save Frame Style** in the **Frame** menu. This will open the **Save Style** dialog, in which you can name and save the style of your active frame as a stylesheet (with the extension `.sty`). Use the Load Frame Style action to retrieve your frame style in another frame or layout.

The following data is saved:

- All plot style information
- Text elements
- Geometries (shapes such as boxes, circles, and polylines)
- Stream positions
- Contour levels

Load Frame Style

To load a saved frame style into your active frame, select **Load Frame Style** in the **Frame** menu. With the **Load Frame Style** dialog that appears, you can load a frame style saved from another frame (with the extension `.sty`) to dictate the frame attributes of your active frame. This will affect only the The Active Frame. To save a frame style, use the Save Frame Style function.

The following data is retrieved from the file and applied to the current frame:

- All plot style information
- Text elements
- Geometries (shapes such as boxes, circles, and polylines)
- Stream positions
- Contour levels

Workspace Management Options

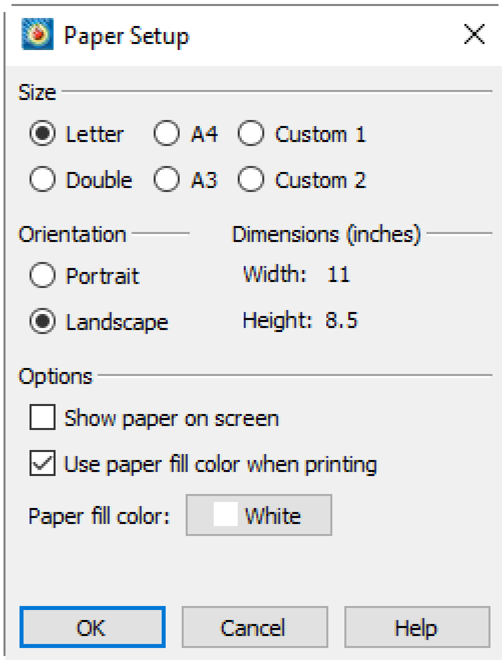
The workspace is the region in which you can create frames. The paper layout is a subset of the workspace and is correlated to the printer settings.

> The paper is hidden by default in Tecplot 360. Select **Show Paper on Screen** in the **Paper Setup** dialog under the **File** menu to include the paper.

Paper Setup

Tecplot 360'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, and color of your paper by going to **File→Paper Setup**.



Paper Size Controls

Tecplot 360 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.

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 Custom sizes see the `$!PAPER` command in the Scripting Guide.

Paper Orientation Controls

Layouts can be landscape or portrait. 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.

Screen Paper Controls

If you are creating plots for display on your screen, you can toggle-off the screen representation of the paper and use the full workspace by deselecting **Show Paper on Screen**.

Dimensions (display only)

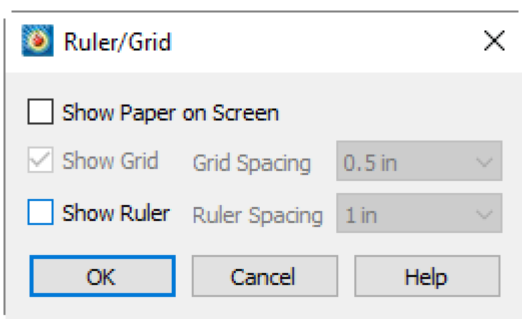
The units displayed in the **Dimensions** region of the **Paper Setup** dialog are determined by the units established in **Options**→**Ruler/Grid**.

Paper Color Controls

You can set up your paper to show any color as a background color (the "paper fill color") on your screen, as well as use that color when printing to a color printer. When you are printing, the paper can be flooded with your specified fill color. (By default, the paper fill color is ignored during printing.) To use the paper fill color when printing, select **Use Paper Fill Color when Printing** from the **Paper Setup** dialog.

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.



To customize the ruler/grid settings, select **Ruler/Grid** from the **Options** menu.

Workspace Grid Controls

Tecplot 360 allows you to select grid spacing from several pre-set sizes in centimeters (cm), inches (in), or points (pt) with the Grid Spacing drop-down menu. You can also specify not to show the grid by toggling-off **Show Grid**.

The grid does not display if **Show Paper on Screen** or **Show Grid** are toggled-off.

Workspace Ruler Controls

You can select the ruler markings from several pre-set sizes in centimeters (cm), inches (in), or points (pt) with the Ruler Spacing drop-down menu. You can also specify not to show the ruler by toggling-off **Show Ruler**. With **Show Ruler** toggled-on, rulers appear on the bottom and right-hand sides of the workspace.

View Modification

Use the **View** menu to adjust the view of the active frame, sidebars, or toolbars, or to adjust the view of the entire workspace. The **View** menu is discussed in the following subsections.

Show a Sidebar or Toolbar

You may close a sidebar by clicking its close box. Once closed, you may reopen it by choosing the desired sidebar from the appropriate menu.

Sidebar	Menu Command
Plot	View→Plot sidebar
Pages	View→Pages sidebar
Frames	View→Frames sidebar
Quick Macro Panel	Scripting→Quick Macros

You may also show or hide a sidebar or a toolbar by right-clicking any toolbar (or the menu bar) and choosing the toolbar or sidebar you wish to show or hide from the context menu.

Redraw Frame

When **Auto Redraw** is toggled-off, click the Redraw button in the Plot sidebar to redraw the active frame.

Zoom

There are two zoom modes: axis (dataset) zooming and paper zooming. To use either, first activate the Zoom tool from the Toolbar.

Plot Zooming

With the Zoom tool selected, drag the magnifying glass cursor to draw a box. The region within the view box will be resized to fit into the frame according to the longest dimension of the view box. If **Snap to Grid** is selected (on the Plot sidebar), you cannot make the zoom box larger than the grid area.

> To return to the previous view, select **Last** (`Ctrl-L`) from the **View** menu or **Undo** (`Ctrl-Z`) from the **Edit** menu.

3D mouse

Tecplot 360 also supports a 3D mouse, an input device that allows zooming (and translation and rotation) of 3D plots with a touch. You do not need to activate the Zoom tool to use the 3D mouse for zooming. Simply push the mouse's cap toward or away from you to zoom.

Hold the `Alt` key while zooming with the 3D mouse to switch the zoom method (see Input Devices for details).

Avoid tilting the mouse cap, as this may induce unintended rotation. Hold down the `Alt` key to switch between zooming by moving the data and by moving the viewer.

Paper Zooming

With the Zoom tool selected, `Shift`-drag the magnifying glass cursor to draw a box about the region that you want to magnify. The plot is resized such that the longest dimension of the zoom box fits into the workspace.

You can fit all frames to the workspace by using the **Fit All Frames to Workspace** option of the **View** menu.

Mouse Zoom and Translation

The middle and right mouse buttons allow you to smoothly zoom and translate your plot. The middle mouse button zooms smoothly, and the right mouse button translates (moves).

3D mouse

Tecplot 360 also supports a 3D mouse, an input device that can be used to zoom and translate 3D plots at any time regardless of the tool or mode selected.

- Push the mouse's cap toward or away from you to zoom. Hold the `Alt` key while zooming with the 3D mouse to switch the zoom method (see Input Devices for details).
- Pull the mouse's cap up or push it down to translate vertically.
- Move the mouse's cap left or right to translate horizontally.

Avoid tilting or rotating the mouse when using these motions to avoid rotating the plot.

Fit Everything

View→Fit Everything (`Ctrl-E`) resizes 3D Cartesian plots so that all data points, and all text and geometries positioned using grid coordinates, are included in the frame. By default, geometries use grid coordinates, while text objects use frame coordinates, so typically this means that Fit Everything fits geometries but not text objects, although if a text object has been set to use grid coordinates, it will be included. Use **View→Data Fit** to consider only data in the resizing.

Long text strings may still extend out of the frame even if positioned using grid coordinates.

Fit Surfaces

View→Fit Surfaces(`Ctrl-F`) resizes 3D Cartesian plots so that all surfaces are included in the frame, excluding any volume zones. If there are no surfaces plotted, **View→Fit Surfaces** fits all data points, text, and geometries into the frame.

Fit to Full Size

View→Fit to Full Size (`Ctrl-F`) resizes non-3D plots so that all zones or lines, geometries, and text fit within the frame.

Data Fit

View→Data Fit resizes the plot so all data points are included in the frame. Text and geometries are not considered. The text and geometries must be set to grid coordinates to be considered in the resizing.

Fit All Frames to Workspace

View→Fit All Frames to Workspace (`Ctrl-Shift-A`) resizes all frames proportionally so the frame(s) fill the workspace either vertically or horizontally.

Nice Fit to Full Size

View→Nice Fit to Full Size (`Ctrl-I`) sets the axis range of 2D Cartesian, XY Line, and Sketch plot types to begin and end on major axis increments. If axes are dependent, the vertical axis length is adjusted to accommodate a major tick mark.

Make Current View Nice

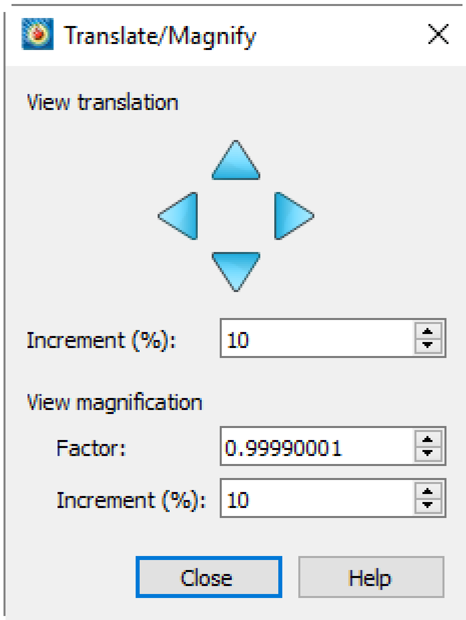
View→Make Current View Nice (`Ctrl-K`) modifies the range on a specified axis of 2D Cartesian, XY Line, and Sketch plots to fit the minimum and maximum of the variable assigned to that axis, then snaps the major tick marks to the ends of the axis. If axis dependency is not independent, this may affect the range on another axis.

Center

Centers the plot within the frame. Only the data is centered; text, geometry, and the 3D axes are not considered. Neither the axes nor the plot is changed in size.

Translate/Magnify

The **Translate/Magnify** dialog (accessed via the **View** menu), allows you to move and resize your plot within a frame. Translating using this dialog moves the image of your data with respect to the active frame. You can translate plots in any direction within a frame.



The following options are available in the **Translate/Magnify** dialog:

Up, Down, Left, Right

Use the arrows to translate (move) the image.

Increment (%)

Controls the step size for each arrow as a percentage of the workspace.

View Magnification - Factor

Change magnification using the up and down arrows to the right of the text field, or enter a value in the text field.

View Magnification - Increment (%)

The step size for each click of the arrows in the Factor field, as a percentage of the current magnification.

The Translate/Magnify tool (located in the Toolbar) allows you to translate/magnify the data within the frame or the entire workspace. Use the Shift key to translate/magnify the workspace instead of the data.

When the Translate/Magnify tool is active, type +/- on your keyboard to increase/decrease the scale of the image.

> To use the Magnify tool on the workspace, hold the `Shift` key and click on the workspace. Then, use the + or - keys on the keyboard to change the magnification of the workspace. Single-click on the data to change the mode back to dataset magnification.

Last

View→Last (`Ctrl-L`) restores the previous view. The **Last** command 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 is allotted four view stacks, one for each plot type. Each view stack stores up to sixteen views, including the active view.

Rotate

Available in the 3D Cartesian plot type only; calls up the **Rotate** dialog for image rotation. For further information, see Three-dimensional Rotation.

3D View Details

Available in the 3D Cartesian plot type only; calls up the **3D View Details** dialog for setting the view position and angle of 3D objects. For further information, see Three-dimensional Rotation.

Copy View

Use the **View→Copy View** menu option to copy the active frame view to the frame view buffer, where it can then be pasted to other frames having the same plot type. The copied view includes all the attributes of the view that are affected by the **View** menu: the amount of zoom, translation and scale, and (in 3D Cartesian plot type) the amount of rotation and perspective projection.

Paste View

View→Paste View pastes a copied view onto the active frame. When you are working with multiple frames attached to the same dataset, it is often useful to make your view changes to one frame and then propagate those changes to the other frames.

Edit Menu

You can duplicate frames, text, and geometries with the cut, copy and paste options of the **Edit** menu (or their keyboard equivalents, `Ctrl-X`, `Ctrl-C`, and `Ctrl-V`). You can also cut objects from one location and paste them into another, or throw them away completely using the Delete option.

Undo

All plot and mapping style modifications can be undone. In addition, you can undo a variety of other plot alterations. As a rule, Tecplot 360 allows undo for reversible 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.
- Creating new frames.
- Moving and copying line maps.
- View operations.
- Some pick operations.
- Streamtrace actions.
- The following data alterations:
    - Deleting zones and variables.
    - Renaming datasets and zones.
    - Creating rectangular or circular zones.
    - Duplication zones.
    - Processing Equations. (Except equations containing derivatives.)

> 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 cannot be undone, the entire undo history for that frame is erased.

Cut

Edit→**Cut** or `Ctrl-X` removes the selected item from the plot and the active dataset (if applicable), and stores the removed item in the Paste buffer.

> Generally, the Cut, Copy, and Paste options work only within Tecplot 360. However, you can copy selected frames and text objects and paste them into other applications. See Clipboard Exporting to Other Applications.

Copy

Edit→**Copy** or `Ctrl-C` stores the selected item in the Paste buffer but leaves it in your plot or the active dataset.

> Generally, the Cut, Copy, and Paste options work only within Tecplot 360. However, you can copy selected frames and text objects and paste them into other applications. See Clipboard Exporting to Other Applications.

Paste

Use **Edit**→**Paste** or `Ctrl-V` to add the contents of the Paste buffer to the active plot. If the object is being copied into the same frame, the new object will be overlaid directly over the original object. Use the **Selector** or the **Adjustor** tool to move the copied item to different locations in the frame.

Pasting from the Paste buffer is allowed only between compatible frames. Attempting to copy an object into a frame that does not hold an appropriate data type results in an error message.

Delete

Remove the selected item from the plot and from the active dataset. Deleted items are not stored in the Paste buffer.

If you cut or clear the only Tecplot 360 frame in the workspace, Tecplot 360 automatically creates a blank frame to replace it.

Send to Back

Pushes the selected item to the bottom of the active draw stack. The plot is drawn on your screen from the bottom of the draw stack to the top; elements lying further down in the stack may be partially obscured by elements higher up. The following types of objects may be pushed: text, geometries, 2D or XY grid areas, and frames.

Bring to Front

Pop the selected item to the top of the active draw stack. The following types of objects may be popped: text, geometries, 2D, or XY grid areas, and frames.

Data Structure

Tecplot 360 accommodates two different types of data: Ordered Data and Finite Element Data.

- **Ordered data** is a set of points logically stored in a one, two, or three-dimensional array, where I, J, and K are the index values within the array. The number of data points is the product of all of the dimensions within the array.
- **Finite-element** data is arranged in two arrays, a variable array and a connectivity matrix. The variable array is a collection of points in 2D or 3D space that are connected into polygonal or polyhedral units called *elements*. The connections between the nodes are defined by the connectivity matrix.

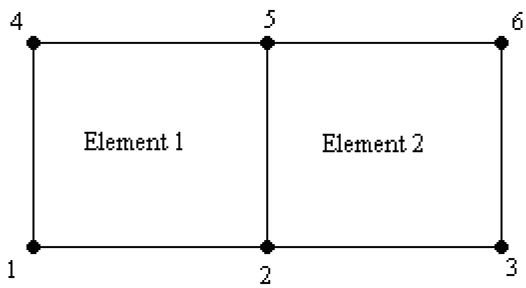
Connectivity List

A connectivity list is used to define which nodes are included in each element of an ordered or cell-based finite element zone. You should know your zone type and the number of elements in each zone in order to create your connectivity list.

The number of nodes required for each element is implied by your zone type. For example, if you have a finite element quadrilateral zone, you will have four nodes defined for each element. Likewise, you must provide eight numbers for each cell in a BRICK zone, and three numbers for each element in a

TRIANGLE zone. If you have a cell that has a smaller number of nodes than that required by your zone type, simply repeat a node number. For example, if you are working with a finite element quadrilateral zone and you would like to create a triangular element, simply repeat a node in the list (e.g., 1,4,5,5).

In the example below, the zone contains two quadrilateral elements. Therefore, the connectivity list must have eight values. The first four values define the nodes that form Element 1. Similarly, the second four values define the nodes that form Element 2.



The connectivity list for this example would appear as follows:

```
ConnList[8] = {4,5,2,1,  /* nodes for Element 1 */
               5,6,3,2}; /* nodes for Element 2 */
```

It is important to provide your node list in either a clockwise or counter-clockwise order. Otherwise, your cell will twist, and the element produced will be misshapen.

Ordered Data

Ordered data is defined by one, two, or three-dimensional logical arrays, dimensioned by IMAX, JMAX, and KMAX. These arrays define the interconnections between nodes and cells. The variables can be either nodal or cell-centered. Nodal variables are stored at the nodes; cell-centered values are stored within the cells.

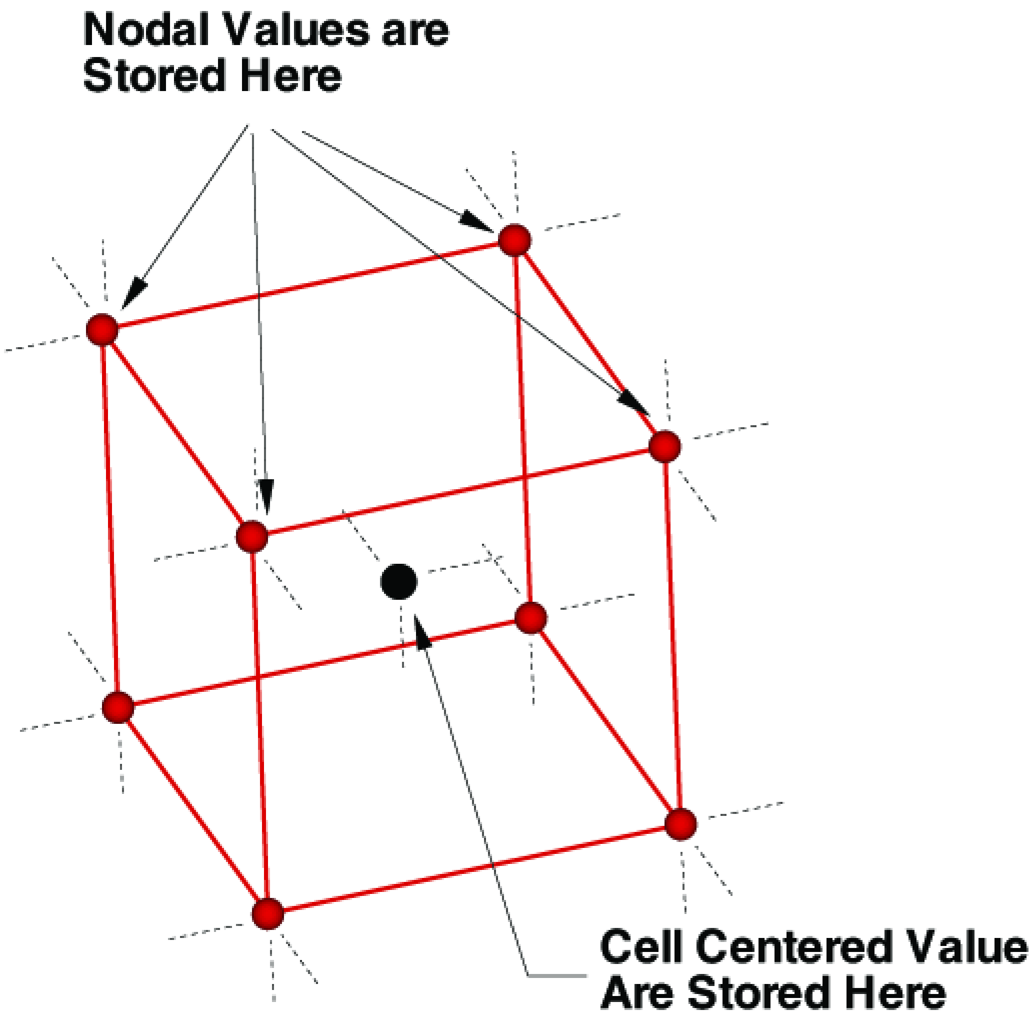
One-dimensional Ordered Data (I-ordered, J-ordered, or K-ordered)

A single dimensional array where either IMAX, JMAX or KMAX is greater than or equal to one, and the others are equal to one. For nodal data, the number of stored values is equal to `IMAX * JMAX * KMAX`. For cell-centered I-ordered data (where IMAX is greater than one, and JMAX and KMAX are equal to one), the number of stored values is (IMAX-1) - similarly for J-ordered and K-ordered data.

Two-dimensional Ordered Data (IJ-ordered, JK-ordered, IK-ordered)

A two-dimensional array where two of the three dimensions (IMAX, JMAX, KMAX) are greater than one, and the other dimension is equal to one. For nodal data, the number of stored values is equal to `IMAX * JMAX * KMAX`. For cell-centered IJ-ordered data (where IMAX and JMAX are greater than one, and KMAX is equal to one), the number of stored values is `(IMAX-1)(JMAX-1)` - similarly for JK-ordered and IK-ordered data.

Three-dimensional Ordered Data (IJK-ordered)



A three-dimensional array where all IMAX, JMAX and KMAX are each greater than one. For nodal ordered data, the number of nodes is the product of the I-, J-, and K-dimensions. For nodal data, the number of stored values is equal to `IMAX * JMAX * KMAX`. For cell-centered data, the number of stored values is `(IMAX-1)(JMAX-1)(KMAX-1)`.

Indexing Nodal Ordered Data

For nodal ordered data, the n-dimensional array of values are treated as a one dimensional array. For example, given an IJK-ordered zone dimensioned by `10x20x30`. To access the value at `I=3`, `J=4`, `K=5` (one based) you would use:

```
IMax       = 10
JMax       = 20
KMax       = 30
I        = 3
J        = 4
K        = 5
NodeIndex = I + (J-1)*IMax + (K-1)*IMax*JMax
```

Indexing Cell-centered Ordered Data

For cell-centered ordered data, the index that represents the cell center is the same as the nodal index that represents the lowest indexed corner of the cell.

For example, the figure below shows an IJ-ordered zone dimensioned 3x4.

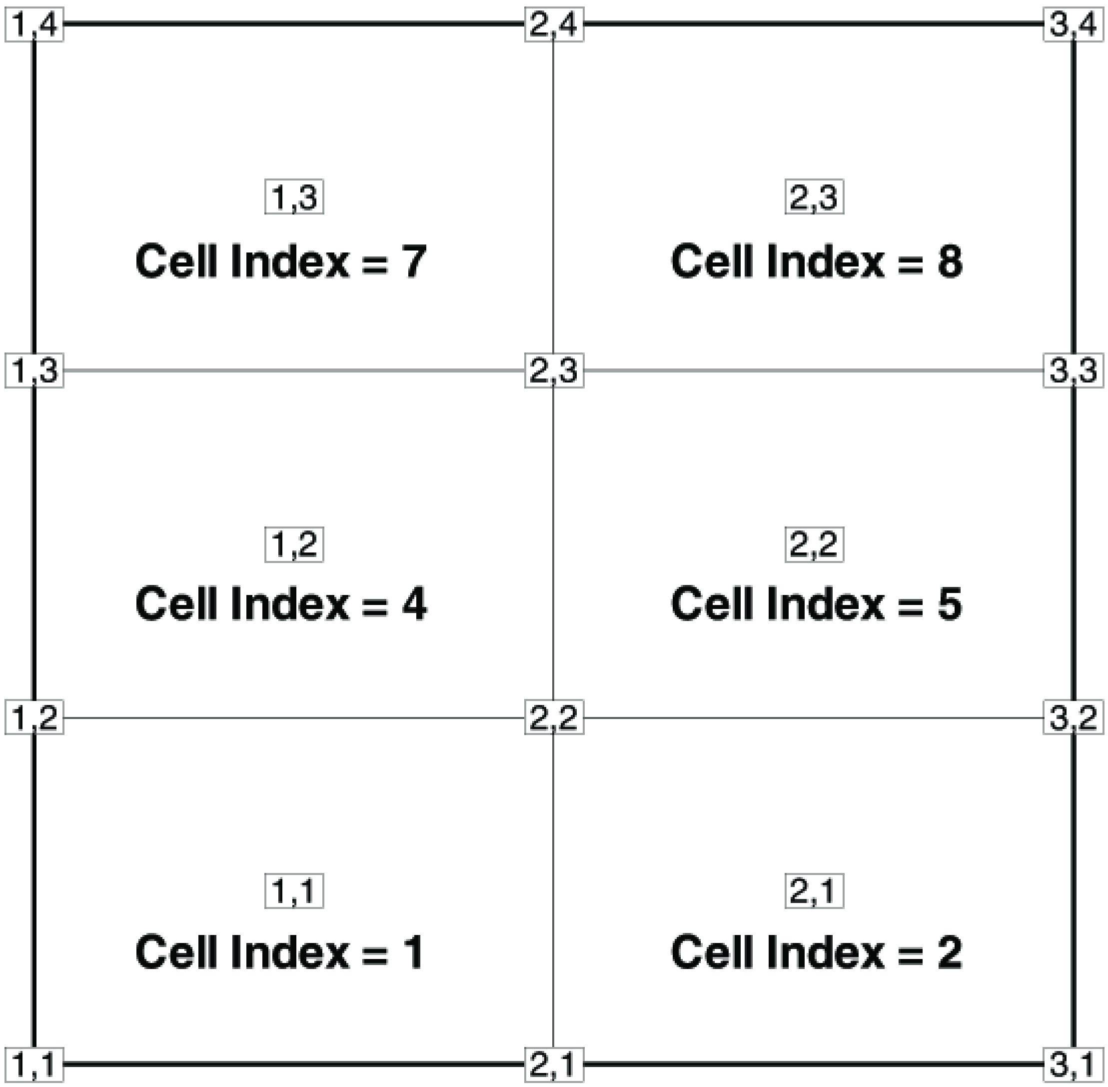


Figure 6. An IJ-ordered zone dimensioned 3x4. Cell index numbers are based on the point number in the lowest corner of the cell.

To access a cell-centered value for the cell in the upper right hand corner, use the following:

```
IMax      = 3
JMax      = 4
KMax      = 1
I      = 2
J      = 3
K      = 1
```

```
CellIndex    = I + (J-1)*IMax + (K-1)*IMax*JMax
```

You'll notice that the equations are exactly the same as with nodal data. As a result there are gaps of unused values at IMax, JMax, and KMax that *must* be left unassigned.

The above equation is generic for 1D, 2D and 3D data. It simplifies for the lower dimensions.

One-dimensional Ordered Data (I, J, or K)

Values 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.

At each node, *N* variables (`V1, V2, ⋯, VN`) are defined. If you arrange the data in a table where the values of the variables (*N* values) at a node are given in a row, and there is one row for each node, the table would appear something like that shown below.

Table 1. Tableof values for I-ordered Nodal Data (typical for XY plots).

V1	**V2**	**V3**	**...**	**VN**	**(Values at node** ***I = 1*****)**
V1	V2	V3	...	VN	(Values at node *I = 2*)
V1	V2	V3	...	VN	(Values at node *I = 3*)
V1	V2	V3	...	VN	
V1	V2	V3	...	VN	
V1	V2	V3	...	VN	(Values at node *I = IMax*)

See XY and Polar Line Plots for more information on XY plots.

IJK-ordered Data Plotting

In one or two-dimensional datasets, all data points are typically plotted. However, for IJK-ordered data you can designate which surface will be plotted by using the **Surfaces** page of the **Zone Style** dialog. You may choose to plot just outer surfaces, or you may select combinations of I, J, and K-planes to be plotted. Refer to Surfaces for in-depth information.

Logical versus Physical Representation of Data

A family of I-lines results by connecting all of the points with the same I-index, similarly for J-lines and K-lines. For IJ-ordered data, both families of lines are plotted in a two-dimensional coordinate system resulting in a 2D mesh. When both the I and J-lines are plotted in a three-dimensional coordinate system, a 3D surface mesh plot results. An example of both meshes is shown below. As you can see, logical data points can transform into an arbitrary shape in physical space.

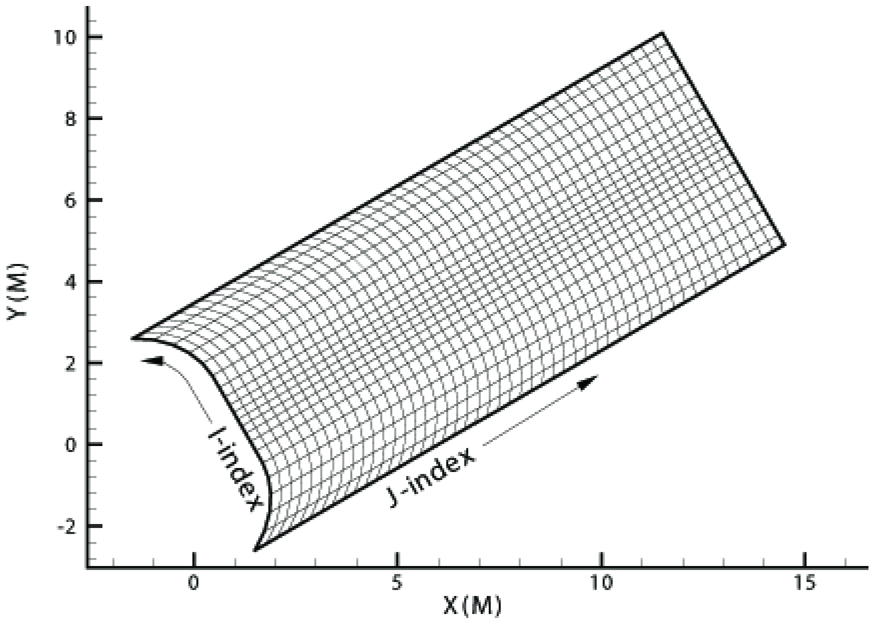


Figure 7. Left, a 2D mesh of IJ-ordered data points. Right, a 3D mesh of IJ-ordered data points.

Finite Element Data

While finite element data is usually associated with numerical analysis for modeling complex problems in 3D 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 dataset, 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 independent of the other elements, so you can group your elements to fit complex boundaries and leave voids within sets of elements. The figure below shows how finite element data can be used to model a complex boundary.

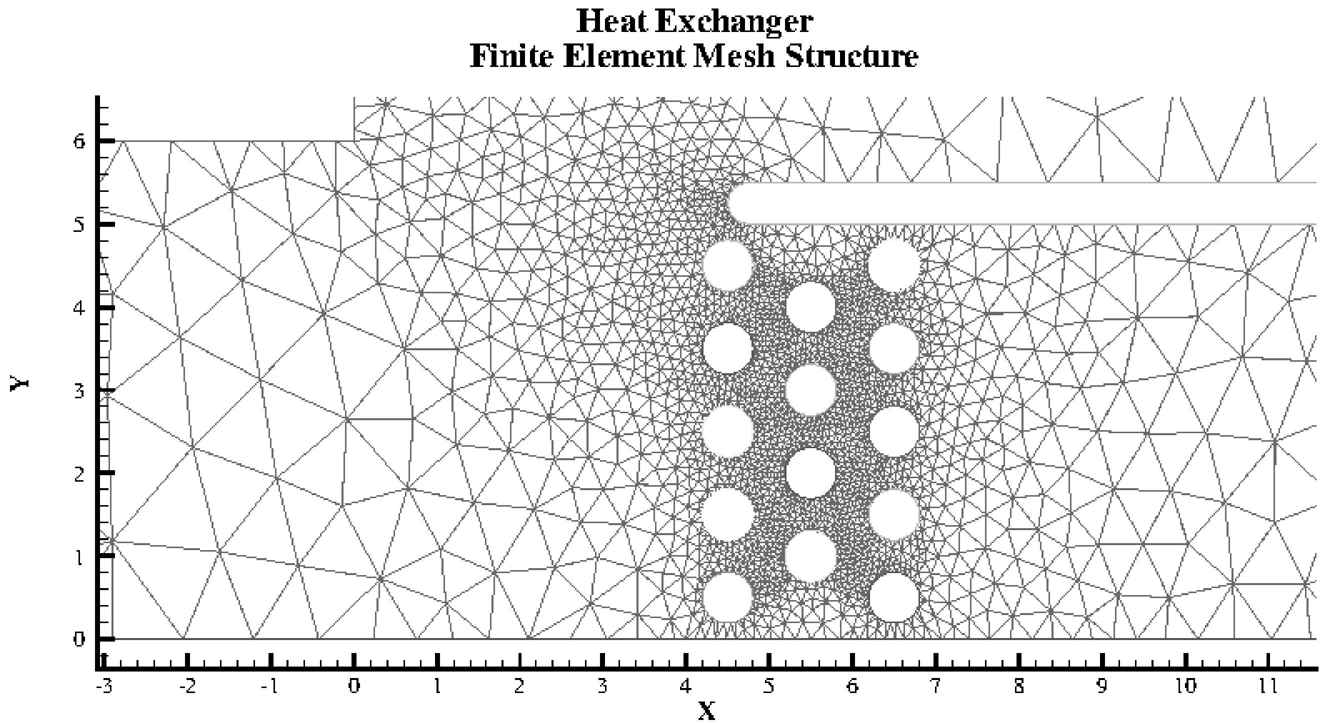
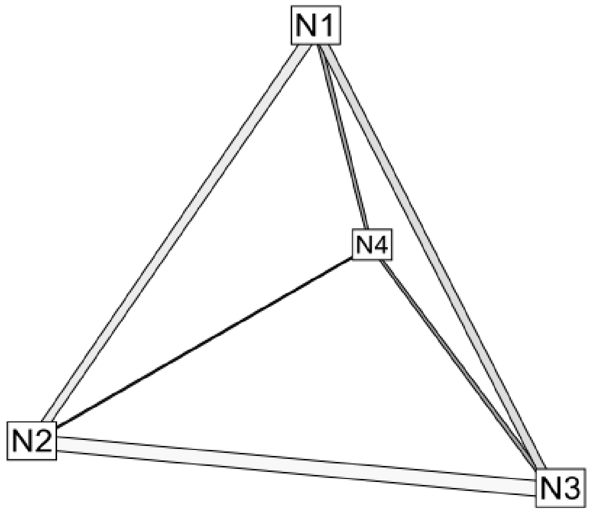
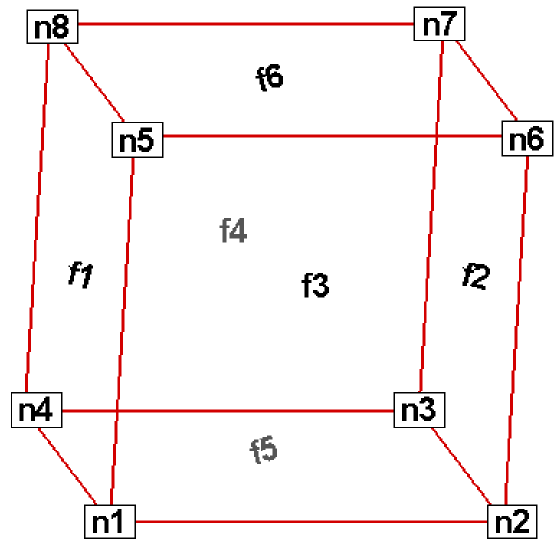


Figure 8. This figure shows finite element data used to model a complex boundary.

Finite element data defines a set of points (nodes) and the connected elements of these points. The variables may be defined either at the nodes or at the cell (element) center. Finite element data can be divided into three types:

- **Line data** is a set of line segments defining a 2D or 3D line. 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.
- **Surface data** is a set of triangular, quadrilateral, or polygonal elements defining a 2D field or a 3D surface. When using polygonal elements, the number of sides may vary from element to element. In finite element surface data, you can choose per zone to arrange your data in three-point (triangle), four-point (quadrilateral), or variable-point (polygonal) elements. The number of points per node and their arrangement are determined by the element type of the zone. If a mixture of quadrilaterals and triangles is necessary, you may repeat a node in the quadrilateral element type to create a triangle, or you may use polygonal elements.
- **Volume data** is a set of tetrahedral, brick, or polyhedral elements defining a 3D volume field. When using polyhedral elements, the number of sides may vary from element to element. Finite element volume cells may contain four points (tetrahedron), eight points (brick), or a variable number of points (polyhedral). The figure below shows the arrangement of the nodes for tetrahedral and brick elements. The connectivity arrangement for polyhedral data is governed by the method in which the polyhedral facemap data is supplied.

Table 2. Connectivity arrangements for FE-volume datasets

<table>
<tr><td>
</td><td>
</td></tr>
<tr><td>Tetrahedral connectivity arrangement</td><td>Brick connectivity arrangement</td></tr>
</table>

In cell-based element types (triangular, quadrilateral, tetrahedal, or brick), points may be repeated to achieve elements containing a smaller number of points. For example, a connectivity list of `n1 n1 n1 n1 n5 n6 n7 n8` (where `n1` is repeated four times) results in a brick-based pyramid element. Any of the points could be repeated; for example, `n1 n1 n3 n3 n5 n5 n7 n7` would also be a valid way to define a pyramidal element in a brick zone.

In Tecplot 360, each FE data zone must be composed exclusively of one element type. However, you may use a different data point structure for each zone within a dataset, as long as the number of variables defined at each data point is the same.

The Data Format Guide provides detailed information about how to format your FE data in Tecplot's text file format.

Finite Element Data Limitations

Working with finite element data has some limitations:

- 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.

Variable Location (Cell-centered or Nodal)

Data values can be stored at the nodes or at the cell centers.

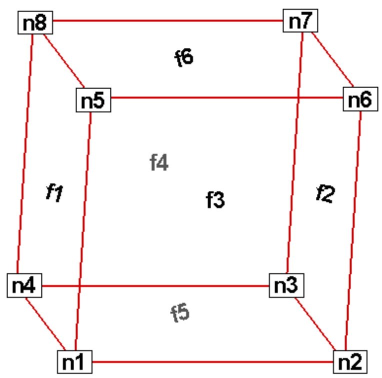
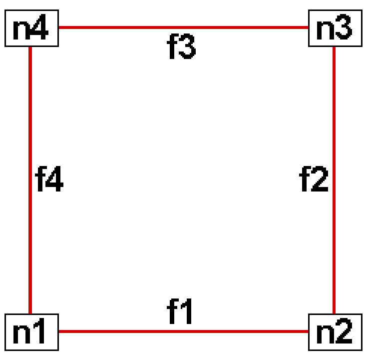
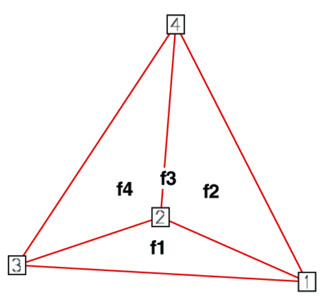
- For finite element meshes, cell-centers are the centers (centroids) of elements.

- For many types of plots, cell-centered values are interpolated to the nodes internally.

Face Neighbors

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 dataset. The face numbers for cells in the various zone types are defined below.

Examples of face neighbors for various zone types.

	**A**: Example of node and face neighbors for an FE-brick cell or IJK-ordered cell.
	**B**: Example of node and face numbering for an IJ-ordered/FE-quadrilateral cell.
	**C**: Example of tetrahedron face neighbors.

The implicit connections between elements in a zone may be overridden, or connections between cells in adjacent zones established, by specifying face neighbor criteria in the data file. Refer to `TECFACE` in the Data Format Guide for additional information.

Working with Unorganized Datasets

Tecplot 360 loads unorganized data as a single I-ordered zone and displays them in XY Mode, by default. An I-ordered zone is considered irregular if it has more than one dependent variable. An I-ordered data set with one dependent variable (i.e. an XY or polar line) is *not* an irregular zone.

To check for irregular data, you can go to the **Data Set Info** dialog (accessed via the **Data** menu). The values assigned to: IMax, JMax, and KMax are displayed in the lower left quadrant of that dialog. If IMax is greater than 1, and JMax and KMax are equal to 1, then your data is irregular.

It is also easy to tell if you have irregular data by looking at the plot. If you are looking at irregular data with the Mesh layer turned on, the data points will be connected by lines in the order the points appear in the data set.

You can organize your data set for Tecplot 360 in the following ways.

- Manually order the data file using a text editor.

> Use the **Label Points and Cells** feature from the **Plot** menu to see if your data set can be easily corrected using a text editor by correcting the values for I, J, and/or K.

- Use one of the **Data**→**Interpolation** options. See Data Interpolation.

Example: Unorganized Three-Dimensional Volume

To use 3D volume irregular data in field plots, you must interpolate the data onto a regular, IJK-ordered zone. To interpolate your data, perform the following steps:

1. Place your 3D volume irregular data into an I-ordered zone in a data file.
2. Read in your data file and create a 3D scatter plot.
3. From the **Data** menu, choose **Create Zone**→**Rectangular**. (**Circular** will also work.)
4. In the **Create Rectangular Zone** dialog, 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) dataset.
6. Click **Create** to create the new zone, and **Close** to dismiss the dialog.
7. From the **Data** menu, choose **Interpolate**→**Inverse-Distance**. Kriging may also be used.
8. In the interpolation dialog, choose the irregular data zone as the source zone, and the newly created IJK-ordered zone as the destination zone. Set any other parameters as desired (see Data Interpolation for details).
9. Select the **Compute** button to perform the interpolation.

Once the interpolation is complete, you can plot the new IJK-ordered zone as any other 3D 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 9 shows an example of irregular data interpolated into an IJK-ordered zone, with iso-surfaces plotted on the resultant zone.

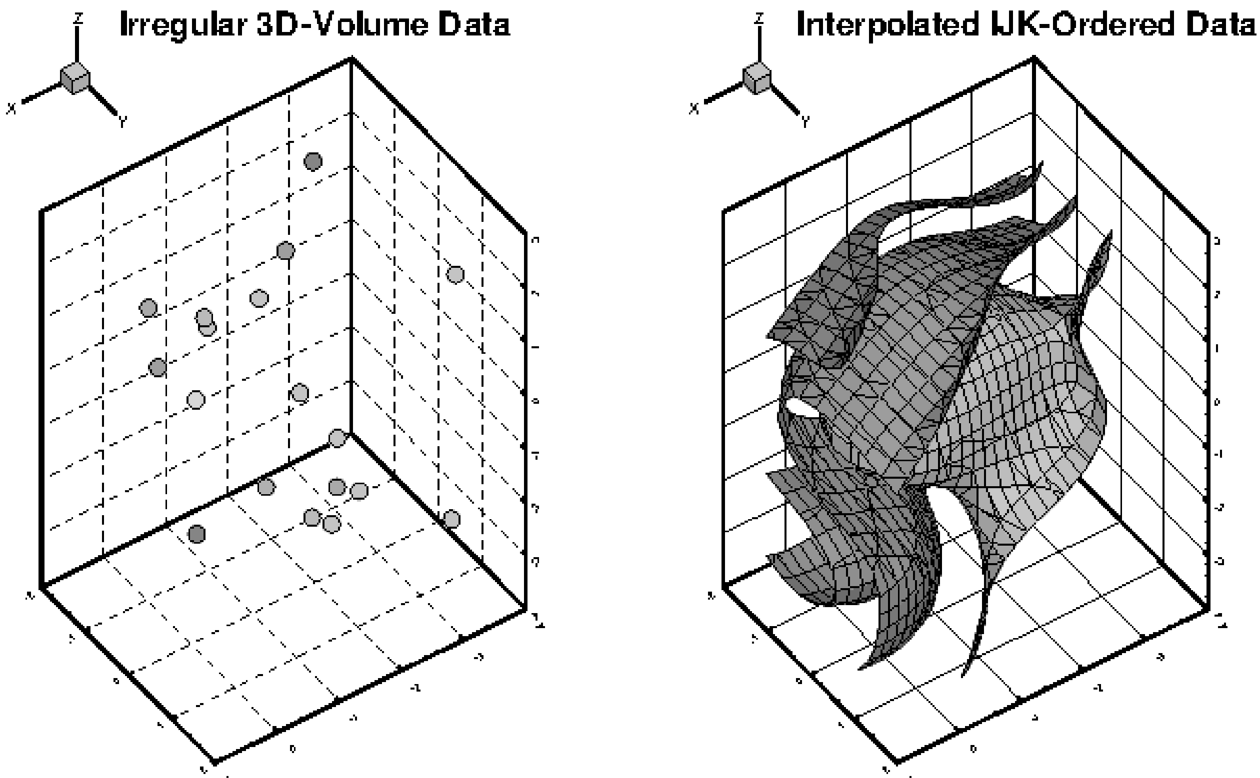


Figure 9. Irregular data interpolated into an IJK-ordered zone.

Part 2: Loading Your Data

Loading Data

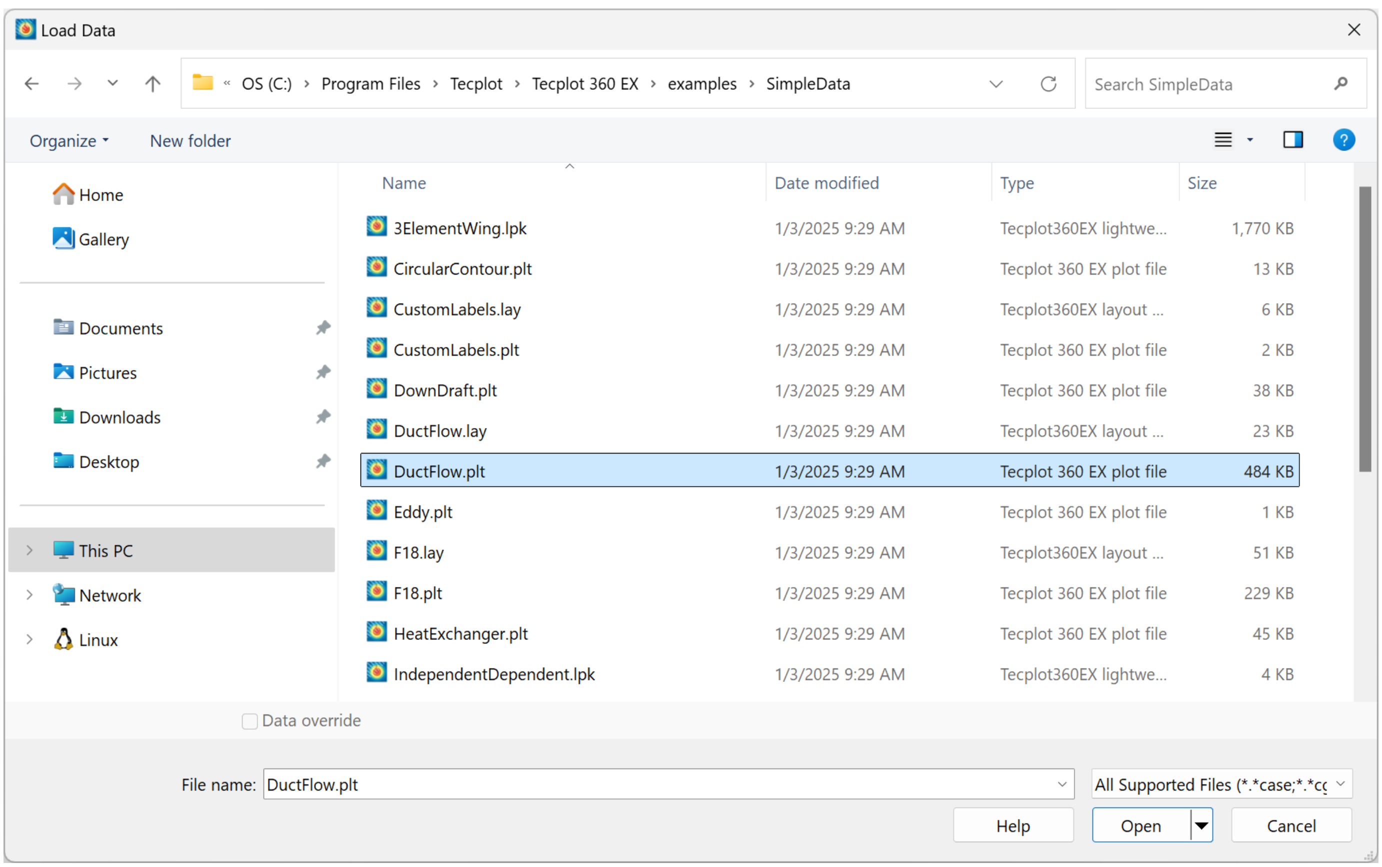
Tecplot 360 can load data from files on your local machine or accessible from a network file share using modules called *loaders*, each of which understands data in a particular format. A number of loaders are provided with Tecplot 360 for commonly-used CFD and general scientific and engineering formats as well as various Tecplot formats. See Loading Data Using a Loader.

You can also load data from Tecplot SZL Server, a lightweight module you can install on a remote Linux host (such as a compute cluster) to get access to your data when it is not practical to move it off the cluster to a local drive or to a network file share, and when you cannot reasonably run Tecplot 360 on the remote host over a remote desktop setup.

To set up Tecplot SZL Server see "Installing SZL Server" in the Tecplot 360 Installation Guide. To open data hosted by SZL Server, see Loading Remote Data using Tecplot SZL Server.

Loading Data Using a Loader

Use the **File→Load Data** command to load a data file using the Load Data dialog, shown below.



Normally, this dialog works slightly differently on Windows and non-Windows platforms. The controls at the bottom of the dialog are different. For example, this is the lower portion of the dialog as seen on

Linux.

On Windows, you can switch to this version of the Load Data dialog by choosing **Options**→**Use Extended Load Data Dialog**. Using this version of the dialog on Windows may be slower to navigate depending on your network; however, you may want to try it if you regularly load multiple data files from more than one directory.

On any platform, you may use the Load Data dialog to navigate to the file you wish to open, or simply type or paste the filename or the full pathname of the file in the File Name field.

> If the file you are loading is a Tecplot-format file, you may also load it by simply dragging it from your file manager (e.g. Windows Explorer or Mac Finder) into the Tecplot 360 workspace, bypassing the Load Data dialog entirely. You can also load data files via the macro language; refer to $!READDATASET in the Scripting Guide for details.

The **Files of Type** menu below the file list allows you to choose a data loader corresponding to the format of data file that you wish to load, or you may select *All Files* to see all the files in the directory and have Tecplot 360 automatically choose a loader for you if possible. When you choose a loader, the file list changes to show only files that can be loaded by that loader.

> If the name of the file you are trying to load has a non-standard extension, type the extension in the File Name field like "`*.foo`", then press Enter to show all files having this filename extension. You may also enter "`*.*`" to show all files regardless of their extension, or choose **All Files** from the **Files of Type** menu to have Tecplot 360 try to determine which loader is needed.

- If the chosen loader needs only a single file, simply navigate to and select the file you want to load in the file list and click **Open**.
- If the chosen loader supports or requires loading more than one file at once, or you are choosing from **All Files** or **All Supported Files**, you may select multiple files. On Windows (using the standard version of the Load Data dialog), you can choose additional files by holding down Control or Shift while clicking the second and subsequent files in the dialog, then clicking **Open**.

  On other platforms (or on Windows using the extended version of the Load Data dialog), the **Add to**

List button becomes enabled. Select each file to be loaded and click **Add to List** to add it to the Additional Files list. You may choose files from more than one directory. You may also remove selected files from the list by clicking the **Remove All** or **Remove Selected** button. Click **Open** to open all the files using the loader you chose.

When using **All Files** or **All Supported Files**, all of the files should be in the same format. If the loader for that format cannot load multiple files, you must load them individually, appending the second and subsequent files.

- Some file formats are very flexible, so you may need to specify exactly how the data was written in order for Tecplot 360 to load it correctly. Some loaders always display an options dialog, since they are needed almost every time. Others only display an options dialog when you request it. On Windows (using the standard dialog), click the menu triangle next to the Open button and choose **Open With Advanced Options** instead of clicking **Open** after selecting your file(s). On non-Windows platforms (or when using the extended dialog on Windows), an **Advanced Options** checkbox becomes enabled and may be selected before clicking **Open**. The loader's options dialog is then displayed before loading the selected file(s).

Available file formats include:

- CGNS Loader
- CONVERGE CGNS File Loader
- CONVERGE HDF5 File Loader
- CONVERGE Out File Loader
- DEM Loader
- DXF Loader
- EnSight Loader
- Excel Loader
- Exodus Loader
- FEA Loader
- FLOW-3D Loader
- FLUENT Loader
- Fluent Common Fluid Files (CFF) Loader
- FVCOM Loader
- General Text Loader
- HDF Loader
- HDF5 Loader
- Kiva Loader
- PLOT3D Loader

- PLY Loader
- Tecplot Data Loader
- Tecplot Subzone Loader
- Tecplot Layout Loader
- Telemac Loader
- Text Spreadsheet Loader
- TRIX Loader
- VTK Loader

Loading by Position

In most CFD data formats, it is valid to have multiple variables with the same name. When loading such a file, Tecplot 360 warns you that this situation exists and offers to load the variables by position: each variable will be identified by the order in which it appears in the file rather than by its name. This may make some aspects of working with the data different or more complicated.

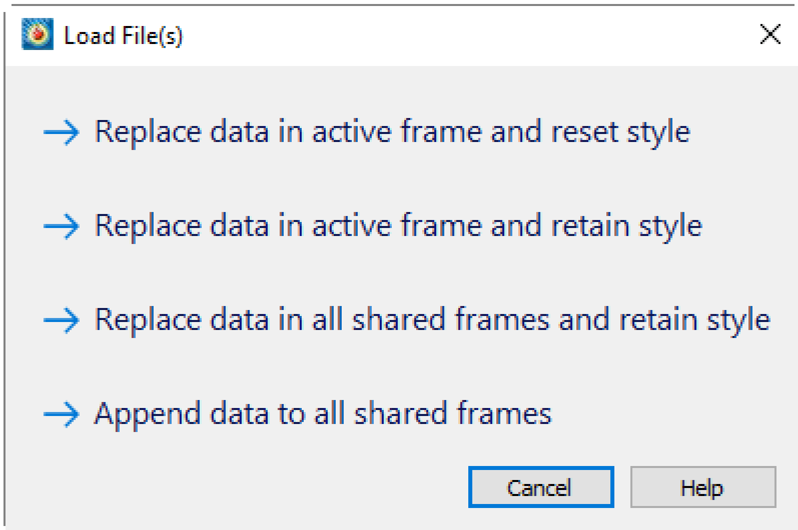
For example, when manipulating data using equations, you *must* use the `V#` syntax to refer to such duplicate variables, rather than the `{VariableName}` syntax, as the latter cannot uniquely identify the variable. (See Data Alteration through Equations.) Macros and Python scripts may also be affected.

When possible, we advise obtaining a data file with a unique and consistent name for each variable, for example by adjusting your solver's configuration.

Appending or Replacing Data

You may append new data to your dataset at any time by choosing **File→Load Data**. You cannot append a grid file that contains only FE connectivity data without appending a solution file along with it. In addition, to append a solution file you must also append a grid file at the same time. In this case, the grid file must be loaded before the solution file.

Some loaders allow you to choose how you wish to append or replace data.



Depending on the circumstances, you may have the following choices:

Replace Data in Active Frame and Reset Style

The new data is loaded in place of the existing data. The plot styles are reset to their defaults, as if the new data was loaded immediately after starting up Tecplot 360.

Replace Data in Active Frame and Retain Style

The new data is loaded in place of the existing data. The plot styles in effect for the existing data are applied to the new data as best as possible and in some cases some style settings may need to be turned off or reassigned. The configuration option `$!COMPATIBILITY USENAMESFORVARIABLEASSIGNMENTS`, when set to TRUE will make style settings match variable assignments based on the variable names, regardless of their position in the old and new datasets. If `$!COMPATIBILITY USENAMESFORVARIABLEASSIGNMENTS` is set to FALSE then the variable offsets from the old style assignments will be used with the new data regardless of the old and new variable names. This option works best when the incoming data is similar to the existing data.

Replace Data in All Shared Frames and Retain Style

This is the same as "Replace Data in Active Frame and Retain Style" except the same operations and rules are applied to all frames that share the same dataset with the active frame.

Append Data to All Shared Frames

The active dataset is shared among multiple frames. Choose this option to append the newly loaded data in all the frames. Default plot styles are used.

Append Data to Active Frame

Choose this option to append the newly loaded data in the current frame.

Replace

When loading a layout, replaces the existing layout with the new one.

Append

When loading a layout, creates new frames for the dataset referenced in the new layout.

Cancel

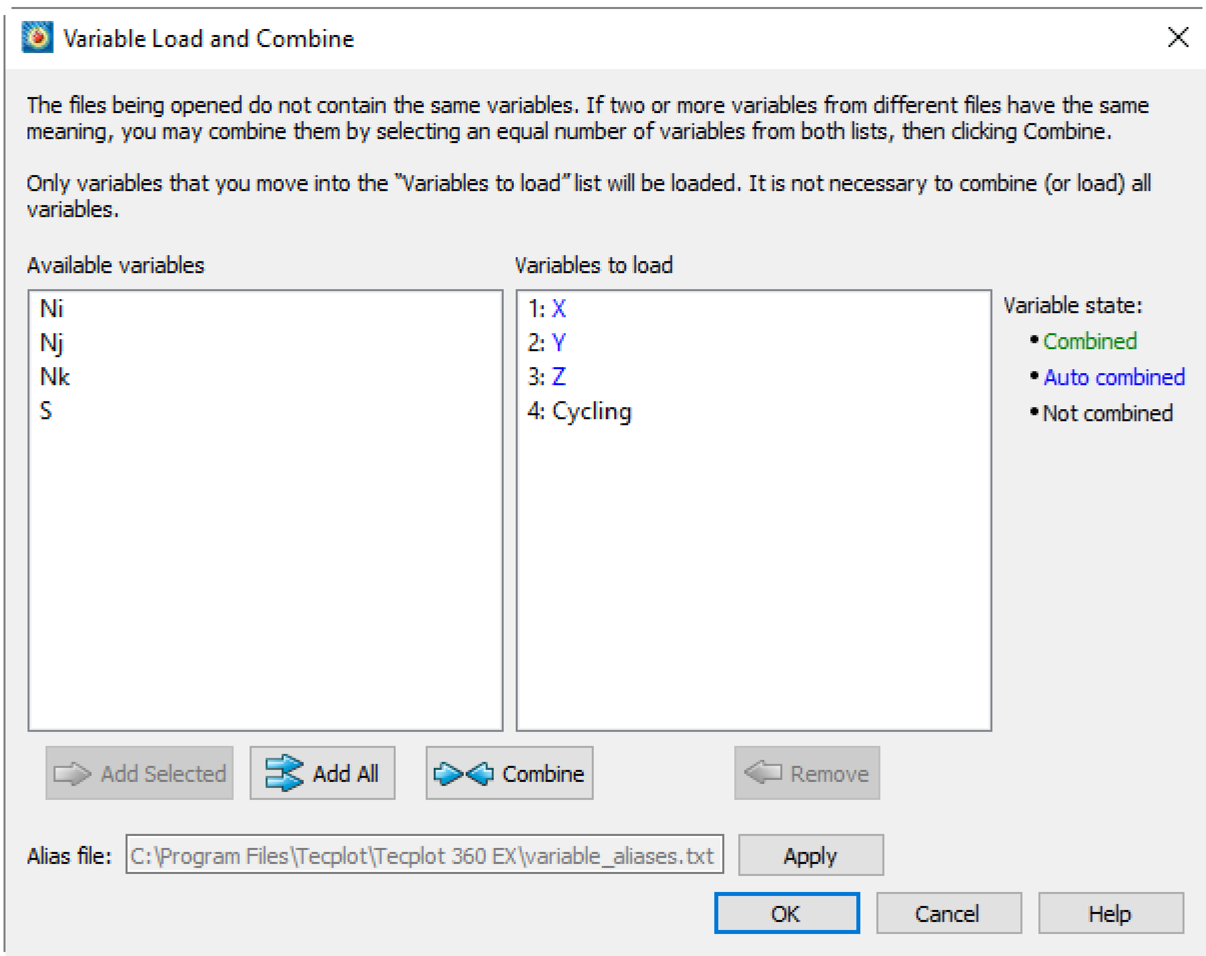
Do not load the data and return to the Tecplot 360 workspace.

Loading and Combining Variables from Multiple Files

In Tecplot 360, you can load multiple data files in two different ways:

- All at once by choosing all the files in the Load Data Files dialog.
- By appending data to the data that has already been loaded.

The Variable Load and Combine dialog appears when loading multiple data files in certain formats, whether you load the files all at once or append new data to previously-loaded data, if the names of the variables in the data files do not match. The dialog allows you tell Tecplot 360 which variables (if any) have the same meanings across the files, as well as which variables should not be loaded at all.



This feature works only with data in the following formats:

- Tecplot - text (`.dat`), binary (`.plt`), and subzone (`.szplt`)
- CGNS

- EnSight
- Excel
- HDF and HDF5
- Text (using the Text Spreadsheet or General Text loader)
- TRIX
- VTK

The Variable Load and Combine dialog appears only when the files you are loading do not all contain exactly the same variable names (or, when appending, if the variable names in the file being loaded do not exactly match those of the data already loaded). When all variable names match in the files being loaded, this dialog does not appear; instead, Tecplot assumes that all the variables have the same meaning in all the files.

> If a file you are loading or appending has multiple variables with the same name, Tecplot 360 asks if you want to load them by position. See Loading by Position. When loading by position, you cannot combine variables (and do not need to).

The dialog allows you to indicate that variables with different names have the same meaning (referred to as *combining* the variables) in two different ways.

- Select variables having the same meaning, one in each list, then click **Combine**.
- Apply a previously-defined variable alias file that indicates which variable names have the same meaning.

The two methods can also be used together. For example, you might first apply a variable alias file that deals with the most common equivalent variables in your typical data sets, then clean up any remaining variables (uncommon variables, or variables that don't always have a well-defined meaning) by hand.

The dialog has the following parts:

- The Available Variables list initially displays all the variables declared in the files you are loading. As you choose variables to combine or load, they move to the Variables to Load list. Any variables remaining in the Available Variables list when you click **OK** are not loaded.
- The Variables to Load list displays variables that are slated to be loaded in the current load operation. If you are appending data, variables that have already been loaded in previous load operations also appear here. Variables are colored according to the following rules:

  Black

  Variable is not combined; it is either already loaded, or will be loaded

  Blue

  Variable appears under the same name in multiple files and has been automatically combined

Green

Variable has different names in different files and has been combined by the user, either using the **Combine** button or by applying a variable alias file

- The **Add Selected** button moves a variable from the Available Variables list to the Variables to Load list, adding it to the set of variables that will be loaded in the current operation. The added variable's name appears in black. By default, variables that are unique to one file are not loaded.
- The **Add All** button moves all variables listed in the Available Variables list to the Variables to Load list, indicating that you want to load all of them in the current operation. The variable names appear in black.
- The **Combine** button combines the variable selected in the Available Variables list with the variable selected in the Variables to Load list, indicating that the variables have the same meaning. The variable is given a name derived from both the selected variables, and this name appears in the Variables to Load list and is colored green. See Combining Variables Manually.
- The **Remove** button removes the selected variable from the Variables to Load list, if possible. If the variable is already part of the current data set (that is, you are appending data, and the variable is from a previous load operation), it cannot be removed.

  When removing a combined variable, all the original variables that were combined are separated and moved back to the Available Variables list, again except for any which have already been loaded from a previous operation.
- The **Apply** button applies a variable alias file to the variables being loaded, combining variables if they are defined in the file as having the same meaning as another variable. See Creating and Using a Variable Alias File.
- Click **OK** to proceed with loading after setting up the variable combining the way you want it.

Variables that have the same name are automatically combined, as this is almost always what you will want to have happen. They appear in blue in the Variables to Load list. If you do not want this to happen, load the files one at a time and, before loading the file with duplicate variable names, rename the existing variables you do not want to combine with newly-loaded variables of the same name. (You can rename variables by double-clicking them in the Data Set Information dialog.

Combining Variables Manually

To combine two variables manually, telling Tecplot 360 that they have the same meaning, you select the first variable in the Available Variables list and the second in the Variables to Load list, then click the **Combine** button. The combined variable is added to the Variables to Load list and given a name consisting of both original variable names separated by a semicolon.

If the variables to be combined are in the Available Variables list, as may be the case when loading all the files at once instead of appending, select one in the Available Variables list and click **Add Selected** to select it to be loaded. Then combine them as above.

If there are more than two variables that have the same meaning, combine the first two, then select the combined variable in the Variables to Load list and combine it with the next variable having the same meaning in the Available Variables list as above.

You may combine multiple sets of variables at a time. If you select more than one variable in the Available Variables list, and the same number in the Variables to Load list, clicking Combine combines the first selected variable from the Available Variables list with the first selected variable from the Variables to Load list, the second with the second, and so on.

You may not combine variables from the same file.

Creating and Using a Variable Alias File

The `variable_aliases.txt` file found in the Tecplot 360 installation directory lists sets of variables that should be considered equivalent. See Custom Files loaded on Startup for information about how variable aliases are loaded.

The `variable_aliases.txt` file must begin with a `!VA 1` line to indicate that it is a variable alias file (format version 1). On subsequent lines, the `character indicates the beginning of a comment; everything following the` on a line is ignored. Variable aliases are specified by listing the equivalent names on a line separated by semicolons. For example:

```
#!VA 1
p;press;pressure
t;temp;temperature # temperature variables in Celsius
m;mach
a;alpha;aoa # angle of attack in radians
b;beta;aos # angle of sideslip in radians
```

Variable names found in this file are case-insensitive. Excess whitespace is ignored.

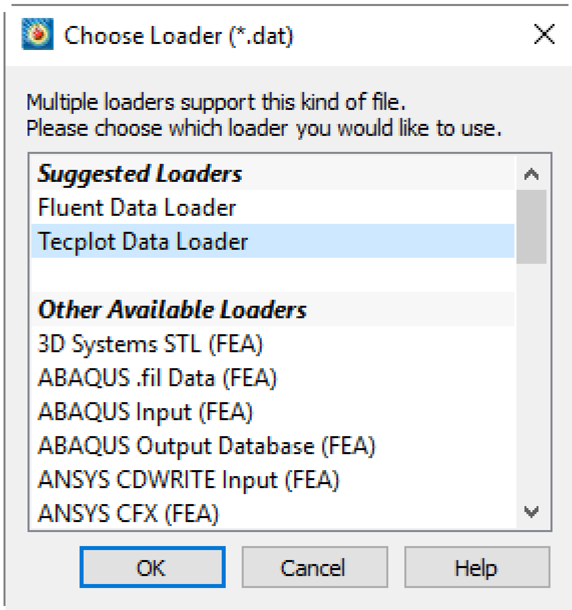
When you click **Apply**, Tecplot 360 combines variables as indicated in `variable_aliases.txt`. Where two or more files contain variables whose names appear on the same line, these variables are combined, and the resulting Tecplot 360 variable is given a name containing the original variable names (as they appear in the datasets being loaded) separated by semicolons. The new variables then appear in the Variables to Load list in the dialog.

For example, if you load two files with the variables "P" and "Pressure," and the lines shown above appear in the `variable_aliases.txt` file, when you click **Apply**, these two variables appear in the Variables to Load list as a single variable called either "P;Pressure" or "Pressure;P" depending on the order in which the files were loaded.

If multiple lines in `variable_aliases.txt` might apply to the data sets being loaded, these conflicts are resolved based on the order in which the files are loaded. For example, if you load three files containing variables "M," "Mach," and "Mass," in that order, and `variable_aliases.txt` contains lines equating "M" with "Mach" and also "M" with "Mass," the resulting data set would have a variable named "M;Mach" and a second variable "Mass." Because the file containing the variable "Mach" was loaded before the file containing the variable "Mass," "M" was combined with "Mach" when that file was loaded and cannot also be combined with "Mass."

You may change the names of combined variables after loading the data by double-clicking the variable in the Data Set Information dialog (**Data Set** → **Data Set Info**).

All Files and All Supported Files



With **All Files** and **All Supported Files**, Tecplot 360 tries to automatically choose a loader based on the names of the files you select.

With **All Files**, you see literally all of the files in the directory you're browsing. With **All Supported Files**, you see the files that Tecplot 360 knows it can load using any of the installed loaders, based on the files' names, but excluding other files. Either choice allows you to load files without having to specify in advance what loader will be used.

These options are particularly useful if you regularly work with multiple data file formats, or (in the case of All Files) if your data files often have unconventional names, such as a nonstandard or absent filename extension.

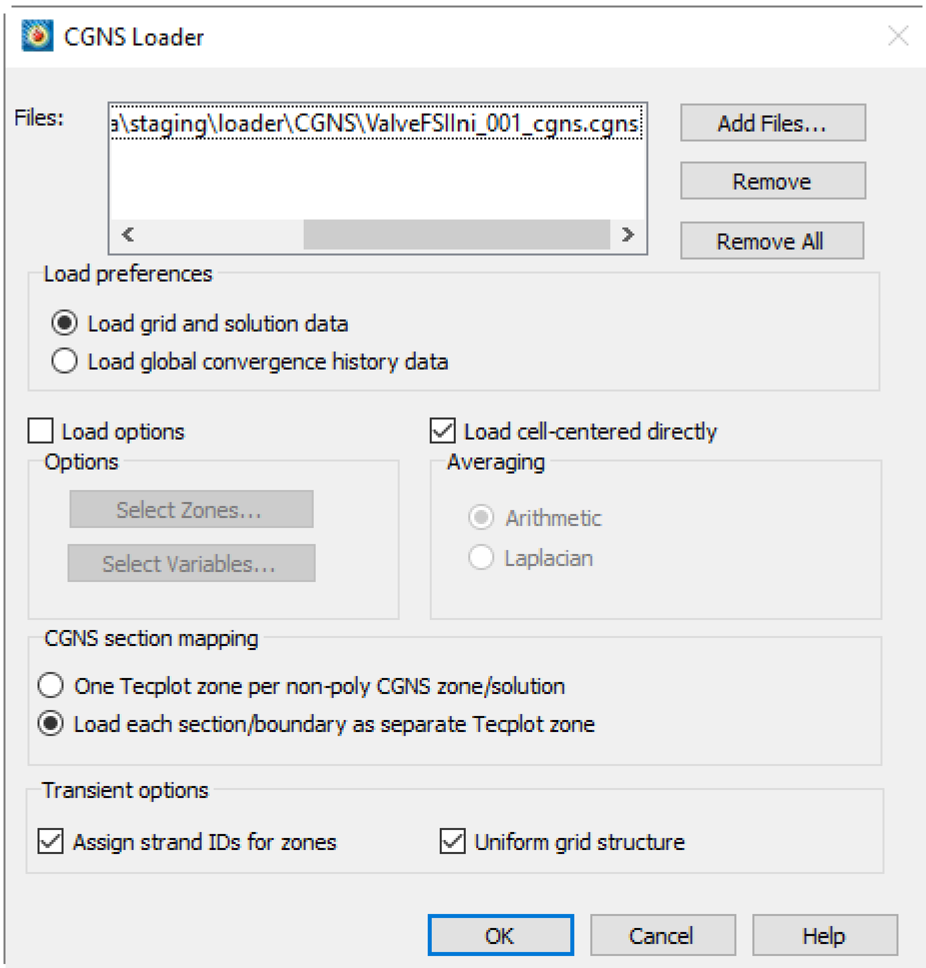
If the names of the selected files unambiguously indicate a single data file format, the appropriate loader is automatically chosen and the data is loaded without further ado. If more than one loader might be able to open the files, or if the files chosen do not have names that are recognized as belonging to a particular loader, the Choose Loader dialog appears. This dialog may recommend one or more loaders in a "Suggested Loaders" section at the top of the list. Choose the loader to be used, then click **OK**.

All selected files must be in the same format.

CGNS Loader

The CGNS Loader supports files created with CGNSLib Version 4.1.2 or earlier with either ADF or HDF5 internal representation. You can choose to load either all or specific bases, zones, and solutions into Tecplot 360 zones. You can also select field variables individually, or define index ranges to load specific sub-zone blocks or planes for structured-grid zones.

The **CGNS Loader** dialog has the following options:



Files

Click the buttons to add or remove files from the list of files to be loaded.

Load Preference

Choose to load either grid and solution data, or global convergence history data. If a file contains both types of information, they must be loaded separately.

Specify Options

Active when a valid file is entered or selected. This option allows you to control the data loaded from your CGNS file, including loading only particular zones, field variables, or partial zones.

If "Specify Options" is not selected, every base, zone, solution, and variable is loaded into Tecplot 360.

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.

Load Cell-centered Data Directly

Toggle-on to load cell-centered data directly (default). When the option is toggled-off, cell-centered data will be averaged to the nodes (using the averaging method specified below).

Averaging

This option is available only if "Load Cell-centered Directly" is not selected. When the field variables are stored at cell centers, either 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 (referred to as sections) for each finite element zone. A zone may contain sections with different cell types and cell dimensions.

One Tecplot zone per non-poly CGNS zone/solution (default)

All non-polytope sections with the same zone cell dimension will be combined into one zone in Tecplot 360.

Load each section/boundary as separate Tecplot zone

A separate zone will be created in Tecplot 360 for each section or boundary regardless of cell dimension.

Transient Options

Assign Strand IDs for zones

Toggle-on to assign Strand IDs to transient zones. Refer to Time Aware for more information on working with transient data.

Uniform grid structure

Toggle-on to use the grid from the first time step for all time steps, saving time and memory. Toggle-off if more than one grid is used.

CGNS Support Notes

The CGNS loader can load IJK-indexed, finite element, and polytope (polygonal/polyhedral) data, the latter denoted with NGON/NFACE sections in the data file. Polygonal sections may not be combined with sections of other types.

CGNS Boundary Conditions can be loaded for both structured and unstructured data with the exception that unstructured boundaries will only be loaded if they have corresponding sections.

Only CGNS bases and zones with valid grids can be read by the CGNS Loader. For unstructured grids, 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. Unstructured sections that are cell-centered and have more cells than are declared in the CGNS Zone_t node will be ignored.

Only vertex and cell-centered field variable locations are supported. Cell-centered data can be read in directly or 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 available. For cell-centered unstructured grids, either Laplacian averaging or arithmetic averaging can be selected to average the cell data to the surrounding nodes.

The CGNS Loader assigns strand IDs for zones by grouping them into time steps based on their solution times. The first zone from each time step is assigned to strand 1, the second to strand 2, and so on until all zones have been assigned to a strand. CGNS Base Names and Zone Names are converted to zone auxiliary data under the keys "CGNS.CGNSBase_t" and "CGNS.Zone_t," respectively.

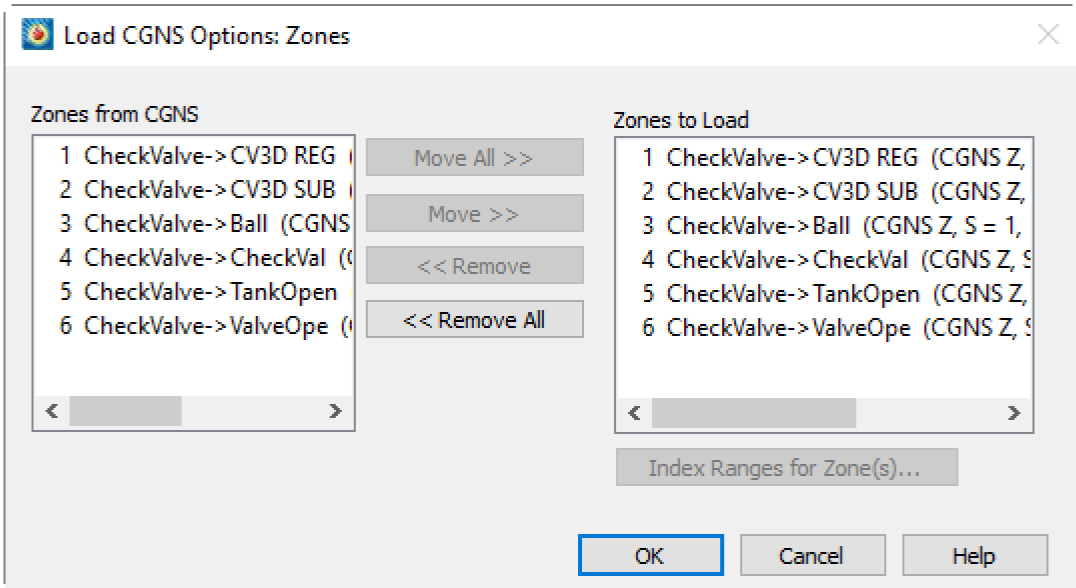
For CGNS files with an HDF5 internal representation, Tecplot 360 uses HDF5 library version 1.12.0.

> The CGNS loader, based on CGNS 4.1 is not compatible with CGNS 3.3 files backed by HDF5 1.10 due to a change to the CGNS library. Ensure your CGNS library is using a compatible version of HDF5. See CGNS bugs here:
>
> cgnsorg.atlassian.net/browse/CGNS-223
>
> cgnsorg.atlassian.net/browse/CGNS-166

> The CGNS 4.1 library has reduced performance (~10%) with ADF backed files. For best performance convert CGNS files to HDF5.

Load CGNS Options: Zones Dialog

Zones in Tecplot 360 are not always equivalent to CGNS zones. The **Load CGNS Options: Zones** dialog allows you to specify zones to load from CGNS data files.



Each solution for a CGNS zone is considered a unique zone in Tecplot 360. The CGNS base (B), zone (Z), and solution (S) hierarchy orders the zones. The integer preceding the word Zone is the internal zone number assigned to that zone. The integer following Zone represents the order the zone was found in the CGNS file.

Table 3 describes the zone description listed in the dialog box. The zone description includes the CGNS hierarchy information. "CGNS B, Z, S =" followed by three integers representing 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.

Table 3. Zone Description in the **Load CGNS: Zones** *dialog.*

`int`	`Zone`	`int`	`{CGNS B, Z, S = x, y, z}`	`[Ordered, FE]`
internal zone number	"Zone"	order in CGNS file	x = Base number y = Zone number z = Structure number	"Ordered" or "FE"

By default, all zones are selected for reading and displayed in Zones to Load. Use the **Move**, **Move All**, **Remove**, or **Remove All** buttons to edit the list.

CGNS Loader Options: Index Ranges Dialog

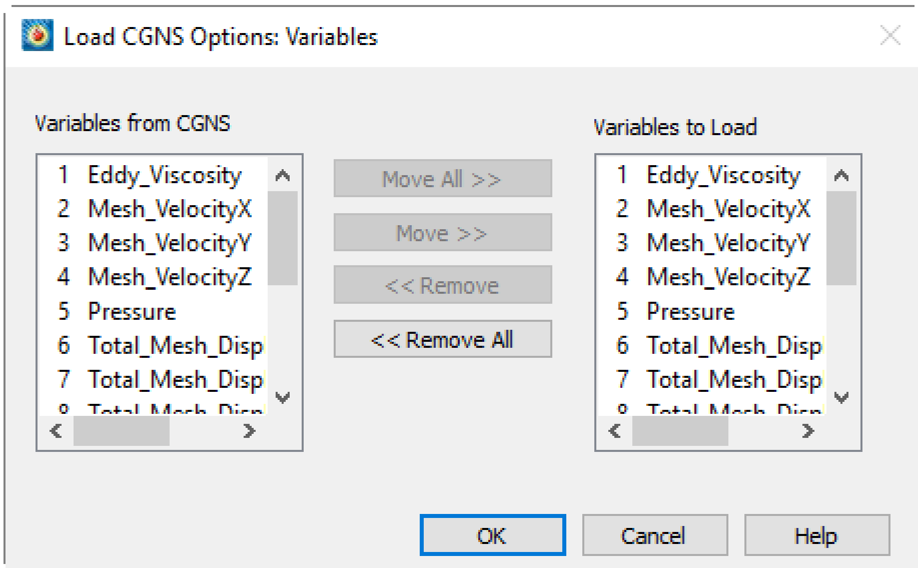
The **Load CGNS: Index Ranges** dialog allows you to specify a sub-set of the selected ordered/structured zone(s) to be loaded, or define a block, plane, or line of points for extraction on loading. To load a partial zone or sub-zone, highlight the zone of interest in **Zones to Load** region of the **CGNS Loader: Zones** dialog, and select the **Index Ranges for Zone(s)** button.

Each index requires Start, End, and Skip values. Start and End points are always loaded. If multiple zones are selected prior to calling up the **CGNS Loader: Index Ranges** dialog, "Mx" (the maximum value for each zone) is the default value for End. You may enter any value for End. However, if the value is greater than the maximum index for a zone, End is replaced by the maximum index.

For multi-dimensional zones, more than one point must be specified to load for the I and J-directions. If the inputs for Start, End, and Skip result in a single point in either direction, an error message appears.

Load CGNS Options: Variables Dialog

The **CGNS Loader: Variables** dialog includes the Variables from CGNS and Variables to Load boxes.



The "Variables from CGNS" list includes all field variables in the CGNS data file, independent of their zone(s). The "Variables to Load" list contains the field variables that have been selected to be loaded into Tecplot 360. Initially, both lists are the same. A variable number is assigned to each CGNS field variable that appears in the "Variables to Load" list.

Because Tecplot 360 requires every zone to have the same number of variables, each zone that is loaded into Tecplot 360 will include every variable in the "Variables to Load" list (regardless of whether the zone included that field variable in the CGNS file). The variables that were not originally in the zone will be set to zero. The field variables that do not appear in the "Variables to Load" list will not have a variable number assigned to them.

Use the **Move**, **Move All**, **Remove**, or **Remove All** buttons to edit the Variables to Load list.

Macro Commands for the CGNS loader

The syntax for loading CGNS data files with the Tecplot macro language is as follows:

```
$!READDATASET
' "STANDARDSYNTAX" "1.0"
"...any of the name value pairs in the following table..." '
DATASETREADER = 'CGNS LOADER'
```

Each name/value pair should be in double quotes. Refer to the Scripting Guide for details on working with the Tecplot macro language.

Keyword	Values	Default	Notes
STANDARDSYNTAX	"1.0"	n/a	Must be the first instruction.
FILENAME_CGNSFILE	"filename"	n/a	The name of the file to load.
FILELIST_CGNSFILES	"filenames"	n/a	Option to load multiple files. Syntax: "FILELIST_CGNSFILES n filenames" where n is the number of files to be loaded and filenames is a list of CGNS files delimited by spaces.
LoaderVersion	"V3"	n/a	The loader version to use; V3 is current
CgnsLibraryVersion	"4.1.2"	n/a	The CGNS library that was used when loading the data. The CGNS loader uses this information to determine whether any version incompatibility exists that may affect how the data file is loaded, and warns the user in that case.
AverageToNodes	"Yes" "No"	"Yes"	Used to average the cell data to the nodes they surround.
AveragingMethod	"Laplacian" "Arithmetic"	"Arithmetic"	If AverageToNodes is set to "yes", specify the AveragingMethod to use.
SectionLoad	"Combine" "SeparateZones"	"SeparateZones"	CGNS files may have multiple node-maps (or *sections*) for each finite element zone. If you specify "Combine", all sections are combined with the zone cell-dimension into one zone. If you specify "SeparateZones", a separate zone is created for each section or boundary regardless of cell dimension.
LoadBCs	"Yes" "No"	"No"	Specify whether to load the boundary conditions. For unstructured data, boundary conditions will always be loaded.
AssignStrandIDs	"Yes" "No"	"Yes"	Set to "Yes" to automatically assign the strand IDs to your data file.
ZoneList	like "Z1, Z2, + Z3-Z7, …"	All zones	Specify the zone number(s) of the zone(s) you wish to load.
VarNameList	like "V1, V2, + V3-V7, …"	All variables	Specify the variable number(s) of the variable(s) you wish to load.

Keyword	Values	Default	Notes
IIndexRange	"Zn, Min, + Max, Skip"	All	If you are loading a subset of zones, you may specify the index ranges for each zone. Specify the zone number, minimum, maximum, and skip value for each index. Set Zn to "0" to apply the index ranges to all zones.
JIndexRange	"Zn, Min, + Max, Skip"	All	
KIndexRange	"Zn, Min, + Max, Skip"	All	
LoadConvergenceHistory	"Yes" "No"	"No"	Loads global convergence history data in the CGNS file, if any, rather than grid and solution data.
UniformGridStructure	"Yes" "No"	"Yes"	If "Yes," the same grid is assumed for all timesteps and only the first timestep's grid is loaded. If "No," the grid is loaded for each timestep.

CONVERGE CGNS File Loader

Use the CONVERGE CGNS loader to load `post*.cgns` files produced by CONVERGE. This loader does not have any Advanced Options and will load all supplied variables. Multiple files may be loaded via the Load Data dialog. Multiple files will be separated into separate zones in Tecplot.

Macro Commands for the CONVERGE CGNS loader

The syntax for loading CONVERGE CGNS data files with the Tecplot macro language is as follows:

```
$!READDATASET ' "STANDARDSYNTAX" "1.0"
"..any of the name value pairs in the following table..." '
DATASETREADER = 'CONVERGE CGNS File Loader'
```

Each name/value pair should be in double quotes.

Keyword	Values	Default	Notes
STANDARDSYNTAX	"1.0"	n/a	Must be the first instruction.
FILENAME_FILE	"filename"	n/a	The name of the file to load.
FILELIST_DATAFILES	"filenames"	n/a	Option to load multiple files. Syntax: "FILELIST_DATAFILES n filenames" where n is the number of files to be loaded and filenames is a list of CONVERGE CGNS files delimited by spaces.

CONVERGE HDF5 File Loader

Use the CONVERGE HDF5 loader to load *post*.h5* files produced by CONVERGE. This loader does not have any Advanced Options and will load all supplied variables. Multiple files may be loaded via the Load Data dialog. Multiple files will be separated into separate zones in Tecplot.

Macro Commands for the CONVERGE HDF5 loader

The syntax for loading CONVERGE HDF5 data files with the Tecplot macro language is as follows:

```
$!READDATASET ' "STANDARDSYNTAX" "1.0"
"..any of the name value pairs in the following table..." '
DATASETREADER = 'CONVERGE HDF5 File Loader'
```

Each name/value pair should be in double quotes.

Keyword	Values	Default	Notes
STANDARDSYNTAX	"1.0"	n/a	Must be the first instruction.
FILENAME_FILE	"filename"	n/a	The name of the file to load.
FILELIST_DATAFILES	"filenames"	n/a	Option to load multiple files. Syntax: "FILELIST_DATAFILES n filenames" where n is the number of files to be loaded and filenames is a list of CONVERGE HDF5 files delimited by spaces.

CONVERGE Out File Loader

Use this loader to load in ASCII `.out` files produced by CONVERGE. This loader does not have any Advanced Options and will load all supplied variables. If CONVERGE `.out` files specify different "Locations" for variables then those locations will be translated into separate zones in Tecplot.

DEM Loader

The DEM Loader allows you to load Digital Elevation Map files that have the same file format as the U.S. Geological Survey's standard DEM format. 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 www.webgis.com/terr_us1deg.html

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. For large DEM files, you may want to set both of these to be 10 or more.

DXF Loader

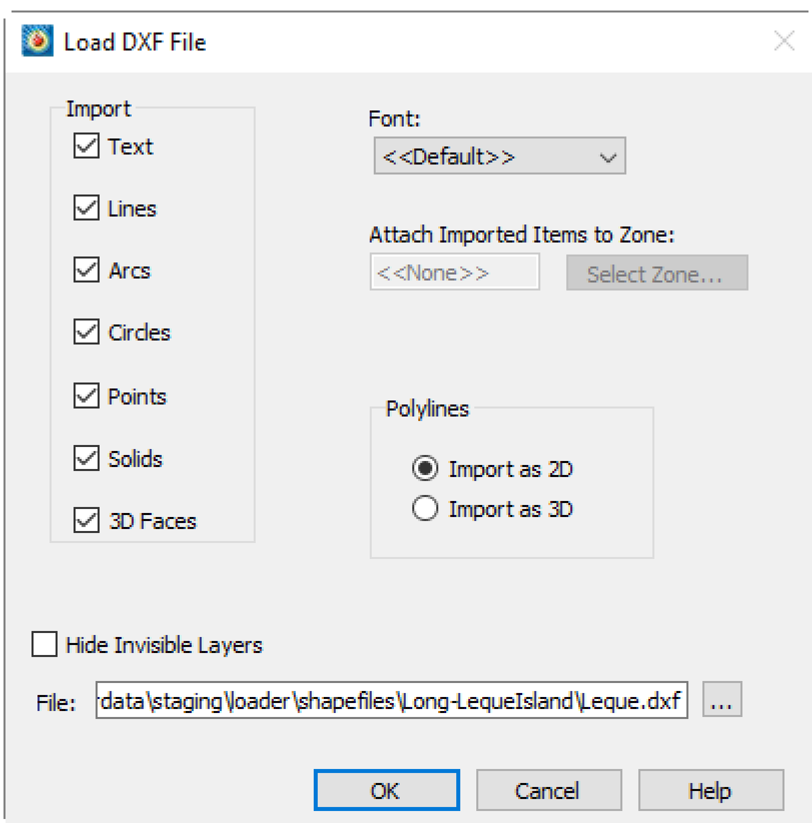
The DXF Loader add-on can import AutoCAD® DXF™ (drawing interchange) files. When importing a file, Tecplot 360 creates an appropriate geometry for each of the following entity types:

- Text
- Lines
- Arcs
- Circles
- Points
- Solid
- 3D faces

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.

Load DXF File Dialog

The **Load DXF File** dialog has a variety of features. 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. (See Font Folders and Fallback for more information on how fonts work with Tecplot 360.)

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 2D 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 3D line geometries for all lines and polylines in the DXF file. To view a 3D DXF file, create or load a 3D zone, import your DXF file, then choose **Fit Everything** from the **View** menu.

Hide Invisible Layers

If this option is checked, objects in layers that are "off" in the DXF file will be imported with the background color.

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 360 are 2D, best results will be obtained by loading "flat" DXF files, such as maps.

Binary AutoCAD® DWG™ are not supported in this release.

EnSight Loader

The EnSight Data Loader allows you to load EnSight Gold and EnSight 6 files of types case (`.case` or `.encase`), geometry (`.geo`), or variable (`.*`). Geometry and variable files can be in either ASCII or binary format, although binary is recommended. Files from earlier versions of EnSight need to be resaved in Gold format using **File→Save→Geometric Entities**. To determine what format the files are in, view the case file and look under the FORMAT section.

EnSight data is stored in a case file, which contains references to all associated geometry and variable files. Loading the case file will load all the associated files. EnSight parts are translated into zones with the caveat that unstructured parts with dissimilar element types (i.e. a volume element and a surface element) will only load the primary element type. Unstructured zone names will be prefixed by the type of zone they represent (point, line, surface, or volume). Vector, tensor, and tensor9 variables are expanded into the appropriate number of variables with the variable name followed by a suffix. Complex 'imaginary' variables will have an 'I' following the name to distinguish them from the 'real' variable.

When loading Ensight data, choose the desired case file, and the associated data files will be loaded.

Macro Commands for the EnSight Loader

The syntax for loading EnSight data files with the Tecplot macro language is as follows:

```
$!READDATASET
' "STANDARDSYNTAX" "1.0" "FILENAME_CASEFILE" "file name"'
DATASETREADER = 'EnSight Loader'
```

Each name/value pair should be in double quotes. Refer to the Scripting Guide for details on working with the Tecplot macro language.

Keyword	Available Value(s)	Default	Notes
STANDARDSYNTAX	1.0	n/a	Must be the first instruction.
FILENAME_CASEFILE	"*filename*"	n/a	The full or relative path of the case file name.
READDATAOPTION	NEW, APPEND, REPLACE	n/a	NEW to create new layout; APPEND to add to existing layout, REPLACE to replace data and retain style

Previous versions of the EnSight loader supported options for loading a subset of the data using IJK skipping, part selection, or variable selection. These options will not generate syntax errors, so old macros will continue to work. However, IJK skipping is ignored, and part/variable selection is supported only for Ensight 6 files.

Excel Loader

The Excel Loader can read numeric data from `.xls` files for Microsoft® Excel® version 5.0 or higher.

The **Excel Loader** is available for Windows platforms only.

The Excel Loader is intended primarily for users who record experimental or simulation data using Excel. Your Excel file must contain only values (no equations). We recommend the use of the Excel add-on from the `Util/Excel` folder as an easier method to open Excel data with Tecplot 360 (see Excel Add-In). Use the Text Spreadsheet loader for delimited files (Text Spreadsheet Loader).

If your spreadsheet is arranged as Table Format or Carpet Format, the Excel Loader is a point-and-click operation. Once you have selected an Excel file to load, the Excel Loader leads you through a series of dialogs, prompting you to specify a variety of attributes, including the data format in the Excel spreadsheet, the variables to be read, and zone information.

> Refer to Excel Add-In for instructions on loading Excel data into Tecplot 360 from Excel.

Spreadsheet Data Formats

The Excel Loader will automatically identify blocks of data in Table Format or Carpet Format. The loader will list blocks of data in standard notation for Microsoft Excel. For example, a block found on worksheet sheet1, cells A1-D8, is listed as follows: `(sheet1! A1:D8)`.

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

Table Format

Use Table format for data that will be plotted in line plots (i.e. data with 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 dataset has the following characteristics:

- The dataset is arranged in one or more adjacent columns.
- Each column is the same length.
- Each cell contains numeric data.
- The first row is a header row containing the variable name for its corresponding column.
- The spreadsheet dataset is imported as a single I-ordered zone in `POINT` format with `N` variables, where `N` is the number of columns in the table.

> 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 contains a number.
>
> The block of data can contain no empty cells. An empty cell will prevent the loader from recognizing the block. You can satisfy this condition by filling blank cells with 0.0.

Figure 10 shows a block of data in table format in Excel.

	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 10. A block of data in table format.

Carpet Format

Use carpet format for spreadsheet data to be plotted in a 2D or 3D Cartesian plot. The carpet formatted dataset, shown in Figure 11, has the following characteristics:

- The spreadsheet dataset is imported as an IJ-ordered zone. See XY and Polar Line Plots.

  In Figure 11, 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.

- 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 dataset was generated from a function f, such that f(X, Y) = V.
- 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	4	8	
4	3	3	6	9	12	
5	4	4	8	12	10	
6						

Figure 11. This carpet table shows values as a simple arithmetic function of X and Y.

Other Formats

The Other format option gives you a great deal of flexibility in loading data into Tecplot 360. 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-format ASCII file.

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 360 handles an unformatted block in an ASCII file. It assumes one zone of I-ordered data in `POINT` format.

Custom format

Using the Custom format option, you can specify characteristics of your dataset. 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, specify 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 you want to read data from an Excel spreadsheet into more than one zone, you must use custom format. By default, all data read is put into a single I-ordered zone.

Excel Loader Restrictions

A block of data is a rectangular group of numbers in the spreadsheet. The Excel Loader places the following restrictions on blocks:

- Carpet and table format (which the loader detects and loads automatically) are 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. A cell containing `X=34` is interpreted by the loader as text, because 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, by pasting your table using the "Paste Special" function, with "values only" selected.
- The spreadsheet file must have been written by Excel Version 5.0 or higher. For newer versions of Excel (Office 2007 and later) that save in `.xslx` format by default, please switch to the older `.xsl` format when saving your file.

Exodus Loader

The Exodus file format is based on the netCDF file structure created by Sandia National labs. The Exodus library functions provide the mapping between FE data objects and netCDF dimensions, attributes, and variables. The accepted Exodus file extensions are `*.e`, `*.e.*`, `*.exo`, `*.exo.*`, `*.exoII`, `*.exoII.*`, `*.g`, `*.g.*`, `*.gen` and `*.gen.*`.

Macro Commands for the Exodus Loader

The syntax for loading Exodus data files with the Tecplot macro language is as follows:

```
$!READDATASET
' "STANDARDSYNTAX" "1.0" "FILENAME_FILE" "filename"'
DATASETREADER = 'Exodus File Loader'
```

Each name/value pair should be in double quotes. Refer to the Scripting Guide for details on working with the Tecplot macro language.

Keyword	Available Value(s)	Default	Notes
STANDARDSYNTAX	1.0	n/a	Must be the first instruction.
FILENAME_FILE	"filename"	n/a	The name of the file to load.
FILELIST_DATAFILES	"n" "file-1" "file-2"… . "file-n"	n/a	Specify the number of solution files, followed by each file name.
INCLUDESIDESETS	"True", "False", "T", "F", "Yes", "No", "Y", "N"	n/a	If name value pair is not present, side sets are not included.

Keyword	Available Value(s)	Default	Notes
ELEMENTBLOCKGROUPING	"ElementDimension" or "ElementBlock"	n/a	ElementDimension: All element blocks containing elements of the same dimension are grouped together into a single zone (one per time step) and, if transient, strand ID. For instance an Exodus dataset containing one or more Hex8 element blocks and one or more Shell4 element blocks is loaded into Tecplot as an FE-brick zone and a FE-quad zone. Element dimension grouping is the default loader behavior if the name/value command is not present in the loader instructions. ElementBlock: All element blocks are assigned individual zones (one per time step) and, if transient, strand IDs.

FEA Loader

Tecplot 360 includes the ability to load input and solution files from many popular finite element analysis (FEA) solvers. Supported formats include:

Solver/File Format	File Name/Extension
3D Systems STL	.stl
ABAQUS Data	.fil
ABAQUS Input	.inp
ABAQUS Output Database	.odb
ANSYS CDWRITE Input	.cdb
ANSYS CFX	.res
ANSYS Result	.rst, .rth, .rfl
LSTC-DYNA Input	.dyn, .k
LSTC-DYNA Taurus State	D3PLOT
MSC/NASTRAN Bulk Data	.bdf
MSC/NASTRAN Output2	.op2
MSC/PATRAN Neutral	.out
OpenFOAM	controlDict

Solver/File Format	File Name/Extension
PTC/Mechanica Design Study	.neu
SDRC/IDEAS Universal	.unv

The file formats supported for each solver's formats are:

Solver	Version
3D Systems STL	all
ABAQUS	up to 2024
ANSYS	up to 2022 R1
ANSYS CFX	up to 2024 R2
LST-DYNA	up to 970.0
MSC/NASTRAN	up to 2018
OpenFOAM	2.0, including compressed files
PTC/Mechanica Wildfire	up to 4.0
SDRC/IDEAS NX	up to Series 11

Format-Specific Notes

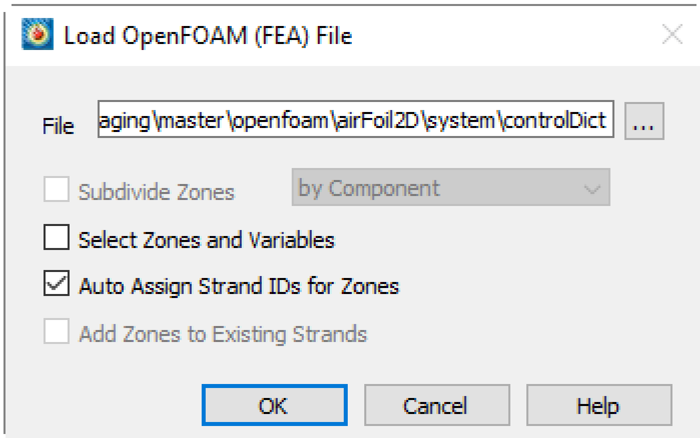
ABAQUS Output Database files are supported on Windows only. Other ABAQUS formats are supported on all platforms.

Abaqus `.odb` files from older versions will be converted to the current format before being loaded through the user interface. However, this conversion will not occur automatically for files loaded with a layout file or via a macro command. To work around this, you will need to convert the older `.odb` file by loading it explicitly and saving the new `.odb` file. Then replace the file names in their layouts/macros with the new names using a text editor.

Von Mises stress data is loaded for NASTRAN data files that contain it. The FEA loader also can load both cell-centered and nodal data from NASTRAN files. This ability was added in Tecplot 360 2016 R3 and may cause macros and layouts that refer to variables by number to choose the wrong variable. A warning will appear when loading such files. To address the warning, re-create the layout or macro, or else update the style settings as necessary and change the !$READDATASET instruction in the layout or macro to include the parameter "FEALoaderVersion" "436".

Loading FEA Data

FEA formats have "(FEA)" appended to the format names in the Load Data dialog. Choosing to open a selected file using the FEA loader's advanced options displays a dialog like the one below. The dialog shown is for the ANSYS Result file format, but all formats use the same dialog (only the file format, displayed in the dialog's title bar, differs)



- Click the **Browse (...)** button to change the file you wish to load. By default, it is the file you selected in the Load Data dialog.

For OpenFOAM cases, load the `controlDict` file in the case's `system` directory. The OpenFOAM loader currently supports polyhedral and non-polyhedral data including surfaces and boundaries, parallel data, and compressed data. If a file contains transient data, this loader loads all time steps including step 0.

Subdividing Zones

Each zone loaded from an FEA file typically represents the entire solution at a particular time step or load increment. Sometimes a solution will consist of many components that you may wish to display individually. To activate this option, choose the "Subdivide Zones" toggle and select the desired subdivision option from the menu. Tecplot 360 provides you with two ways to subdivide zones: by Component and by Element Type.

Subdividing Zones by Component

Some FEA file formats include the ability to identify components or sub-regions. If this information is available, you may direct Tecplot 360 to apply it by selecting the "by Component" option. Components within each solution step will be identified by sequentially numbered zone names in Tecplot 360, for example, "Component 1 Step 1 Incr 1," "Component 2 Step 1 Incr 1," and so on.

Subdividing Zones by Element Type

If component information is not available in a solution file, the above option will produce only one component per solution step and increment. In this case, it may still be possible to achieve the desired effect if sub-regions in the solution are represented by different element types, such as shell elements and brick elements. Selecting "by Element Type" from the subdivision option menu creates a separate Tecplot zone for each element type present in the solution file. Tecplot zone names will then represent each element type, for example, "Quadrilaterals Step 1 Incr 1" and "Tetrahedrals Step 1 Incr 1." This makes it easy to operate on individual components or sub-regions in Tecplot 360's **Zone Style** dialog by selecting the desired zones by name.

Selecting Zones and Variables to Load

See Selecting Zones and Variables to Load.

Auto Assign Strand IDs for Zones

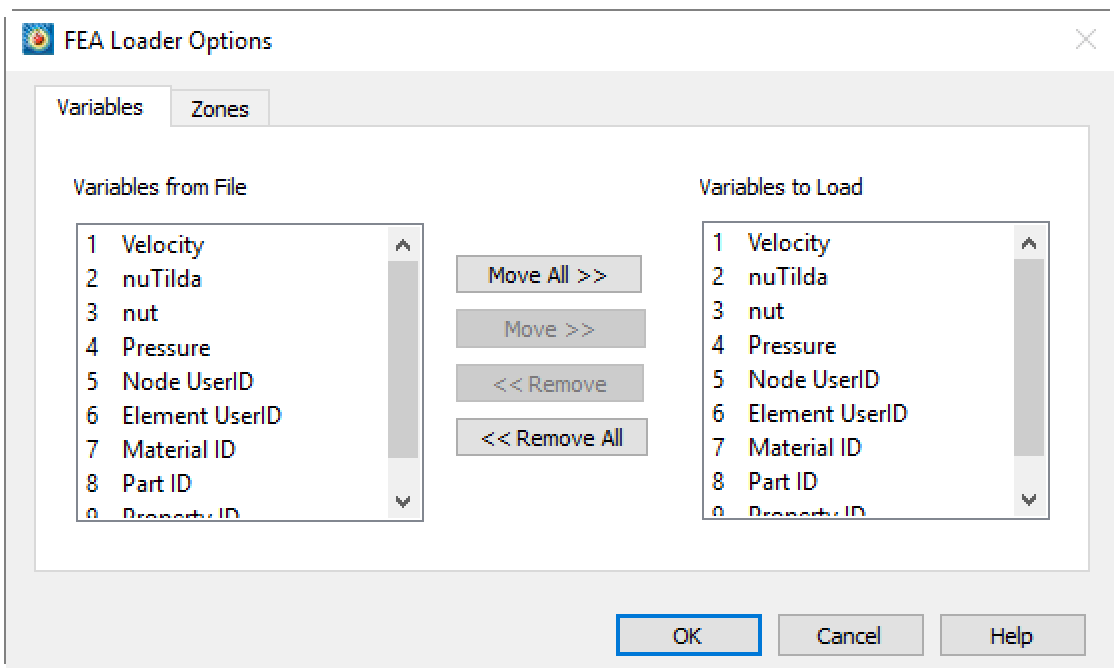
Regions or components of solutions throughout an unsteady solution are tracked by Strand IDs. All zones that represent a particular region or component are assigned the same Strand ID. Selecting this option directs Tecplot 360 to assign Strand IDs to the loaded zones. This ensures that only the zones representing the chosen solution time are displayed in Tecplot 360. Zones that do not have Strand IDs assigned are displayed at all solution times. See also Time Aware.

Add Zones to Existing Strands

If you are appending data to an existing dataset, select Add Zones to Existing Strands to append the new zones to existing strands. This is appropriate where the new data represents the same regions or components as are represented in the existing dataset, such as an additional solution time level of an unsteady solution.

Selecting Zones and Variables to Load

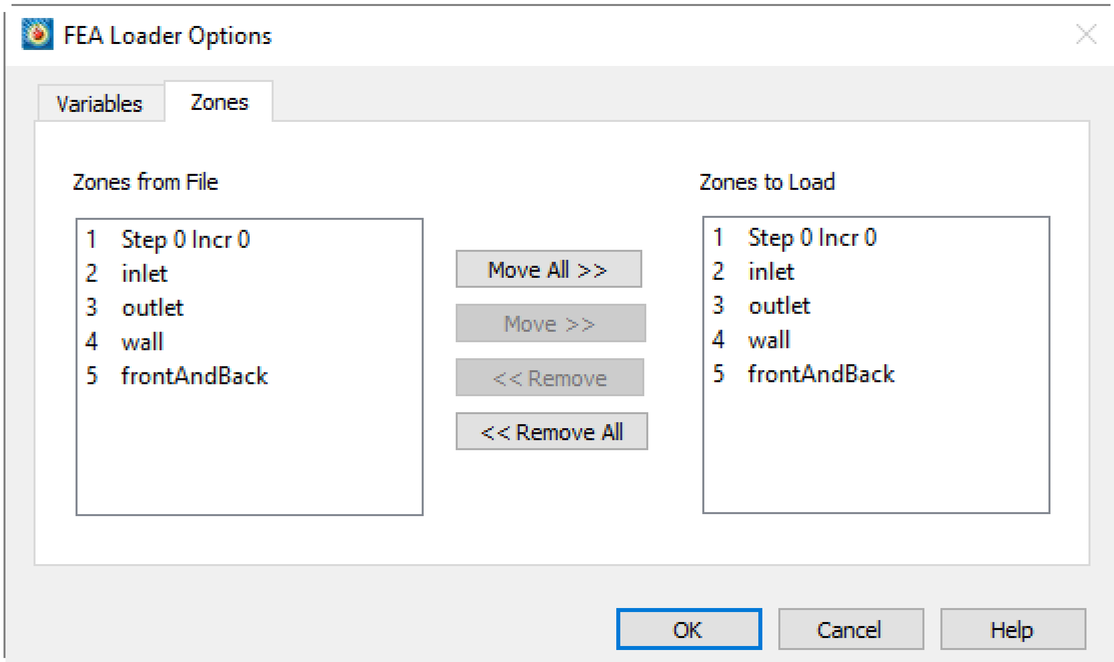
By default, Tecplot 360 loads all zones and variables present in the solution file, unless multiple steps or increments are present. In this case, Tecplot 360 will not load step 0 increment 0 (which normally has no solution data associated with it). If you wish to load step 0 increment 0, or a subset of the available zones or variables, choose the "Select Zones and Variables" toggle in the main loader dialog. When you then click OK, the **FEA Loader Options** dialog will be displayed, as shown below:



Use the **Move All**, **Move**, **Remove**, and **Remove All** buttons to add or subtract zones or variables from the list.

The **Variables** page is displayed above. The **Zones** page displays the zone list. If you elect to subdivide zones, the zones will be subdivided in the list. The figure below shows a zone list where "Subdivide

Zones by Component" has been chosen:



When you have selected the zones and variables you wish to load, select **OK**.

The resulting Tecplot zones for each step and increment in the file will be named accordingly in Tecplot 360, beginning with Step 1 Incr 1. The precise meanings of "Step" and "Increment" are solver and problem-dependent, but normally correspond to time steps in unsteady cases, load increments in steady-state cases, or frequencies or vibrational modes in harmonic analyses.

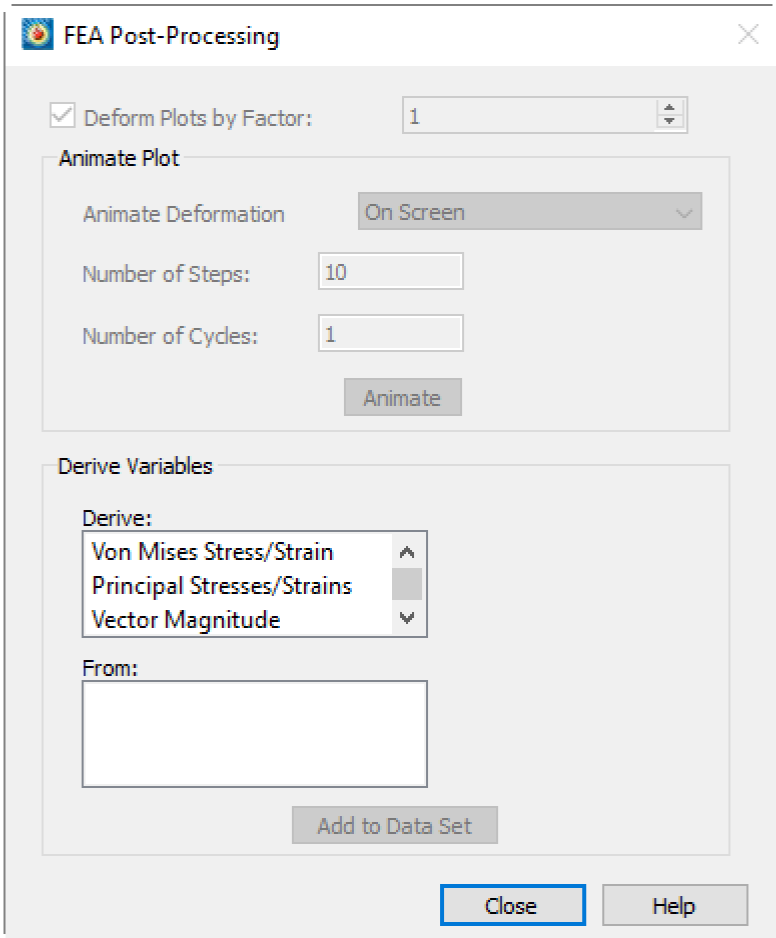
Appending Finite Element Data to an Existing Dataset

If you wish to add a finite element solution to data you have already loaded, select "Add to current data" set in the **Load Data File Warning** dialog, which appears after you have selected the file and zones and/or variables to load.

Zones from the file will be added sequentially at the end of the current zone list, and new variables, if any, will be appended to the current variable list. The new zones will not be plotted initially. To plot the appended zones, select them in the **Zone Style** dialog, toggle on the **Zone Show** checkbox, and choose **Activate**.

Post-processing Finite Element Data

When you load an FEA solution into Tecplot 360, the **FEA Post-processing** dialog is displayed if Advanced Options checkbox is turned on in the initial Load Data dialog (unless you are appending to an existing solution). You may re-display it at any time by selecting "FEA Post-processing" from the **Tools** menu.



The following formats will trigger the automatic display of the FEA Post-Processing dialog:

- ABAQUS
- ANSYS Input
- ANSYS Results
- ESI/PAM-CRASH
- LSTC/DYNA
- MSC/Nastran
- MSC/Patran
- PTC/Mechanica
- SDRC/IDEAS

The three sections of this dialog allow you to: deform the plot using deformation read from the solution file, animate the deformation, and derive new variables from the solution variables.

Deforming the FEA Plot

Finite element solutions commonly include deformations calculated from applied loads. When a solution is initially read in, the un-deformed geometry is displayed. If the file contains deformation data, you can display the deformed geometry by toggling-on "Deform Plots by Factor". The deformation factor is displayed to the right of this toggle. You may enter the deformation factor in this text field, or use the up or down arrows next to it to change it. By default, the Deform Plot by Factor toggle is checked and the field is set to "1".

Deriving New Variables from an FEA Solution

FEA solutions may consist of various types of stress and strain, or gradients of scalar quantities such as temperature. The lowest section of this dialog allows you to calculate certain other quantities of interest that may be derived from these basic solution variables. For tensor quantities such as stress and strain, the principal stresses or strains plus Von Mises stress are available. For vector quantities, the vector magnitude may be calculated. Choose the derivation you want in the Derive list, and a list of candidate source variables in the solution will be displayed in the From list. Choose the source variable and click **Calculate** to add the desired quantity to the dataset. If Tecplot 360's Calculate-on-demand feature is active, the variable will only actually be calculated when it is displayed. In this case, you may notice no delay when you select **Calculate**, but some delay later when you choose to display the variable by selecting it, for example, as the contour variable.

Macro Commands for the FEA loader

You may also load FEA data files with the Tecplot macro language. The syntax is as follows:

```
$!READDATASET
' "STANDARDSYNTAX" "1.0"
"...any of the name value pairs in the following table..." '
DATASETREADER = 'FEA LOADER' See List below
```

The value for the DATASETREADER parameter should match the name of the loader as shown in the Select Import Format dialog, or as listed below.

- ANSYS® CDWRITE Input (FEA)
- ANSYS® Results (FEA)
- ABAQUS Input (FEA)
- ABAQUS .fil Data (FEA)
- ABAQUS Output Database (FEA)
- LSTC/DYNA Input (FEA)
- LSTC/DYNA Taurus State Database (FEA)
- MSC/NASTRAN Bulk Data (FEA)
- MSC/NASTRAN Output2 (FEA)
- MSC/Patran Neutral (FEA)
- OpenFOAM (FEA)

- PTC/Mechanica Design Study (FEA)
- SDRC/IDEAS Universal (FEA)
- 3D Systems STL (FEA)

Each name/value pair should be in double quotes.

Keyword	Available Value(s)	Notes
STANDARDSYNTAX	1.0	Required as the first instruction.
FEALoaderVersion	"424", "435", "436", "443", "446", "450", "452", "461", "66051", "131842"	Indicates the version of the loader that recorded the macro. Later versions of the loader emulate older versions where necessary. Current version is "131842".
Append	"Yes" or "No"	Specify whether to append the current dataset with the FEA file(s).
FILENAME_File	"filename"	Specify the full or relative path of the file name.
SubdivideZonesBy	"DoNotSubdivide" "Component" "ElementType"	Specify method of zone division.
AutoAssignStrandIDs	"Yes" or "No"	Set to "Yes" to have Tecplot 360 assign the strand IDs.
AddToExistingStrands	"Yes" or "No"	Available only if Append is set to "Yes".
ZoneList	"Z1,Z3,Z6-Z8,…"	Specify the list of zones to load. You may specify a comma-separated list or use a range (-).
VarNameList	"VarName1 + VarName2" + …	Specify the list of variables to load. Use the + symbol between each variable name.
InitialPlotType	"Cartesian3D" "Cartesian2D"	Set the initial plot type.
ShowFirstZoneOnly	"Yes" or "No"	Specify whether to show only the first zone.
BoundaryZoneConstruction	"Reconstructed" or "Decomposed"	Set how the boundary zones are constructed. Available for OpenFOAM datasets only.
IncludeSolutionDependentVariables	"Yes" or "No"	Specifies whether Solution Dependent Variables should be loaded. Available for Abaqus ODB datasets only. Defaults to "No" if omitted.

Example

The following example loads "myfile.odb" with the Abaqus Output Database loader. Zones 1 & 2 are loaded, along with the following variables: external force, stress, material ID, and part ID.

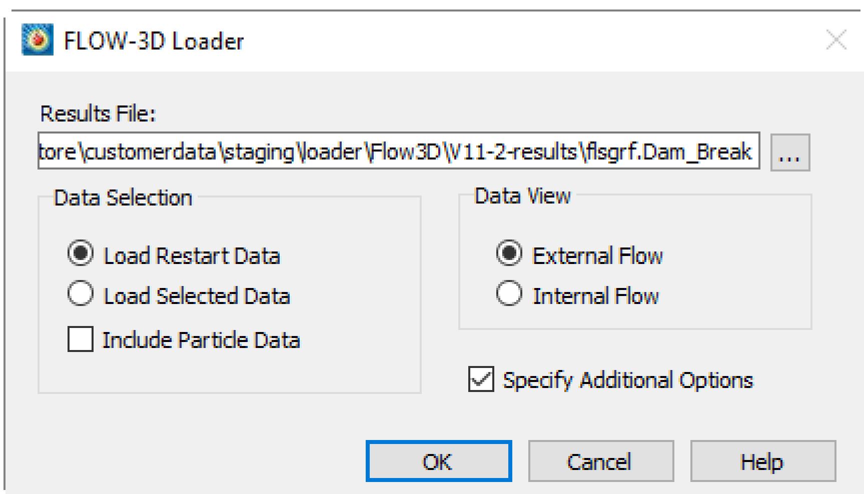
```
$!READDATASET
'"STANDARDSYNTAX" "1.0"
"FILENAME_File" "myfile.odb"
"SubdivideZonesBy" "Component"
"AutoAssignStrandIDs" "Yes"
"ZoneList" "1-2"
"VarNameList" "External Force"+"Stress"+"Material ID"+"Part ID"
"InitialPlotType" "Cartesian3D"
"ShowFirstZoneOnly" "No"'
 DATASETREADER = 'ABAQUS Output Database (FEA)'
```

FLOW-3D Loader

The FLOW-3D loader allows you to load restart and selected FLOW-3D data files into Tecplot 360. This loader supports Flow3D 11.0.1.15.

> When working with FLOW-3D data in Tecplot 360, we recommend linking the solution time between frames using Frame→Frame Linking.

The **FLOW-3D Loader** dialog has the following options:



File

Use the browse button **[...]** to launch the **Read FLOW-3D Data File** dialog which will allow you to navigate to the data file you would like to load. You may load data files with the flsgrf extension only. Alternatively, you may type the full path of the data file in the File text field.

When loading a parallel (MPI) results file, selecting flsgrf1 loads the data for all files. Selecting `flsgrf`# will load a single file's data, where # is an integer greater than 1.

Data Selection

Use the Data Selection region of the dialog to specify whether to load restart or selected data. You may also opt to Include Particle Data or to select a Data Subset.

Load Restart Data

Select this option to load restart data into Tecplot 360. Restart data contains every simulation variable at a small number of time steps.

Load Selected Data

Select this option to load selected data. Selected data typically includes fewer variables than restart data. However, it usually has a larger number of time steps. Selected data is used to output variables of interest at many time steps without bloating the output file with "uninteresting" variables.

> Selected data is available in the file only when you request it before the simulation run.

Include Particle Data

Toggle-on "Include Particle Data" to load the particle data from your data file.

Data View

Use the Data View region of the dialog to specify whether to view the data as external or internal flow. This option affects how the solid surfaces are drawn at block boundaries. For external flows, surfaces are drawn only at blocked boundaries in the mesh. This option is recommended for solutions that involve flow around obstructions in free space. For internal flows, surfaces are drawn around open space in the mesh, and blocked surfaces are eliminated. This option is recommended for solutions that involve flow into an enclosed volume, such as casting results.

Specify Additional Options

Select the "Specify Additional Options" toggle to launch the FLOW-3D Loader Options dialog after selecting **OK** on the **FLOW-3D Loader** dialog. The FLOW-3D Loader Options dialog allows you to load a subset of zones and/or variables from the data file. The Options page of the dialog allows you to specify transient options, specify boundary cell options, and calculate the complement of F.

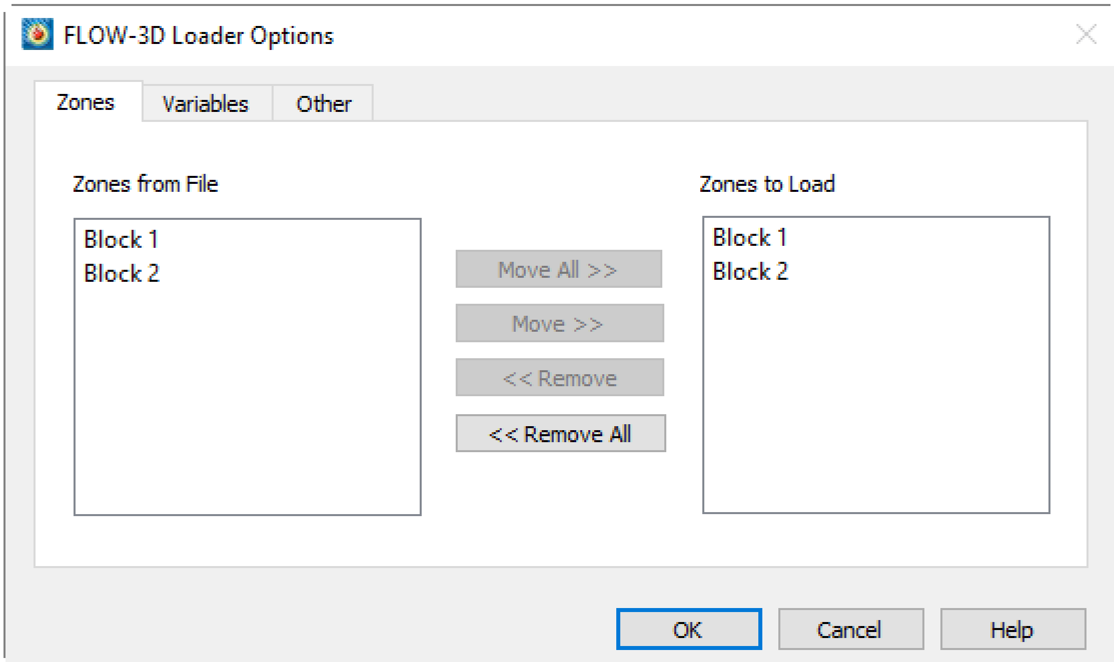
FLOW-3D Loader Options

The **FLOW-3D Loader Options** dialog is launched when you select the "Advanced Options" toggle in the initial Load Data dialog.

Zones Page

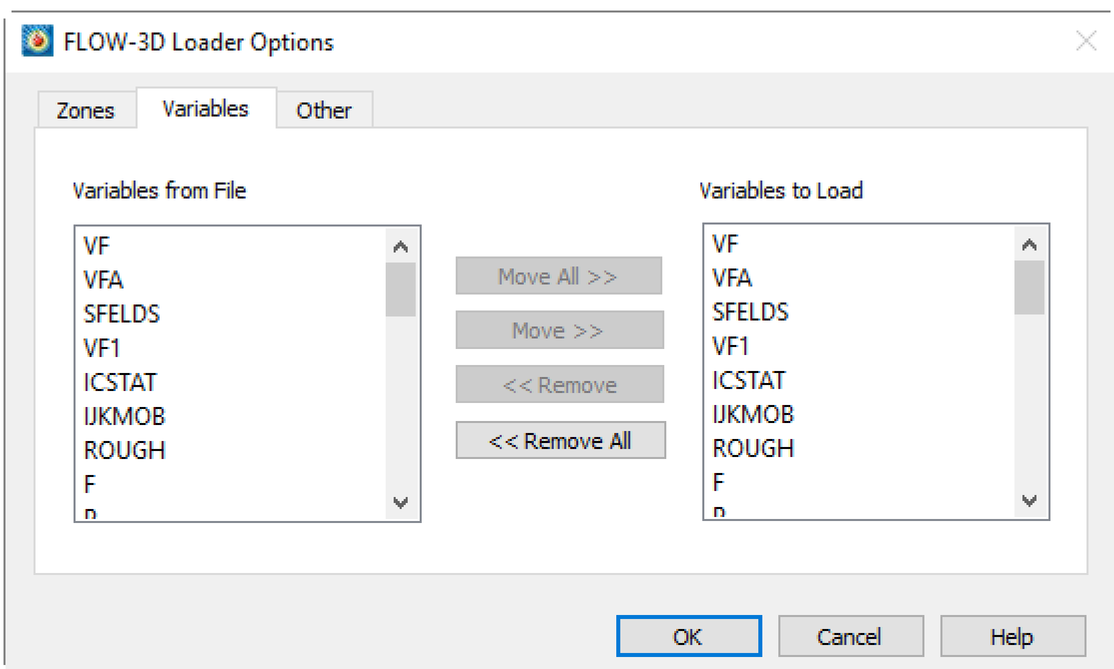
Use the Zones page of the dialog to select which zones from your dataset to load into Tecplot 360. The box on the left lists the available zones, and the box on the right lists the variables selected to load into

Tecplot 360.



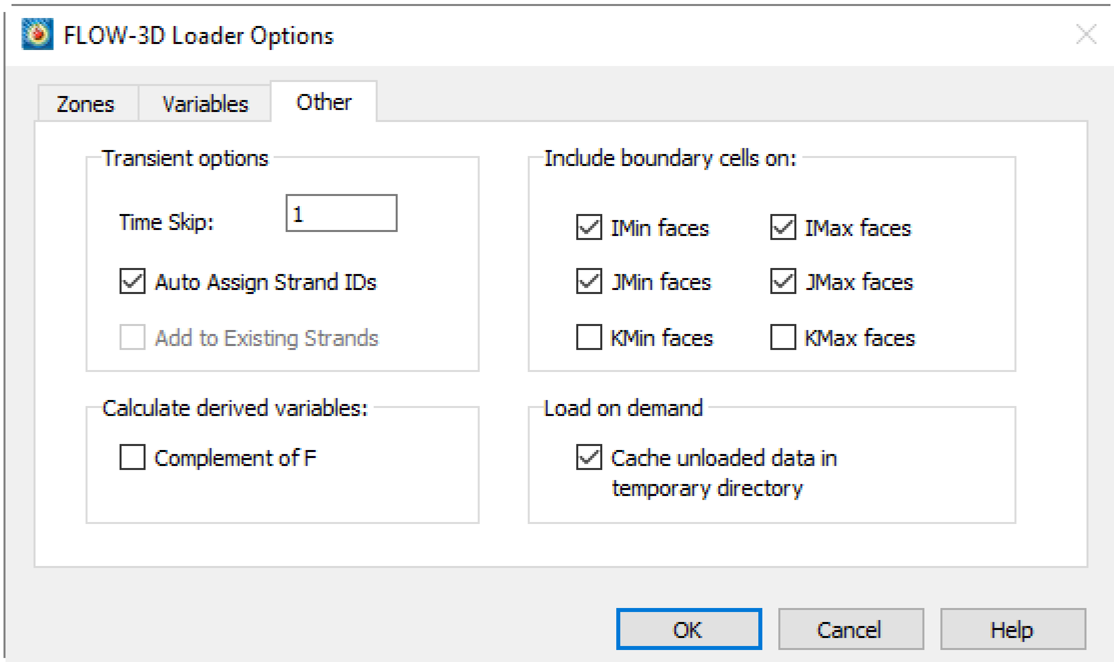
Variables Page

Similarly, use the Variables page of the dialog to select which variables to load.



Other Page

Use the Other page of the dialog to specify transient options, boundary options, and whether to calculate the complement of F.



The page has the following options:

Transient Options

Use the Transient Options region of the dialog to specify:

Time Skip

Specify the interval between each loaded time step. A value of one loads all time steps, a value of two loads every other time step, and so forth.

Auto Assign Strand IDs

Select this option to allow Tecplot 360 to assign the Strand IDs to your data. Regions or components of solutions throughout an unsteady solution are tracked by Strand IDs. All zones that represent a particular region or component are assigned the same Strand ID. Selecting this option directs Tecplot 360 to assign Strand IDs to the loaded zones. This ensures that only the zones representing the chosen solution time are displayed in Tecplot 360. Zones that do not have Strand IDs assigned are displayed at all solution times. See also Time Aware.

Add Zones to Existing Strands

If you are appending data to an existing dataset, select Auto Assign Strand IDs to Zones in order for Tecplot 360 to append the new zones to existing strands. This is appropriate where the new data represents the same regions or components as are represented in the existing dataset, such as an additional solution time level of an unsteady solution.

For more information on working with transient data, refer to Time Aware.

Calculate Derived Variables

Use the Calculate Derived Variables region of the dialog to opt to include the *Complement of F*. The complement of F is calculated as:

$$\text{Complement of F} = \{1 - f\} * \{vf\}$$

where f is the fluid fraction and vf is the volume fraction.

The Fluid Surface, where Fluid Surface $= \{vf\} \cdot \{f\}$, is always calculated and added to the dataset.

Include Boundary Cells On

Use the boundary cell region of the dialog to specify whether to load boundary cells on the I, J, or K extrema. An additional layer of boundary cells will be loaded on the given side of each block for each extremum selected.

Load on Demand

Toggle-on "Cache unloaded data in temporary directory" to enable Tecplot 360 to create a temporary directory to cache the data. The data in the temporary directory is formatted such that it may be quickly read back into Tecplot 360 as needed.

FLOW-3D Macro Commands

The `$!READDATASET` macro command is extended for the FLOW-3D loader with the following options:

Keyword	Values	Default	Notes
StandardSyntax	1.0	None	
FILENAME_File	Path to flow-3d results file	None	Specifies the path to the file to load.
DataGroup	"Selected" or "Restart	"Restart"	Specifies which data group to load from the file.
IncludeParticleData	"Yes" or "No	"No"	"
DeriveCompOfF	"Yes" or "No	"No"	"
Append	"Yes" or "No	"No"	"
AutoAssignStrandIDs	"Yes" or "No	"Yes"	"
ZoneList	Set of zone numbers to load from the file.	All	FLOW-3D refers to these as "blocks." Really, they are a set of zones that will belong to the same StrandID.
VarNameList	Set of variable names to load from the file	All	X, Y, Z, and "Fluid Surface" are always loaded

Keyword	Values	Default	Notes
DataView	"Internal" or "External"	External	Specify whether to view the data as an internal or external flow solution.
IncludeBoundaryCells	"YES/NO" "YES/NO" "YES/NO" "YES/NO" "YES/NO" "YES/NO"	"Yes" for all entries	Specify 6 boolean values for including boundary cells. The values are use the following order for the boundary cells: IMIN, IMAX, JMIN, JMAX, KMIN, KMAX

FLOW-3D Auxiliary Data

The following auxiliary data is added to the dataset by the loader:

Auxiliary name	Value
Common.UVar	Number of variable "U"
Common.VVar	Number of variable "V"
Common.WVar	Number of variable "W"
Common.VectorVarsAreVelocity	TRUE
Common.PressureVar	Number of variable "P"
Common.DensityVar	Number of variable "RHO"
Common.TemperatureVar	Number of variable "TN"
Common.StagnationEnergyVar	Number of variable "RHOE"
Common.TurbulentKineticEnergyVar	Number of variable "TKE"

Auxiliary Data can be viewed on the Aux Data Page of the Data Set Information dialog (accessed via the **Data** menu).

FLUENT Loader

The FLUENT® Data Loader reads FLUENT Version 5 and newer case (`.cas`) and data (`.dat`) files up to and including version 14.0. To load files from earlier versions of FLUENT, you must first import them into FLUENT 5 or newer, then resave them in the newer format.

Particle data may also be loaded from an accompanying XML file. If loading a single `.dat/.cas` pair, all sections of particle data in an accompanying XML file are loaded. If loading multiple `.dat/.cas` pairs, one particle section is loaded from the XML file for each `.dat/.cas` pair loaded. If the XML file is invalid, particle data will be skipped, but the rest of the data will continue to be loaded.

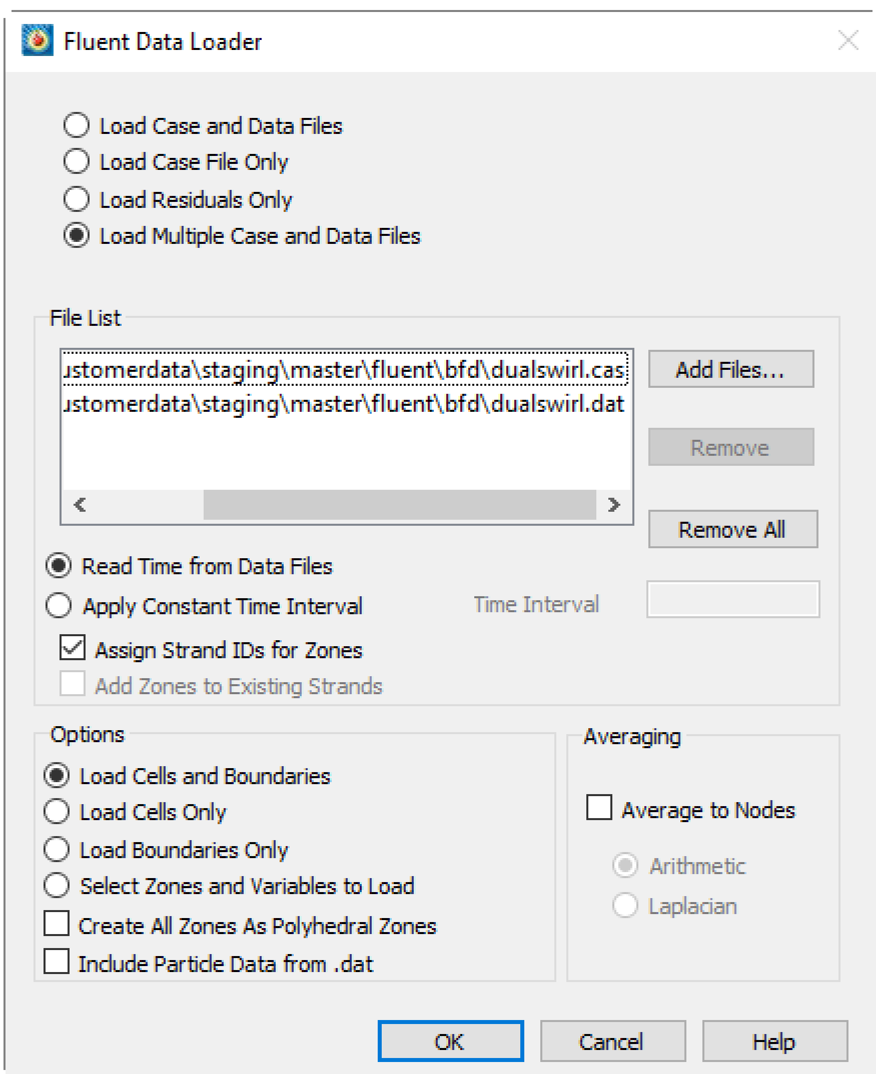
When possible, and assuming the user has the necessary permissions to create new files in the directory containing the FLUENT files, the FLUENT loader creates an index file for each case and data file it loads. These files are stored in a subdirectory called `tecplot-derived` located in the same directory as the case and data files and are used to load the data faster in future sessions. The FLUENT loader assumes that any files it finds in the `tecplot-derived` directory relate to the case and data files found in the parent directory. If the FLUENT data files are updated, delete the files in the `tecplot-derived` directory so that the FLUENT loader will regenerate them. (Be sure all Tecplot 360 sessions using the files are closed first.)

Tecplot 360 does not automatically calculate CFD variables from your existing FLUENT data variables. You may add the variables to your plots by performing calculations via the **Analyze** menu. Refer to CFD Analysis for details.

Additionally, Tecplot 360 does not perform the same wall-boundary calculations that are performed by FLUENT. Instead, the cell-centered data will be extrapolated to the boundary.

See also Time Aware for information on working with transient datasets in Tecplot 360.

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 (DEFAULT)

Displays the File List form in the dialog. You can load matched pairs of case and data files, or one case 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 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 360 does not use this data, but other add-ons may.

Flow Solution is Unsteady/Time Interval

The FLUENT data loader saves the problem time of each solution as the solution time variable. There are two options for determining the time to save for each one: (1) reading the flow-time entry from each `.dat` file, or (2) applying a constant time interval to successive `.dat` files.

Read Time from Data Files

If this option is selected, Tecplot 360 reads the flow-data parameter from each `.dat` file. If no `.dat` files are included (i.e. only `.cas` files are loaded), the solution time variable will not be created for the zones.

Apply Constant Time Interval

If this option is selected, Tecplot 360 applies the time interval specified in the Time Interval text field to zones created from successive `.cas` or `.dat` files. The zones from the first `.cas`/`.dat` files are given time 0. Times for successive files are calculated by incrementing the time of the previous files by the specified time interval.

Assign Strand IDs for Zones

Toggle-on to have Tecplot 360 assign Strand IDs to transient zones. Common strand IDs will be assigned to each cell or boundary zone with matching FLUENT zone IDs.

Add Zones to existing Strands

Toggle-on to add the appended zones to StrandIDs in the current dataset.

Add Zones to existing Strands is available only when the current dataset is being appended and Assign Strand IDs for Zones is toggled-on.

Time Interval

If "Apply Constant Time Interval" is selected, the time interval entered in the text field is included.

For the load options other than 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 360.

Load Cells Only

Loads only the cell (solution) zones. Each zone will be displayed as a separate zone in Tecplot 360.

Load Boundaries Only

Loads only the boundary zones. Each zone will be displayed as a separate zone in Tecplot 360.

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.

Create All Zones As Polyhedral

Select this option to load all FLUENT zones as Tecplot 360 polytope (polygonal or polyhedron) zones. We recommend you select this option, as converting all zones to polyhedral zones eliminates the possibility of hanging nodes and holes in your iso-surfaces or slices. In this case, the number of faces per element is derived from the element-type, and the number of nodes per face is derived from the face-type. The existence of hanging nodes (determined from the existence of a cell-tree and/or face-tree section) adds to the number of faces in the element and the number of nodes in the face that contains the hanging node. Since polygons must have at least 3 nodes, line segment elements will not be converted. When this option is not selected, only FLUENT polytope zones will become Tecplot 360 polytope zones. In this case, if hanging nodes are encountered, Tecplot 360 will create larger faces, compress connectivity, and expand face neighbors.

Include Particle Data from .dat

Some FLUENT simulations include the effects of discrete particles, such as sand grains or water droplets, in the `.dat` file. 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 360 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.

Newer versions of FLUENT store the particle data in a separate `.xml` file. To load this data, use the Load Multiple Case and Data Files mode and simply add any `.xml` files to the file list.

Average to Nodes

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 360, especially slicing. 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. However, you have the option to average the data to the nodes using Arithmetic or Laplacian averaging. Arithmetic averaging is faster, but calculates values at hanging nodes (nodes in the center of a cell face or edge) only from those cells where the node is a corner. This can lead to discontinuous contours. Laplacian averaging option takes additional neighboring cells into account, and results in smoother contours when hanging nodes are present. By default, non-grid variables are stored at cell centers, consistent with FLUENT.

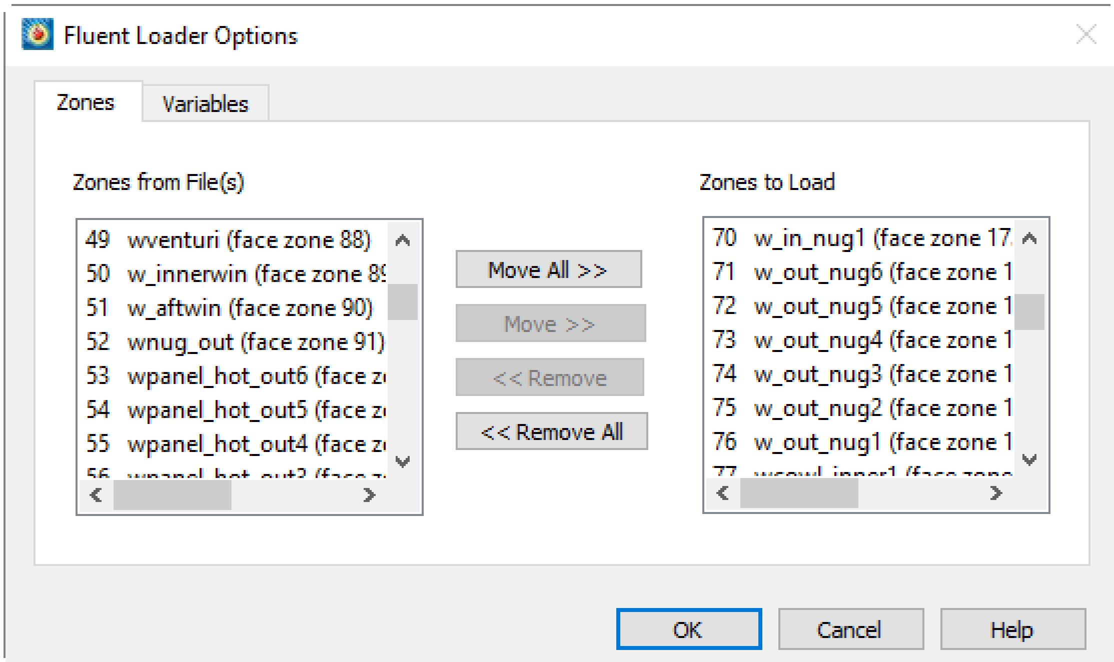
Arithmetic

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, select only those zones and variables you wish to load from the files **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 select **OK**. Use the **Move**, **Move All**, **Remove**, or **Remove All** buttons to edit the Zones/Variables to Load lists.

Macro Commands for the FLUENT loader

The syntax for loading FLUENT data files with the Tecplot macro language is as follows:

```
$!READDATASET
' "STANDARDSYNTAX" "1.0"
"...any of the name value pairs in the following table..." '
DATASETREADER = 'FLUENT DATA LOADER'
```

Each name/value pair should be in double quotes. Refer to the Scripting Guide for details on working with the Tecplot macro language.

Keyword	Available Value(s)	Default	Notes
STANDARDSYNTAX	1.0	n/a	Must be the first instruction.
Append	"Yes" "No"	"No"	Specify whether to append the current dataset with the FLUENT file(s).
LoadOption	"CaseAndData" "CaseOnly" "ResidualsOnly" "MultipleCaseAndData"	n/a	Specify whether to load case and data files, a case file only, residual data only, or multiple case and data files.

Keyword	Available Value(s)	Default	Notes
FILENAME_CaseFile	"*filename*"	n/a	Specify the full or relative path of the case file name. Used if the LoadOption is CaseAndData or CaseOnly.
FILENAME_DataFile	"*filename*"	n/a	Specify the full or relative path of the data file name. Used if the LoadOption is CaseAndData or ResidualsOnly.
FILELIST_Files	"n" "file1" "file2"… "filen"	n/a	Specify the number of files, followed by each file name. Only available if the LoadOption is MultipleCaseAndData.
UnsteadyOption	"ReadTimeFromDataFiles" "ApplyConstantTime Interval"	"ReadTimeFromDataFiles"	Only available if LoadOption is MultipleCaseAndData. For "ApplyConstantTimeInterval", the TimeInterval parameter is required.
TimeInterval	"<double→"	"1.0"	Specify the value of the time interval. Only available if the UnsteadyOption is set to ApplyConstantTimeInterval.
AssignStrandIDs	"Yes" "No"	"No"	Only available if LoadOption is MultipleCaseAndData.
AddZonesToExisting Strands	"Yes" "No"	"No"	Only applicable when Append is set to "yes".
GridZones	"CellsAndBoundaries" "CellsOnly" "BoundariesOnly" "SelectedZones"	"CellsAndBoundaries"	If "SelectedZones" is specified, either the ZoneList parameter, the VarNameList parameter, or both parameters are required.
ZoneList	"Z1,Z2,…Z3-37"	all zones	Specify the list of zones to load. You may specify a comma-separated list or use a range (-). This option is only available if GridZones is set to SelectedZones.
VarNameList	"V1+V2+V3"+….	all variables	Specify the list of variables to load. Use the + symbol between each variable number. This option is only available if GridZones is set to SelectedZones.

Keyword	Available Value(s)	Default	Notes
IncludeParticleData	"Yes" "No"	"No"	Available only for CaseAndData and MultipleCaseAndData load options. Applies only to particle data in the `.dat` file, if loading particle data from an XML file, the XML file should simply be included as one of the files to be loaded in FILELIST_Files.
AllPolyZones	"Yes" "No"	"No"	Not available if the load option is ResidualsOnly. Set to "Yes" to convert all zones to Tecplot polytope zones (polyhedral or polygonal).
AverageToNodes	"Yes" "No"	"Yes"	Specify whether to average the cell-centered data to the grid nodes.
AveragingMethod	"Arithmetic" "Laplacian"	"Arithmetic"	Specify the averaging method to use. Available only if AverageToNodes is set to "yes".
LoadAdditional Quantities	"Yes" "No"	"No"	Loads additional quantities, such as residuals, which may be stored in the file.

Fluent Common Fluid Files (CFF) Loader

The "Fluent Common Fluid Files Loader" is a loader based on ANSYS's latest, HDF5 file format for its Fluent solver. Loader instructions, following Tecplot's Standard Syntax, are given to Tecplot's $!ReadDataset command to control how the case and data files are loaded.

Platform support

Windows

Full support

Linux

Full support.

Mac

The loader is based on a binary library from ANSYS that is not supported on the Mac.

Macro Commands for the Fluent Common Fluid Files loader

The syntax for loading Fluent Common Fluid Files with the Tecplot macro language is as follows:

Note that the StandardSyntax 1.0 name/value pair option must be the very first instruction. All other name/value(s) options can be specified in any order.

```
$!ReadDataSet
' "Standard Syntax" 1.0
...any of the name value pairs in the following table... '
DataSetReader = "Fluent Common Fluid Files Loader"
```

Each name/value pair should be in double quotes. Refer to the Scripting Guide for details on working with the Tecplot macro language.

Keyword	Available Value(s)	Default	Notes
STANDARDSYNTAX	1.0	n/a	Must be the first instruction.
FileList_DataFiles	n "file1" "file2"... "filen"	n/a	**REQUIRED**. The FileList_DataFiles command expects an integer count representing the number of files to follow. The count is followed by file paths to one or more case files, where each case file is followed by zero or more data files. Case files must have the extension `.cas.h5` and data files must have the extension `.dat.h5`.

Keyword	Available Value(s)	Default	Notes
SolutionTimeSource	Auto SteadyState ConstantTimeInterval	Auto	**Auto**: Loader first looks for and uses solution time specified in the simulation settings of the Fluent Cff data files. If that information isn't present in all supplied data files, the loader looks for solution time embedded in the file names. If solution time cannot be determined, the loader assigns strands and a constant time interval, starting at zero and incrementing by one, if there is a single case file, or if there are case and data file combinations, otherwise the loader assigns static stand IDs and a solution time of zero to all zones. **SteadyState**: Loader assigns static strand IDs and a solution time of zero to all zones. **ConstantTimeInterval**: Loader assigns strands and a constant time interval, starting at zero and incrementing by one.
IncludeInteriorFaceZones	True, Yes, T, Y, False, No, F, N	False	**True, Yes, T, Y**: Interior face zones used to build the cell zones are also loaded as independent face zones, and added to the end of the other loaded face zones, such as walls, symmetry, etc. **False, No, F, N**: Interior face zones are not loaded as independent face zones, however, they are used to construct the cell zones.

Keyword	Available Value(s)	Default	Notes
IncludeParticleZones	True, Yes, T, Y, False, No, F, N	True	**True, Yes, T, Y**: If particle data exists in the `.dat.h5` file(s) it is loaded and represented as additional I-ordered zones and nodal variables. **False, No, F, N**: Particle data is not loaded even if it exists in the `.dat.h5` file(s).

Example

`$!ReadDataset` command using the "Raw String" formatting, which allows the command to be spread over multiple lines and to contain quotation marks and other characters without escaping them. Raw strings begin with `R"(` and end with `)"` allowing the string within those delimiters to contain any character that doesn't match the closing delimiter:

```
$!ReadDataset R"(
StandardSyntax 1.0
FileList_Datafiles 6
"/path/to/my/data/vessel-15.cas.h5"
"/path/to/my/data/vessel-15.30001.dat.h5"
"/path/to/my/data/vessel-15.30002.dat.h5"
"/path/to/my/data/vessel-15.30003.dat.h5"
"/path/to/my/data/vessel-15.30004.dat.h5"
"/path/to/my/data/vessel-15.30005.dat.h5"
SolutionTimeSource ConstantTimeInterval
IncludeInteriorFaceZones True
IncludeParticleZones False
)"
DatasetReader = "Fluent Common Fluid Files Loader"
```

FVCOM Loader

The FVCOM Loader allows you to import netCDF files output from FVCOM into Tecplot 360. Currently, classic netCDF and netCDF-4 formats are supported. The loader imports one or more FVCOM history outputs with the same topology and variable structure into a single Tecplot dataset, creating one zone for each time step. FVCOM history outputs with differing topology or variable structure can be combined but only through appending.

The X and Y grid variables are loaded directly from the file and projected to each Z position, which is derived from the zeta, h and siglev variables. The grid variables are chosen based upon the Coordinate System specified in the FVCOM attribute data. If no Coordinate System is provided, Cartesian is assumed.

The following auxiliary data is added to the data by the loader:

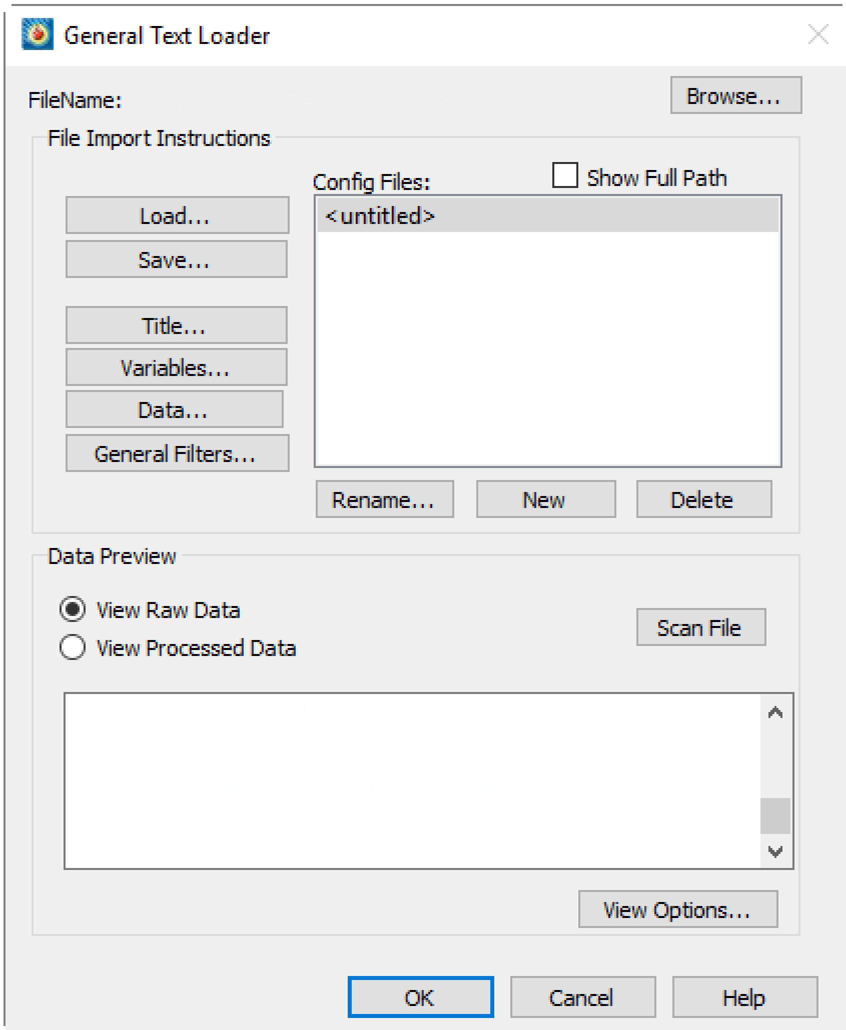
Auxiliary Name	**Cartesian**	**Spherical**
Common.XVar	Number of variable "x"	Number of variable "lon"
Common.YVar	Number of variable "y"	Number of variable "lat"
Common.ZVar	Number of variable "z"	Number of variable "z"

The vertical level and layer variables are added for visualization of FVCOM model layers. FVCOM variables stored at the nodes of each sigma level are loaded as nodal variables and variables stored at the elements for each sigma layer are loaded as cell-centered variables. FVCOM variables stored at the nodes of each sigma layer are interpolated to the volume cell-centers ignoring any FVCOM specified boundary conditions. Variables located at the elements of each sigma level are ignored. Attributes for the file and each variable are loaded into dataset and variable auxiliary data respectively.

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 dataset title information or indicate specific places to read them from in your data file. 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 following options are available:



Filename

The name of the file to be loaded. This is automatically filled in with the name of the file selected in the Load Data dialog, but may be changed using the **Browse** buton.

Titles

Launches the Dataset Title dialog, which allows you to specify dataset title properties.

Variables

Launches the Variable Import Instructions dialog which allows you to specify dataset variable properties.

Data

Launches the General Text Loader: Data dialog which allows you to specify dataset field properties.

General Filters

Launches the General Text Loader: Filters dialog which allows you to specify general filters when reading your file.

Configuration File List

This list shows available configuration files. Configuration files can be edited using a text editor, although this is not usually necessary and is not recommended. The format of these files is listed on the Configuration page.

Load

Loads a single configuration file from any location.

Save

Saves a single configuration file to any location.

Rename

Renames a configuration file.

Delete

Deletes a configuration file.

New

Creates a new, untitled configuration file.

Data Preview

View Raw Data

This displays the data exactly how it looks in the file without any processing.

View Processed Data

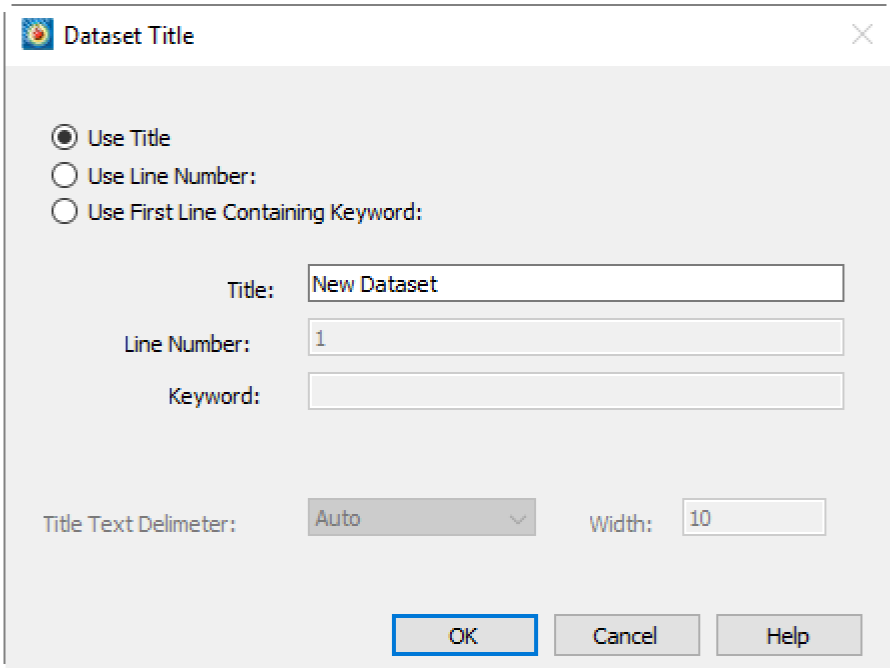
This displays the processed and filtered data that will be loaded.

View Options

Launches the General Text Loader: View Options dialog.

Dataset Title

The **Dataset Title** dialog allows you to specify options for General Text Loader titles.



Use Title

Manually enter the dataset title, rather than have General Text Loader scan the file for it.

Use line number

Enter the line number of the dataset title in the file. The General Text Loader skips white space on the line until text, and then reads until the delimiter indicated is found. To include spaces in the title, enclose them in double quotes.

Use first line containing keyword

Enter a keyword for the dataset title line. The title will read the first line containing this keyword (case insensitive). General Text Loader searches for a title on this line in the following order, (unless the delimiter is specified as fixed):

1. First, it will look for any text enclosed in double quotes. If it finds this, then the enclosed text will be read as the title.
2. If no text in double quotes is found, the first non-white space text after the keyword ending with the indicated delimiter will be used.

Text Delimiter

The text delimiter indicates when the end of text has been reached. You can set it to one of the following:

Auto

Space, tab, comma, semicolon.

Fixed

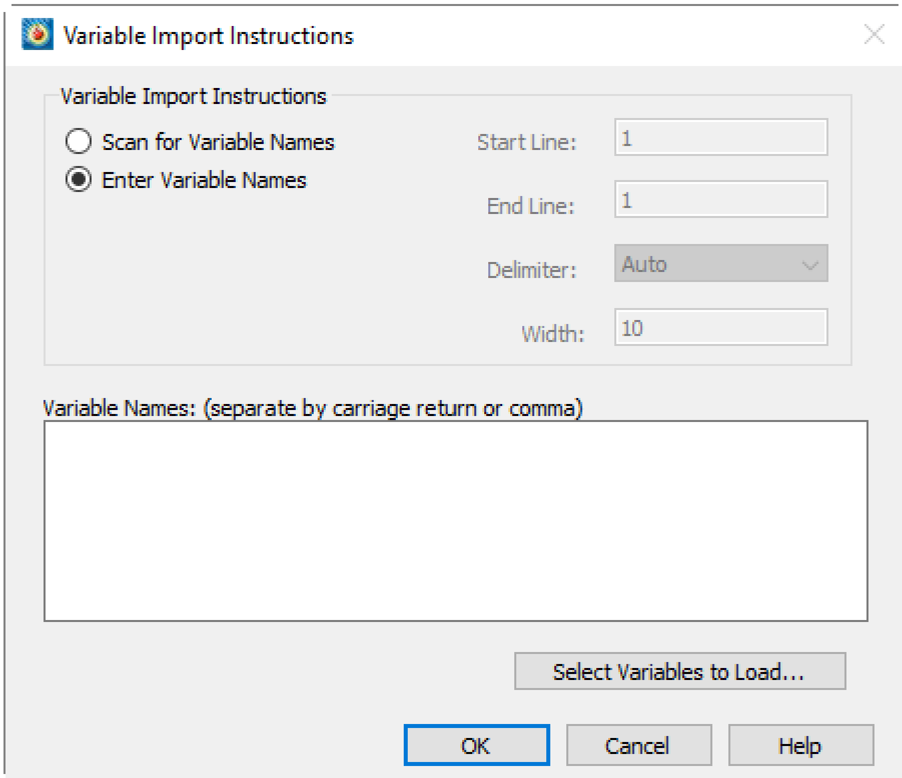
Each width number of characters on the line is a token field. White space is removed from the beginning and end of the field.

Width

If the delimiter is fixed, enter the width of each field here.

Variable Import Instructions

The **Variable Import Instructions** dialog of the General Text Loader allows you to scan for the location of the variable names in the data file, and enter which variables to load.



Scan for variable names

Specify the following:

Start line

Enter the starting line of variable names in the file.

End line

Enter the ending line of the variable names in the file. This is typically the same as the starting line.

Delimiter

The delimiter indicates when the end of each variable name has been reached. You can set it to one of the following:

Auto

Space, tab, comma, semicolon.

Fixed

Each 'width=n' number characters on the line is a variable. White space is removed from the beginning and end of the field. For example, if the line length is 60 and the width is ten, the columns 1-10, 11-20, 21-30, and so forth, are variable names. Spaces are removed from the beginning and end of the variable names.

Width

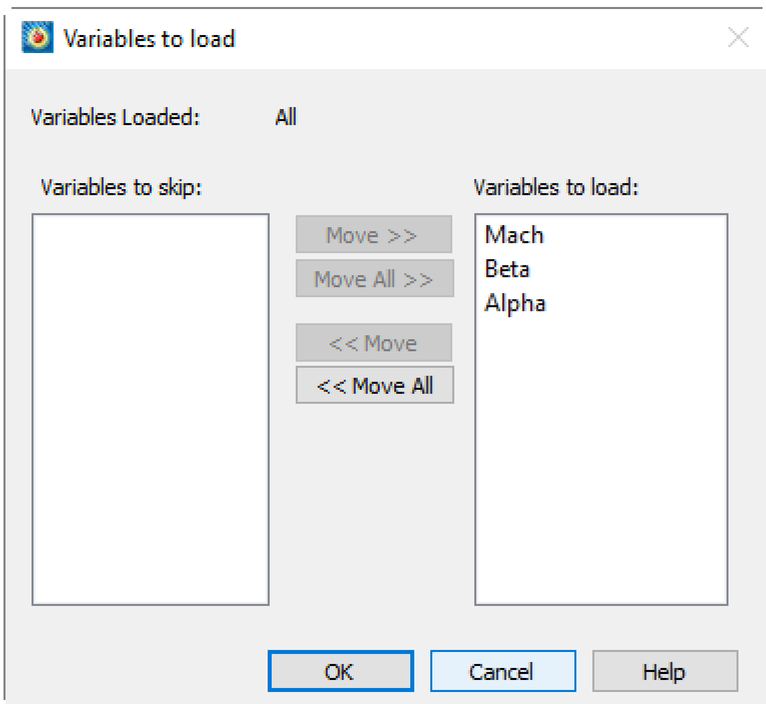
If the delimiter is fixed, enter the width of each field here.

Enter Variable Names

Select this option to enter a list of variable names in the dialog box. Variable names should be separated by carriage returns.

Select Variables to Load

Launches the **Variable to Load** dialog.



Variables to Skip

Displays a list of variables that will be skipped.

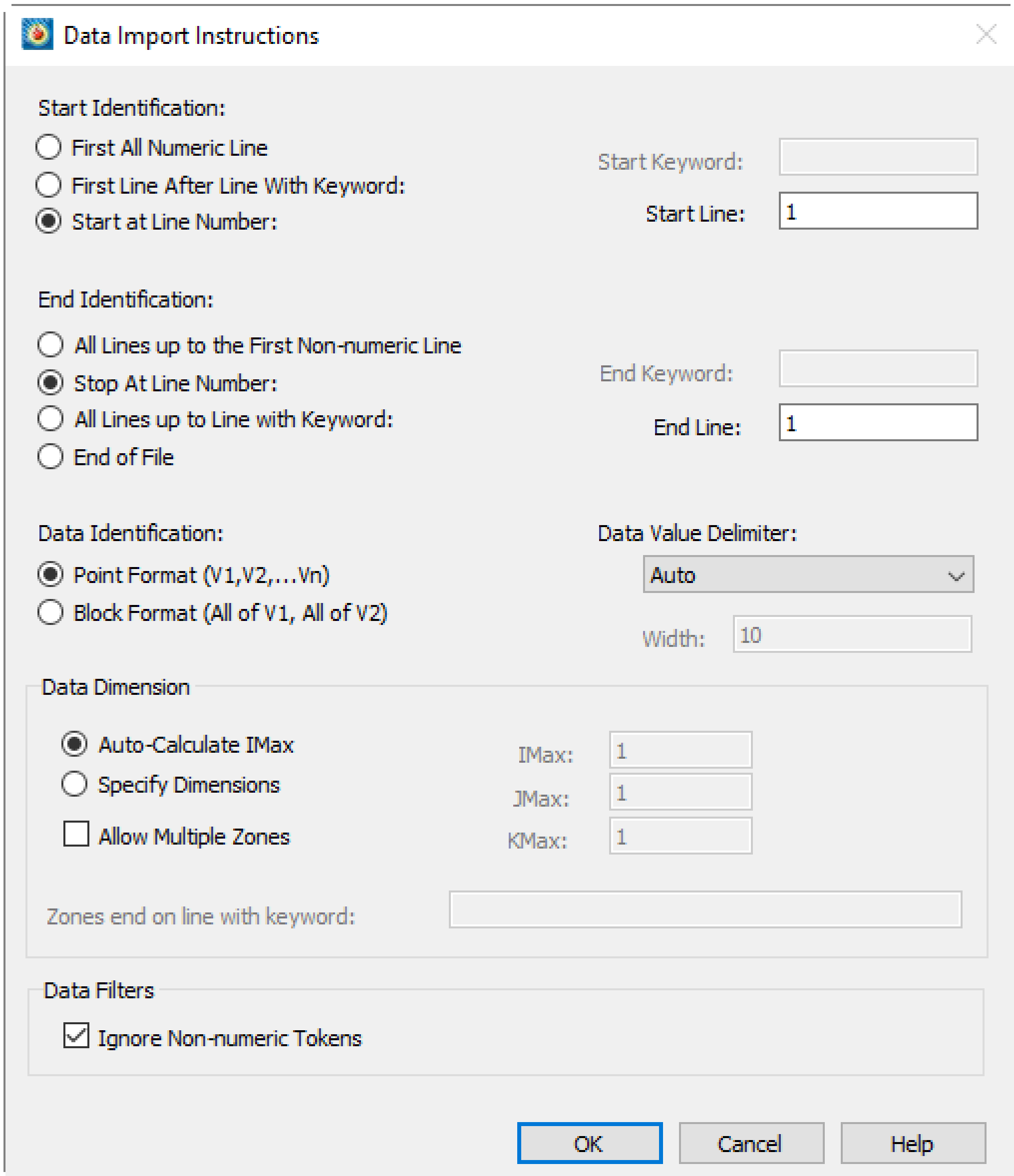
Variables to Load

Displays a list of variables that will be loaded.

Use the **Move**, **Move All**, **Remove**, or **Remove All** buttons to edit the "Variables to Load" list.

General Text Loader: Data

The **Data Import Instructions** dialog of the General Text Loader allows you to specify the location of the data names in the data file, and what data to load.



Start Identification

First all-numeric line

Select if the data begins at the first line of a file that contains only numbers. If you have specified multiple zones, all non-numeric lines will be skipped at the beginning of each zone.

First line after line with keyword

Select if the data begins at the first non-blank line after the line containing the specified keyword. The keyword is case insensitive.

Start at line number

Select to specify the line number where the data begins. Blank lines are ignored in the data section.

End Identification

All lines up to first non-numeric line

Select if the data ends at the first non-blank line containing any text.

Stop at line number

Select to specify the line number where the data ends.

All lines up to line with keyword

Select if the data ends at the first line before the line with the specified keyword. The keyword is case insensitive.

End of file

Select if the data ends at the end of file.

Data Identification

Point format

In this format all values of all variables are given for the first point, then the second point, etc.

Block format

In this format all values for the first variable are given, then all values for the second variable, etc.

Data value delimiter

The data value delimiter indicates when the end of a data value has been reached. You can set it to one of the following:

Auto

Space, tab, comma, semicolon.

Fixed

Each 'width=n' number characters on the line is a token field. White space is removed from the beginning and end of the field. For example, if the line length is 60 and the width is ten, the columns 1-10, 11-20, 21-30, and so on, are token fields.

Width

If the delimiter is 'fixed', enter the width of each field here.

Data Dimension

If the data dimensions are entered, General Text Loader adds zones as necessary depending on the number of data points found in the file. There must be an equal number of data points for each zone (equal to the product of the IJK dimensions).

Auto-Calculate IMAX

The I-dimension is calculated based on the number of data points found. J and K-max are set to one.

Specify Dimensions

Specify the I, J, and K-dimensions for the data. There must be enough data points found in the file to match the indicated dimensions.

Allow Multiple Zones

If checked, General Text Loader will attempt to read more than one zone from the data file.

Zone ends on line with keyword

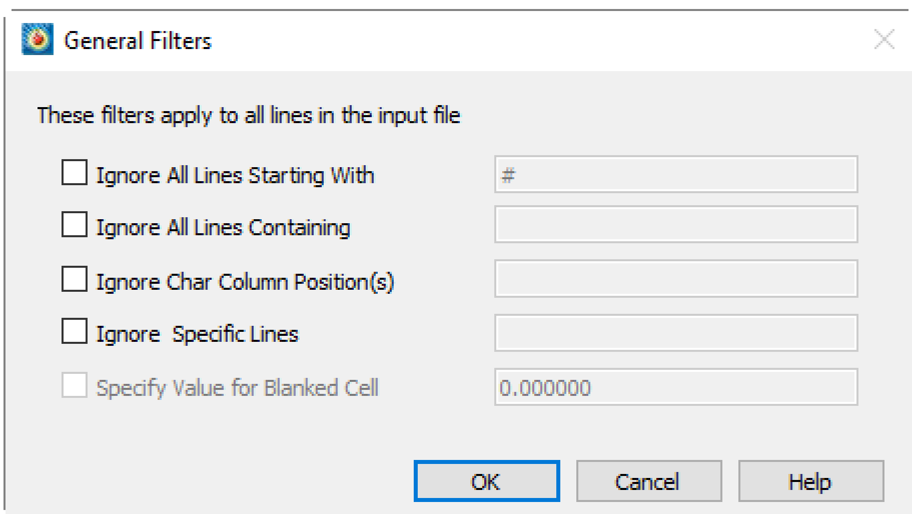
If Allow multiple zones is selected and Auto-calculate IMax is selected, then you must enter a keyword here to mark the end of one zone and the beginning of the next. Zones are ended when a line containing this text is found.

Ignore non-numeric tokens

If checked, then any non-numeric information in the data sections is ignored. If not checked, General Text Loader displays an error if any non-numeric data is found in the data section.

General Text Loader: Filters

Use the **General Filters** dialog of the General Text Loader to filter the data file.



Ignore All Lines Starting With

If checked, all lines beginning with the entered string are ignored.

Ignore All Lines Containing

If checked, all lines containing the indicated text are ignored.

Ignore Character Column Position(s)

If checked, then the entered columns are ignored when scanning the file. Columns are entered as a single number or a hyphenated range, one or more of which may be separated by commas.

> If there are tabs in the data file, they are not expanded in this filter. For example, if column 1 is a tab and you wish to skip column 2, you should enter 2, even though a

text editor will show more than one space after expanding the tab.

Ignore Specific Lines

If checked, entered lines are ignored when scanning the file. Lines are entered as a single number or a hyphenated range, one or more of which may be separated by commas. You may also use "end" to specify the last line of the file.

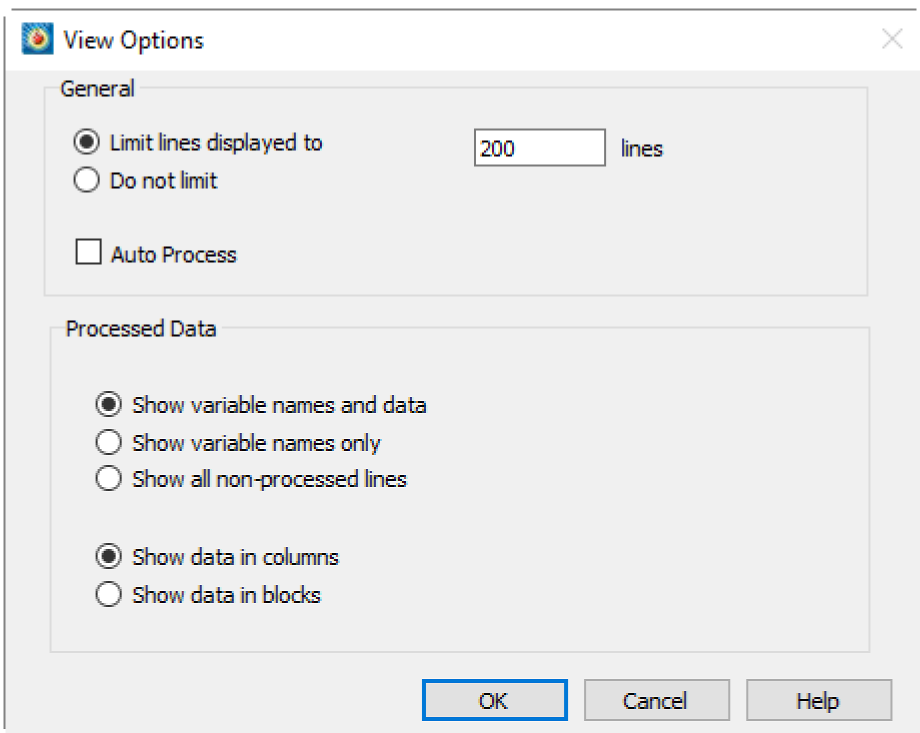
Specify Values for Blanked Cell

If checked, you can specify a value which the loader uses for blank cells.

This option is only available if the data delimiter is a comma or semicolon. You can change the data value delimiter using the **Data Import Instructions** dialog.

General Text Loader: View Options

The **View Options** dialog of the General Text Loader allows you to specify the data display.



General Options

Limit lines displayed

Limits the number of lines displayed in the preview window. For large files, you may want to set this to a number less than the total number of lines. The fewer number of lines, the faster the preview display.

Do not limit

If you select this toggle, the entire file will be displayed in preview mode.

Auto Process

If selected, General Text Loader automatically refreshes all information about the file whenever any loader settings are changed. For very large files (multi-megabyte), this option is not recommended, since re-scanning a large file can be time consuming.

Processed Data

Show variable names and data

If selected, variable names and processed data will be displayed in the preview window.

Show variable names only

If selected, only variable names will be displayed in the preview window.

Show all non-processed lines

If selected, all lines which will not be loaded will be displayed in the preview window.

Show data in columns

Shows the data in columns where each column is a variable.

Show data in blocks

Shows the data in blocks where each block is a variable.

General Text Loader Configuration File

A configuration file contains all of the instructions that tell the General Text Loader how to load a particular type of text file. This information is collected from the dialog fields and written to a file when you click **Save** on the main dialog. The configuration file format is similar to the Tecplot macro language. Configuration files for the general text loader are ASCII text files which use a command set that can describe all loading instructions. Normally you do not need to edit these files, as they are automatically written by the loader when you select New on the main dialog.

Editing these configuration files by hand is not recommended.

General Text Loader CONFIGFNAME Command

When reading a dataset using General Text Loader, instead of specifying individual parameters in `$!READDATASET`, you may use the `CONFIGFNAME` command. This consists of:

```
CONFIGFNAME = <string>
VERSION = <integer>
# version of the template file (default is 100)
# Note: changing the version number may cause unpredictable behavior
TITLE
{
   SEARCH = [NONE|LINE|KEYWORD] # default = NONE
     NAME = <string>
```

```
# default = "New Dataset", ignored if SEARCH is not NONE
     LINE = <integer> # 1-based, ignored if SEARCH is not LINE
  KEYWORD = <string> # ignored if SEARCH is not KEYWORD
DELIMITER = [AUTO|TAB|SPACE|SEMICOLON|COMMA|FIXED]
    WIDTH = <integer> # Valid only if DELIMITER = FIXED
}

VARIABLES
{
    SEARCH = [NONE|LINE]
     NAMES = <string> # ignored if SEARCH is SCAN
                       # <string> is a comma separated string
    LOADED = <all|n1,n2,...nn> # list of variables to be loaded
 STARTLINE = <integer> # 1-based, ignored if SEARCH=NONE, default = 1
 {
   STARTID = [FIRSTNUMERICLINE | LINE | KEYWORD]
   {
     KEYWORD = <string> # ignored if STARTID is not KEYWORD

     ENDLINE = <integer> # 1-based, ignored if SEARCH=NONE, default = 1
   DELIMITER = [AUTO|TAB|SPACE|SEMICOLON|COMMA|FIXED]
       WIDTH = <integer> # Valid only if DELIMITER = FIXED
   }
   DATA
   {
     IGNORENONNUMERICTOKENS = <boolean> # default = TRUE
     IMPORT
     LINE = <integer>
     # 1-based, ignored if STARTIDENTIFICATION is not LINE
   }
   ENDID = [FIRSTNONNUMERICLINE | LINE | KEYWORD]
   {
     KEYWORD = <string> # ignored if ENDID is not KEYWORD
        LINE = <integer> # 1-based, ignored if ENDID is not LINE
   }
      FORMAT = [POINT|BLOCK] # default POINT
   DELIMITER = [AUTO|TAB|SPACE|SEMICOLON|COMMA|FIXED]
       WIDTH = <integer> # Valid only if DELIMITER = FIXED
 }

 DIMENSION
 {
    AUTO=<boolean> # default = TRUE
    IMAX=<integer> # ignored if AUTO = TRUE, default = 1
    JMAX=<integer> # ignored if AUTO = TRUE, default = 1
    KMAX=<integer> # ignored if AUTO = TRUE, default = 1
    USEMULTIPLEZONES = <boolean> # ignored if AUTO = TRUE, default false
    KEYWORD=<string> # ignored if USEMULTIPLEZONES = FALSE
```

```
 }
}
GLOBALFILTERS # filters are applied cumulatively, so lines matching
# any of the criteria are filtered
{
COMMENT = <string> # ignore lines beginning with <string>
NUMBER = <integer> # ignore all lines starting with line number
<integer>
KEYWORD = <string> # ignore all containing <string> (case insensitive)
COLUMNS = <list> #<list> is a comma separated list of number ranges
# example: "1-80,100-end", etc. Must be in double quotes
ROWS = <list> # same as above
USEBLANKCELLVALUE = <boolean> # if TRUE, then the value of blank cells
is BLANKCELLVALUE
BLANKCELLVALUE = <double> # blank cell value. Ignored if
USEBLANKCELLVALUE is FALSE
}
```

Where `<string>` is a file name or file path. Settings will be loaded from the file name specified in `<string>`. This command is only allowed in conjunction with the `$!READDATASET` command as described below. It may not be used inside a configuration file.

For example, instead of:

```
$!READDATASET '"C:\test.txt" "VERSION=100 FILEEXT=\"*.txt\"
FILEDESC=\"general text\" "+""+"TITLE{SEARCH=NONE NAME=\"New
Dataset\" LINE=1 DELIMITER=AUTO WIDTH=10
}"+""+"VARIABLES{"+"SEARCH=LINE LOADED= All STARTLINE=1 ENDLINE=3
DELIMITER=SEMICOLON WIDTH=5
}"+""+"DATA"+"{"+"IGNORENONNUMERICTOKENS=TRUE
IMPORT"+"{"+"STARTID=LINE {"+"LINE=4
}"+""+"ENDID=FIRSTNONNUMERICLINE {"+"LINE=1 }"+""+"FORMAT=IJKPOINT
DELIMITER=AUTO WIDTH=1 }"+""+"DIMENSION"+"{"+"AUTO=TRUE
CREATEMULTIPLEZONES=FALSE
}"+"}"+"GLOBALFILTERS{"+"USEBLANKCELLVALUE=TRUE
BLANKCELLVALUE=0.000000 }"'
DATASETREADER = 'General Text Loader'
```

Using the `CONFIGFNAME` command, you can write:

```
$!READDATASET ' "myfile.dat"
"CONFIGFNAME=c:\config_files\myconfig.lgc" '
# contains all of the instructions in the example above
DATASETREADER='General Text Loader'
```

Components of the Configuration File

All General Text Loader configuration files must start with the line:

```
#!TECPLOT_LOADGEN
```

Instruction Syntax

Each instruction file contains commands which describe the loading instructions.

Comments

Any text following `#` to the end of the line is ignored.

String Format

The `<string>` parameter must be enclosed in double quotes. You can include a double quote character in the string by preceding it with a backslash `\`. For example:

- "This is a normal string"
- "This is a \"quote\" inside a string"

List Format

The <list> parameter type is defined as one or more number ranges, separated by commas, enclosed in double quotes. A number range may be a single number or two numbers separated by a dash. Optionally, you may use "end" to indicate the last valid number. For example:

- "1"
- "1,2-7,3"
- "10-end,3,2-5"

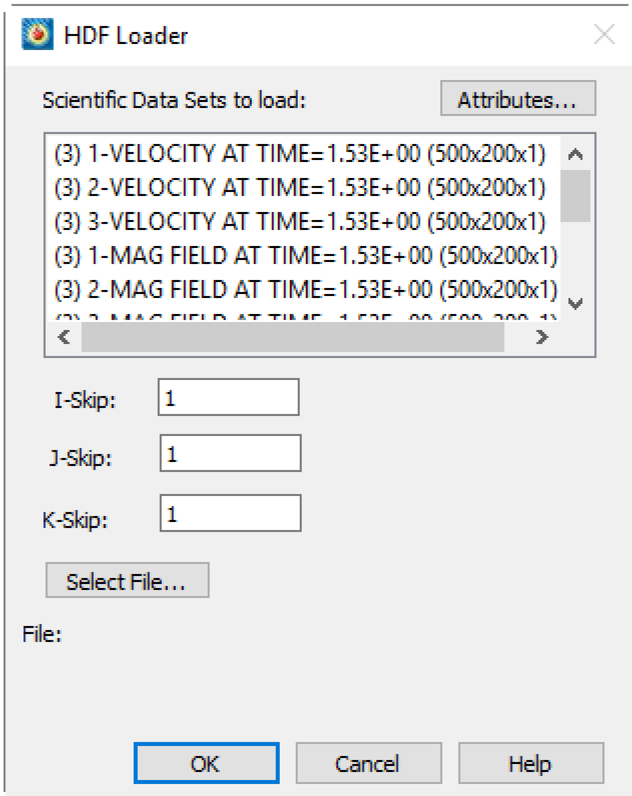
Command List

The commands in the file may appear in any order, and any command may be divided into any number of lines (that is, all white space, including carriage returns, is ignored).

HDF Loader

The HDF Loader add-on can load 1D, 2D, and 3D Scientific Data Sets (SDS) from HDF files.[1] When a dataset from an HDF file is imported, the file is scanned and a list of all SDS in the file is displayed in the Scientific Data Sets to load portion of the **HDF Loader** dialog. Select one or more SDS to import. **Every SDS that you select must have the same dimension.** A rectangular I, IJ, or IJK-ordered zone (for 1, 2, or 3D data, respectively) is created for each SDS that you select to load.

The **HDF Loader** dialog has the following options:



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 one 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

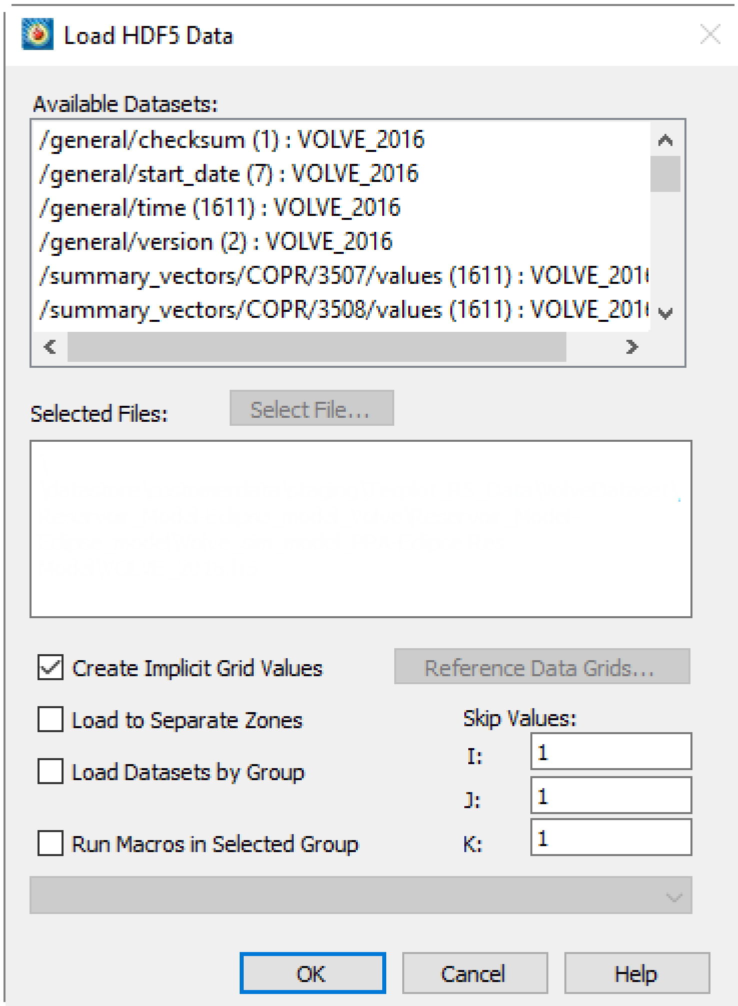
Displays attributes of each SDS found, such as number type, rank, label, and so on.

You may also click **Select File** to choose a different file.

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.

HDF5 Loader



The HDF5 loader add-on allows you to import general HDF5 files into Tecplot 360. The loader provides a mechanism for importing generic data from multiple HDF5 datasets or groups. The HDF5 loader will load datasets within user selected groups, load one or more user selected datasets to one zone, load multiple user selected datasets to multiple zones, execute macros after data has been loaded, create implicit X, Y, and Z grid vectors as needed, sub-sample loaded data, and reference user selected vectors for X, Y, and Z grids. Datasets must be ordered data. The HDF5 library used is version 1.12.0.

Data Selection

One or more files may be opened if all selected files have an identical hierarchy. Hierarchy information for the selected HDF5 files is displayed in the form: `/group/[group]…/dataset` - the dimension of each dataset is displayed immediately following the dataset name. In this window, one or more datasets or groups may be selected for loading.

Importing/Loading Data

Datasets may be loaded using one of three methods: 1) Loading Multiple Datasets to One Zone (default), 2) Loading Multiple Datasets to Separate Zones, or 3) Loading Datasets by Group.

Loading Multiple Datasets to One Zone (default)

Loading multiple datasets to one zone is the default method of importing HDF5 files. Using this method, the HDF5 loader will create one zone with N variables, where N is the number of HDF5 datasets selected. Selected datasets may have one to three dimensions. The dimension of loaded Tecplot 360 variables will match the I, J, and K values of the selected datasets. Variable names are assigned the corresponding names of selected datasets. **All selected datasets must have equivalent dimensions.**

To import your data, select one or more datasets from the **Available Datasets** window. All selected datasets must be identical in dimension; dataset dimensions are shown immediately to the right of dataset names in the **Available Datasets** window.

Loading Multiple Datasets to Separate Zones

Using this method, the HDF5 loader will create N zones, where N is the number of datasets selected in the **Available Datasets** window. Each zone contains one variable per selected dataset where each dataset must have one to three dimensions. The I, J, and K values of each Tecplot 360 zone will match the dimensionality of each selected dataset. Variable and zone names are automatically assigned. **Dimensionality may vary between selected datasets.**

To import your data using this method, select the "Load to Separate Zones" toggle. Select one or more datasets from the **Available Datasets** window. One zone will be created for each selected dataset and each zone will contain exactly one variable (unless you selected Create Implicit Grid Values or Reference Data Grids).

Loading Datasets by Group

Using this method, the HDF5 loader will create N zones with M variables, where N is the number of groups selected in the **Available Datasets** window and M is the number of datasets in each group. The I, J, and K indices of the Tecplot 360 variables will be equivalent to the respective dimension of selected datasets. Datasets in any selected group must be equal in dimension; however, datasets may be unequal in dimension between groups. When selecting multiple groups, all groups must contain an equal number of datasets and dataset names must be identical between groups. The HDF5 loader will only load datasets within the root directory or within a subgroup, i.e., the HDF5 loader will not load data within nested groups.

To import your data using this method, select the "Load Datasets by Group" toggle. Press **Select File** to open a HDF5 file. Select one or more groups from the **Available Datasets** window; all groups must contain an equal number of datasets where all datasets have identical names between groups. The number of selected groups determines the number of zones that load into Tecplot 360. Zone names will match the name of the corresponding group. Variable names will match the respective dataset name.

Each zone will include as many variables as datasets per selected groups.

Additional Options

Additional options may be specified when loading HDF5 data into Tecplot 360. These options include: Using Macros, Sub-Sampling Data, Referencing Data Grids, and Grid Generation.

Using Macros

Macros may be defined within a HDF5 vector and placed in any group. Each character string in the selected vector must be a valid one-line Tecplot macro. Macros are executed in the order encountered after all data are loaded.

To run a macro defined as a character vector in your HDF5 file, select the "Run Macros in Selected Group" toggle. Select the macro you want to execute from the **Select Macro** pull-down menu. Your macro will run after your data has been successfully loaded into Tecplot 360.

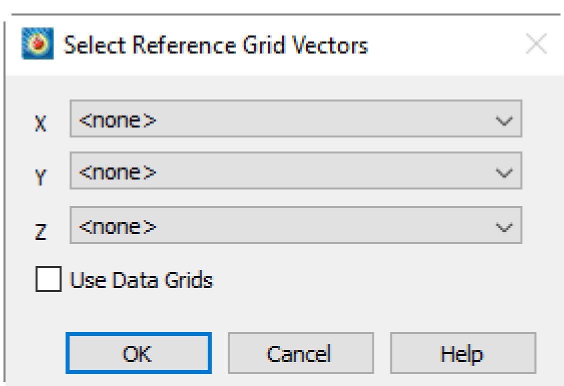
Sub-Sampling Data

The HDF5 loader will sub-sample the first, second, and third dimensions of loaded datasets respectively as defined by the user. The default skip-value is 1. When specifying non-unitary skip values, the dimensionality of all selected datasets must be equivalent. Datasets will be sub-sampled using the user defined I-Skip, J-Skip, and/or K-Skip values – skip values must be whole numbers.

To sub-sample data in the first, and/or second, and/or third dimensions of selected datasets, change the respective I-Skip, and/or J-Skip, and/or K-Skip values located in the **HDF5 loader** dialog. If the skip-values are non-unitary then selected datasets must have equivalent dimensions.

Referencing Data Grids

The HDF5 loader allows users to specify X, and/or Y, and/or Z grid vectors. Selected vectors are used for plotting all zones. Vectors are of dimension 1 and length M. The X grid vector length must be equal to the first dimension of selected datasets, the Y grid vector length must be equal to the second dimension of selected datasets, and the Z grid vector length must be equal to the third dimension of selected datasets. The number of selected grid vectors must equal the rank of selected datasets.



To define the grid vectors, first toggle off "Create Implicit Grid Values" option, then click the **Reference**

Data Grids button. In the Select Refernce Grid Vectors dialog that appears, select the vector you want to use as the corresponding grid using the menus. You MUST toggle on "Use Data Grids". The number of grid vectors you specify must equal the rank of selected datasets.

Grid Generation

The HDF5 loader can automatically create X, Y, and Z grid vectors as necessary for selected datasets. Grid vectors will be of length equal to the corresponding dimension.

To automatically create X, Y, and Z grid vectors, accept the default setting of "Create Implicit Grid Values" in the **HDF5 loader** dialog. The grid vectors will be created upon loading your data into Tecplot 360.

Kiva Loader

The Kiva loader imports GMV format files that were exported from Kiva.

Select Input Files

From this button, multiple files can be selected in the **Read Kiva/GMV File** dialog. Those that are in GMV format will be added to the list of Kiva/GMV files. Once files are added to this list, they will remain in the list throughout the Tecplot 360 session, unless the **Clear List** button is selected.

File Selections

Use the File Selection options for long file lists. Identify the first file to load by entering a number in the Start field, and the last file to load by entering a number in the Stop field.

Enter a value of 2 in the Skip field to load every other file, or 3 or greater to skip more files. To see the list selections updated according to the values in the Start, Stop, and Skip fields, click the **Apply Skip** button. At any time, you can choose to Select All or Deselect All files.

Velocity Vector

Identify the naming convention for your velocity vectors.

Loading Options

IsDouble

Allows greater precision for your data values.

LoadParticleData

Adds a zone for any files containing particle data.

Select Variable to Load

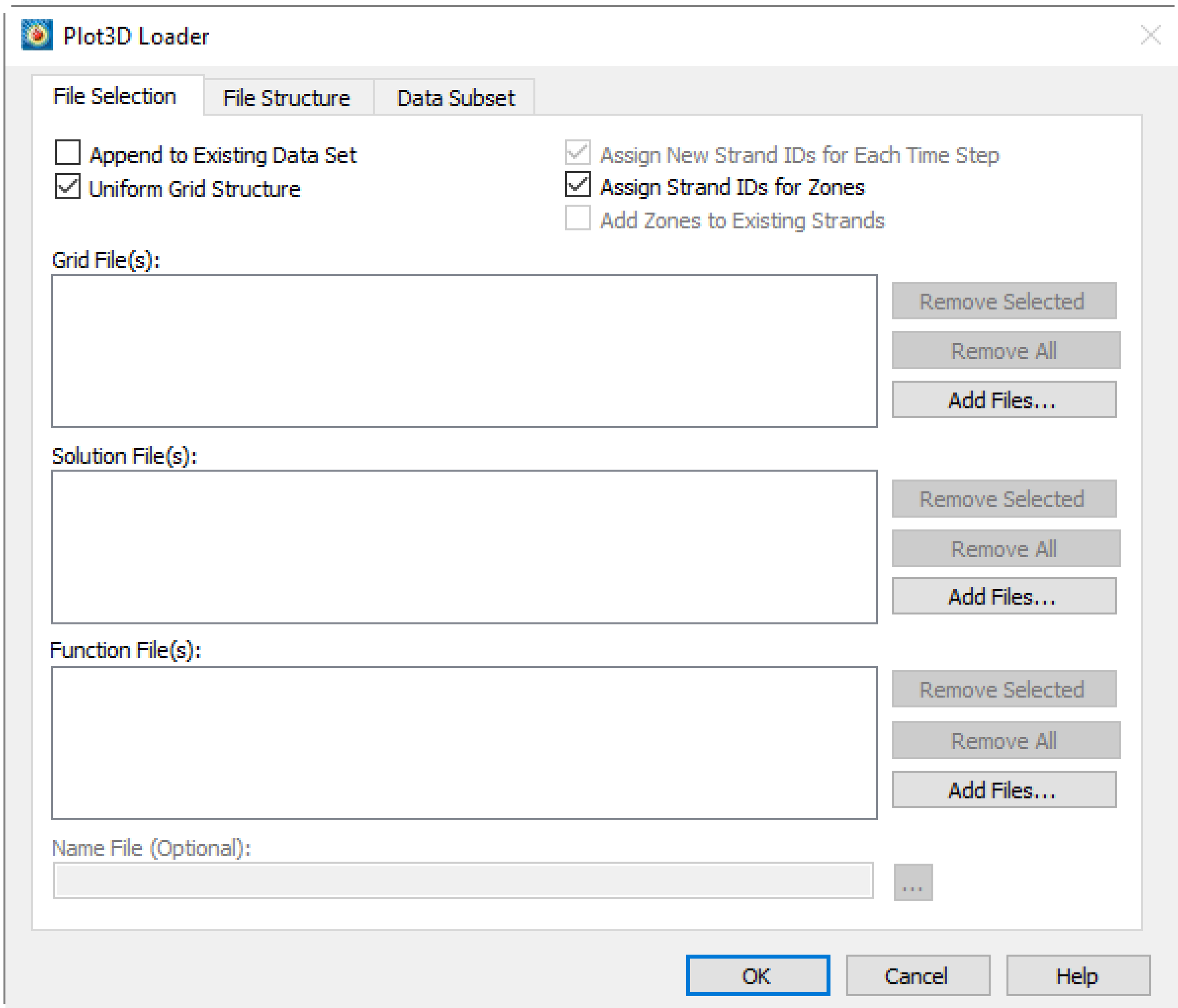
After clicking the **OK** button with one or more files selected, the **Select Variables** dialog appears. Clear All allows only variables X, Y, and Z to be loaded. You can use the **Select All** button to load all variables, or you can highlight variable names in the list.

PLOT3D Loader

The PLOT3D Loader add-on can import data files formatted in PLOT3D format, developed by Pieter Buning at the NASA Ames Research Center.

File Combinations

Use the File Selection page of the **PLOT3D Loader** dialog to specify whether to load just grid files, both the grid and solution files, or just solution files. When loading grid and solution files together, either one grid file must be loaded (in which case it is assumed that all solution files use this grid), or a number of grid files equal to the number of solution files must be loaded (in which case it is assumed that there is a grid file for each solution file). If multiple grid files are loaded, all must use the same organization (binary or ASCII, etc.). Only the first file is scanned for organization.



Choosing both solution (or function) and grid files will allow you to optionally specify a name file as well. The name file contains names to replace either the solution or function variable names on a 1-to-1 basis for as many names as are in either file.

If a boundary file exists, it must have a name of the form "`gridfilenamewithextension.fvbnd`" and be in FieldView 1.4 format to be automatically loaded.

You may append the files being loaded to the current data set by toggling-on the Append option. The files being appended should have the same number of zones as data that has already been loaded. If some zones are static and some are transient, load the static zones and then append the transient zones.

If you are loading more than one grid file and they all have the same structure, make sure the Uniform Grid Structure option is toggled-on for best performance. If you are unsure, toggle-off this option to make sure that the data can be loaded without errors.

This page also allows you to automatically assign Strand IDs for transient zones. An option to add zones to current strands can be used if you are appending additional time steps to existing transient data. Finally, you can have new strand IDs generated for each time step if you want to manage the style of each time step's data independently.

The Plot3D loader attempts to use numbers in filenames as solution times for transient data. Both integers and decimal fractional values are detected in filenames, and values may be negative. If multiple numeric fields are found in the data set's filenames, the one with a unique value in each filename being loaded is taken as the solution time; if multiple such fields are found, the rightmost one is used.

Some solvers that write data in PLOT3D format use the same solution time for all time steps. To accommodate such solvers, the PLOT3D loader will attempt to extract solution times from filenames whenever the solution time information contained in the actual data set is inconsistent or contradictory, even if the loader has not been explicitly instructed to do so.

The following table describes what the PLOT3D loader does in all six grid/solution scenarios:

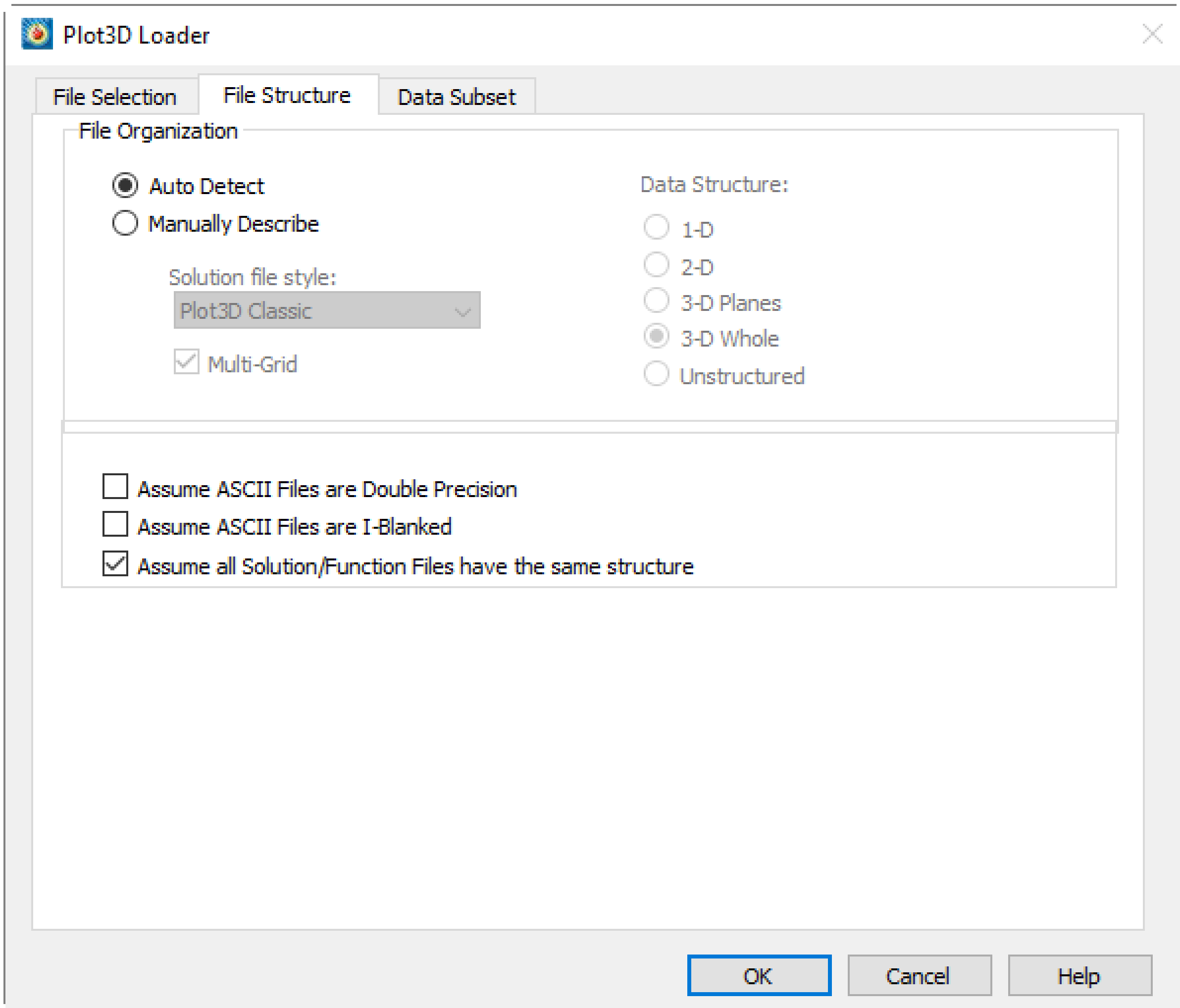
Load	**Not Appending**	**Appending**
Grid Only	Existing dataset is deleted and zones (one per grid) are loaded.	New zones are added (one per grid). Solution variables in new zones are zeroed out.
Grid and Solution	Existing dataset is deleted and zones (one for each grid in each solution file) are loaded. Each set of zones loaded shares spatial variables with the first set of grids loaded.	Same as "Not Appending" except original dataset is preserved. Existing dataset must have at least as many variables as the number needed by the incoming data.

Load	Not Appending	Appending
Solution Only	A dataset must already be present. The existing dataset is reduced to contain the same number of zones as there are grids in each incoming solution file. Solution variables in the first solution file replace the solution variables 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 dataset is preserved. Existing dataset must have at least as many variables as the number contained in incoming solution file. Spatial variables are shared with last *n* original zones where *n* is the number of grids in each incoming solution file.

```
  When loaded from the user interface, the loader attempts to extract solution times
from the filenames. If the function filenames have embedded numeric values that can be
interpreted as solution times, the function files will be treated as if they are
solution files, otherwise the variables in the specified function files are added to
the zones defined in the previously loaded solution files. When loaded from a macro,
this same treatment of function files is achieved by adding the
"EXTRACTTIMEFROMSOLFILENAMES" "YES" name/value pair to the loader instructions given to
the $!ReadDataSet command.
```

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 PLOT3D Loader can automatically 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. You may also specify the file format manually.

To enhance performance when loading multiple solution/function files, the primary solution/function file can be chosen to represent the structure of all subsequent files. To activate this option, toggle-on "Assume all Solution/Function Files have the same structure". You must determine if this is appropriate.

Special File Conditions

Unstructured Data Files

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.

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

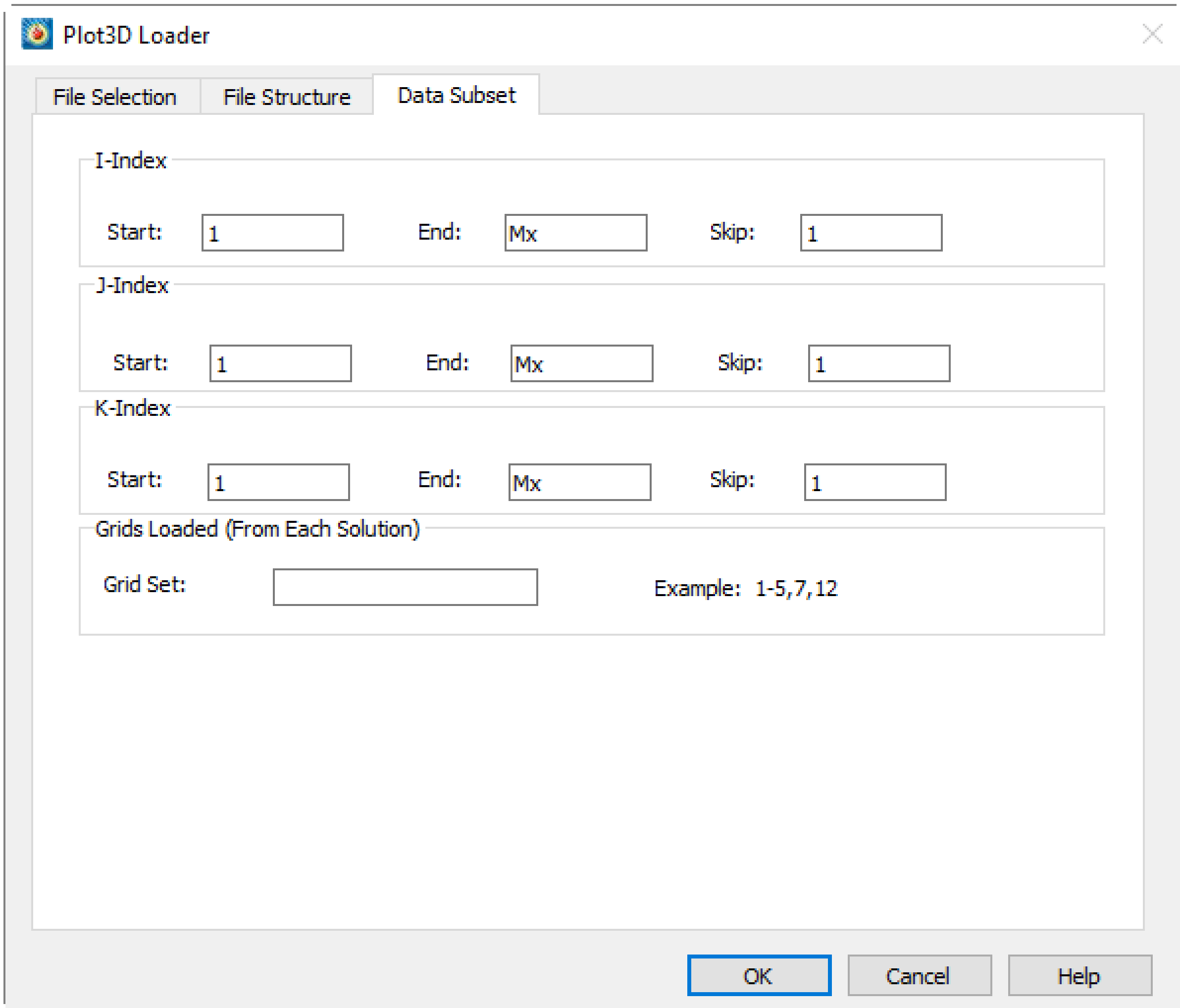
Pure Binary Files

The following pure binary files (binary files without record markers) require special attention:

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

PLOT3D Data Subsets

The Data Subset page of the PLOT3D Loader allows you to: read subsets of ordered zones within the files, specify the desired beginning and ending index values to read, and enter a skip value for each index direction. A skip value of one results in every value in the specified index range being read. A skip of 2 reads every second value, and so on.



Macro Language Commands for the Plot3D Loader

The syntax for loading PLOT3D data files with the Tecplot macro language is as follows:

```
$!READDATASET
' "STANDARDSYNTAX" "1.0"
"...any of the name value pairs in the following table..." '
DATASETREADER = 'PLOT3D Loader'
```

Each name/value pair should be in double quotes:

Keyword	**Value(s)**	**Default**	**Notes**
STANDARDSYNTAX	1.0	n/a	Must be the first instruction.
FILELIST_FUNCTIONFILES	"n" "file-1" "file-2"… . "file-n"	n/a	Specify the number of function files, followed by each file name.
FILELIST_SOLUTIONFILES	"n" "file-1" "file-2"… . "file-n"	n/a	Specify the number of solution files, followed by each file name.

Keyword	Value(s)	Default	Notes
FILELIST_GRIDFILES	"n" "file-1" "file-2"... . "file-n"	n/a	Specify the number of grid files, followed by each file name.
FILENAME_NAMEFILE	"filename"	n/a	The associated name file.
IINDEXRANGE	"Min, Max, Skip"	all	The index ranges for the data to be loaded, minimum, maximum, and skip value.
JINDEXRANGE	"Min, Max, Skip"	all	
KINDEXRANGE	"Min, Max, Skip"	all	
APPEND	"Yes" or "No"	"No"	Whether to append the read data to the existing dataset.
ASCIIISDOUBLE	"Yes" or "No"	"No"	Whether the ASCII data format is double.
ASCIIHASIBLANK	"Yes" or "No"	"No"	Whether the ASCII file contains blanks.
AUTODETECT	"Yes" or "No"		Whether to autodetect the PLOT3D file format variant. If set to "No," the DATASTRUCTURE, ISMULTIGRID, and STYLE keywords are used to specify the file format.
DATASTRUCTURE	"1D", "2D", "3DP", "3DW", or "UNSTRUCTURED"	none	Required if `AUTODETECT` is "No," otherwise ignored.
ISMULTIGRID	"Yes" or "No"	none	Required if `AUTODETECT` is "No," otherwise ignored.
STYLE	"PLOT3DCLASSIC" or "PLOT3DFUNCTION" or "OVERFLOW" or "SWIFT"	none	Required if `AUTODETECT` is "No," otherwise ignored.
ASSIGNSTRANDIDS	"Yes" or "No"	"No"	Whether to automatically assign strand IDs.
ADDTOEXISTINGSTRANDS	"Yes" or "No"	"No"	Whether to add data to existing strands.
UNIFORMGRIDSTRUCTURE	"Yes" or "No"	"Yes"	Whether the grid structure is the same for all time steps.
SOLUTIONSSHARESTRUCTURE	"Yes" or "No"	"No"	When loading multiple files whether the primary solution file represents the structure of all subsequent files. Will improve performance.

Keyword	Value(s)	Default	Notes
EXTRACTTIMEFROMSOLFILE NAMES	"Yes" or "No"	"No"	If set to YES for a grid file and one or more solution files or one or more function files without solution files, and the solution or function filenames have embedded numeric values that can be interpreted as solution times, the loader extracts the embedded solution time from the filenames for use as the solution times. Additionally, function files are treated as transient data instead of adding the function file variables to the existing zones. For solution files only, if the option is set to NO, or not part of the loader command, then auto extraction takes place, detecting if there are duplicated solution times, and if duplicates are found, extracting the solution time from the filenames. For function files only, if the option is set to NO, not part of the loader command, or the solution times could not be extracted from the filenames, the function files are not treated as transient data and the variables are added to the existing zones.

PLOT3D Auxiliary Data

The following auxiliary data is created by the PLOT3D Loader:

Auxiliary Name	Assigned To
Common.ReferenceMachNumber	Dataset and Individual Zones[2]
Common.AngleOfAttack	Dataset and Individual Zones[2]
Common.ReynoldsNumber	Dataset and Individual Zones[2]
Common.IsBoundaryZone	Individual Zones
Common.BoundaryCondition	Individual Zones
Common.DensityVar	Dataset
Common.UVar	Dataset
Common.VVar	Dataset

Auxiliary Name	Assigned To
Common.WVar	Dataset
Common.StagnationEnergyVar	Dataset
Common.GammaVar	Dataset
Common.TurbulentKineticEnergyVar	Dataset
Common.TurbulentDissipationRateVar	Dataset
Common.VectorVarsAreVelocity	Dataset
Common.SpeedOfSound	Dataset
G[3]	Individual Zones
B[3]	Individual Zones
T[3]	Individual Zones
I[3]	Individual Zones
H [3]	Individual Zones
H1[3]	Individual Zones
H2[3]	Individual Zones

PLY Loader

Use this loader[4] to load 3D triangular surface files with the `.ply` extension. This format is often used to store surfaces generated from tessellation of 3D range measurement data. Files may be either ASCII or binary, but must contain both vertex and face elements (sections). This loader is included in your Tecplot 360 installation.

PVD Loader

The PVD Data Loader loads VTK (Visualization Toolkit) files which have the filename extension .pvd. The PVD format is essentially a header file that collects together other VTK XML-based data files, such as VTI, VTP, VTR, VTS, VTU, etc.

Opening a single PVD file loads all supported files embedded within it. All zones created from a PVD file will be assigned the same strand ID.

The PVD loader does not have the advanced options dialog but supports advanced options via the macro language by augmenting the PVD Data Loader instructions in the `$!READDATASET` command of a layout or macro file. Multiple PVD files can loaded at once.

The PVD Data Loader will read the `<FieldData>` section of the files and add any scalars it finds to the zone's auxiliary data. The loader can be instructed to use time scalar values for zone solution times if all of the files have a scalar named "time" or "timevalue". The PVD loader can optionally detect a consistent pattern of incrementing numeric values in the names of the files themselves and use them for the zone's solution time. If the loader cannot follow the instruction it is given, because not all of the files have the "time" scalar or because there is no consistent pattern of values in the filenames, it will not assign a solution time or strand ID.

To tell the loader how to assign solution times, add the name/value pair `SOLUTIONTIMESOURCE <setting>` to the loader instructions. The settings are listed below.

Macro Language Commands for the PVD Data Loader

The syntax for loading PVD data files with the Tecplot macro language is through the `$READDATASET` command as follows:

```
$!READDATASET
' "STANDARDSYNTAX" "1.0" "...one or more PVD Data Loader instructions..."'
DATASETREADER = 'PVD Data Loader'
```

The PVD DATA Loader instructions are in name/value pairs and each name/value pair should be independently surrounded by double quotes. Refer to the Scripting Guide for details on working with the Tecplot macro language.:

Keyword	Value(s)	Default	Notes
STANDARDSYNTAX	"1.0"	n/a	Must be the first instruction.
FILENAME_FILE	"filename"	n/a	Specify a single file name.
FILELIST_DATAFILES	"n" "file-1" "file-2"… . "file-n"	n/a	Specify the number of solution files, followed by each file name.
SOLUTIONTIMESOURCE	"FromFieldData", "FromFilename", "Auto", "None"	"Auto"	"Auto": Favors the "time" scalar for the solution time over the numbers embedded in file names, "None": Does not assign solution time or Strand ID, "FromFieldData": Uses the "time" scalar for solution time, "FromFileName": Uses the solution time embedded in the file names.

Tecplot Data Loader

This section describes the process for loading Tecplot-format data files with the extensions `.dat` for ASCII files and `.plt` for binary files. Some other products that write Tecplot binary files use the filename extension `.bin` or `.tec`, and these are also recognized by the Tecplot-Format loader. (Refer to the Data Format Guide for information on outputting data into Tecplot 360 file format.)

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

Generate a Tecplot-format ASCII data file

Read the file into Tecplot 360 and work without conversion. If the dataset is altered, save it as an ASCII data file. This method works well for smaller datasets where the convenience of an ASCII file outweighs any inefficiencies.

Generate a Tecplot-format ASCII data file and convert it to a binary file with Tecplot 360

Read it into Tecplot 360, 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 datasets where ASCII inefficiencies are noticeable. See Data File Writing.

Generate a Tecplot-format ASCII data file, then convert it to a binary file with Preplot

Preplot, a utility program included with Tecplot 360, converts ASCII to binary Tecplot-format 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 the Data Format Guide for a description of Preplot.)

Generate a Tecplot-format binary data file

Read the binary data file into Tecplot 360 and work without conversion. You must use routines provided by Tecplot as part of the TecIO library to write Tecplot-format binary files from C or FORTRAN programs. (See the Data Format Guide for complete details.)

See also Tecplot Subzone Loader for information on loading the newer subzone file format (`.szplt`), which is optimized for working interactively with larger files. It is straightforward for developers to upgrade software that writes Tecplot binary files using the TecIO library to write subzone files.

Tecplot Data File Loading

The Tecplot Data Loader (accessed via **File→Load Data**) allows you to load ASCII (`.dat`) and binary (`.plt`) Tecplot-format data files. For both kinds of files, Tecplot 360 supports full data files, grid files, and solution files, where the file types are defined as follows:

Full

Full files contain both grid and solution data. Data files produced for Tecplot products versioned 2006 and earlier are treated as full data files. Full files can be loaded in any order.

Grid

Grid files contain static data for all zones. They have at least one variable or FE connectivity; they may contain both variables and connectivity simultaneously.

Solution

Solution files contain time-varying data for all zones in the file.

You may load more than one Tecplot format file at a time by selecting multiple files in the Load Data dialog. In fact, this is required when loading solution data because the corresponding grid file must be loaded at the same time.

Loading Grid and Solution Data Files

When you are loading grid and/or solution files, please keep the following in mind:

- Each solution file must be read in *after* the grid file it is associated with has been loaded.
- If you load multiple grid and solution files into Tecplot 360, the order the files are listed will be used to determine which grid is used for which solution file(s). For example, if you load a set of grid and solution files in the following order:
  - Grid A
  - Solution A
  - Solution B
  - Solution C
  - Grid B
  - Solution D

Grid A will be shared for Solution Files A, B, and C. Grid B will be used for Solution File D.

> You may select multiple files to load by holding the Shift key. However, the order is not always preserved with this method. We strongly recommend that you use the **Add to List** button when loading grid and solution data to make sure the files are added in the order you require.

- You may load a grid file with variables or variables and connectivity without loading a solution file. However, you may not load a grid file that contains only FE connectivity data.

Tecplot Subzone Loader

The Subzone Loader (SZL or "Sizzle") allows you to visualize large data files on typical engineering workstations by loading only the data required for a particular operation. Unlike the previous load-on-demand capability, Subzone Load can load partial zones instead of entire zones. This capability requires a new file format, `.szplt`, which can be loaded using the Tecplot Subzone Loader.

Data can be converted to the `.szplt` format manually, by loading it into Tecplot 360 and then writing it in SZL format (see Data File Writing). For information on batch-converting files to SZL format, see Batch Converting to SZL Format.

The Tecplot Subzone Loader is used for loading `.szplt` files stored on your local disk or on a standard network share (that is, a share that is supported by your computer’s operating system). To load files from a Linux remote host on which Tecplot SZL Server is installed, see Loading Remote Data using

Tecplot SZL Server.

Macro Commands for the Tecplot Subzone Loader

The syntax for loading Tecplot Subzone data files with the Tecplot macro language is as follows:

```
$!READDATASET
' "STANDARDSYNTAX" "1.0"
"...any of the name value pairs in the following table..." '
DATASETREADER = 'Tecplot Subzone Data Loader'
```

Each name/value pair should be in double quotes. Refer to the Scripting Guide for details on working with the Tecplot macro language.

Keyword	Available Value(s)	Default	Notes
STANDARDSYNTAX	"1.0"	n/a	Must be the first instruction.
FILENAME_FILE	"filename"	n/a	Specify a single file name.
FILELIST_DATAFILES	"n" "file-1" "file-2"... . "file-n"	n/a	Specify the number of files, followed by each file name.

Tecplot Layout Loader

The Tecplot Layout loader is used for loading Tecplot layouts (`.lay`) and layout packages (`.lpk`). Layouts include only links to the original data files; layout packages include the subset of data needed to view the layout. See Layout Files, Layout Package Files, Stylesheets for more details on creating these files.

Usually it is most convenient to load these kinds of files using the Welcome screen, the **File→Open Layout** or **File→Recent Layouts** commands, or double-clicking the file in your file manager (e.g. Windows Explorer). However, you can also load layouts and layout packages by choosing **File→Load Data** and selecting this loader in the Load Data dialog.

This loader has no options; simply choose the desired `.lay` or `.lpk` file, and it will be opened.

Telemac Loader

The Telemac Data Loader reads Telemac `SERAFIN`- and `SERAFIND`-formatted (.slf, .srf, .sel or .res) and boundary (.cli or .conlim) files. You may select multiple `SERAFIN` and `SERAFIND` files, but only a single boundary file for a single load.

To load a boundary file, you must either load it along with one or more `SERAFIN[D]` files, or else append it to an existing Telemac data set. It does not include its own X or Y variables, but rather contains references to node numbers in a corresponding `SERAFIN[D]` file. It is loaded as a single I-ordered zone. If loaded along with one or more `SERAFIN[D]` files, the boundary zone's X and Y values will be extracted from the first zone of the first `SERAFIN[D]` file. If appended to existing data, the boundary zone's X and Y values will be extracted from the first zone of the data set.

Telemac solution (SERAFIN) files may be successfully interactively loaded with the Telemac loader even if the file suffix (.slf, .srf, .sel or .res) has been stripped off. However, boundary files must have extension (.cli or .conlim) in order to load.

SERAFIN[D] files contain variable names, which are used for the resulting data set variables. Boundary files contain no variable names, and add the following variables (unless noted below, refer to Telemac documentation for their meaning):

BC-h

The boundary condition code for depth (LIHBOR)

BC-u

The boundary condition code for u (LIUBOR)

BC-v

The boundary condition code for v (LIVBOR)

BC-Qs

The boundary condition code for tracer (LITBOR)

BC-open

Indicator for closed or open boundary-0 for closed, 1 for open.

Macro Commands for the Telemac Data Loader

The syntax for loading Telemac data files with the Tecplot macro language is through the $READDATASET command as follows:

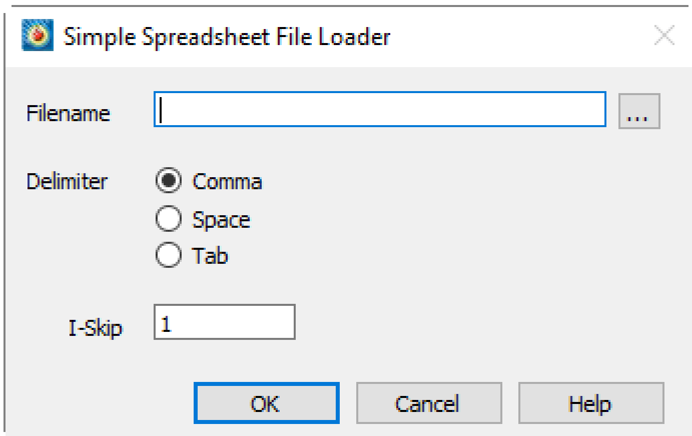
```
$!READDATASET
' "STANDARDSYNTAX" "1.0" "...one or more Telemac Data Loader
instructions..."'
DATASETREADER = 'Telemac Data Loader'
```

Keyword	Value(s)	Default	Notes
STANDARDSYNTAX	"1.0"	n/a	Must be the first instruction.
FILELIST_TELEMAC	"n" "file-1" "file-2"... . "file-n"	None	Specify the number of Telemac SERAFIN and SERAFIND files to load, followed by each file name.

Keyword	Value(s)	Default	Notes
FILENAME_BOUNDARY	"filename"	None	Specify a boundary file to load. Must either be appened to existing Telemac data or loaded along with one or more `SERAFIN[D]` files.

Text Spreadsheet Loader

The Text Spreadsheet Loader add-on allows you import simple data from ASCII files. Select the delimiter and I-skip (if necessary) from the **Simple Spreadsheet File Loader** dialog.



the **Text Spreadsheet Loader** dialog has the following options:

Filename

Enter the path to the file you would like to load.

Delimiter

Choose whether your data is separated by a comma, space, or tab.

I-Skip

Select an I-skip value. A value of 1 loads all values, a value of 2 loads every other value, and so on.

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
```

Text Spreadsheet Loader Limitation

The first line must contain all of the variable names. See How to Create a Plot

TRIX Loader

The TRIX Data Loader loads TRIX files, which have the filename extension `.trix`. TRIX is an XML-based unstructured file format used by Cart3D and related solvers and is a subset of the Visualization Toolkit VTU format. Only triangle surface data can be loaded. The TRIX loader has no advanced options and can load only one file at a time.

VTI Loader

The VTI Data Loader loads structured data VTK (Visualization Toolkit) files which have the filename extension .vti. The VTI format is an XML-based format representing a topologically and geometrically regular data. Examples include volumes (voxel data) and pixmaps. The representation supports images up to three dimensions.

The VTI loader does not have the advanced options dialog but supports advanced options via the macro language by augmenting the VTI Data Loader instructions in the `$!READDATASET` command of a layout or macro file. Multiple VTI files can loaded at once.

The VTI Data Loader will read the `<FieldData>` section of the files and add any scalars it finds to the zone's auxiliary data. The loader can be instructed to use time scalar values for zone solution times if all of the files have a scalar named "time" or "timevalue". The VTI loader can optionally detect a consistent pattern of incrementing numeric values in the names of the files themselves and use them for the zone's solution time. If the loader cannot follow the instruction it is given, because not all of the files have the "time" scalar or because there is no consistent pattern of values in the filenames, it will not assign a solution time or strand ID.

To tell the loader how to assign solution times, add the name/value pair `SOLUTIONTIMESOURCE <setting>`

to the loader instructions. The settings are listed below.

Macro Language Commands for the VTI Data Loader

The syntax for loading VTI data files with the Tecplot macro language is through the `$READDATASET` command as follows:

```
$!READDATASET
' "STANDARDSYNTAX" "1.0" "...one or more VTI Data Loader instructions..."'
DATASETREADER = 'VTI Data Loader'
```

The VTI DATA Loader instructions are in name/value pairs and each name/value pair should be independently surrounded by double quotes. Refer to the Scripting Guide for details on working with the Tecplot macro language.:

Keyword	Value(s)	Default	Notes
STANDARDSYNTAX	"1.0"	n/a	Must be the first instruction.
FILENAME_FILE	"filename"	n/a	Specify a single file name.
FILELIST_DATAFILES	"n" "file-1" "file-2"... . "file-n"	n/a	Specify the number of solution files, followed by each file name.
SOLUTIONTIMESOURCE	"FromFieldData", "FromFilename", "Auto", "None"	"Auto"	"Auto": Favors the "time" scalar for the solution time over the numbers embedded in file names, "None": Does not assign solution time or Strand ID, "FromFieldData": Uses the "time" scalar for solution time, "FromFileName": Uses the solution time embedded in the file names.

VTK Loader

The VTK Data Loader loads unstructured VTK (Visualization Toolkit) files which have the filename extension .vtu. The VTU format is an XML-based format used by solvers. The loader is designed to work with ASCII, inline-binary, and appended formats with "raw" or "base64" encoding and either big or little-endian byte ordering, with or without ZLib compression. The supported VTK cell types are Vertex, Line, Triangle, Pixel, Quad, Tetra, Voxel, Hexahedron, Wedge, and Pyramid. These types are loaded into Tecplot 360 as ordered-point, FE-Lineseg, FE-Triangle, FE-Quad, FE-Tetra, and FE-Brick data. The VTK loader has no advanced options dialog but does provide advanced options that can be set via the macro language by augmenting the VTK Data Loader instructions in the `$!READDATASET` command of a layout or macro file.

Multiple VTU files can loaded at one time or appended to an existing dataset. Multiple VTU files loaded at one time must all have the same number of variables (VTK DataArray elements) and the same variable names in the same order.

The VTK Data Loader can also load .pvtu and .vtm files. A .pvtu file is a parallel VTU file that references one or more VTK data files and assembles them together to form a single zone, one for each .pvtu file. A .vtm file is a multi-block file that references one or more VTK data files and assembles them into multiple zones, one for each data file referenced.

The VTK Data Loader will read the <FieldData> section of the files and add any scalars it finds there to the zone's auxiliary data. If all of the files have a scalar named "time" or "timevalue" in the <FieldData> section, the loader can use the "time" scalar variable for the zone's solution time. If the loader is instructed to use the "time" scalar for solution time, it will not add the "time" scalar to the zone's auxiliary data. The VTK loader can optionally detect a consistent pattern of incrementing numeric values in the names of the files themselves and use them for the zone's solution time. If the loader cannot follow the instruction it is given, because not all of the files have the "time" scalar or because there is no consistent pattern of values in the filenames, it will not assign a solution time or strand ID.

To tell the loader how to assign solution times, add the name/value pair SOLUTIONTIMESOURCE <setting> to the loader instructions. The settings are listed below.

Macro Language Commands for the VTK Data Loader

The syntax for loading VTU data files with the Tecplot macro language is through the $READDATASET command as follows:

```
$!READDATASET
' "STANDARDSYNTAX" "1.0" "...one or more VTK Data Loader instructions..."'
DATASETREADER = 'VTK Data Loader'
```

The VTK DATA Loader instructions are in name/value pairs and each name/value pair should be independently surrounded by double quotes. Refer to the Scripting Guide for details on working with the Tecplot macro language.:

Keyword	**Value(s)**	**Default**	**Notes**
STANDARDSYNTAX	"1.0"	n/a	Must be the first instruction.
FILENAME_FILE	"filename"	n/a	Specify a single file name.
FILELIST_DATAFILES	"n" "file-1" "file-2"... . "file-n"	n/a	Specify the number of solution files, followed by each file name.

Keyword	Value(s)	Default	Notes
SOLUTIONTIMESOURCE	"FromFieldData", "FromFilename", "Auto", "None"	"Auto"	"Auto": Favors the "time" scalar for the solution time over the numbers embedded in file names, "None": Does not assign solution time or Strand ID, "FromFieldData": Uses the "time" scalar for solution time, "FromFileName": Uses the solution time embedded in the file names.

VTP Loader

The VTP Data Loader loads unstructured data VTK (Visualization Toolkit) files which have the filename extension .vtp. The VTP format is an XML-based format representing a geometric structure consisting of vertices, lines, polygons, and/or triangle strips. Point and cell attribute values (e.g., scalars, vectors, etc.) also are represented.

> Only particle and polygonal (VTK_POLYGON) data is currently supported by the VTP loader.

The VTP loader does not have the advanced options dialog but supports advanced options via the macro language by augmenting the VTP Data Loader instructions in the `$!READDATASET` command of a layout or macro file. Multiple VTP files can loaded at once.

The VTP Data Loader will read the `<FieldData>` section of the files and add any scalars it finds to the zone's auxiliary data. The loader can be instructed to use time scalar values for zone solution times if all of the files have a scalar named "time" or "timevalue". The VTP loader can optionally detect a consistent pattern of incrementing numeric values in the names of the files themselves and use them for the zone's solution time. If the loader cannot follow the instruction it is given, because not all of the files have the "time" scalar or because there is no consistent pattern of values in the filenames, it will not assign a solution time or strand ID.

To tell the loader how to assign solution times, add the name/value pair `SOLUTIONTIMESOURCE <setting>` to the loader instructions. The settings are listed below.

Macro Language Commands for the VTP Data Loader

The syntax for loading VTP data files with the Tecplot macro language is through the `$READDATASET` command as follows:

```
$!READDATASET
```

```
' "STANDARDSYNTAX" "1.0" "...one or more VTP Data Loader instructions..."'
DATASETREADER = 'VTP Data Loader'
```

The VTP DATA Loader instructions are in name/value pairs and each name/value pair should be independently surrounded by double quotes. Refer to the Scripting Guide for details on working with the Tecplot macro language.:

Keyword	**Value(s)**	**Default**	**Notes**
STANDARDSYNTAX	"1.0"	n/a	Must be the first instruction.
FILENAME_FILE	"filename"	n/a	Specify a single file name.
FILELIST_DATAFILES	"n" "file-1" "file-2"… . "file-n"	n/a	Specify the number of solution files, followed by each file name.
SOLUTIONTIMESOURCE	"FromFieldData", "FromFilename", "Auto", "None"	"Auto"	"Auto": Favors the "time" scalar for the solution time over the numbers embedded in file names, "None": Does not assign solution time or Strand ID, "FromFieldData": Uses the "time" scalar for solution time, "FromFileName": Uses the solution time embedded in the file names.

VTR Loader

The VTR Data Loader loads structured data VTK (Visualization Toolkit) files which have the filename extension .vtr. The VTR format is an XML-based format representing a rectilinear grid where the axes are parallel to the coordinate axes, and the cells are rectangular or rectangular cuboids (in 3D). Unlike a regular grid, e.g. VTI, the spacing between grid points can vary. The representation supports grids up to three dimensions.

The VTR loader does not have the advanced options dialog but supports advanced options via the macro language by augmenting the VTR Data Loader instructions in the `$!READDATASET` command of a layout or macro file. Multiple VTR files can loaded at once.

The VTR Data Loader will read the `<FieldData>` section of the files and add any scalars it finds to the zone’s auxiliary data. The loader can be instructed to use time scalar values for zone solution times if all of the files have a scalar named "time" or "timevalue". The VTR loader can optionally detect a consistent pattern of incrementing numeric values in the names of the files themselves and use them for the zone’s solution time. If any of the files are missing the "time" scalar or there is not a consistent pattern of values in the filenames, the loader will not assign a solution time or strand ID.

To tell the loader how to assign solution times, add the name/value pair `SOLUTIONTIMESOURCE <setting>` to the loader instructions. The settings are listed below.

Macro Language Commands for the VTR Data Loader

The syntax for loading VTR data files with the Tecplot macro language is through the `$READDATASET` command as follows:

```
$!READDATASET
' "STANDARDSYNTAX" "1.0" "...one or more VTR Data Loader instructions..."'
DATASETREADER = 'VTR Data Loader'
```

The VTR DATA Loader instructions are in name/value pairs and each name/value pair should be independently surrounded by double quotes. Refer to the Scripting Guide for details on working with the Tecplot macro language.:

Keyword	Value(s)	Default	Notes
STANDARDSYNTAX	"1.0"	n/a	Must be the first instruction.
FILENAME_FILE	"filename"	n/a	Specify a single file name.
FILELIST_DATAFILES	"n" "file-1" "file-2"… . "file-n"	n/a	Specify the number of solution files, followed by each file name.
SOLUTIONTIMESOURCE	"FromFieldData", "FromFilename", "Auto", "None"	"Auto"	"Auto": Favors the "time" scalar for the solution time over the numbers embedded in file names, "None": Does not assign solution time or Strand ID, "FromFieldData": Uses the "time" scalar for solution time, "FromFileName": Uses the solution time embedded in the file names.

VTS Loader

The VTS Data Loader loads structured data VTK (Visualization Toolkit) files which have the filename extension .vts. The VTS format is an XML-based format representing a geometric structure represented by a regularly indexed array of points. Unlike orthogonal grids, such as VTI or VTR, the VTS data format allows representation of geometries with arbitrary positions.

The VTS loader does not have the advanced options dialog but supports advanced options via the macro language by augmenting the VTS Data Loader instructions in the `$!READDATASET` command of a layout or macro file. Multiple VTS files can loaded at once.

The VTS Data Loader will read the `<FieldData>` section of the files and add any scalars it finds to the zone's auxiliary data. The loader can be instructed to use time scalar values for zone solution times if all of the files have a scalar named "time" or "timevalue". The VTS loader can optionally detect a consistent pattern of incrementing numeric values in the names of the files themselves and use them for the zone's solution time. If the loader cannot follow the instruction it is given, because not all of the files have the "time" scalar or because there is no consistent pattern of values in the filenames, it will not assign a solution time or strand ID.

To tell the loader how to assign solution times, add the name/value pair `SOLUTIONTIMESOURCE <setting>` to the loader instructions. The settings are listed below.

Macro Language Commands for the VTS Data Loader

The syntax for loading VTS data files with the Tecplot macro language is through the `$READDATASET` command as follows:

```
$!READDATASET
' "STANDARDSYNTAX" "1.0" "...one or more VTS Data Loader instructions..."'
DATASETREADER = 'VTS Data Loader'
```

The VTS DATA Loader instructions are in name/value pairs and each name/value pair should be independently surrounded by double quotes. Refer to the Scripting Guide for details on working with the Tecplot macro language.:

Keyword	**Value(s)**	**Default**	**Notes**
STANDARDSYNTAX	"1.0"	n/a	Must be the first instruction.
FILENAME_FILE	"filename"	n/a	Specify a single file name.
FILELIST_DATAFILES	"n" "file-1" "file-2"… . "file-n"	n/a	Specify the number of solution files, followed by each file name.
SOLUTIONTIMESOURCE	"FromFieldData", "FromFilename", "Auto", "None"	"Auto"	"Auto": Favors the "time" scalar for the solution time over the numbers embedded in file names, "None": Does not assign solution time or Strand ID, "FromFieldData": Uses the "time" scalar for solution time, "FromFileName": Uses the solution time embedded in the file names.

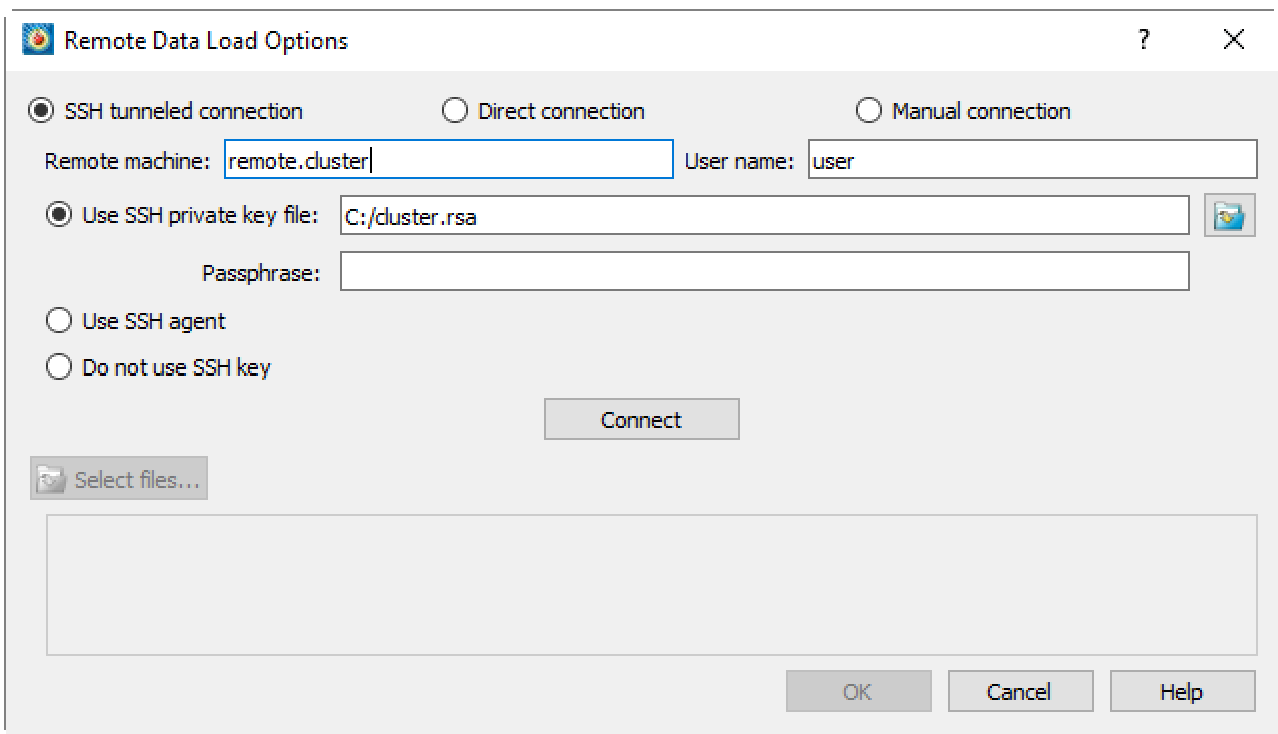
Loading Remote Data using Tecplot SZL Server

Tecplot SZL Server is a lightweight server that you can install on a remote Linux host (such as a computer cluster) to get access to your data when it is not practical to move the data off the cluster to a local drive or to a network file share, and when you cannot reasonably run Tecplot 360 on the remote host using a remote desktop setup. Usually this is due to slower network speeds between the remote host and the visualization workstation.

On some linux machines, due to differences in OpenSSL libraries, the Load Remote Data (SZL Client) add-on may not load by default, and the Load Remote Data option will not appear in the File menu. If you observe this, try using the --use-openssl command line argument when starting Tecplot Tecplot 360. If this is not successful, please contact Tecplot Support for further assistance.

Tecplot SZL Server runs only on Linux hosts and serves only SZL data, since it is the subzone loading capability of SZL format that makes it practical to visualize your data over slower connections. Clients can be any platform supported by Tecplot 360.

You can load SZL data from a remote host using Tecplot SZL Server using the **Load Remote Data** command on the Tecplot 360 **File** menu. Choose **Load Remote Data** command on the Tecplot 360 **File** menu to open the Remote Data Load Options dialog.



If the Load Remote Data option is not in the File menu you may need to install libssh2 (Linux only).

The first time you use Load Remote Data on Windows, Windows may display a dialog

> asking you to allow Tecplot 360 to listen for connections. You must grant this permission to allow Tecplot 360 to load remote data.

The Remote Data Load Options dialog offers three ways to connect to the remote SZL Server:

- **SSH Tunneled Connection** - Uses a secure shell (SSH) connection to carry the data from the server to the workstation so it can be visualized in Tecplot 360. As long as you can connect to the remote host via SSH, you can visualize data from a SZL Server running on that host; the SSH connection is all that is needed. This minimizes any possible network and workstation complications.
- **Direct Connection** - May offer better performance than a SSH tunnel, but with less security, because the data is not encrypted. (A SSH connection still must be established to start the server, but the actual data is sent to the workstation without encryption.) The remote host must be able to establish a connection to arbitrary ports on the visualization workstation, which may require setup from a network administrator or may even be disallowed entirely at your site.

  You might use Direct Connection mode if your network is already secure (for example, when using a VPN or WAN) and the encryption provided by SSH would be redundant, or if performance is paramount.
- **Manual Connection** - This mode allows you to establish a SZL Server session by manually issuing a command on the remote host. Each time you establish a manual connection, Tecplot 360 provides you with an appropriate command that can be used for this purpose.

  By default, SZL Server connection is made directly from the remote host to the workstation as in Direct Connection mode described above. However, you can also establish an SSH tunnel manually and run your SZL Server session through it. See Manual Connection Mode for details.

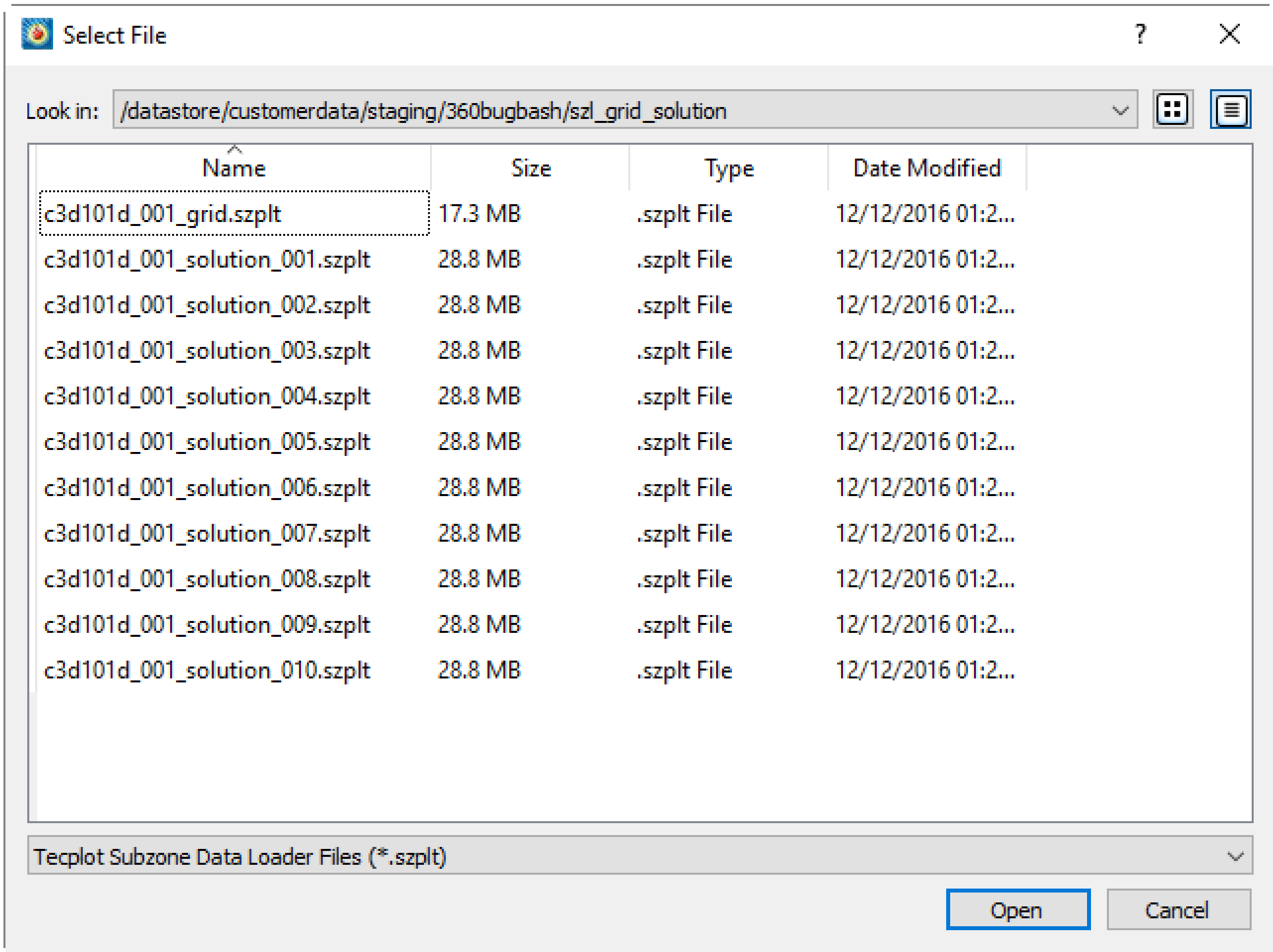
If you are using either of the first two modes, choose it in the dialog, then specify:

- The hostname or IP address of the remote host
- Your username on that host
- How you wish Tecplot 360 to authenticate to the remote host, using one of these options:
  - Your private key file (specify the file by entering the path or by clicking "..."). If your key is protected by a passphrase, enter it in the field provided.
  - A key manager: ssh-agent (or PuTTY’s Pageant utility on Windows).
  - Without a key (you will be prompted for a password when you connect)

Then click the **Connect** button to establish the connection. This may take a few seconds.

> If you are using Manual Connection mode, you will instead see a small window indicating the command you must execute on the remote host to establish the connection. See Manual Connection Mode for details on using this mode.

Once the connection has been established, the **Select Files** button becomes enabled. Click **Select Files** to choose a file to open. The Select File dialog appears.

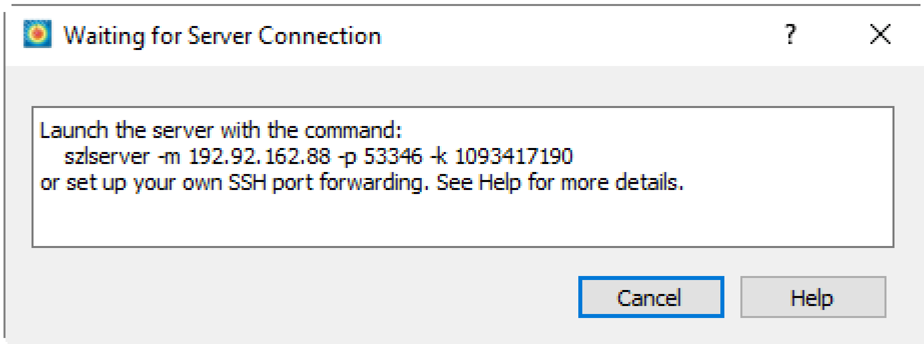


All remote files open at any one time must be opened from the same directory on the same remote host. You cannot append data using remote files.

After you choose the files you wish to open and click **Open**, you return to the Load Remote Data dialog with those files listed in the bottom panel. Click **OK** to proceed with opening the files. Tecplot 360 opens the files and displays the initial plot.

Manual Connection Mode

In Manual Connection mode, you establish an SSH connection to the remote host yourself, then start the server and tell it how to connect back to the Tecplot 360 client running on your workstation. This is useful for unusual network configurations and for those who require complete control.



When you click **Connect** in the Remote Data Load options with the mode set to Manual Connection, Tecplot 360 displays the Waiting For Server Connection dialog. This dialog remains open until you establish the connection from the remote host. To do this, you must issue a command on the host to start SZL Server and tell it how to connect to Tecplot 360 on the client workstation.

The `szlserver` command shown in the dialog can be copied and pasted into your remote command line session (established with the `ssh` command on Linux or Mac, or a GUI client like PuTTY on Windows). Usually, you will establish a new SSH connection, but it is possible to use an existing one (see the Note at the end of this section). For this example, we will establish a new connection.

When invoked using the provided `szlserver` command, SZL Server establishes a direct, unencrypted connection to the workstation (as in Direct Connection mode). This requires that arbitrary ports on your workstation be accessible from the server, which may require network administrator support or be disallowed entirely at your site.

If it is not possible for the server to connect to the workstation, or if you wish the session to be encrypted, you can manually tunnel the connection over an SSH connection. You will need to know the port number on which Tecplot 360 is waiting for the connection. This is the number following the `-p` flag in the `szlserver` command shown in the Waiting For Server Connection dialog.

You should note the session key, which is the number following `-k`. The session key is a random number that is changed for each connection; it is used to verify that the connection is being made to the correct workstation. When you issue the `szlserver` command on the remote host, you must use the session key displayed in the Waiting For Server Connection dialog.

In the example dialog shown here, Tecplot 360 is listening on port 49767. So you need to establish a tunnel from port 49767 on the remote host to port 49767 on your workstation.

> You may use a different port number on the remote host if the port you wish to use is already in use on that system, but this is quite rare. If there is a port conflict, it is probably best to close the Load Remote Data Options dialog and try again; Tecplot 360 chooses a different port each time you open the dialog. Alternatively, specify 0 for the remote port; `ssh` will choose an available port for you and tell you what it is.
>
> Valid port numbers range from 1024-65535. Ports with numbers less than 1024 can only be opened by the remote host's root user.

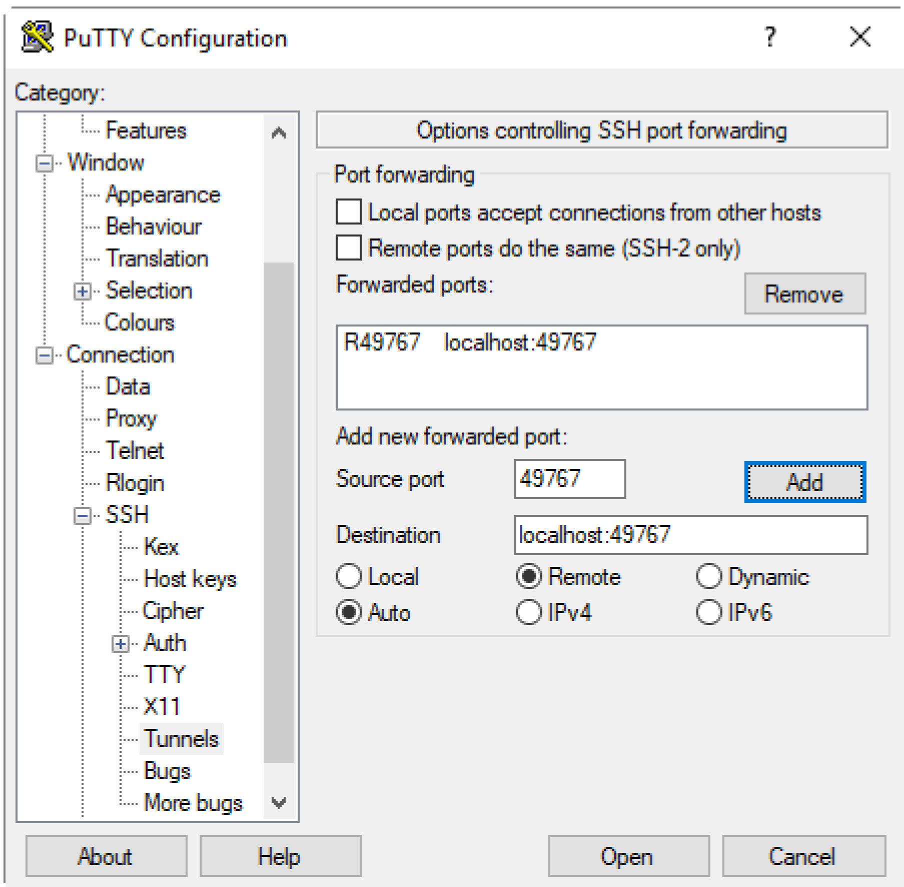
On Linux or Mac systems, you can create the tunnel using the `-R` option with the `ssh` command line client. In this command, `localhost` refers to the workstation running Tecplot 360, the client of the `ssh` connection (the hostname is resolved on the client end).

```
ssh -R 49767:localhost:49767 username@remote.host
```

If you are using a server port number different from the workstation port, the server port number should be the first number after the `-R`. For example, if you have decided to use port 4242 for the server end of the tunnel, the ssh command would be:

```
ssh -R 4242:localhost:49767 username@remote.host
```

On Windows, you can set up port forwarding in PuTTY on the **Connections** → **SSH** → **Tunnels** page.



Enter (in this example) `49767` for the source port, `localhost:49767` as the destination, and choose the Remote radio button. (If you are using a different port number on the server than on the workstation, it goes in the Source Port field.) Then click **Add** to add the tunnel to the forwarded ports list. When you click **Open** to establish the SSH connection, the tunnel is created along with it.

One you have established an SSH connection with the remote host, incorporating a tunnel, you can issue the `szlserver` command on the remote host. However, instead of specifying the address of the client workstation after `-m`, specify `localhost`. This will instruct SZL Server to connect to the host's end of the tunnel, which will forward the connection to the workstation.

```
szlserver -m localhost -p 49767 -k 1443240310
```

If you are using a different port number on the remote host, specify this after `-p`. For example, if you are using port 4242 for the remote end of the tunnel, use:

```
szlserver -m localhost -p 4242 -k 1443240310
```

> On Windows, PuTTY lets you add tunnels to an existing connection. Simply click the icon in your SSH terminal's upper left corner, choose **Change Settings** from the menu, and navigate to **Connection**→**SSH**→**Tunnels** as if you were setting up a new connection. From there, you can remove any obsolete port forwarding and add new tunnels as needed.
>
> On Linux and Mac, you can type `~C` to escape into a mini command line from an established session, then enter an -R option exactly as you would on the command line to set up a port forward (e.g. `-R 49767:localhost:49767`). To cancel an existing tunnel, enter `-KR 49767` (for example) using the remote port number of the tunnel. Press Enter to exit the mini command line and return to your SSH session.
>
> This feature may not be available depending on SSH client configuration. If it is not available, simply open a new terminal window and establish a new SSH connection incorporating the desired tunnelling.

Interactive Authentication and Unattended Operation

Depending on your organization's network and security policies, you might need to enter information to prove your identity when connecting to a SZL Server via SSH. This information might include:

- An account password, if you are not using a key pair
- A passphrase for a protected private key or key store
- A security code from an authentication token or app (such as Google Authenticator)

Or, if you are using Manual Connection mode, you must enter a command on the remote host in order to establish the connection.

Tecplot 360 prompts you to enter any necessary authentication information or take other action to establish the connection to SZL Server as necessary. When running Tecplot 360 interactively—with its full graphical user interface—a dialog appears to request your password or to ask you to perform other steps.

When you record a connection to a SZL Server in a macro, the most that is ever stored in the macro is the path to your private key file. The key itself and its passphrase are never stored; likewise, if you are using password authentication, the password is not stored. Therefore, for batch operations—such as running a macro from the command line—the Linux and Mac versions of Tecplot 360 will prompt you to enter this information in the terminal window if it is necessary.

Similarly, the macro file does not store any manual steps you do when establishing a connection via Manual mode, because these steps are done outside Tecplot 360. So when connecting via the Manual connection mode in batch mode, you will be instructed in the terminal to establish the connection, and Tecplot 360 will wait for the connection to be established.

The Windows version of Tecplot 360 detaches from the text console and cannot read or write to the command line window. Therefore, the `$!READDATASET` macro instruction fails in batch mode on Windows if any additional information or action is needed to connect to the remote host. (The file `batch.log` will include an error message in this case.) You must use Tecplot 360 in interactive mode to be able to open such data sets on Windows.

True unattended operation requires use of the SSH Tunneled or Direct Connection modes with SSH key pairs without passphrases or other interactive requirements. In this case, all platforms can access the data files in batch mode without difficulty.

Part 3: Plotting Data

Plot and Data Handling

How to Create a Plot

The basic steps for creating a plot in Tecplot 360 are:

1. Define your dataset using one of the following methods:
    1. Use the Load Data command from the **File** menu. Please refer to Loading Data for information on working with a specific data loader.
    2. Use the **Open Layout** command from the **File** menu to load linked layout or layout package files. (See Layout Files, Layout Package Files, Stylesheets for more information on layout files.)
    3. Use any combination of the options in the **Create Zone** submenu of the **Data** menu or the **Insert** menu to create your datasets directly within Tecplot 360. Zone Creation and Text, Geometries and Images for more information.
2. Select the Plot Type (3D, 2D, XY Line, Polar Line, or Sketch) from the Plot sidebar. Refer to Field Plots for information regarding 3D and 2D plots, XY and Polar Line Plots for information on XY Line and Polar Line plots, and Text, Geometries and Images for information on sketch plots.
3. Toggle-on any mapping or zone layers from the Plot sidebar. The following mapping and zone layers are available:
    - Mesh Layer
    - Contour Layer
    - Shade Layer
    - Vector Layer
    - Scatter Layer
    - Edge Layer
    - Map Layer
    - Symbols Map Layer
    - XY Line Error Bars
    - XY Line Bar Charts

The layers available in the Plot sidebar are dependent upon the active plot type.

In order to view surfaces on your plot, open the **Zone Style** dialog, select the **Surfaces tab**, and use the **Surfaces to Plot** drop-down menu to choose an available surface plotting option.

The **Zone Style** dialog is available by either **double-clicking** on your plot, selecting the **Zone Style** button from the Plot sidebar, or selecting **Plot→Zone Style** from the Menu bar.

4. Use the options in the **Plot** menu (such as **Blanking** or **Axis**) to customize how your data is displayed. Refer to Three-dimensional Plot Control or Axes for additional information.
5. Use the options in the **Data** menu (such as **Alter→Specify Equations** or **Interpolate**) to alter the dataset. Refer to Data Operations and CFD Analysis for additional information.
6. **[3D only]** Toggle-on zone effects (**Translucency** and **Lighting**). Refer to Translucency and Lighting for details.
7. Use the **Zone Style** or **Mapping Style** dialogs to opt zones in and out of plot layers or the entire plot. Refer to Field Plot Modification and the Zone Style Dialog and Mapping Style and Creation, respectively, for details.
8. **[2D or 3D only]** Add derived objects (Slices, Streamtraces or Iso-Surfaces). Use their respective buttons: to customize any derived objects.

You are not limited to working with only one plot at a time. You can create multiple plots on a single page using frames and frame linking, and you can create multiple pages. See Frames for more information.

Data Journaling

On occasion you may modify data prior to making a final plot. Some (but not all) of the data operations mentioned in this chapter modify data. Tecplot 360 "journals" the data modifications you make in the form of macro instructions as well as performing them. If you then save a layout file, the layout file can reference the original data and include the instructions necessary to reconstruct the plot. This way, the original data file is not modified. You can also generate a new plot using a different data file by reusing the layout. (Layout files are text files and may be modified in an editor.)

If you perform an operation that Tecplot 360 is unable to journal, you are prompted to save the dataset to a new file when you save a layout file. This is necessary for the layout to reproduce exactly what you have in your plot.

The following operations are included in the data journal:

- Data alteration except for derivatives in equations.
- Creation of rectangular zones, circular zones.
- Slice extraction of a primary slice through volume or surface zones for X, Y, Z, or arbitrary planes.
- Zone duplication and mirroring.

- Zone and variable deletion.
- Fourier transform.

Data Sharing

In order to conserve computer memory and disk space, Tecplot 360 shares variables between zones whenever possible. Variable sharing typically occurs with any of the following scenarios:

- When a variable is calculated for two or more zones, Tecplot 360 determines if the results will be the same in the different zones, and shares the variable where appropriate. See Equation Syntax and Variable Sharing Between Zones.
- When zones are duplicated, all variables are shared between the source zones and their duplicates. See Zone Duplication.
- When mirrored zones are created, the mirrored zone shares all variables with the original. See Mirror Zone Creation.
- When a data loader supporting data sharing loads a variable that is identified for two or more zones, that variable is shared among the specified zones. This often occurs with time dependent data, where the physical coordinates (e.g. X, Y, and Z) are typically the same for all time steps.

> If a zone is altered (independently of zones it is sharing data with), any variable that is changed will no longer be shared.

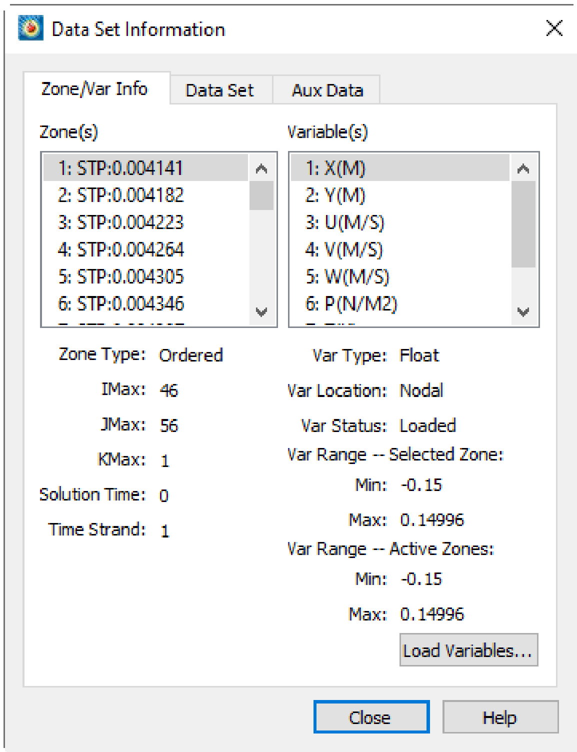
Variable sharing and connectivity sharing (for finite element zones) can also be established in a Tecplot data file by supplying the appropriate parameters to the `TECZNE` function when writing the file (see the Data Format Guide). In addition, multiple solution files can share the same grid file.

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 dataset, including the dataset title, zone and variable names, and the minimum and maximum values of a selected variable. You can modify the dataset title, zone, and variable names of any dataset.

Zone/Variable Info Page

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



Zone(s)

Lists all zones by number, with their titles.

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:

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.

Solution Time

Displays the solution time for the selected zone (see also Time Aware).

Time Strand

Displays the strand ID for the selected zone (see also Time Aware).

Variable(s)

Lists all variables by number, with their names.

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 Status

Use the Var Status field in the dialog to determine the status of the current variable. The variable status can indicate the variable passivity and if the variable is "loaded" or "not loaded."

Var Range - Selected Zones

Displays the Min and Max values for the selected variable in the selected zones. When working with subzone data, one or more of the values displayed may be estimates, which is indicated by `(est)` next to the affected values.

Var Range - Active Zones

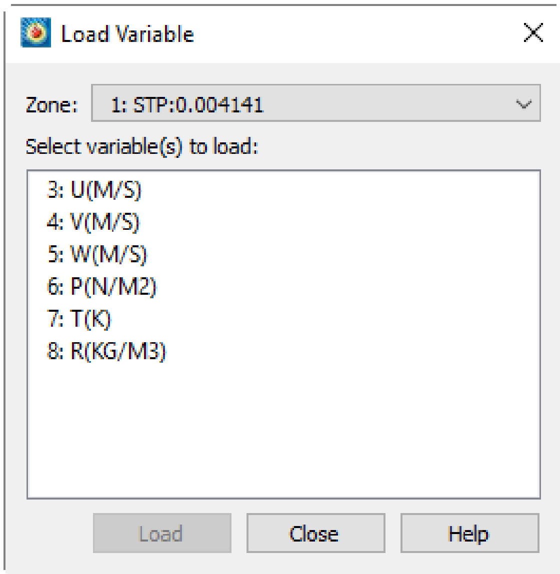
Displays the Min and Max values for the selected variable for all active zones. When working with subzone data, one or more of the values displayed may be estimates, which is indicated by `(est)` next to the affected values.

Load Variables

If a variable was not initially loaded, "Not Loaded" will be displayed in **Var Range** portions of the dialog. Use the **Load Variables** button to display the Load Variable dialog, with which you can load any variables from your dataset that were not initially loaded.

Load Variable

The Load Variable dialog allows you to load variables into memory if they are not currently loaded. Choose the zone containing the variables you wish to load using the Zone menu at the top of the dialog, then select one or more variables from that zone.

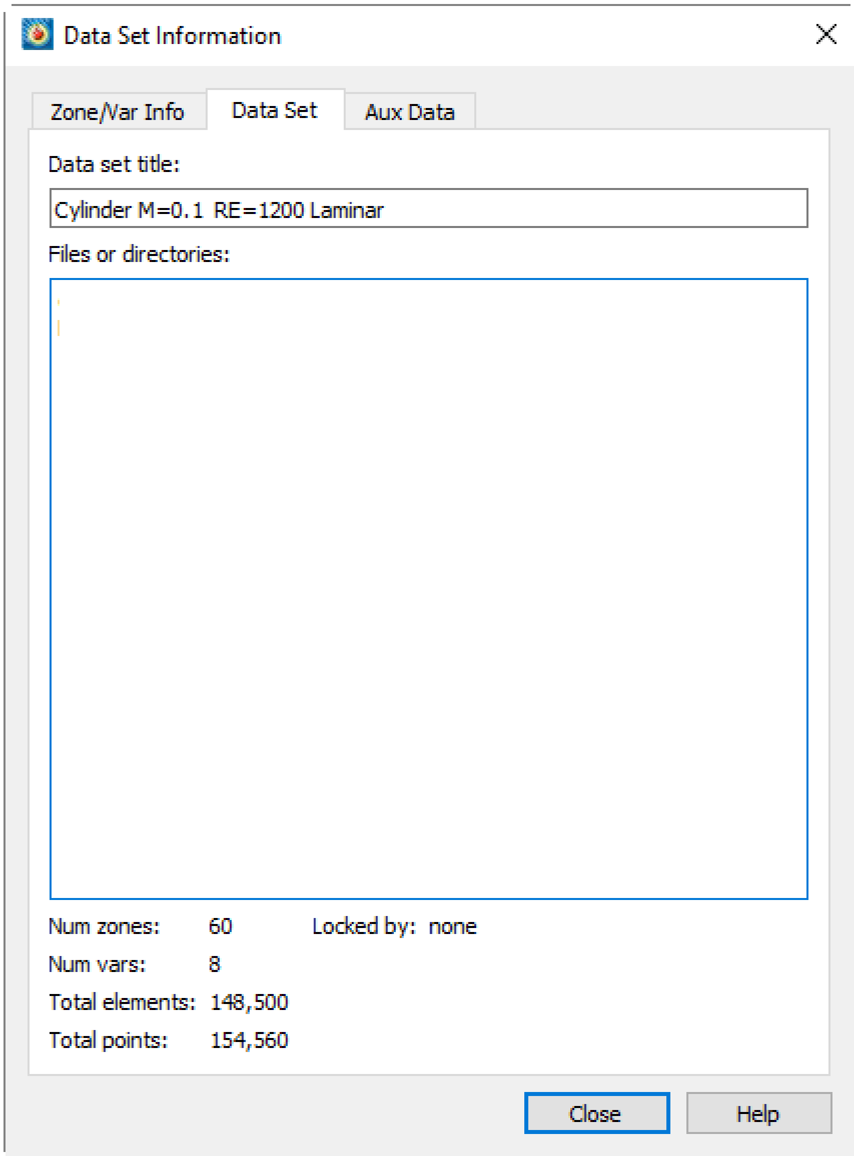


- Click once to select only a single variable
- Hold the `Shift` key while clicking a second variable to select a contiguous range of variables
- Hold the `Control` key while clicking to toggle individual variables on and off
- Hold the `Control` key while clicking to toggle individual variables on and off
- Press `Control-A` to select all variables

Click the **[Load]** button to load the selected variables. See Load On Demand for more information about this function.

Data Set Page

The Data Set page provides information about your data set, including: title, filenames, number of zones, variables, elements, and lock status.



The following information can be found on the Data Set page:

Data Set Title

Enter a title for the current dataset, 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 dataset.

Files or directories

Lists the names and paths of all external data files making up the current dataset. Some file formats use directories as containers; these will list only the directory name.

Num Zones

Number of zones in the dataset.

Num Vars

Number of variables in the dataset.

Total Elements

Total number of elements in the dataset.

Total Points

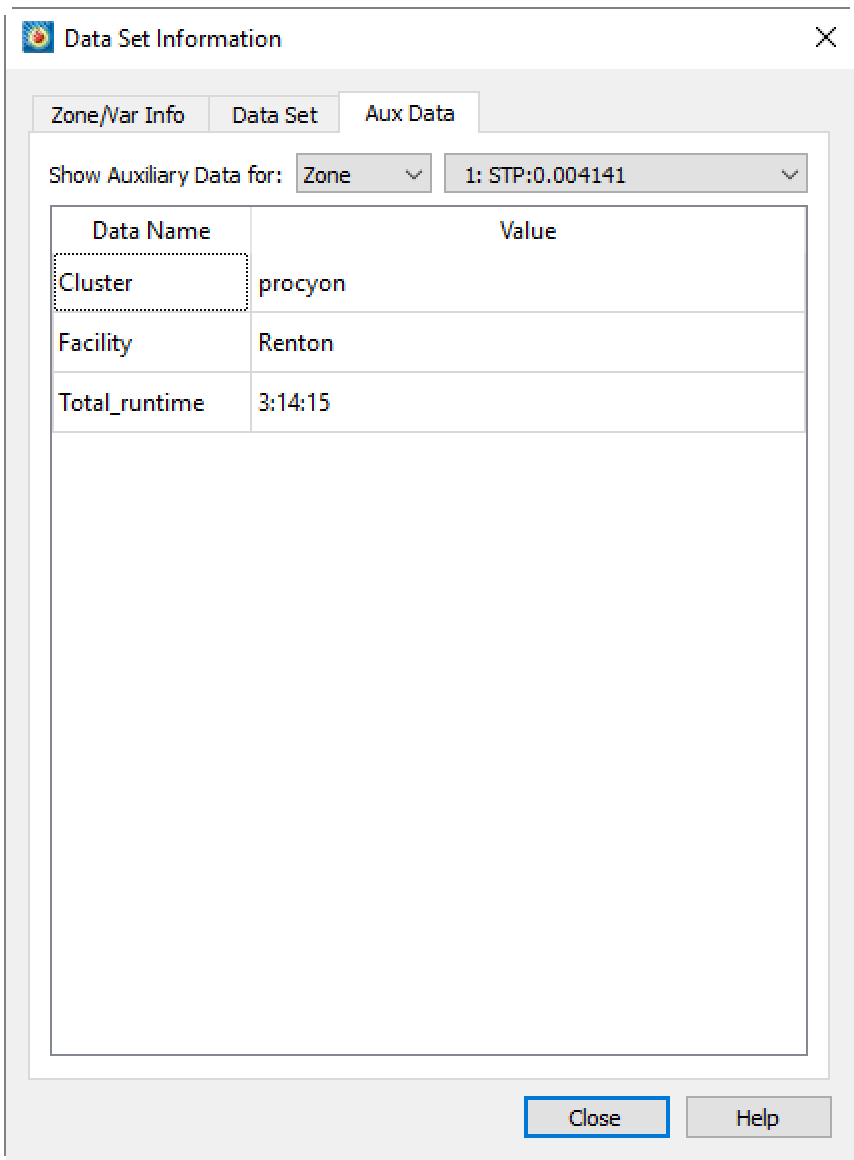
Total number of points in the dataset.

Locked By

This field will inform you if the current dataset has been locked by an add-on. Add-ons can lock a dataset which in turn prevents you from deleting zones or deleting the last frame associated with the dataset.

Aux Data Page

Auxiliary data is metadata attached to zones, datasets, or frames. Auxiliary data is added to the plot via the data file.



The Auxiliary Data page has the following information:

Show Auxiliary Data

Use the drop-down to display auxiliary data for zones, datasets, frames, or names.

Data Name/Value

Displays the names and values of any auxiliary data.

Refer to the Data Format Guide for information on creating Tecplot 360 data files that include auxiliary data.

Color Chooser

Each attribute of your plot can be set to a different color or color type using the **Color Chooser**. Use the **Select Color** dialog to apply contour groups, a basic color palette, or RGB coloring to the selected plot attribute.

There are three types of color assignments:

Contour Groups

The Contour Variables (Multi 1, 2, 3, 4, 5, 6, 7, and 8) are defined in the Contour & Multi-Coloring Details. The Contour Variables are typically used for coloring mesh, contour, vector, and scatter layers.

RGB Coloring

Select RGB to use RGB coloring established in **Plot→RGB Coloring→Variables/Range**. RGB coloring is used to illustrate the relationship between two or three variables in your dataset, by setting R, G, and B to each of the variables.

Basic Color Palette

Use the basic color palette to apply a single, constant color to a plot attribute.

For example, you can create a 3D field plot with a contour layer (with colors defined by a contour variable), an edge layer (with colors from the basic color palette), and a vector layer (with colors defined by RGB vectors).

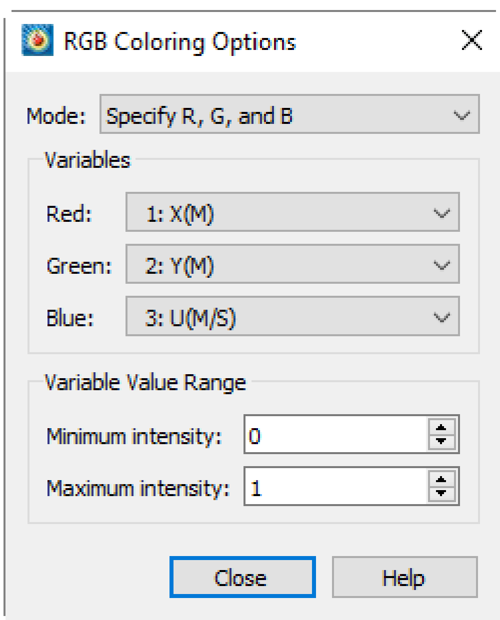
RGB Coloring

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 or vector direction plots. RGB coloring may be used for each field plot object: 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.

Exported Vector-based Files Limitation in RGB Coloring. Vector-based export files such as WMF cannot 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.

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 (accessed via **Plot→RGB Coloring→Variables/Range**). The **RGB Coloring Options** dialog (shown below) has the following options:



Mode

You can either specify all three variables or specify two of the three variables and calculate the third. The third variable is calculated using the following formula *f(R)+f(G)+f(B)=1.0* (assuming f() is a function that maps R,G,B values into [0,1.0]).

Variables

Assign the variables which supply the values for the color components, as specified in the RGB Mode.

Variable Value 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**.

Basic Color Palette

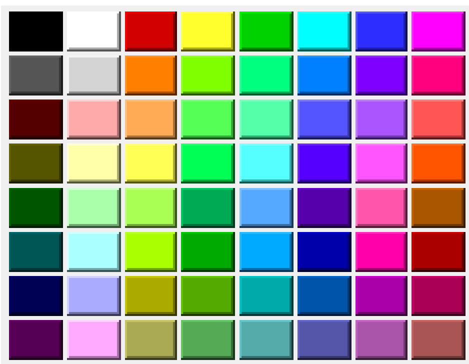


Figure 12. The Basic Color Palette region of the **Select Color** *dialog.*

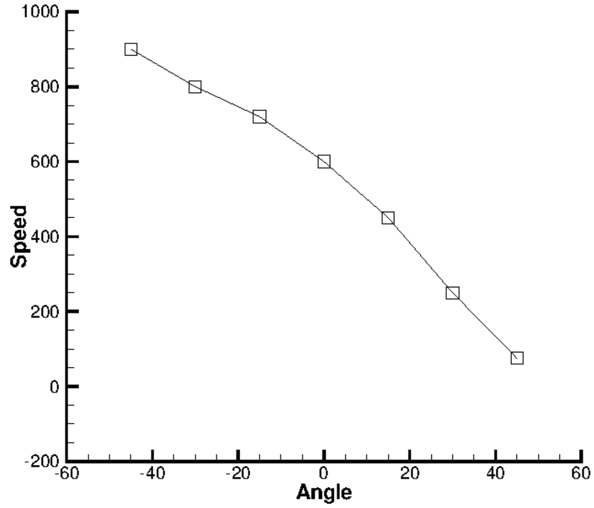
Use the Basic Color Palette to define a constant color to the selected plot attribute(s).

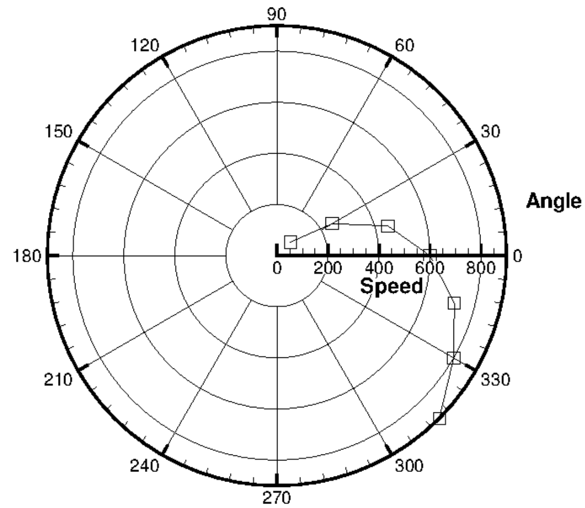
XY and Polar Line Plots

A line plot is the simplest type of graph you can produce with Tecplot 360. A typical line plot includes a dependent variable (usually the vertical axis for XY plots) and an independent variable (usually the horizontal axis for XY plots). Each line on the line plot represents one series of data points, where each data point is defined by its independent and dependent variable values. A series of data points is referred to as a mapping (or map, for short).

Tecplot 360 supports two types of line plots, XY plots and Polar plots. XY plots are plotted on Cartesian coordinates using X & Y as the independent and dependent variables (See Axis Variable Assignment). XY plots can include line, symbols, bar and/or error bar layers. Polar plots are plotted on polar coordinates using Theta and R values. Polar plots can include line and/or symbol layers.

An example of XY and Polar Line plots is shown below.





Line plots are usually created from one-dimensional, I-ordered data. 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 can also create line plots from two or three-dimensional data in the IJ or IJK-ordered structure, or from finite element data by selecting "XY Line" from the plot type menu in the Plot sidebar. If "XY Line" is selected, finite element data sets will be treated as I-ordered (the connectivity list is ignored), IJ-ordered datasets will be treated as a family of J-sets of I-ordered data, and IJK-ordered datasets will be treated as K-planes of J-families of lines. Use the Indices page of the **Mapping Style** dialog to select different ranges and skip intervals for the I, J, and K-indices. See this section on ijk indices later in this chapter for more information.

When you first create a line plot, the Create Mappings dialog appears automatically so you can choose the mappings to be created (see Mapping Creation). Names, colors, symbol types, and line patterns are automatically assigned 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 select the **Mapping Style** button on the Plot sidebar.

Mapping Style and Creation

Line plots 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 referred to as mappings. Mappings are defined for each frame; the same dataset can have a different set of mappings in each frame it is attached to.

Mappings can include any combination of the following mapping layers:

Lines

Can be drawn as linear segments or curve that fit the data points.

Symbols

Each data point is represented by a symbol.

Error Bars (XY Only)

Error bars are drawn for each data point. The error bar value is determined by a third variable.

Bar Charts (XY Only)

Each data point is represented by a vertical or horizontal bar.

XY Line plots can have up to five x-axes and five y-axes simultaneously. Polar Line plots can have only one Theta-axis and only one R-axis.

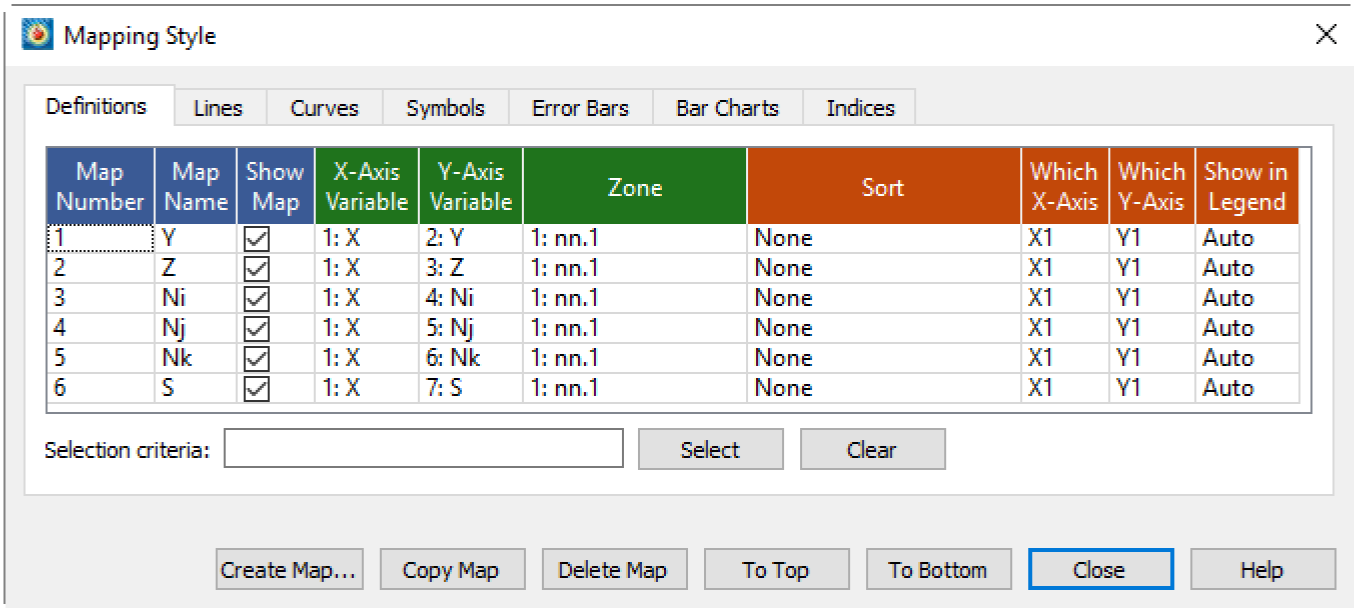
Open the **Mapping Style** dialog by clicking the Mapping Style button on the Plot sidebar to set attributes for lines and symbols and, in XY Line plots, bar charts and error bars. Initially, any linemaps selected in the workspace are selected in the Mapping Style dialog. You may also double-click a line in your plot to open the dialog with that mapping selected. You can set the style of any mapping independently of all other mappings.

Each page of the **Mapping Style** dialog is divided into three color-coded regions. The blue columns (Map Number, Map Name, and Show Map) apply to the mapping itself and are repeated on each page. The green and orange columns represent primary and secondary settings specific to the selected mapping page.

You can also use the line map context menu and toolbar to change some of these settings quickly and easily by right-clicking on a line on your plot. The toolbar allows you to turn on or off the line, symbol, and error bars for the selected line map by clicking the icons. You can also adjust the attributes of these (for example, line color, symbol shape, or error variable) using drop-down menus to the right of each icon.

Mapping Definitions

Existing mappings are edited with the **Plot** menu’s **Mapping Style** dialog. From the Definitions page of the **Mapping Style** dialog, you can 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 assign particular X and Y-axes to XY-line plots.



Some settings are represented by checkboxes, which can be toggled on and off by clicking them. Other settings require a right-click. In general, select the mapping or mappings you want to change, then right-click the selection in the column of the setting to be edited. Most of the time this will activate a pop-up menu; in other cases, a dialog appears. You may change mappings whether they are shown on the plot or not (activated or deactivated).

Map Number

Displays the number of each map. This cannot be edited.

Map Name

Double-click the map name to edit it. Type Enter to complete the edit or Escape to cancel without saving the edit.

Show Map

Each mapping can be opted in and out of a plot by toggling the checkbox or by right-clicking and choosing one of the following options:

Activate

Turns selected mappings on.

Deactivate

Turns selected mappings off.

Show Selected Only

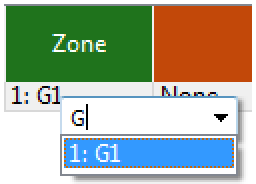
Turns on selected mappings, and turns off all other mappings.

Invert

Switches the current activation settings for the selected map(s).

Axis Variables

The choice of variables is the heart of the mapping. Each mapping is defined by two variables: X and Y in XY Line plots and Theta and R in Polar Line plots. You may change the variables assigned to a mapping by right-clicking or double-clicking.



Zone

Each mapping uses variable values from a specified zone. If your dataset has multiple zones, specify the zone for each mapping by right-clicking or double-clicking the displayed zone name.

The pop-up field chooser appears. Display a list of all zones by clicking the downward-pointing arrow next to the filter field, or start typing to display a filtered list of the zones whose name contains what you type.

Sort

By default, mappings are sorted by the order in which they occur in the data file. You can change this order with the Sort option on the Definitions page of the **Mapping Style** dialog.

Choose from one of the following Sort 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 Specific Variable

Select a variable from the **Select Variable** dialog. The points of the selected mappings are sorted in ascending order of the values of this variable.

> Only Line Segment and Parametric Spline curve types are affected by the Sort options. Splines are always sorted by the independent variable. See Curve Types for more information on curve types.

Which Axes

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 by right-clicking.

The ranges and scales for each axis are defined in the **Axis Details** dialog (accessed via the **Plot→Axis**).

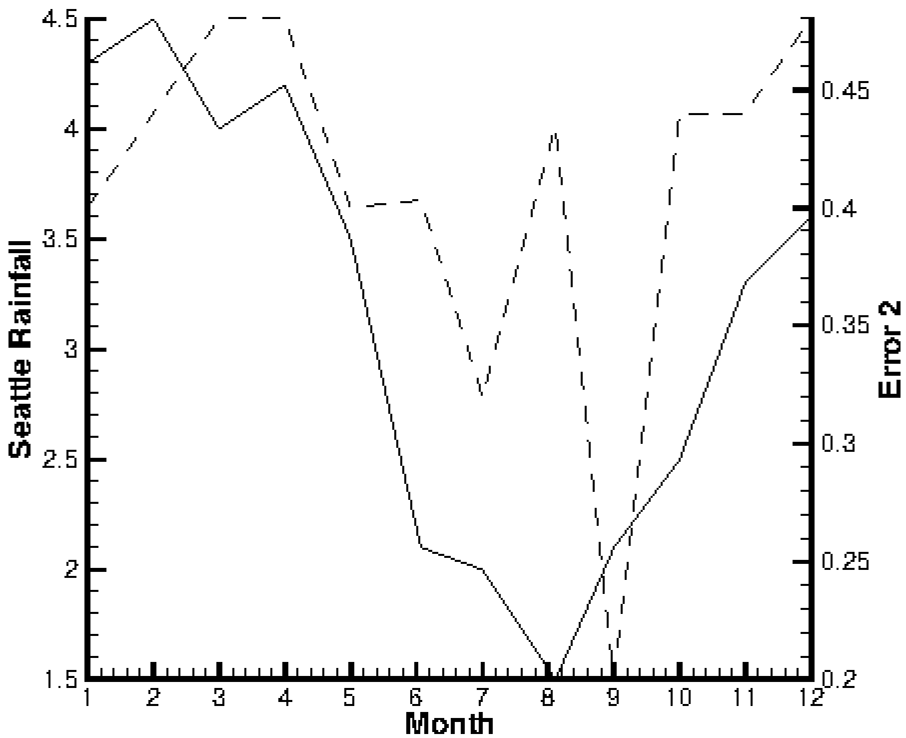


Figure 13. An XY Line plot using two Y-axes.

By default, the X1 axis is placed 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 13, 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.

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.

Show in Legend

By default, all active mappings appear in the line legend. However, the legend only lists mappings with identical entries once. (See Line Legend for details on the Line Plot Legend.) The **Show in Legend** button has 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 mappings would result in the same entry in the legend, only one entry is shown.

Selection Criteria

Enter a wildcard pattern and click **Select** to select one or more mappings based their name. In wildcard patterns, most characters match themselves, but the * and ? characters have special meaning.

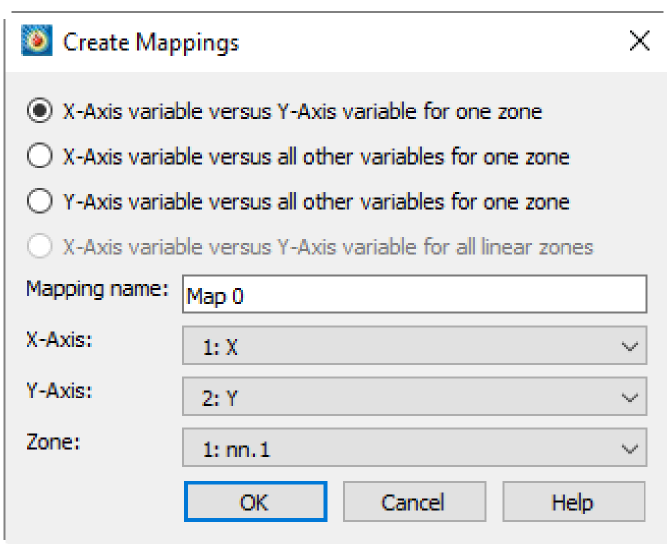
Character	Meaning
?	Matches any single character.
*	Matches any number of characters, including none.

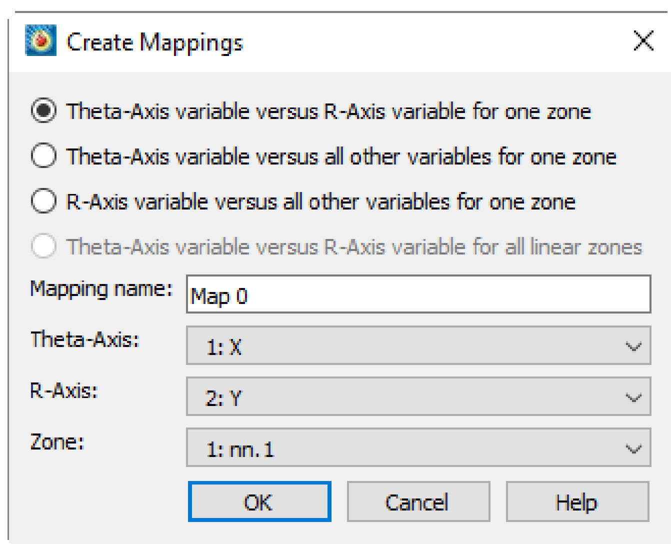
Clear

Clear the selection and the pattern field.

Mapping Creation

To define a new mapping, select the **Create Map** button in **Mapping Style** dialog. The dialog also appears automatically the first time you choose the XY Line or Polar Line plot type. As shown below, the dialog is slightly different for the two plot types.





In XY Line plots, you have the following options:

X-axis Variable versus Y-axis Variable for One Zone default)

Add a single mapping with one X and one Y-variable for one zone.

X-axis Variable versus All Other Variables for One Zone

Create a new set of mappings using one variable as the X-variable and each of the other variables in the zone as Y-variables.

Y-axis Var versus All Other Variables for One Zone

Create a new set of mappings using one variable as the Y-variable and each of the other variables in the zone as X-variables.

X-axis Variable versus Y-axis Variable for All Linear Zones

Define one map for each zone, with the specified X-axis and Y-axis variables. If you choose this option, you specify only the X-axis and Y-axis variables.

The options for polar line plots are the same as above, but with respect to the Theta-axis and R-axis variables.

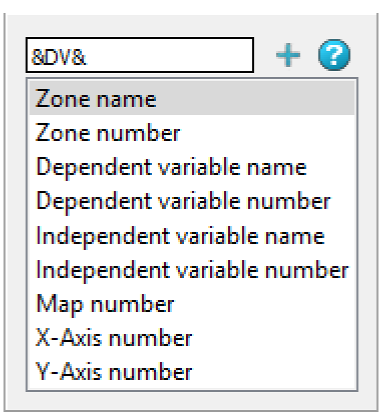
Specify the mapping name and the axis variables in the bottom section of the dialog. The default name is "Map *n*," where *n* is the number of the mapping to be created. If you are creating a mapping for one zone, you also select the zone here.

When you first load an ordered dataset, some mappings are automatically created for you. If your dataset has more than two variables, mappings are created that associate the first variable with each of the other variables for the first zone only.

> Selecting variables in a 3D finite element zone may require significant time, since the variable must be loaded over the entire zone. XY and Polar line plots are best used with linear or ordered data, or with two-dimensional finite element data. The defaults used in the Create Mappings dialog favor the types of zones that yield good performance.

Mapping Names

Tecplot 360 automatically assigns a name to each mapping. The nature of the name depends on 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. In either case, Tecplot 360 assigns each mapping a special name (`&ZN&` or `&DN&` that is replaced with the zone or variable name when the name is displayed.



You can modify any mapping's name by right-clicking the name of the mapping in the Mapping Style dialog. A small window pops up to allow you to enter a new name for the mapping, as shown here (note the default name of `&DN&` in our example; our data has multiple dependent variables, and the name of each is used as the mapping name).

You may add other dynamic text placeholders like `&DN&` to the displayed mapping name by clicking one of the options below the edit field, then clicking `+` (or simply double-clicking). The placeholders will be replaced with the indicated value when the name is displayed. By combining static text with these placeholders, you can construct a name in any format you like.

The placeholders available in this window are:

Zone name

Adds `&ZN&` to the mapping name. This will be replaced with the actual name of the zone assigned to that mapping.

Zone number

Adds `&Z#&` to the mapping name. This will be replaced with the actual number of the zone assigned to the mapping.

Independent variable name

Adds `&IV&` to the mapping name. This will be replaced with the actual name of the independent variable assigned to that mapping.

Independent variable number

Adds `&I#&` to the mapping name. This will be replaced with the actual number of the independent variable assigned to the mapping.

Dependent variable name

Adds `&DV&` to the mapping name. This will be replaced with the actual name of the dependent variable assigned to that mapping.

Dependent variable numbe

Adds `&D#&` to the mapping name. This will be replaced with the actual number of the dependent variable assigned to the mapping.

Map number

Adds `&M#&` to the mapping name. This will be replaced with the actual number of the mapping.

X-Axis number

Adds `&X#&` to the mapping name. This will be replaced with the actual number of the X-axis assigned to that mapping for XY Line plots. (This option is not available for Polar Line plots.)

Y-Axis number

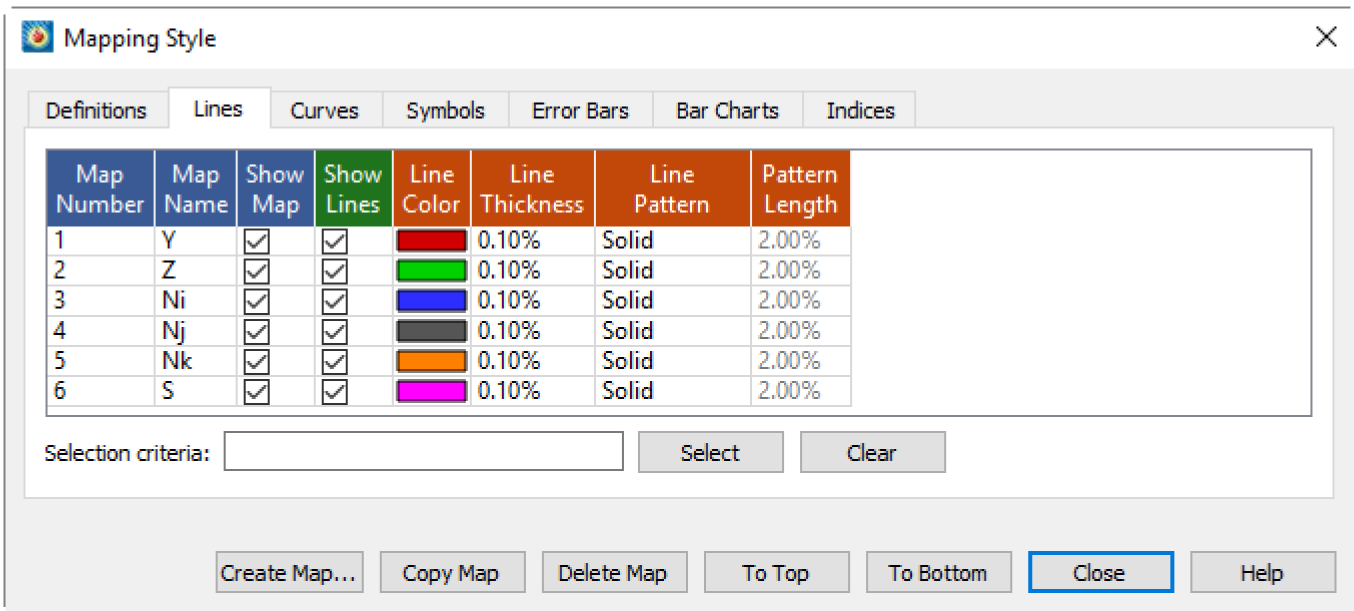
Adds `&Y#&` to the mapping name. This will be replaced with the actual number of the Y-axis assigned to that mapping for XY Line plots. (This option is not available for Polar Line plots.)

Map Layer

The Line map layer is available for both XY and polar line plots. Activate the layer by toggling-on "Lines" in the Plot sidebar. When the Lines map layer is on, the dataset is represented by a connected line for each mapping, which may be either a simple collection of line segments connecting all the data points, or a curve fitted to the original data.

Line Attributes

The Lines page of the **Mapping Style** dialog (accessed via the Plot sidebar or **Plot→Mapping Style**) is shown below.



The first two columns, Map Number and Map Name, list the mapping number and the mapping name. The Map Show field lists which mappings are currently active. These columns behave the same as the corresponding columns on the Definitions page (see Mapping Definitions). The remaining columns of the Lines page of the **Mapping Style** dialog contain specific line attributes.

In order for the changes made in the Lines page to be visible in your plot, the Lines mapping layer must be toggled-on in the Plot sidebar.

Show Lines

This option allows you to turn off lines for selected mappings, while keeping both the selected mappings and the Lines map layer active overall.

Line Color

Set line color for line plots. Right-clicking displays the Color Chooser.

Line Thickness

Set the thickness of lines. Right-clicking allows you to choose from preset line widths of 0.02, 0.10, 0.40, 0.80, or 1.50 percent, or enter any thickness using a dialog.

Line Pattern

Right-click to choose line patterns for line plots.

Pattern Length

Set the pattern length for patterned lines. Pattern length is measured as a percentage of the frame height for one complete cycle of the pattern.

> For information on using the controls at the bottom of the Mapping Style dialog to select mappings by name, see the description of these at the end of Mapping Definitions.

Curve Types

Tecplot 360 offers a variety of curve-fits and spline fits. By specifying the curve type, you control how the data points are connected (see Map Layer).

Set the type of curve plotted for a mapping by right-clicking the desired Curve Type entry on the **Curves** page of the **Mapping Style** dialog.

Tecplot 360 offers the following curve types:

Line Segment (No Curve-fit)

A series of linear segments connect adjacent data points. In XY Line plots, these will be line segments.

Linear Fit

A linear function is fit to the data points. In XY Line plots, this will be a straight line.

Polynomial Curve-fit

A polynomial of order N is fit to the data points (where $1 \le N \le 10$, for N=1 a Linear Fit is done).

Exponential Curve-fit

An exponential curve-fit that finds the best curve of the form $Y = e^{b*lnX+c}$ (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.

Power Curve-fit

A power curve fit that finds the best curve of the form $Y=e\_^{b*lnX+c}$ (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 gives you control over the initial and final slopes of the curve.

Parametric Spline

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 additional sorting of the points is performed; the sorting specified on the Definitions page of the **Zone Style** dialog is used for the order of the data points. This spline may result in a multi-valued function (of either or both axis variables).

Extended Curve-fit

Uses a curve-fit supplied by an add-on. These curve-fits may be provided by Tecplot 360, a third party, or written by users. The functionality of each extended curve-fit is defined by its creator.

Linear Fit, Polynomial Fit, Exponential Fit, and Power Fit are all determined by using a least squares algorithm. Examples of each curve-fit type are shown in Figure 14.

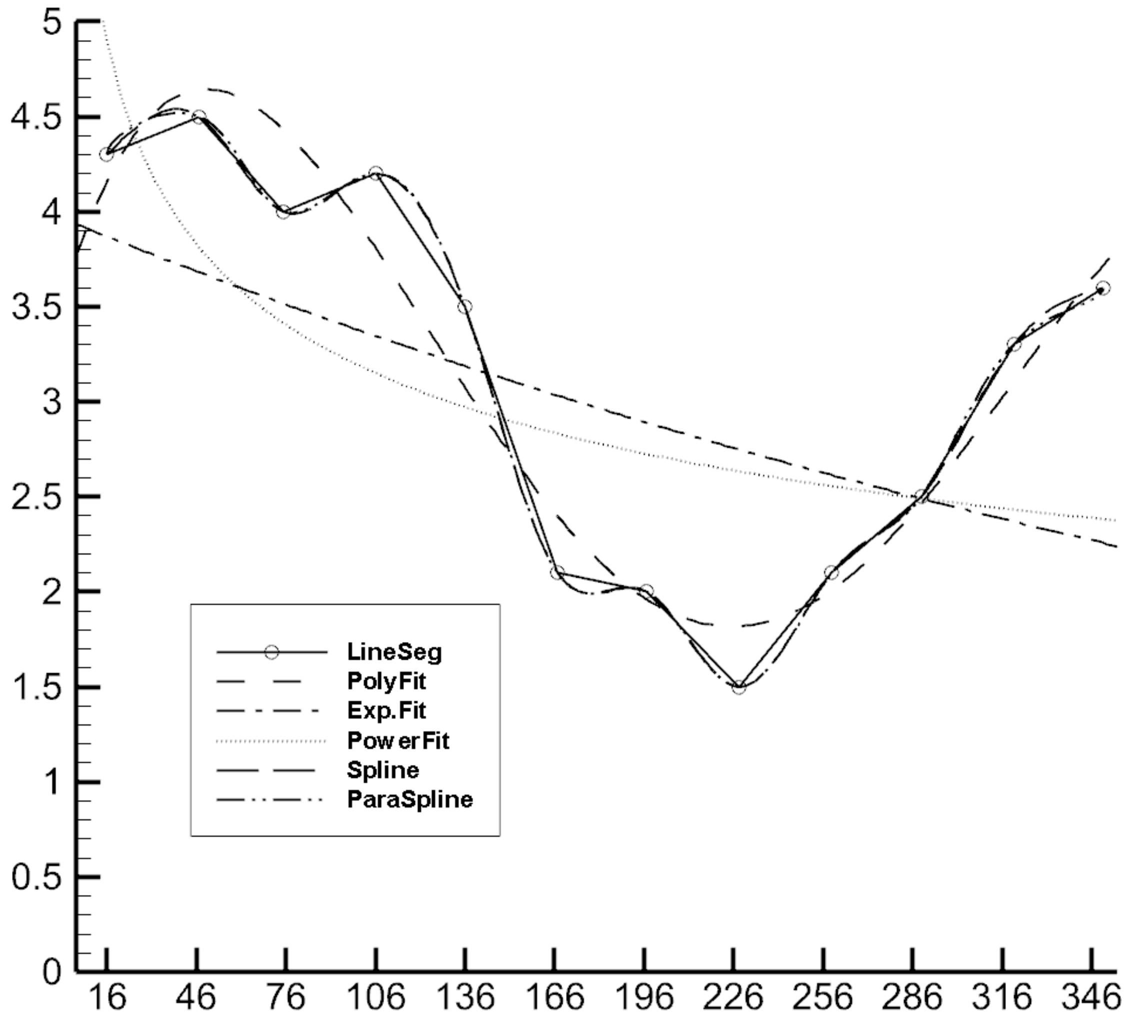


Figure 14. Tecplot 360's curve-fit types.

The Curves page also contains fields for controlling the following attributes:

Dependent Variable

Right-click to choose how curve fits and splines are interpreted. Dependent Variable has no effect on mappings of the Line Segment curve type.

Curve Points

Controls the number of points used to draw curve fits and splines. Right-click to choose a preset number of points or to enter your own. Raising the number of points increases the accuracy of the curve but also increases plotting time and the size of print files.

Curve Setting

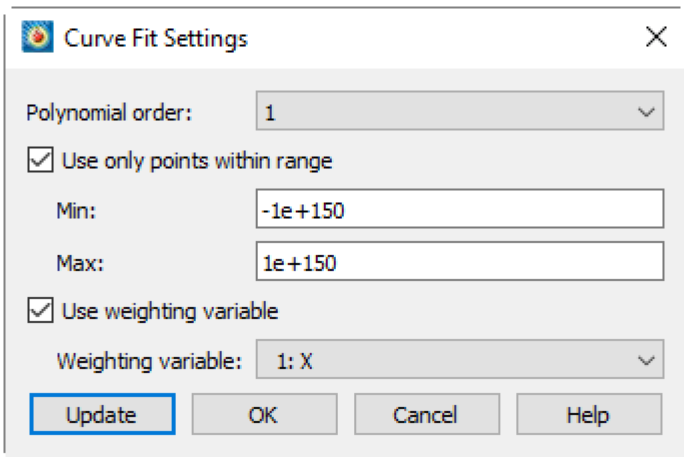
Displays options specific to the curve type, such as weighting for curve fits or starting derivatives for splines. Right-click to set these options.

For information on using the controls at the bottom of the Mapping Style dialog to select mappings by name, see the description of these at the end of Mapping Definitions.

Linear Fit

Tecplot 360 fits the data to a linear function 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, right-click in the Curve Type column on the Curves page of the **Mapping Style** dialog and select "Linear Fit." Then right-click in the Curve Setting column and choose Settings to display the Curve Fit Settings dialog, shown below.



Polynomial Order

This is shown on the dialog, but should always be "1" for a linear fit. If you change this from 1, the curve type is changed to Polynomial Curve-fit.

To limit the points used in the mapping(s)

select "Use Only Points Within Range", and enter minimum and maximum values.

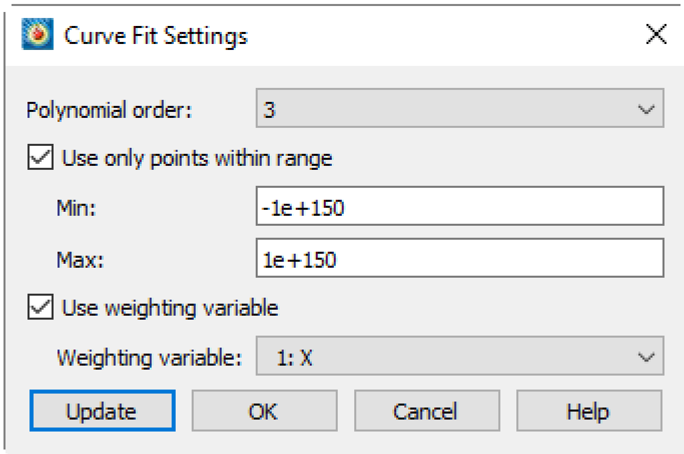
To assign a curve weighting variable

select "Use Weighting Variable", and select the variable from the drop-down. For more information on curve weighting, see Curve-fit Weighting Variables.

Polynomial Curve-fit

Tecplot 360 uses a standard least-squares algorithm to fit data to a polynomial function. You specify the order of the polynomial (from one to ten), and the polynomial for which the sum of the squared differences from the data points is a minimum is calculated.

To fit a polynomial function to your data, right-click in the Curve Type column on the Curves page of the **Mapping Style** dialog and select "Polynomial Fit."Then right-click in the Curve Setting column and choose Settings. The Curve Fit Settings dialog appears.



By default, this option fits a cubic polynomial, using all the points in the mapping and weighting them equally.

Polynomial Order Drop Down

Select the desired polynomial order (1 to 10). An order of 2 is a quadratic polynomial, an order of 3 is a cubic polynomial, etc. If you select 1, the curve type is set to Linear Fit, as a polynomial of order 1 is a linear function. (See Linear Fit.)

To limit the points used in the mapping(s)

Select "Use Only Points Within Range", and enter minimum and maximum values.

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 Curve-fit Weighting Variables.

Exponential Curve-fit

Tecplot 360 fits the data to an exponential function using the standard least-squares algorithm.

The dependent-variable values must be either all positive or all negative.

For XY plots (where X is the independent variable)

Tecplot 360 finds the best curve of the form:

$Y=e^{b*X+c}$ (equivalent to $Y=a*e^{b*X}$ where $a=e^{c}$).

Similarly when Y is the independent variable.

For Polar plots (where Theta is the independent variable)

Tecplot 360 finds the best curve of the form:

$R = \pm e^{(b\theta+c)}$

or

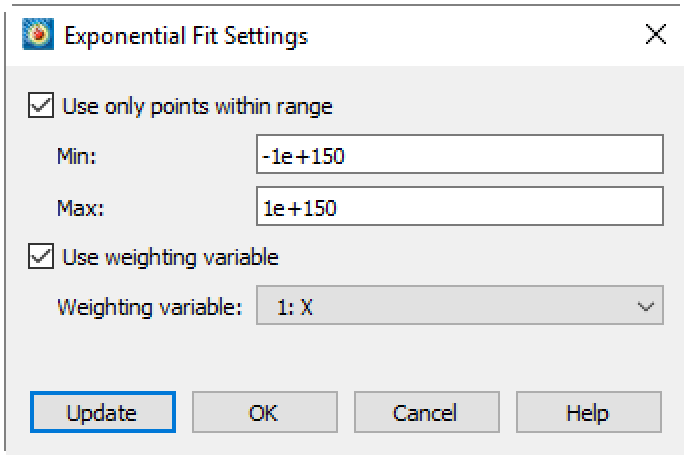
$R = \pm\alpha e^{(b*\theta)}$

Similarly when R is the independent variable.

To fit an exponential function to your data, right-click in the Curve Type column on the Curves page of the **Mapping Style** dialog and select "Exponential Fit."

By default, this option uses all the data points in the mapping, weighting them equally.

Use the **Exponential Fit Settings** dialog dialog (accessed by right-clicking in the Curve Setting page on the Curves page of the **Mapping Style** dialog) to specify different settings. The dialog is shown below.



To limit the points used in the mapping(s)

Select "Use Only Points Within Range", and enter minimum and maximum values.

To assign a curve weighting variable

Select "Use Weighting Variable", and choose the variable from the drop-down. For more information on curve weighting, see Curve-fit Weighting Variables.

Power Curve-fit

Tecplot 360 fits 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. Data points with zero or negative independent values are ignored.

For XY plots (where X is the independent variable)

Tecplot 360 finds best curve of the form:

$Y=e^{b*\ln X+c}$ (equivalent to $Y=a*X^b$ where $a=e^c$).

Similarly, when Y is the independent variable.

For Polar plots (where Theta is the independent variable)

Tecplot 360 finds the best curve of the form:

$R = \pm e^{b*\ln(\theta)+c}$

or

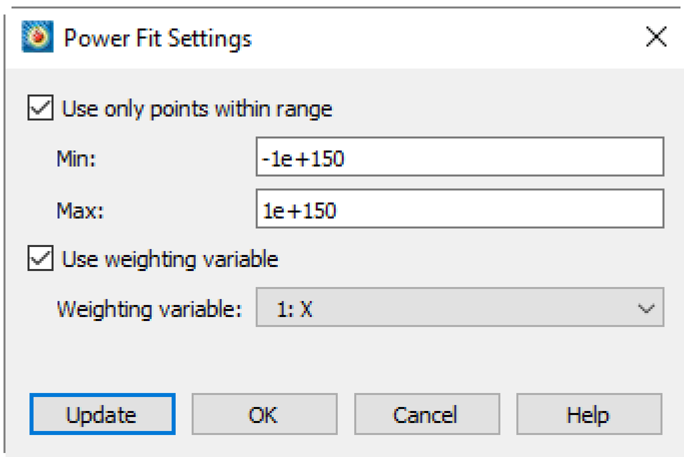
$R = \pm \alpha * \theta^{b}$

Similarly, when R is the independent variable.

To fit a power-curve function to your data, right-click in the Curve Type column on the Curves page of the **Mapping Style** dialog and select "Power Curve."

This option uses all the data points in the mapping, weighting them equally.

Use the **Power Fit Settings** dialog (accessed via the [Curve Settings] button) to specify different settings. The dialog is shown below.



To limit the points used in the mapping(s)

Select "Use Only Points Within Range", and enter minimum and maximum values.

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 Curve-fit Weighting Variables.

Spline

A spline is a mathematical function defined to link a specified set of points with a function that 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.

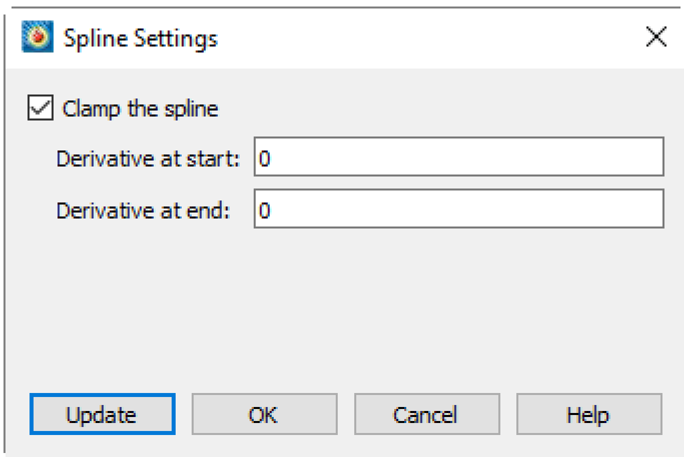
Splines can be natural or clamped: natural splines are twice-differentiable at the end points and the second derivative is zero at those points, while clamped splines have known first-derivatives at the boundary points. Before plotting the spline, the data points are sorted in increasing value along the independent axis.

The Sort option of the Definitions page of the **Mapping Style** dialog has no effect on splines.

To fit a spline function to your data, right-click in the Curve Type column on the Curves page of the **Mapping Style** dialog and select "Spline." By default, this option fits a natural cubic spline.

To specify a clamped spline:

1. Right-click in the Curve Setting column on the Curves page of the **Mapping Style** dialog and choose **Settings**.
2. In the **Spline Settings** dialog (shown below), select "Clamp the Spline", and enter values for the derivative at the start and end of the spline.



Parametric Spline

The 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 θ&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 the order the data points appear in the data file. See Mapping Definitions for a discussion of sorting.

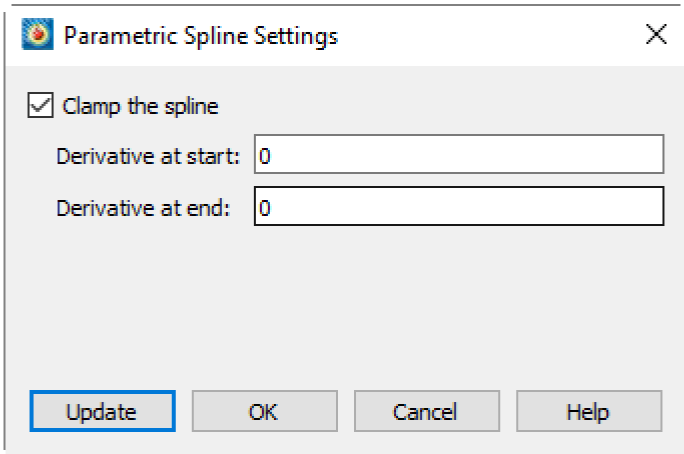
To fit a paraspline function to your data, right-click in the Curve Type column on the Curves page of the **Mapping Style** dialog and choose "ParaSpline."

By default, This option fits two natural cubic splines to the data point index.

To specify a clamped spline:

1. Select Para Spline from the Curve drop-down menu, and then select Curve Settings.

2. In the **Parametric Spline Settings** dialog (shown below), select "Clamp the Spline", and enter values for the derivative at the start and end of the spline.



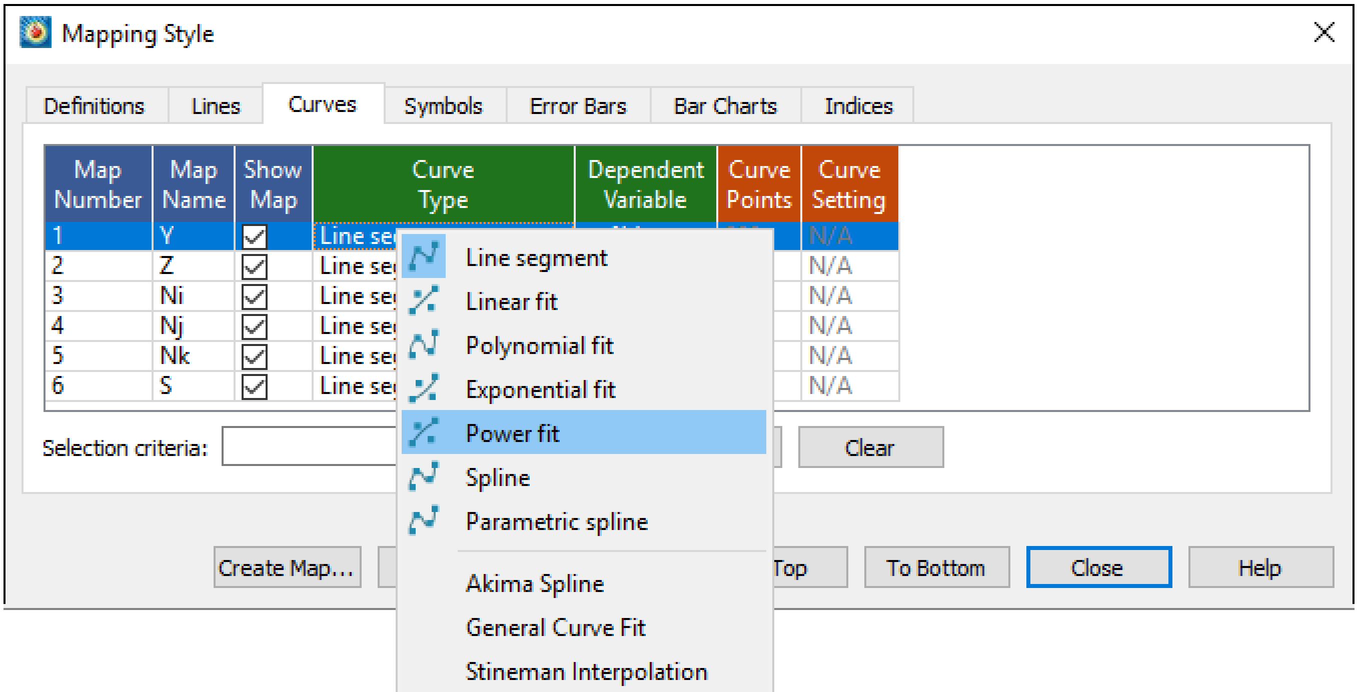
For the XY Line plot type, the derivatives are either *dy/dx* or *dx/dy* depending on the Function Dependency for the mapping. Tecplot 360 calculates *dx/ds* and *dy/ds* from these values (where *s* is the parametric variable). For the Polar Line plot type, the derivatives are either *dR/dTheta* or *dTheta/dR* (depending on the Function Dependency for the mapping), and *dR/ds* and *dTheta/ds* are calculated from these values (where *s* is the parametric variable). See Dependent and Independent Variables for a full description of the Function Dependency option.

Extended Curve-fit

Tecplot 360 add-ons can provide new curve-fit types. These curve types are called extended curve-fits. These curve-fits may be provided by Tecplot 360, a third party, or written by users. The functionality of each extended curve-fit is defined by its creator.

To fit an extended curve to your data:

1. Use the Curves tab of the **Mapping Style** dialog to select the mappings for which you want to apply an extended curve-fit.
2. Right-click **Curve Type**, and select an option from the drop-down.
3. The extended curve fit options are located below the separation line (shown below).



Extended Curve fits can also be selected by the right-click context menu on a line plot. Simply right-click on the line you wish to change, hover over Curve Type, and a similar drop-down menu will appear as above.

Three extended curve fit add-ons are supplied with Tecplot 360:

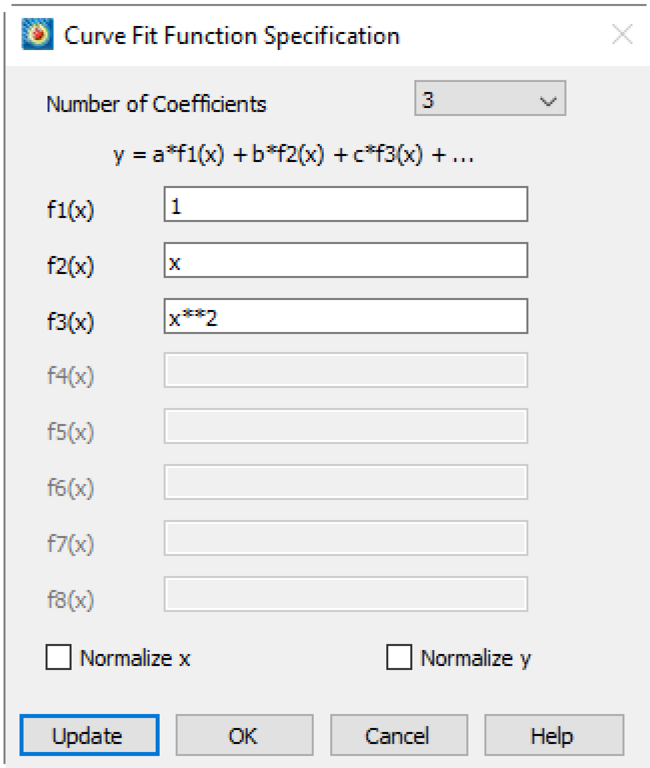
Akima

The Akima spline is an alternative that exhibits less dramatic overshoots and undershoots than the classical spline. The slopes at the end of each segment are computed using a nonlinear average of the segment slopes.[5] The Akima spline is always unclamped. There are currently no options available for the Akima spline.

- Extended Curve-Fit - General
- Extended Curve-fit - Stineman

Extended Curve-Fit - General

The General Curve Fit add-on fits an equation composed of a linear combination of user-specified sub-functions to the data in the specified map. The optional parameters can be accessed by selecting the **Curve Settings** option on the **Curves** page of the **Mapping Style** dialog.



The curve fit computes (least squares) the optimal curve fit coefficients by multiplying these sub-functions.

The following options are available:

Number of Coefficients

Specify the number of coefficients (and number of sub-functions) for the desired curve fit. The default is three. You must specify a sub-function for each coefficient in the text fields labeled f1(x) through fn(x), where n is the number of coefficients.

f1(x) through f8(x)

Enter the sub-functions for the curve fit using the syntax described in Data Alteration through Equations.

In these equations use the variable x as the independent variable, even if x is specified as the dependent variable in the **Dependent Variable** option of the **Mapping Style** dialog.

Normalize X

Causes the curve to be fit using a normalized independent variable. In particular, the independent variable will be translated and scaled to vary from zero at the smallest value of the independent variable to one at the largest value of the independent variable. For most curves other than polynomials, this option will alter the shape of the curve fit. It is useful when you get the "Rank reduced for at least one curve fit" warning message, but otherwise is not recommended.

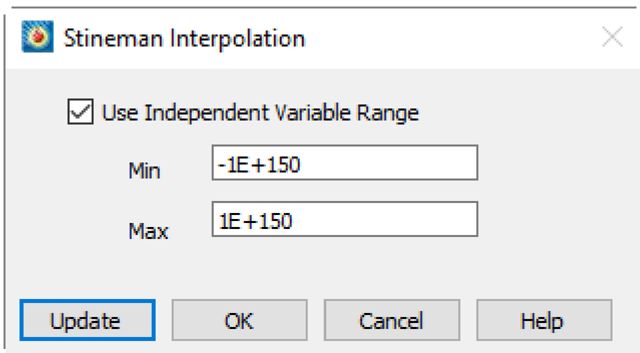
Normalize Y

Causes the curve to be fit using a normalized dependent variable. In particular, the dependent variable will be translated and scaled to vary from zero at the smallest value of the dependent variable to one at the largest value of the dependent variable. For most curves other than polynomials, this option will alter the shape of the curve fit. It is useful when you get the "Rank reduced for at least one curve fit" warning message, but otherwise is not recommended.

Extended Curve-fit - Stineman

This method of interpolation generates a curve that will never have more inflection points than are clearly required by the given set of data points. The interpolating curve passes through the data points and exactly matches the computed slopes at those points[6]

The optional parameters can be accessed by selecting the **Curve Settings** option on the **Curves** tab of the **Mapping Style** dialog.



Line Segment (No Curve-fit)

By default, a series of linear segments are drawn between each set of points for the XY Line plot type.

To turn off curve fits for your data and use linear segments between points:

1. The Curves page of the **Mapping Style** dialog, select the mappings you want to show as linear segments.
2. Right-click in the Curve Type column on the **Curves** page of the **Mapping Style** dialog and select "Line Segments."

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 lines segments are drawn in the order the data points appear in the data file. See Mapping Definitions for a discussion of sorting.

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 by right-clicking in the **Dependent Variable** column on the Curves page of the **Mapping Style** dialog.

For the XY Line plot type, the default setting is *y=f(x)* (you may change the value to *x=f(y)*). With *y=f(x)*, the X-axis variable is the independent variable and the Y-axis variable is the dependent variable. With *x=f(y)*, the Y-axis variable is 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 15.

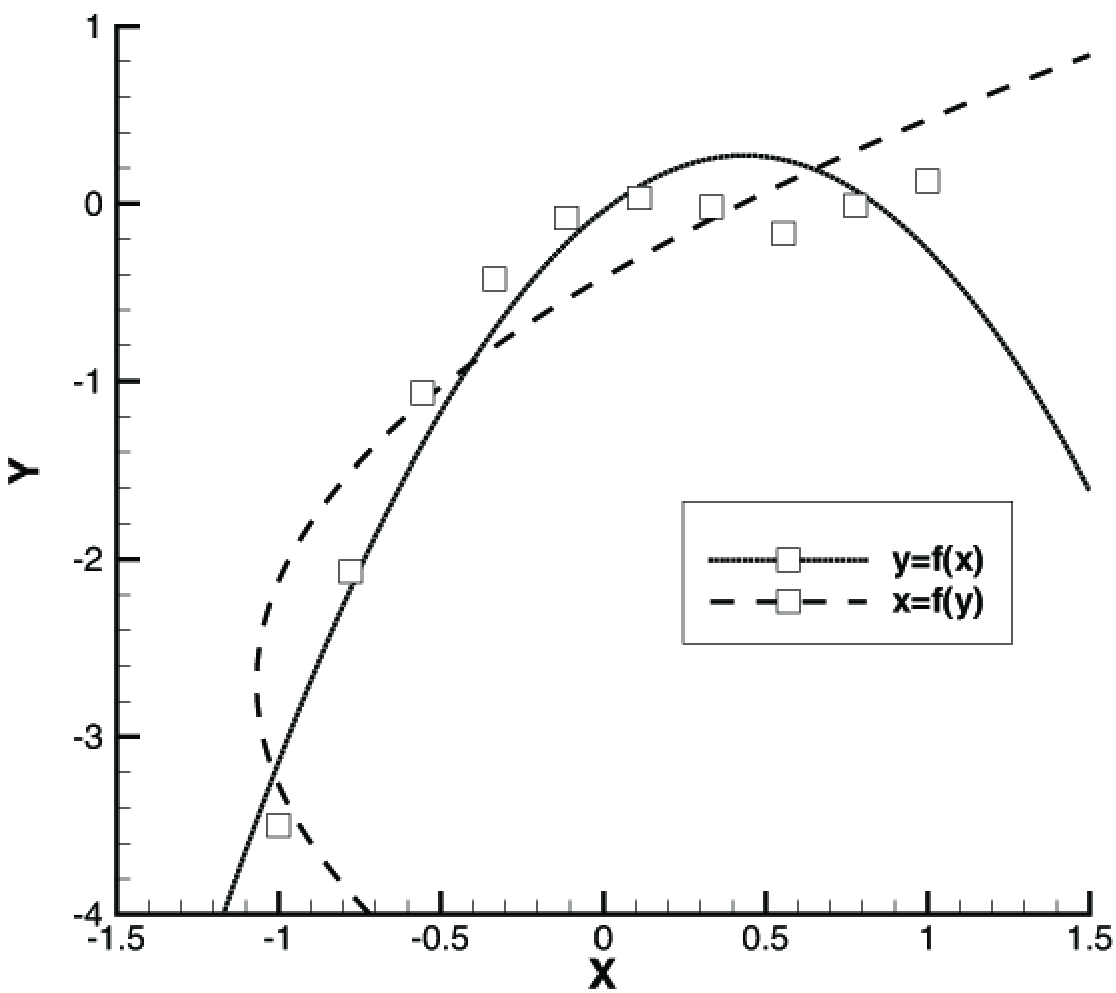


Figure 15. XY Line plot type dependencies.

Similarly for Polar Line plots, the default setting is *R=f(Theta)* (you may change the value to *Theta=f(R)*). With *R=f(Theta)*, the Theta-axis variable is the independent variable and the R-axis variable is the dependent variable. With *Theta=f(R)*, the R-axis variable is the independent variable and the Theta-axis variable is the dependent variable.

To change the dependency setting:

1. From the Curves page of the **Mapping Style** dialog, select the mappings to change.
2. Right-click in the Dependent Variable column on the Curves page of the **Mapping Style** dialog choose either **R=f(Theta)** or **Theta=f(R)**.

> 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 XY Line Bar Charts.

Curve-fit Weighting Variables

Linear and polynomial 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.** Therefore if your data contains outliers that would affect the overall curve fit, adding a zero weight at the point of the outlier will remove it from the overall curve fit calculation.

The weighting coefficients must be integers in the range of zero to 9,999. Weighting coefficients defined as floating-point numbers are truncated. That is, a weighting coefficient of 1.99 is truncated to 1.0.

Curve Information

You can view information about curve-fits and splines by right-clicking the curve in the plot and choosing an item from the Curve Details submenu. The available functions include

Show On Plot

Displays information about the curve directly on the plot, including the zone, the variables, the Goodness of Fit, and other information specific to the curve type (such as the polynomial for the Polynomial Fit type). This information is in a standard text element, and you can reposition it as usual, or change its appearance (its font, size, and color) using the **Text Details** dialog. It is automatically updated if you change the curve fit settings.

Write Curve Points to File

Write the points of the curve to a Tecplot-format data file (`.dat`).

Write Curve Details to File

Writes the same information displayed by Show On Plot to a text file. Note that this text file includes some Tecplot text formatting tags such as `<sup>`.

Goodness of Fit

R^2 is displayed in the curve details for linear, polynomial, exponential, and power curve fits. It is a statistical calculation that measures the success of the curve-fit in modeling the variation of the data.

R^2 is defined as the ratio of the sum of the squares of the regression (SSR) and the total sum of the squares (SST).

$$SSR = \sum_{i=1}^{n} W_i (y_{curvefit_i} - y_{mean})^2$$

$$SST = \sum_{i=1}^{n} W_i (y_i - y_{mean})^2$$

$$R^2 = \frac{SSR}{SST}$$

Where:

Identifier	Represents
SSR	sum of the squares of the regression
SST	total sum of the squares
W_i	the value of the weight variable at index i
y_i	the value of the dependent variable at index i
y_{mean}	the mean value of the dependent variable y
$y_{curvefit\_i}$	the value computed using the curve-fit at the i-index value of the independent variable (x_i).
i	current index number
n	total number of data points

R-square can take any value between zero and one, with a value closer to one indicating a better fit.

A fundamental error term in least-squares curve fits is the sum of the squares residual (SSE), defined by

$$SSE = \sum \lim_{i=1}^{n} W_i (y_{curvefit_i} - y_i)^2$$

This is the number that is minimized when computing the curve-fit coefficients. Using the equation SST = SSE + SSR, R^2 can be related to SSE:

$$R^2 = 1 - \frac{SSE}{SST}$$

Using this form to compute R^2, it is easier to see that an R^2 closer to one (SSE=0) indicates a better curve-fit.

Goodness of Fit Residutal Degrees of Freedom Adjustment

One problem with R^2 is that it will always indicate a good curve-fit when the number of data points, n, equals the number of degrees-of-freedom, m. (For example, a quadratic curve-fit through three data points.) In this case, the curve passes through all data points so SSE=0 and r-square=1. However, there are no other data points so, in reality, no estimate can be made on the quality of the curve fit away from the specified data points. In general, any time m (degrees-of-freedom) is close to n (number of data points), r-square will overstate the quality of the curve fit. For this reason, we include the second goodness-of-fit parameter: degrees-of-freedom adjusted R^2:

$$R^2_{dof} = 1 - \frac{SSE(n-1)}{SST(m-n)}$$

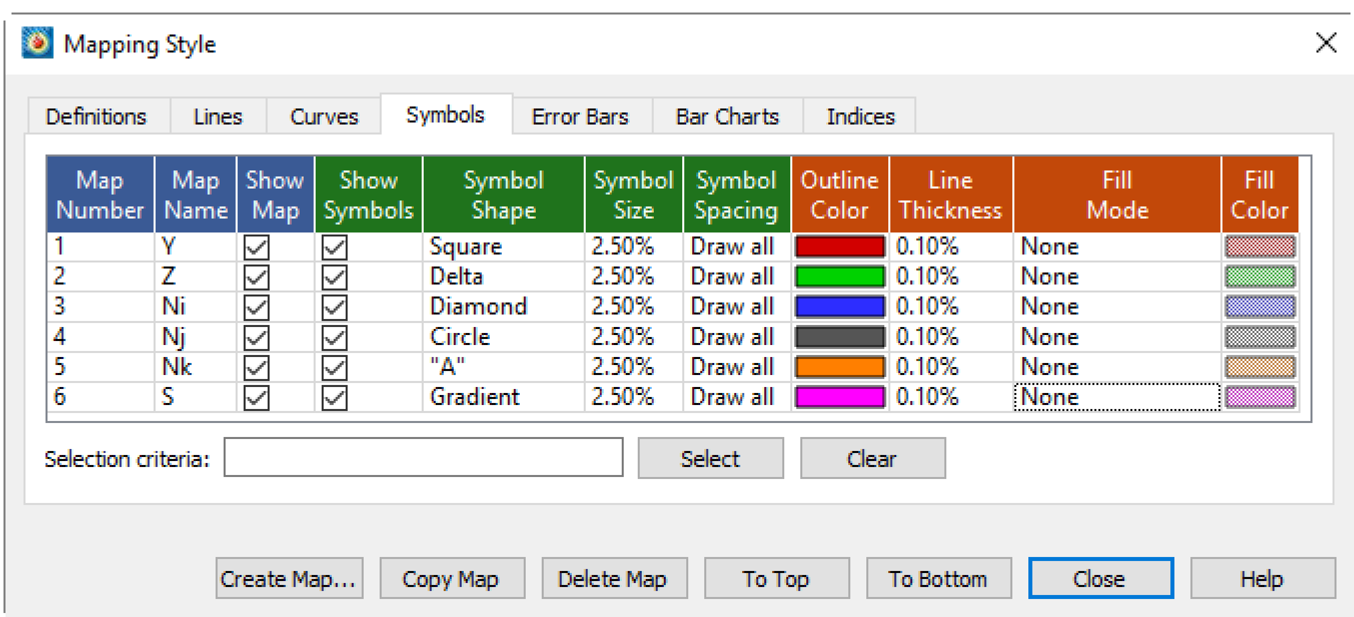
Like the standard R^2, R^2_{dof} will vary from zero to one with values closer to one indicating a better curve fit. R^2_{dof} will be less than R^2 when the degrees-of-freedom are close to the number of data points, but will be nearly equal to R^2 when the number of data points is significantly greater than the degrees-of-freedom.

Symbols Map Layer

The Symbols map layer is available for both XY and polar line plots. Activate the layer by toggling-on "Symbols" in the Plot sidebar. When the Symbols map layer is on, each data point is represented by a symbol on the plot. For each mapping, you may choose the plotting symbol used, and whether to use filled or plain symbols.

Symbol Attributes

Use the Symbols page of the **Mapping Style** dialog (shown below) to modify the attributes of the Symbols layer.



The first two columns list the mapping number and name. The Show Map column indicates currently active mappings. These columns behave the same as the corresponding columns on the Definitions page (see Mapping Definitions).

The remaining columns of the Symbols page of the **Mapping Style** dialog contain specific symbol attributes: Symbol Show, Symbol Shape, Symbol Size, Symbol Spacing, Outline Color, Line Thickness, Fill Mode, and Fill Color.

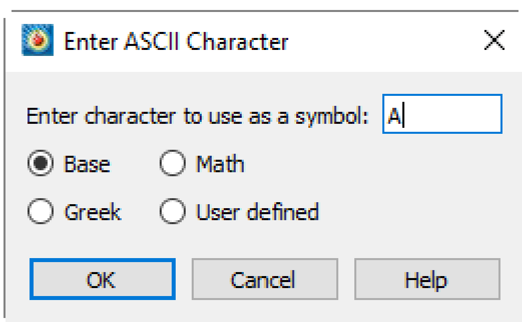
In order for the changes made on the Symbols page to be visible in your plot, the Symbols mapping layer must be toggled-on in the Plot sidebar.

Show Symbols

Allows you to turn off symbols for selected mappings, while keeping both the selected mappings and the Symbols map layer active overall.

Symbol Shape

Right-click and choose a symbol type. In addition to the predefined symbols, you may use any ASCII character by selecting *Character*. Enter the ASCII character to use as a symbol in the **Enter ASCII Character** dialog (shown below), and select a font from which to display the symbol. See Character Indices in Tecplot 360 for further information on the symbols available in each character set.



You can change the base font in the Scatter Size/Font dialog, accessible via **Plot**→**Symbol Font**. See Scatter Size/Font.

Symbol Size

Right-click and choose the symbol size for your line plotting symbols. Symbol size is measured as percentage of the frame height. You can choose a preset size or enter your own.

Symbol Spacing

Right-click to specify the spacing between symbols. 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 a pre-set value.

Draw All

Symbols are drawn at every data point.

ISkip=2, 3 or 4

Symbols are drawn every second, third, or fourth data point.

Enter Distance=1, 2 or 3%

Symbols are drawn at the first data point and subsequently at data points that are at least one, two, or three percent of the frame height distant from the previously plotted data point.

Enter Index

Enter an index skip between symbols.

Enter Distance

Enter a distance between symbols in frame units.

Outline Color

Right-click to choose a color using the Color Chooser.

Line Thickness

Right-click to choose the thickness of lines used to draw the plotting symbols. You may either enter a value or use a pre-set value.

Fill Mode

Right-click to specify the Fill Mode:

None

The symbols are not filled.

Use Line Color

The symbols are filled with the same color specified in Outline Color and appear as 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.

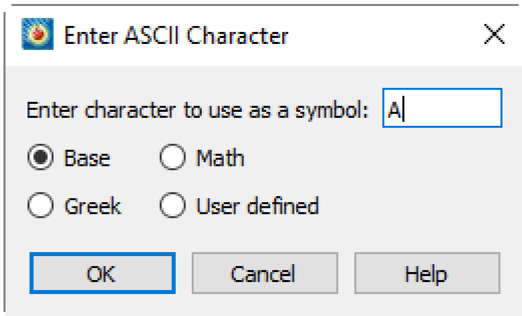
Fill Color

If the Fill Mode is set to "Use Specific Color", right-click in this column to choose the fill color in the Color Chooser.

> For information on using the controls at the bottom of the Mapping Style dialog to select mappings by name, see the description of these at the end of Mapping Definitions.

Enter ASCII Character

Use the **Enter ASCII Character** dialog to specify an ASCII character to use as a plotting symbol in either a field scatter plot or an XY symbol plot. You enter the desired character, and then choose the character set from which to draw the character. See Character Indices in Tecplot 360 for further information on the symbols available in each character set.



This dialog has one text field for specifying the ASCII character and four option buttons representing the available character sets, as follows:

Enter Character to use as a Symbol

Enter the desired ASCII character in this text field.

Base

Select this to use the English-text character set as the source of the plotting character.

Math

Select this to use the math character set as the source of the plotting character.

Greek

Select this to use the Greek character set as the source of the plotting character.

User-defined

Select this to use the user-defined character set as the source of the plotting character. See Custom Character and Symbol Definition.

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. Each mapping can be associated with only one error bar variable. If you want to assign multiple error bar variables to a mapping, create a copy of the mapping for each error bar variable.

An example plot with error bars is shown in Figure 16.

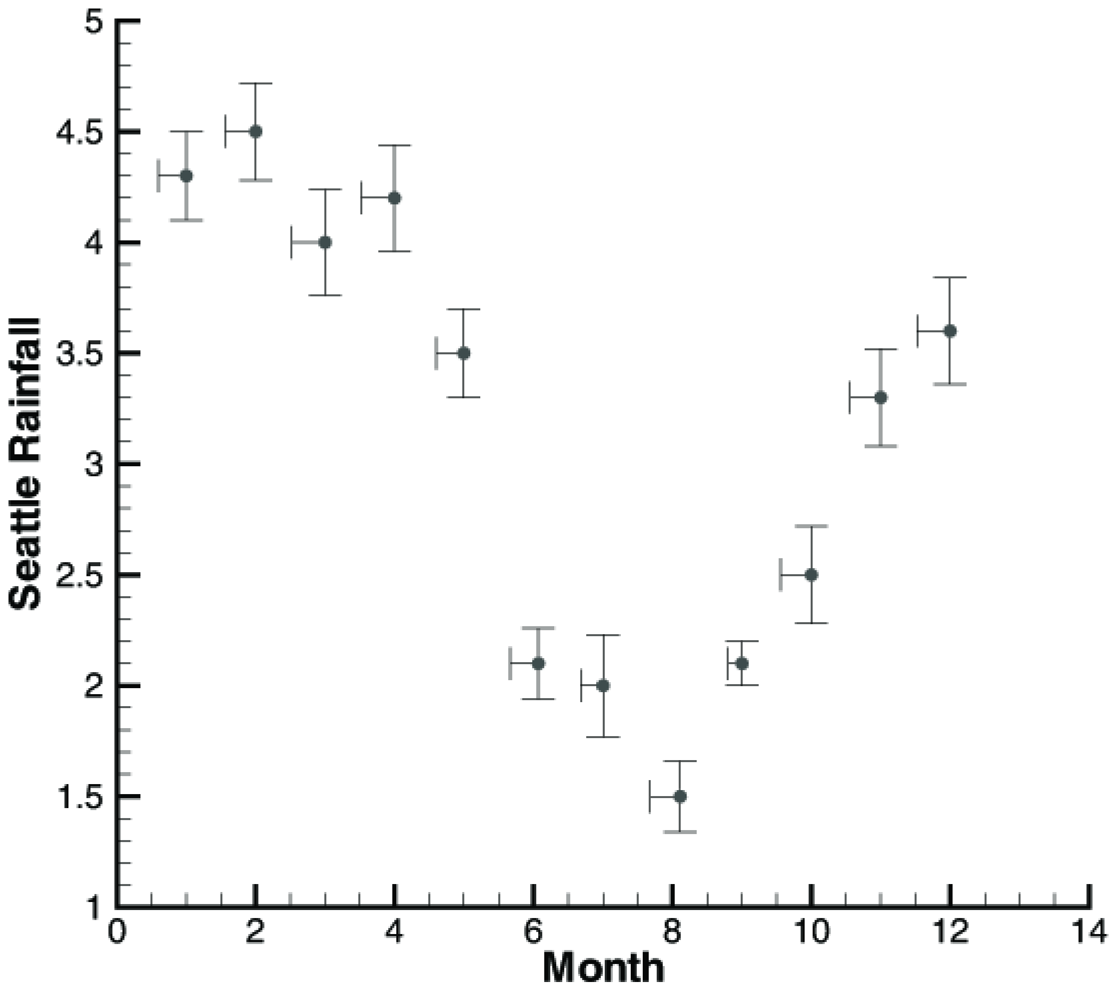


Figure 16. An XY Line plot with symbols and error bars.

You can use any variable in your dataset 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.

NOTE: If error bar values are not included in your original dataset, you may create error variables using Tecplot 360'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" via **Data→Alter→Specify Equations**. See Data Operations.

Select Variable

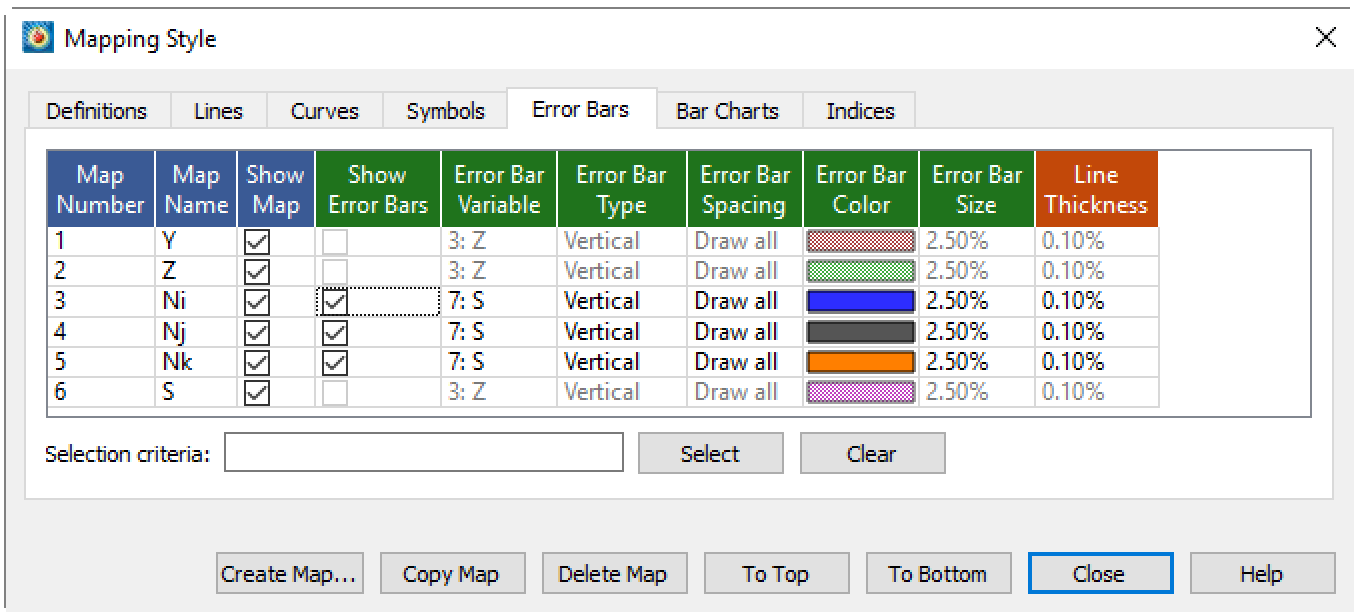
Use the **Select Variables** dialog to choose:

- A single variable, as when assigning a variable to the X or Y-axis in an mapping. The text and labels will vary with the particular action being performed, but the operation of the dialog is the same in all cases. Select a variable from the drop-down of the dataset's variables and select **OK**.

- Two variables, as when assigning 2D axis variables or choosing 2D vector components. The text and labels will vary with the particular action being performed, but the operation of the dialog is the same in all cases. For each of the two variables required, select a variable from the drop-down of the dataset's variables.
- Three variables, as when assigning 3D axis variables or choosing 3D vector components. The text and labels will vary with the particular action being performed, but the operation of the dialog is the same in all cases. For each of the three variables required, select a variable from the drop-down of the dataset's variables.

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 (shown below).



In order for the changes made on the Error Bars page to be visible in your plot, the Error Bars mapping layer must be toggled-on in the Plot sidebar.

The first two columns list the mapping number and name. The Show Map column indicates currently active mappings. These columns behave the same as the corresponding columns on the Definitions page (see Mapping Definitions). The other settings are:

Show Error Bars

Indicates whether error bars are displayed for this mapping.

Error Bar Variable

Right-click or double-click to choose the error bar variable.

Error Bar Type

Right-click to choose from seven types of error bars.

Up

Extends upward for positive values (and downward for negative values) of the error bar variable.

Down

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.

> 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.

Error Bar Spacing

Right-click to specify the spacing between error bars. 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, 3 or 4

Error bars are drawn every second, third or fourth data point.

Distance=1, 2, or 3%

Error bars are drawn at the first data point and subsequently at data points that are at least one, two or three percent of the frame height distant from the previously plotted data point.

Error Bar Color

Right-click to specify the error bar line color in the Color Chooser.

Error Bar Size

Right-click to specify the size of the crossbar. Crossbar size is measured as a percentage of frame height. You may either choose a preset value or enter an exact value yourself.

Line Thickness

Right-click to specify the line thickness of the error bars. The error bar line thickness is measured as a percentage of frame height.

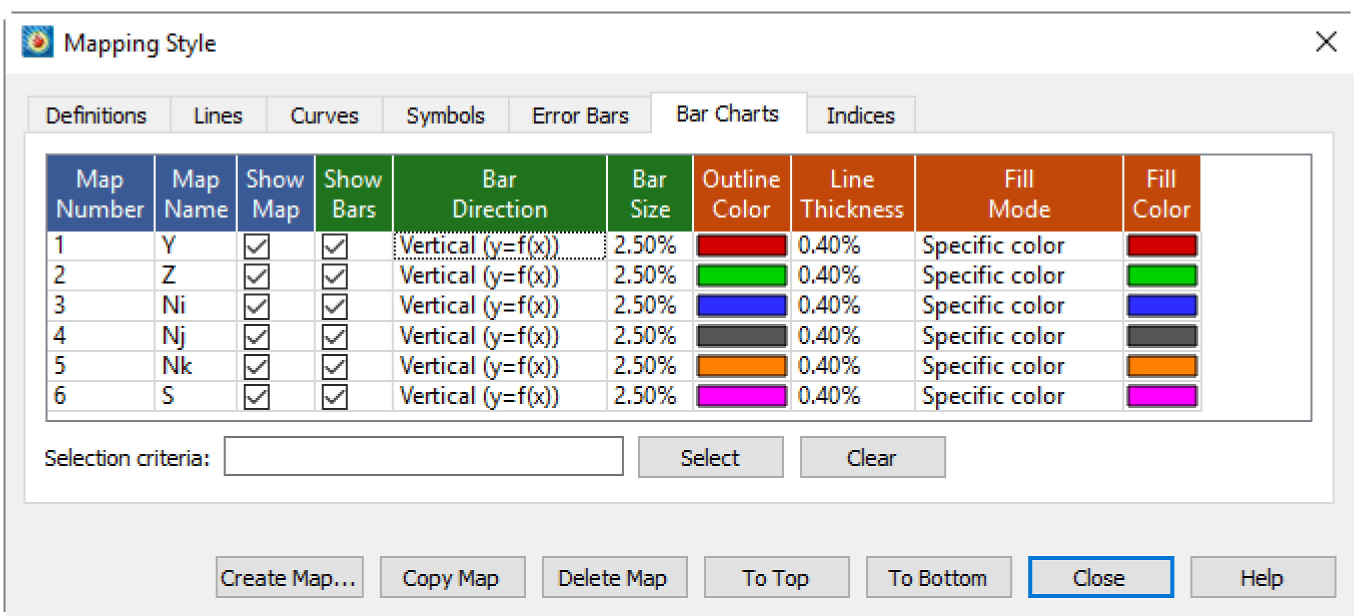
For information on using the controls at the bottom of the Mapping Style dialog to select mappings by name, see the description of these at the end of Mapping Definitions.

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. You can create bar charts by activating the Bars map layer on the Plot sidebar.

Bar Chart Attributes

The style of the bar chart is controlled on the Bar Charts page of the **Mapping Style** dialog, shown below.



The first two columns list the mapping number and name. The Show Map column indicates currently active mappings. These columns behave the same as the corresponding columns on the Definitions page (see Mapping Definitions).

Show Bars

Toggle on or off the bar chart for this mapping.

Bar Direction

Right-click to change between vertical or horizontal bars.

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 Dependent and Independent Variables.

To modify other attributes (*Bar Size, Outline Color*, *Line Thickness*, *Fill Mode*, *Fill Color*) on the Bars page, follow the same procedures used to set Symbol Attributes.

> For information on using the controls at the bottom of the Mapping Style dialog to select mappings by name, see the description of these at the end of Mapping Definitions.

I, J, and K-indices

Each mapping can show either I, J, or K-varying families of lines. By default, Tecplot 360 displays the I-varying family of lines. Figure 17 shows the family of I-varying lines for Zone 1 of the data.

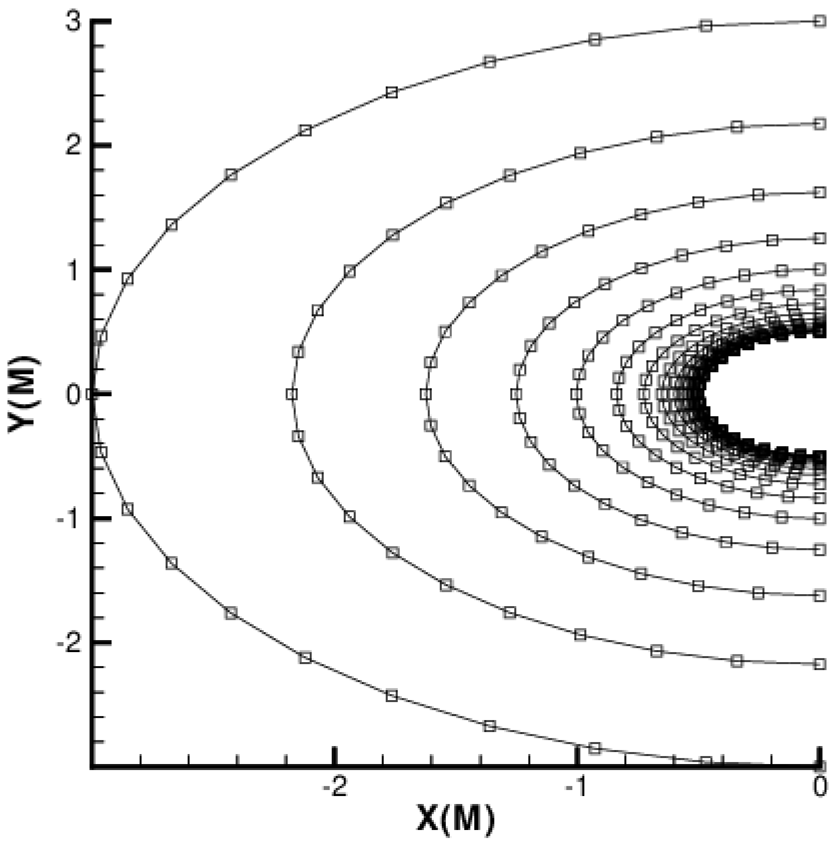
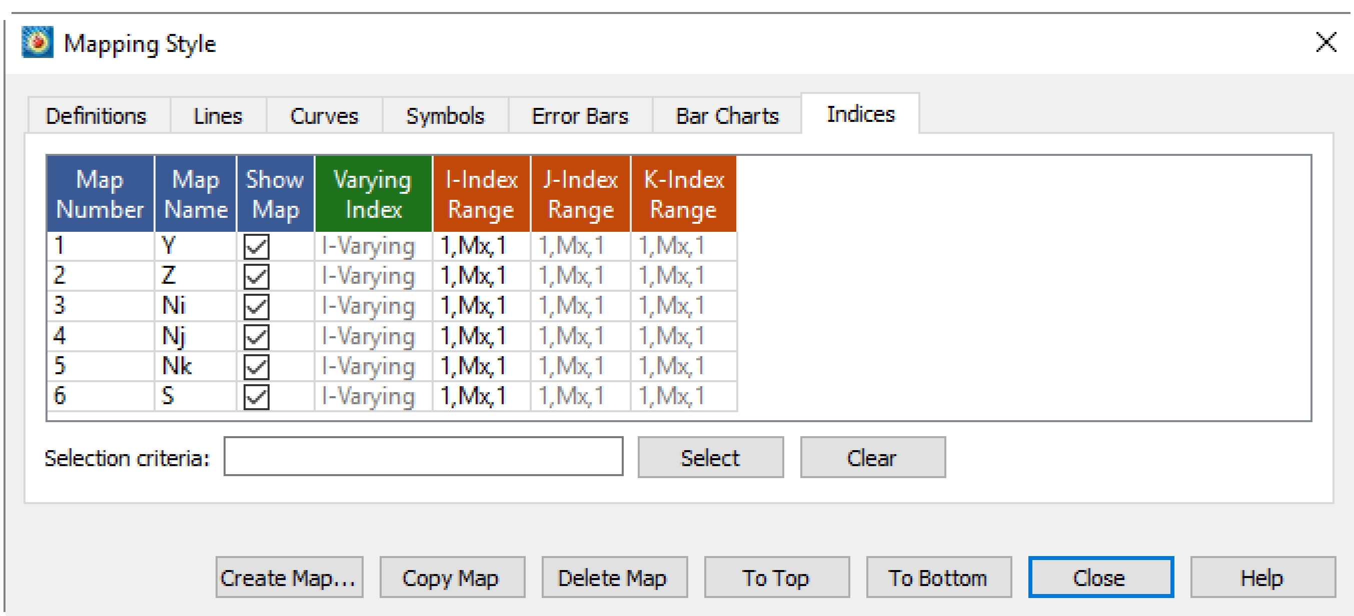


Figure 17. A family of I-varying lines for the cylinder data.

You can change the family of lines using the Indices page of the **Mapping Style** dialog as shown below.



The first two columns list the mapping number and name. The Show Map column indicates currently active mappings. These columns behave the same as the corresponding columns on the Definitions page (see Mapping Definitions).

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 360 which points to include in each line, and the index ranges for the other indices tell Tecplot 360 which lines in the family to include. Thus, you may use this option for selecting a subset of an I-ordered zone to plot.

Varying Index

To choose the varying index, and thus specify the family of lines to be drawn, right-click the Varying Index column and choose the desired family (I, J, or K-varying). K-varying is only available if the mapping is using an IJK-ordered zone.

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. Right-click and choose Enter Range to specify the range.

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.

For information on using the controls at the bottom of the Mapping Style dialog to select mappings by name, see the description of these at the end of Mapping Definitions.

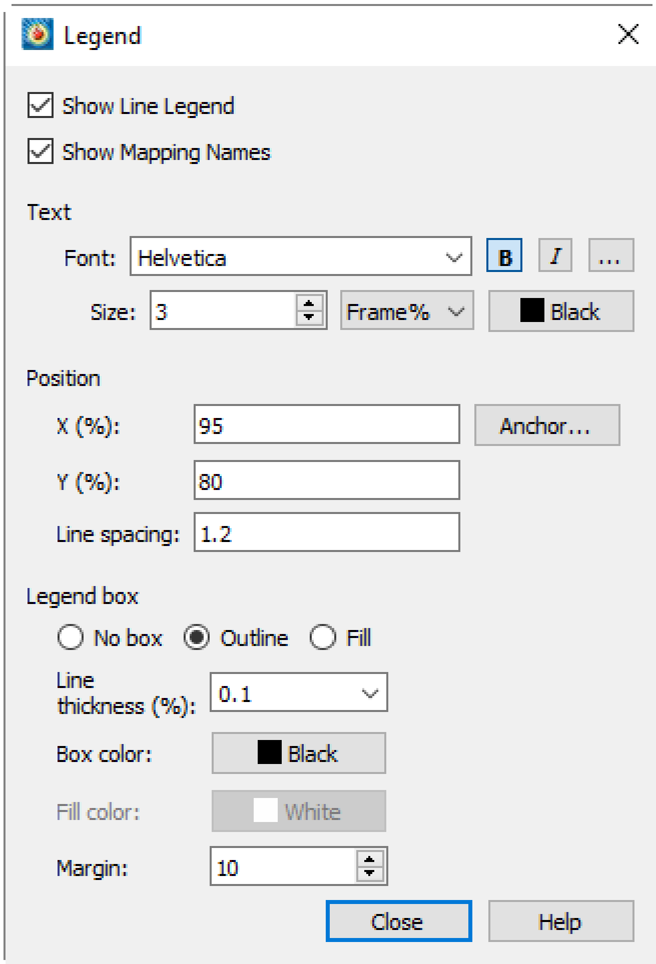
Line Legend

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 360 removes redundant entries.

To include the line plot legend, open the **Line Legend** dialog (accessed via the **Plot** menu) and toggle-on "Show Line Legend".

The **Line Legend** dialog has the following additional options:



Show Mapping Names

Toggle-on or off to include mapping names in the legend.

Text

Format the text for the legend by choosing a color, font, and size. (See Font Folders and Fallback for more information on how fonts work with Tecplot 360.)

Position

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.

Line Spacing

Spacing between items in the legend as a multiple of the font size.

Anchor

You may also specify the anchor location of the legend using the **Anchor Alignment** dialog. By default, the legend is anchored in the top right.

Legend Box

Choose No box, Outline, or Fill mode. If the legend box mode is Outline or Fill, 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

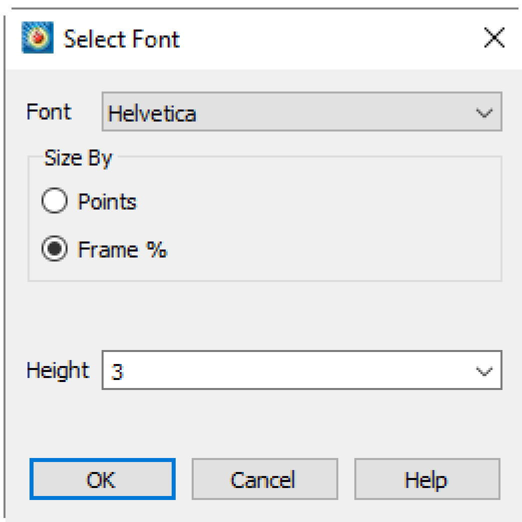
Choose a color for the legend box fill (Fill mode only).

Margin

Specify the margin between the legend text and legend box as a percentage of the text height.

Select Font

Use the **Select Font** dialog for your font preferences.



The available options are:

Font

Choose a typeface from the drop-down. Some typefaces also have bold, italic, or bold/italic variants listed.

Not all fonts have Bold and/or Italic variants. For fonts that do not have these styles, the **B** and/or *I* buttons may have no effect.

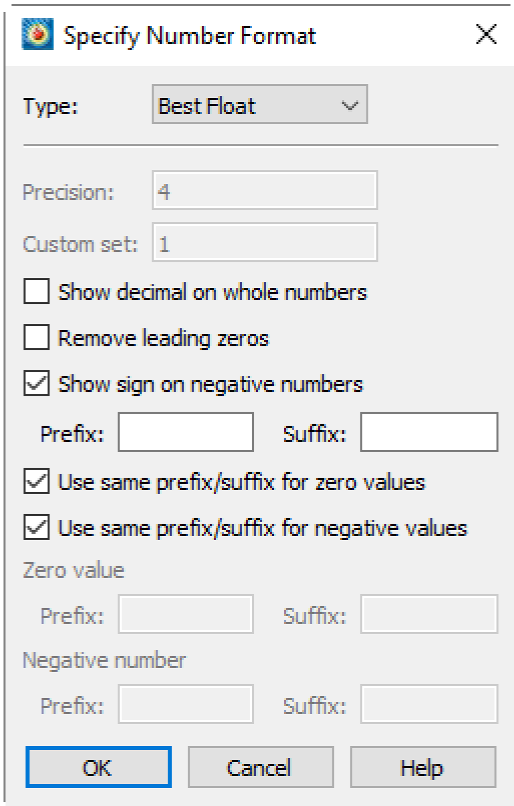
Size by

Choose to size fonts by points or percentage of frame height, or in certain cases by axis percentage.

Height

Select font height from the drop-down.

Specify Number Format



Not all fonts have Bold and/or Italic variants. For fonts that do not have these styles, the **B** and/or *I* buttons may have no effect.

Format

Choose the format for numbers in the legend from the drop-down:

Integer

Display the number as an integer; if the exact value is not an integer, it is truncated.

Float

Display the number as a floating-point number. The value is shown to the number of decimal places specified in the Precision field.

Exponent

Display the number using FORTRAN exponential format (for example, 1.0125E + 02). The number of decimal places is specified using the Precision field.

Best Float

Display the number as a floating-point number, with its exact form determined by Tecplot 360.

Range Best Float

Tecplot 360 selects the best floating-point representation of the tick mark labels, taking into account the range of values on the axis. (Available only for axis labels.)

Superscript

Display the number in scientific notation, using a number times a power of ten. The number of decimal places shown is specified using the Precision field.

Custom

Not a number format at all, Custom specifies that a set of custom labels (specified by number in the Custom Set field) should be used in the contour legend. The first label in the set is used for the value one, the second label for two, and so on. All non-integer numbers are rounded to the nearest integer. If the number of levels exceeds the number of custom labels, the labels are reused cyclically as needed. For example, if you have defined the custom labels Mon., Tue, Wed, Thu, Fri, Sat, and Sun, then a value of eight would display Mon, nine would display Tue, and so on.

Time/Date

You can specify a Time/Date format for your labels by selecting Time/Date from the Format drop-down menu. See Time/Date Format Options for more information on specifying your labels in Time and/or Date format.

Precision

(Float, Exponent, or Superscript only) - Enter the number of decimal places each number is to show.

Custom Set (Custom only)

Enter the number of the set of custom labels. You define custom label sets as records in standard Tecplot-format data files.

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 360 using the Prefix/Suffix text fields. Tecplot 360 allows you to specify separate prefixes and suffixes for zero values and negative values as well.

Field Plots

Field plots are 2D Cartesian or 3D Cartesian plots. The axes in a field plot are all independent variables. In Tecplot 360, field plots can be created using any combination of the following zone layers:

- Mesh Layer
- Contour Layer
- Vector Layer
- Scatter Layer
- Shade Layer
- Edge Layer

By default, 2D and 3D field plots are initially displayed with Mesh and Edge zone layers:

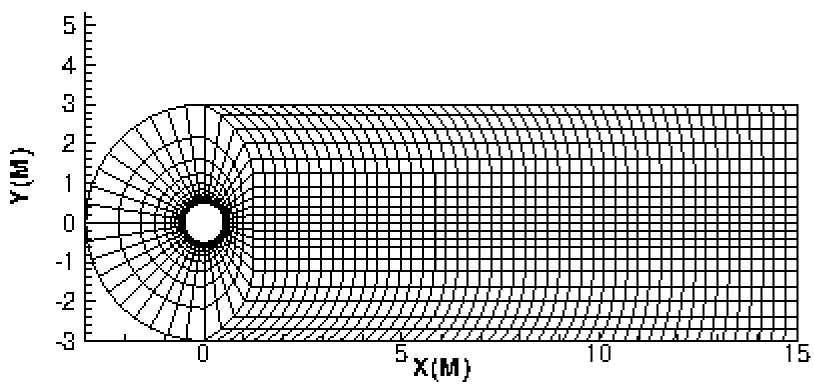


Figure 18. A 2D mesh and edge plot.

3D field plots may be enhanced with lighting effects and translucency (Translucency and Lighting).

Bounding boxes are displayed in 3D plots for volume-only zones when the plot contains only such zones and the zones themselves are not otherwise visible due to having no style. These bounding boxes may be turned off in the Options menu.

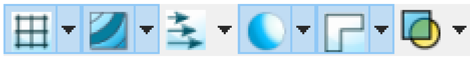
Field plots may also contain any combination of the following objects (which are derived from the values in the dataset):

- Iso-Surfaces
- Slices
- Streamtraces

This chapter discusses the plot attributes that are common to all of the plot layers.

Field Plot Modification and the Zone Style Dialog

Once you have loaded your data, you can modify field plot attributes using the **Zone Style** dialog or the context toolbar.



The context toolbar appears above the context menu when you right-click a zone in your field plot. This toolbar allows you to turn on or off the grid, contour, vector, shade, edge, and translucency layers for the selected zone(s). Additionally, you may adjust frequently-used style settings for each layer using the drop-down menu to the right of each, for example selecting a color for the grid (or choosing a variable by which to color it).

The **Zone Style** dialog may be opened by clicking the Zone Style button in the Plot sidebar or from the **Plot** menu. Initially, any zones selected in the workspace are selected in the Zone Style dialog. You may also double-click any zone in your plot to open the dialog with that zone selected.

> Field plots containing transient data work slightly differently from static datasets in the Zone Style dialog. If a zone contains no data at the current time step, that entire line in the Zone Style dialog is grayed out. See Time Aware for more information on working with transient datasets.

The following pages are available in the **Zone Style** dialog, each representing a plot layer:

Mesh

See Mesh Layer.

Contour

See Contour Layer.

Vector

See Vector Layer.

Scatter

See Scatter Layer.

Shade

See Shade Layer.

Edge

See Edge Layer.

Points

See Points.

Surfaces

See Surfaces.

Volume

See Derived Volume Object Plotting. (3D only)

Effects

See Translucency and Lighting.

The values in the **Zone Style** dialog can be modified in place. For example, some settings, such as Show Zone, are represented by checkboxes, and can simply be clicked to toggle their state. Other settings allow you to right-click to display a context menu or other method for changing them. These allow you to change the setting for multiple zones at once by selecting the desired zones (hold down Shift or Control while clicking), then right-clicking in the column for the setting you wish to change.

> Each page of the **Zone Style** dialog is divided into three color-coded regions. The blue columns apply to the zone itself and are the same on all pages of the dialog. The green and orange columns represent primary and secondary settings specific to the corresponding plot layer.

The following attributes in the **Zone Style** dialog apply to the zone in general and appear on all pages of the dialog.

Zone Number

The number of each zone.

Zone Name

The name of each zone. Strands are indicated by an "*" after the name.

> For transient data, the first zone of the strand applicable to the current time step is displayed in the Zone Name and Zone Number columns.

Group Number

Displays the zone's group number. Double-click to change the number.

Show Zone

By default, all zones are displayed. Turn zones or groups of zones on or off by toggling the checkboxes in this column on or off.

Select Zones by Pattern

Enter a wildcard pattern in the Selection criteria text box and click **Zones** or **Groups** to select one or more zones based their name or group number. Selecting Zones will match the selection criteria with the Zone Name while selecting Groups will match the selection criteria with the Group Number. In wildcard patterns, most characters match themselves, but the * and ? characters have special meaning.

Character	Meaning
?	Matches any single character.
*	Matches any number of characters, including none.

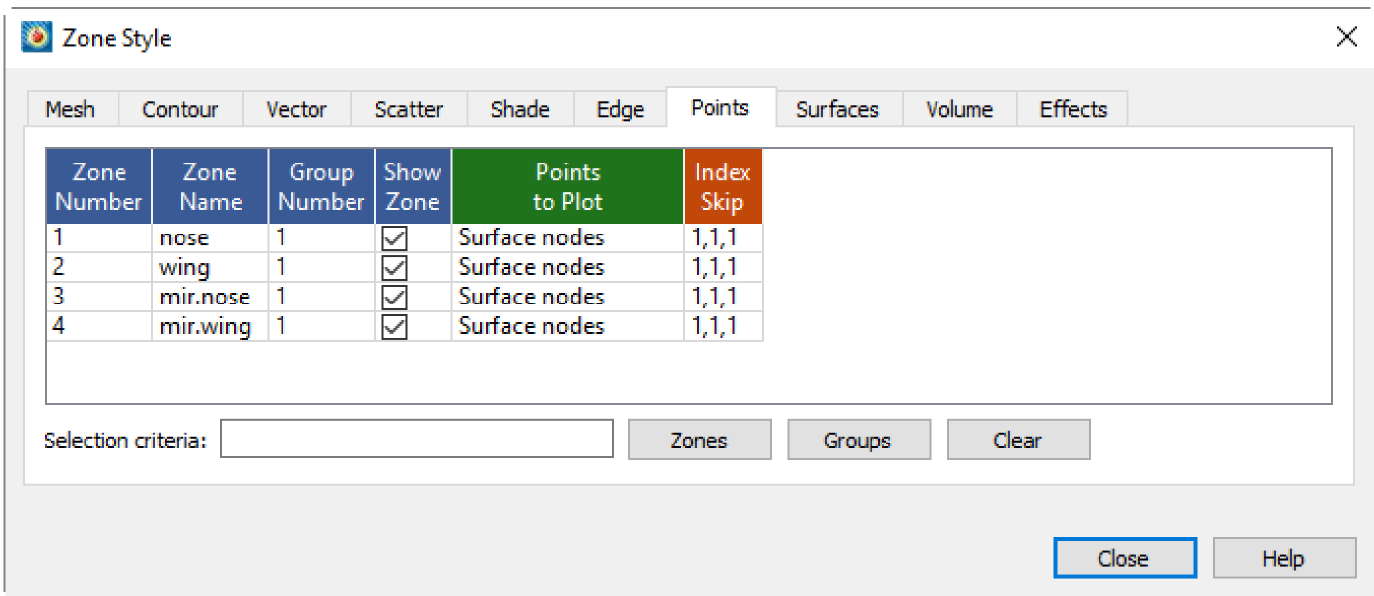
Clear

Clear the selection and the Selection criteria field.

The remaining columns in the **Zone Style** dialog are dependent upon the active page and are discussed in their corresponding sections.

Points

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 (shown below).



For information on using the controls at the bottom of the Zone Style dialog to select zones by name, see the description of these at the end of Field Plot Modification and the Zone Style Dialog.

Figure 19 shows a plot where zone 1 is plotting scatter symbols only on one plane (J=5) and zone 2 is plotting all symbols.

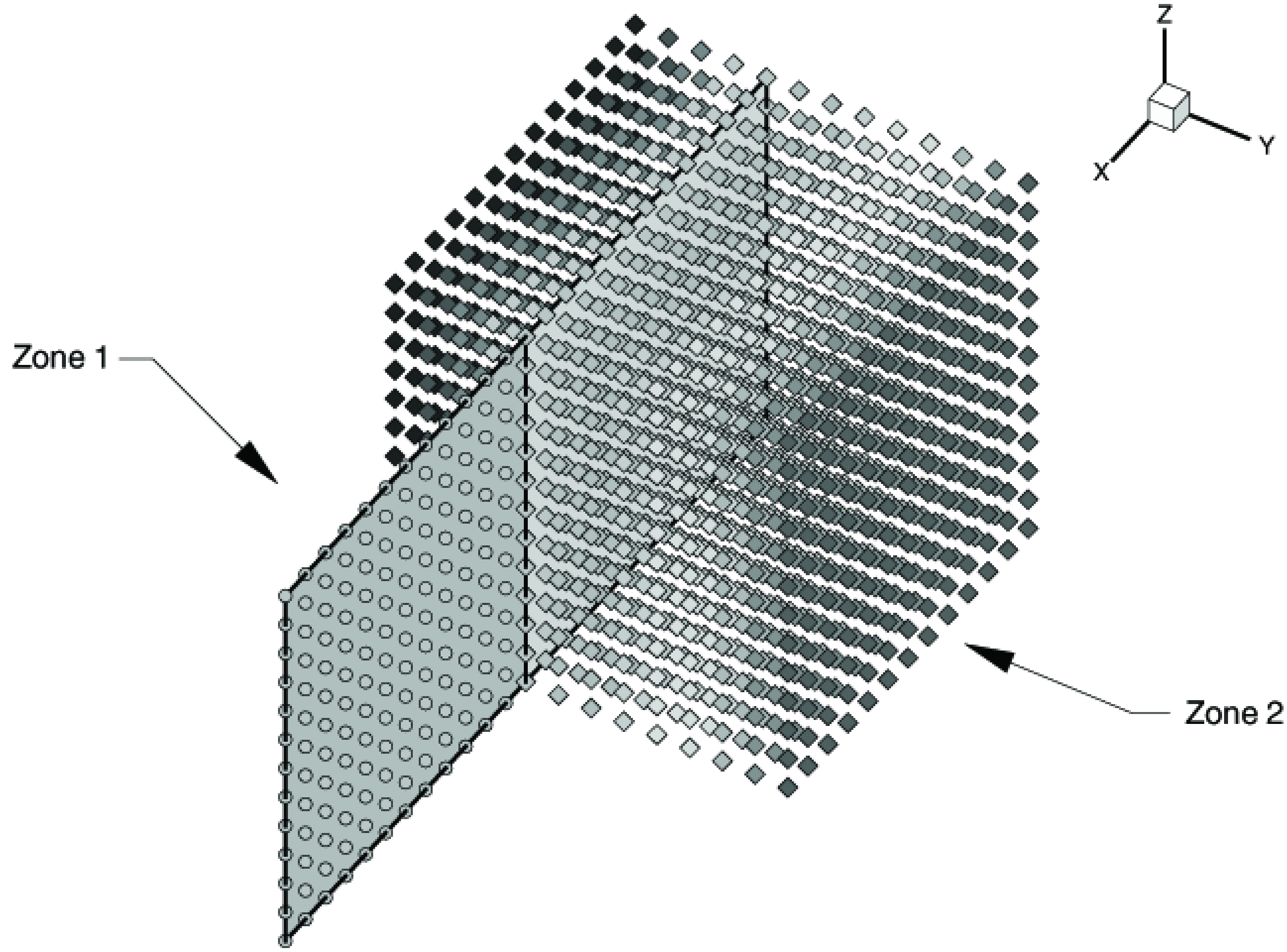


Figure 19. A plot showing two zones set to show only J-planes equal to five, with scatter symbols plotted on the surface in zone 1 and all symbols in zone 2.

Points to Plot

Right-click to select how the points are plotted:

Nodes on Surfaces

Draws only the nodes that are on the surface of the zone.

All Nodes

Draws all nodes in the zone.

All Connected

Draws all the nodes that are connected by the node map. Nodes without any connectivity are not drawn.

Cell Centers Near Surfaces

Draws points at the cell centers which are on or near the surface of the zone.

All Cell Centers

Draws points at all cell centers in the zone.

Index Skip

Right-click to specify the skip intervals for the I, J, and K-indices. The menu options are as follows:

No Skip

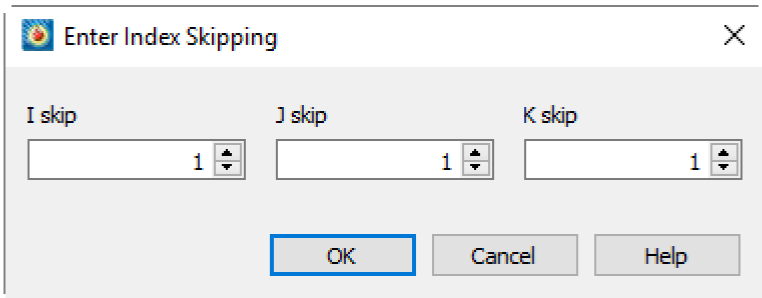
Set the I, J, and K-skip intervals to one; plot all points or vectors.

Enter Skip

Specify I, J, and K-skip intervals on the **Enter Index Skipping** dialog.

Enter Index Skipping

Use this dialog to enter the skip intervals for the I, J, and K indices.

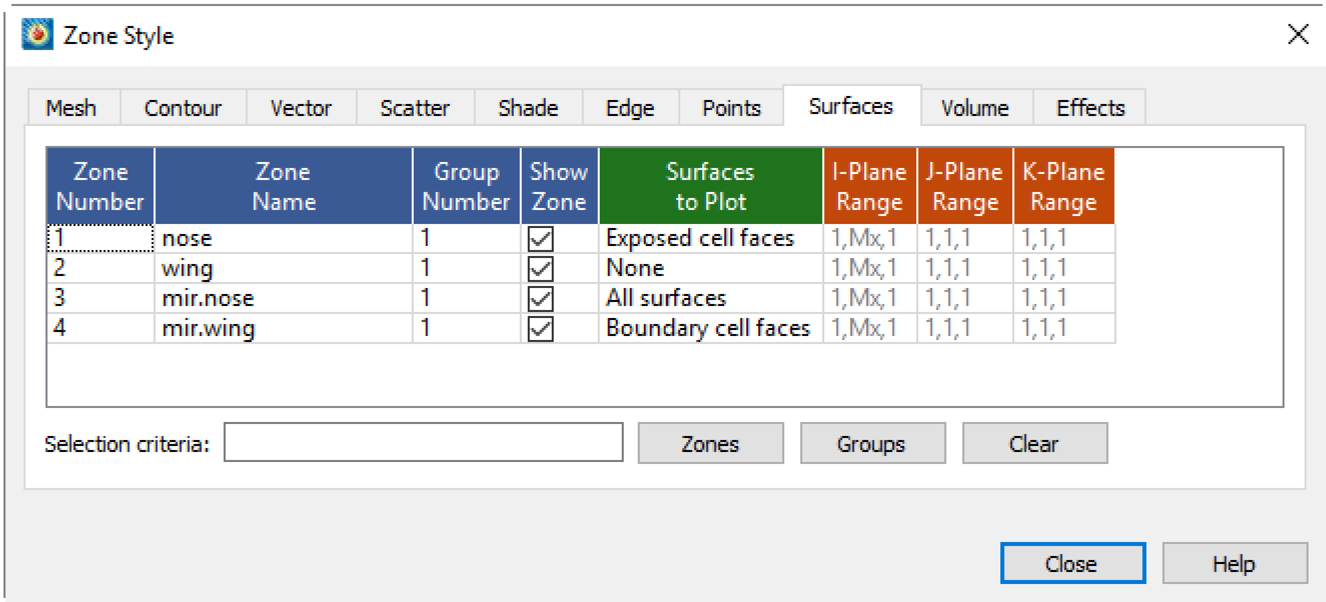


A skip value of 1 for an axis indicates that every data point along this access will be used; a skip value of 2 indicates every second data point, a skip value of 3 indicates every third data point, and so on.

> For irregular and finite element data, only the I-Skip has an effect. I-skip will allow you to skip through nodes in the order they are listed in the data file.

Surfaces

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. In Tecplot 360 you may choose which surfaces to plot for volume zones from the Surfaces page of the **Zone Style** dialog (accessed by double-clicking on a zone via the Plot sidebar, or via **Plot**→**Zone Style**.



For information on using the controls at the bottom of the Zone Style dialog to select zones by name or by group number, see the description of these at the end of Field Plot Modification and the Zone Style Dialog.

Right-click in the Surfaces to Plot column to choose one of the following:

None

None of the volume zone surfaces are plotted (edges still appear). This is the default Surfaces setting for your plot.

Boundary Cell Faces

Plots all surfaces on the outside of the volume zone. This includes:

IJK-ordered data

The minimum and maximum I, J, and K-planes are plotted.

Finite element volume data

All faces that do not have a neighbor cell (according to the connectivity list) are plotted.

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 shows plots of a volume zone with Surfaces to Plot set to "Boundary Cell Faces": without blanking, with value blanking, and with IJK-blanking. See Blanking for information on working with Blanking.

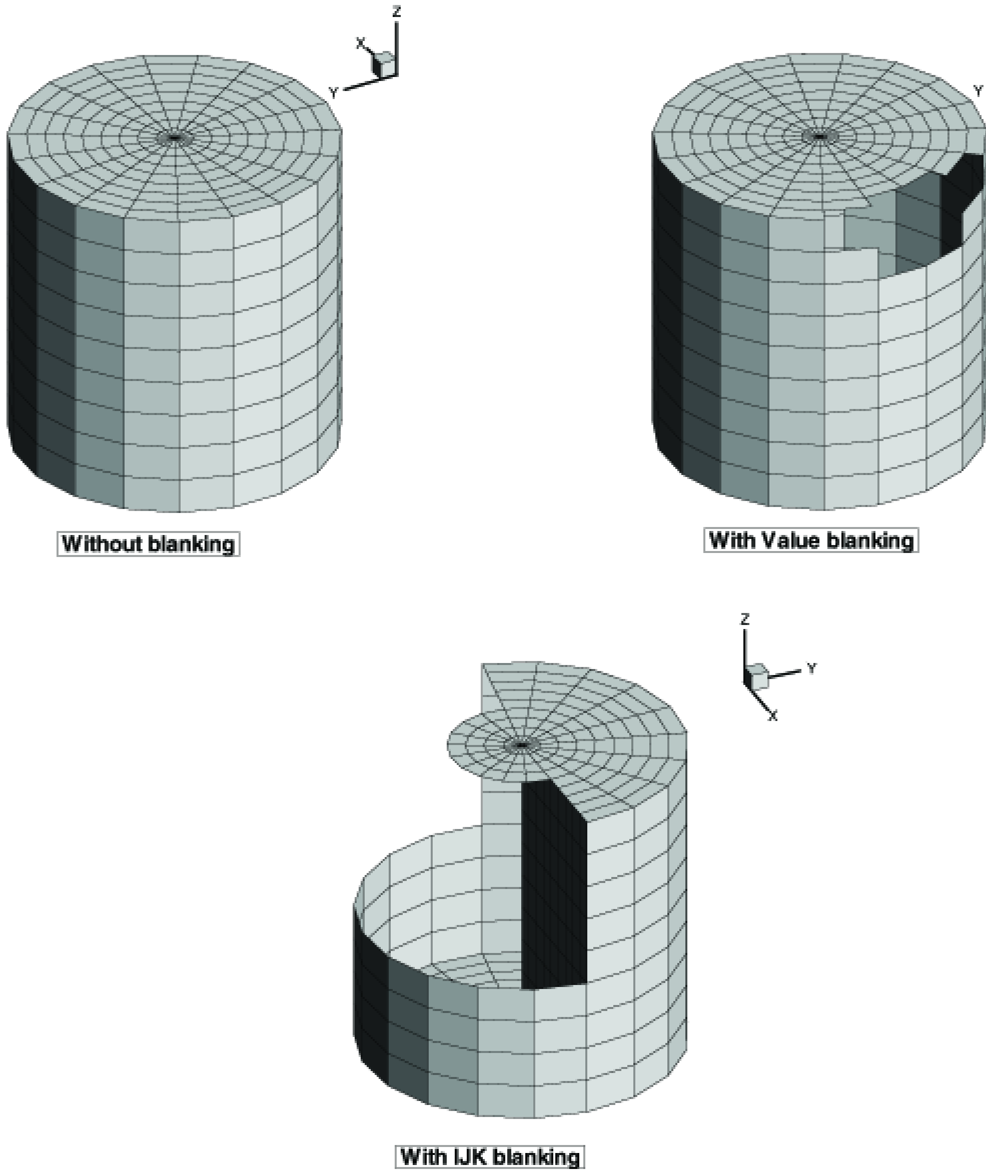


Figure 20. Boundary Cell Face plotting without blanking, with value-blanking, and with IJK-blanking.

Exposed Cell Faces (default)

This setting is similar to the "Boundary Cell Faces" setting, unless value blanking is active. When value blanking is used, the outer cell faces between blanked and non-blanked cells and the outer surfaces of the data are drawn. Figure 21 shows a plot of a volume zone with Surfaces to Plot set to "Exposed Cell Faces" with and without value blanking. See Blanking for information on working with Blanking.

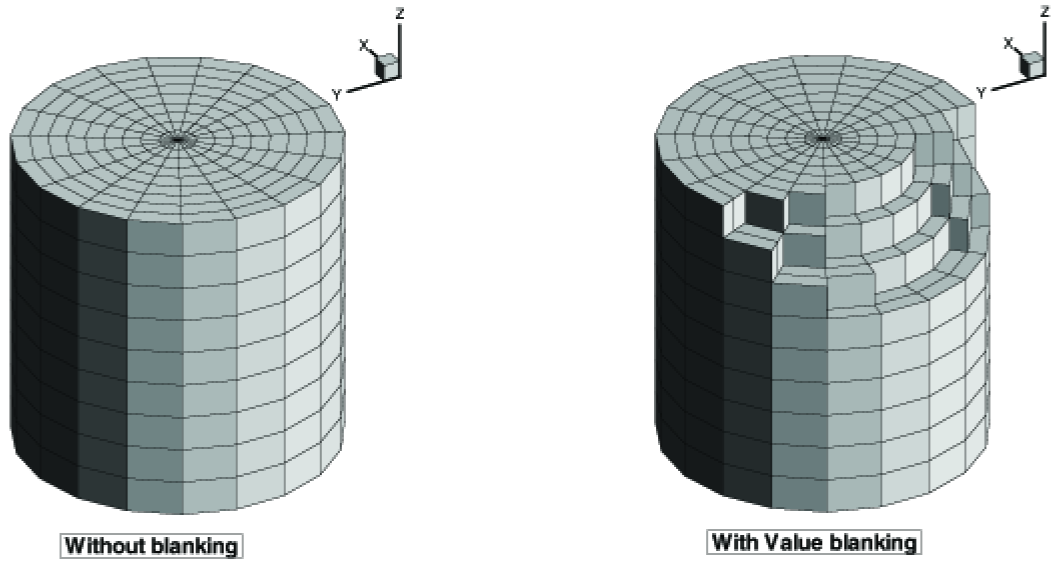


Figure 21. Examples of plots where **Surfaces to Plot** *has been set to* **Exposed Cell Faces** *with (left) and without (right) value-blanking.*

Planes Settings (I, J, K, IJ, JK, IK, and IJK-planes)

Plots the appropriate combination of I, J, and/or K-planes. The planes are determined by the Range for each plane, which can be set by right-clicking in a range column. **These settings are available only for IJK-ordered data.** Figure 22 shows a number of examples of plotting I, J, and K-planes.

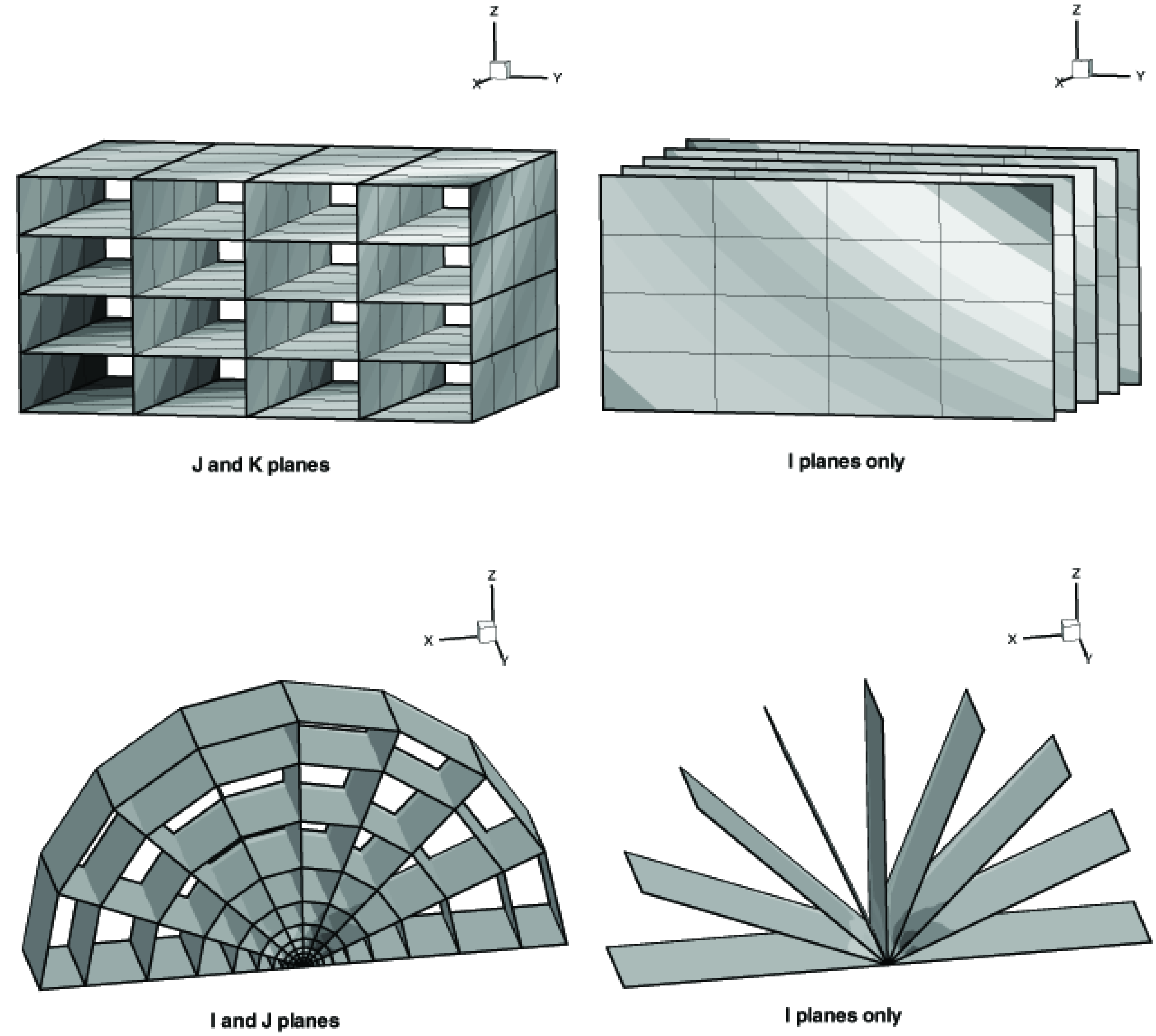


Figure 22. Examples of plotting I, J, and K-planes.

Every Surface (Exhaustive)

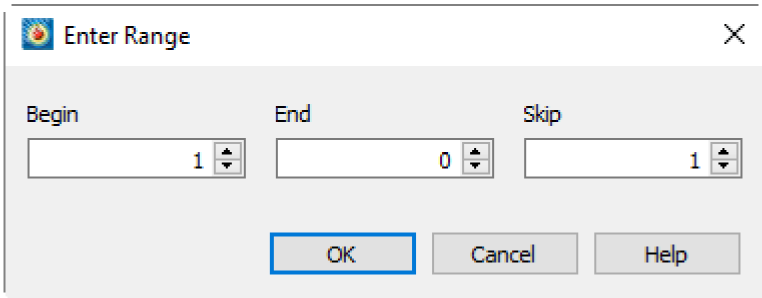
This setting will plot every face of every cell in volume data. It is not recommended for large datasets. Unless the surfaces are translucent, the plot will appear the same as the Exposed Cell Faces setting.

Ranges

The Range for I, J, or K Planes options allow you to specify the ranges for the corresponding planes. Right-click in the appropriate Range column to enter the range as the beginning plane, the ending plane (0 indicates the maximum index, -1 the next highest, and so on) and the skip factor in the **Enter Range** dialog. Use a skip factor of two to display every other plane, a skip factor of three to display every third plane, and so on.

Enter Range

The Enter Range dialog allows you to specify the I, J, or K index range in the Mapping Style or Zone Style window or when performing an integration.



Specify the beginning and end of the range. You may specify 0 to represent the maximum index, -1 to represent the next highest, and so on.

You may also specify a skip value. A skip value of 1 considers every point in the range; 2 considers every other point, and so on.

Derived Volume Object Plotting

The Volume page of the **Zone Style** dialog allows you to specify whether or not to show streamtraces, iso-surfaces, or slices for the selected zone(s). Right-click in the appropriate column to choose yes or no. Figure 23 shows a plot with two zones where streamribbons and an iso-surface have been excluded from zone 2.

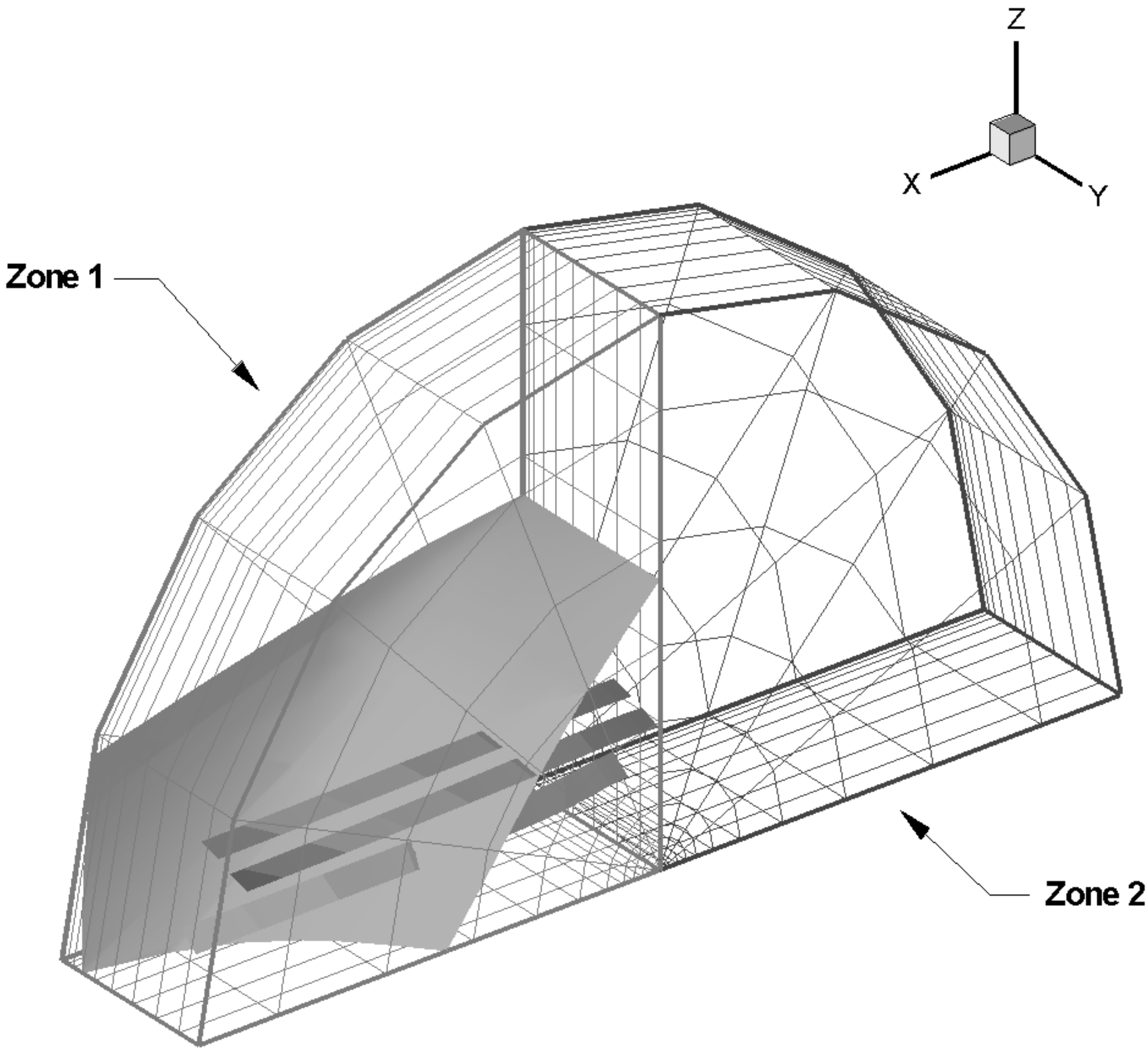


Figure 23. A plot where streamribbons and an iso-surface have been excluded from zone 2.

Time Aware

For transient datasets, you can use the Tecplot 360 interface to display your data at a given time or to animate your data over time. The zones loaded into Tecplot 360 can be linked to a specific solution time, and the active solution time is used to determine which zones are displayed.

For the following definitions, consider the following fictitious dataset:

Table 4. Sample Time Aware Dataset

Zone	Time	StrandID
1	n/a	n/a

Zone	Time	StrandID
2	0.0	2
3	0.18	3
4	0.22	1
5	0.25	2
6	0.28	1
7	0.32	3
8	0.38	2
9	0.42	1
10	0.52	1
11	0.57	2
12	0.58	3
13	0.62	1
14	n/a	n/a

Transient zones

Zones associated with time. The transient zone(s) displayed in the current frame are dependent upon the current solution time. Zones 2-13 in Table 4 are transient zones.

Static zones

Zones not associated with time. They are displayed regardless of the current solution time. Zones 1 and 14 from Table 4 are static.

Current Solution Time

The value that determines which transient zones are displayed in the current frame.

Strand

A series of transient zones that represent the same part of a dataset at different times. Zones 2, 5, 8, and 11 in Table 4 all have the same StrandID and therefore, they are part of the same strand.

StrandID

An integer value defined for each transient zone. The StrandID of a given zone is determined by the data loader.

> Changes made in the Zone Style dialog to any zone in a given StrandID are propagated to all zones with that StrandID. See also Field Plot Modification and the Zone Style Dialog.

Relevant Zone

Only "relevant zones" are plotted at a given solution time. Tecplot provides two different policies for determining what zones are relevant at any given solution time. Please refer to `$!GlobalTime TransientZoneVisibility` macro command in the Scripting Guide for more information. The default option is, `ZonesAtOrBeforeSolutionTime`. The rules for the different transient zone visibility options are as follows:

`ZonesAtOrBeforeSolutionTime`

For each strand at a given solution time, the zones shown are those that are at the solution time, within a tolerance. Except for the very first or last time step, if no zones exist at the given solution time, zones shown are those from the first prior solution time showing zones.

`ZonesAtSolutionTime`

For each strand at a given solution time, the zones shown are those that are at the solution time, within a tolerance.

Static zones are always considered relevant. Refer to Figure 24.

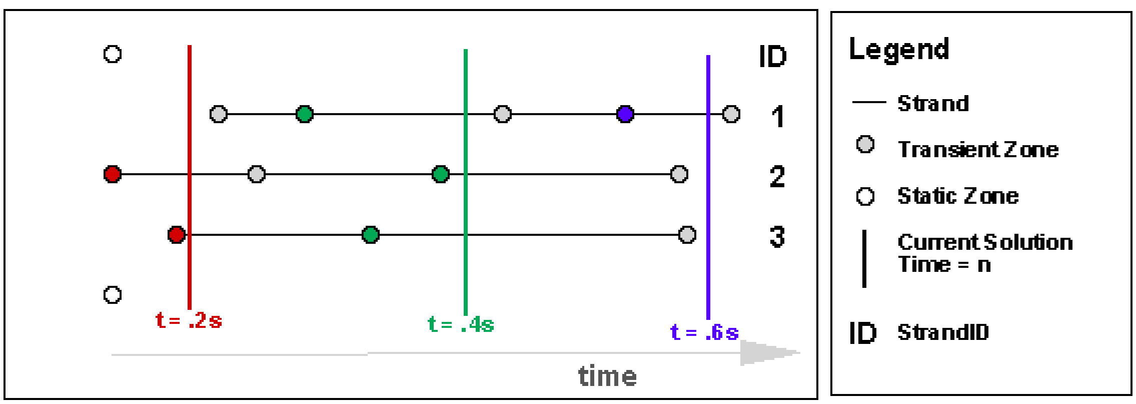


Figure 24. An illustration of how relevant zones are determined (based on the data in the figure using the default transient zone visibility, ZonesAtOrBeforeSolutionTime). For a given solution time, the relevant zones ONLY are displayed in the plot. NOTE: static zones are always considered relevant zones.

`t = .2s`

The red-colored transient zones and both static zones are plotted. NOTE: no zones from the first strand are represented because the strand is not defined at that time.

`t = .4s`

The green-colored transient zones and both static zones are plotted.

`t = .6s`

The blue-colored transient zones and both static zones are plotted. NOTE: no zones from the second and third strands are represented because the strands are not defined at that time.

Data Point and Cell Labels

Field Plots

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).

Line Plots

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 both the values (`X&Y` or `Theta & R`) for the data point. For example, Figure 25 shows an XY Line plot with each data point labeled with its X-Y value pair.

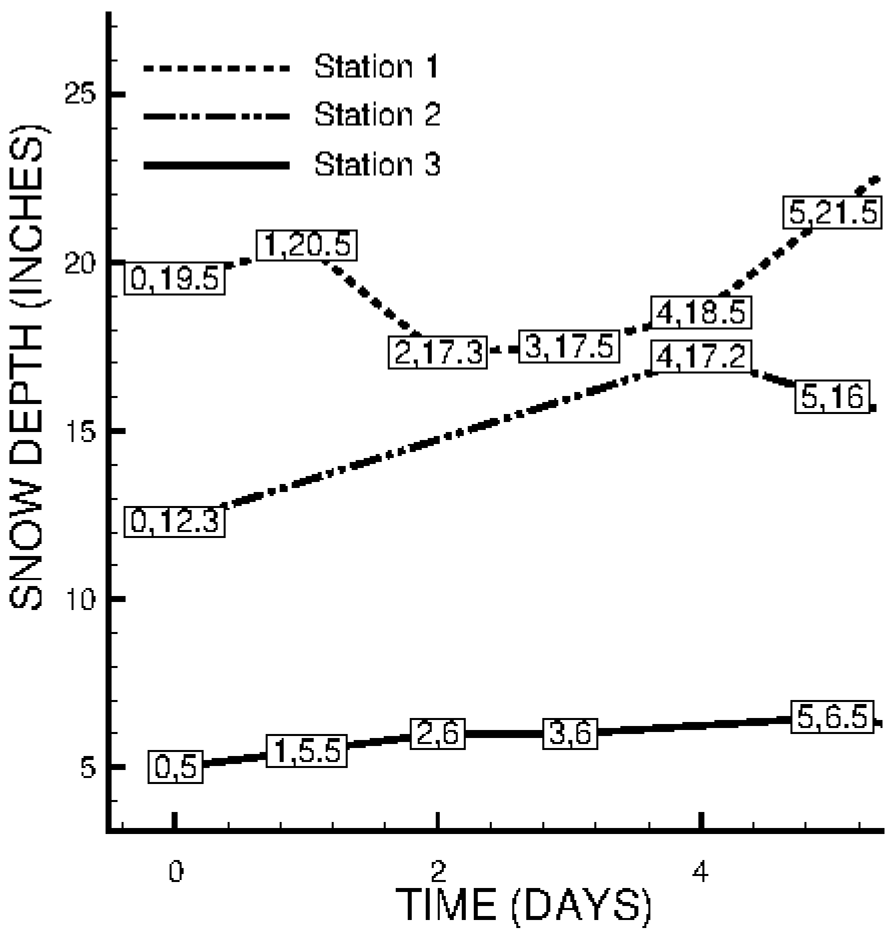
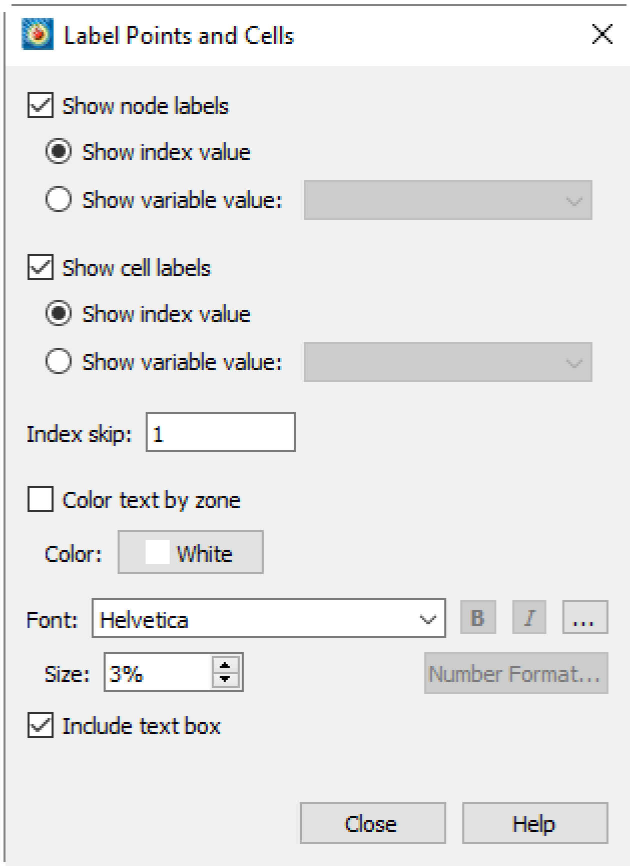


Figure 25. An XY-line plot with data labels.

To add data labels to your plot, go to **Plot**→**Label Points and Cells** dialog (accessed via the **Plot** menu). The **Label Points and Cells** dialog has the following options:



Show Node Labels

Toggle-on to show node labels. Select either Index Value or Variable Value.

> Index values are not displayed for subzone data sets (`.szplt`) since these values cannot currently be reliably determined with this file format.

Show Cell Labels

Toggle-on to show cell labels. Select either Index Value or Variable Value.

Index Skip

If labeling by index values, select an index skip.

Color Text by Zone/Map

Toggle-on and choose the color, font, font size, and number format for the labels.

Include Text box

Toggle-on "Include Text Box" to include a box around each label.

Three-dimensional Plot Control

You can view any type of data as a 3D plot by selecting 3D Cartesian from the plot type drop-down menu in the Plot sidebar. IJK-ordered data and finite element volume data are displayed in 3D automatically.

Three-dimensional plots can be manipulated with the following controls, which can be accessed via the **Plot** menu:

Reset 3D Axes
: Reset the 3D axis sizes and the 3D origin of rotation.

Three-dimensional Axis Limits
: Control the data and axis aspect ratios for 3D plotting.

Three-dimensional Orientation Axis
: Control the optional 3D orientation axis, which displays the current orientation of the three axes in the workspace.

Light Source
: Control the light source position, as well as the intensity of the light, the background light, and the surface color contrast. See Three-dimensional Light Source for more details.

Advanced 3D Control
: Specify the default lift fraction for 3D lines, symbols, and tangent vectors, as well as the 3D sorting algorithm for the plot.

The following controls can be accessed via the **View** menu.

The Rotate Dialog
: Control the 3D orientation of the plot.

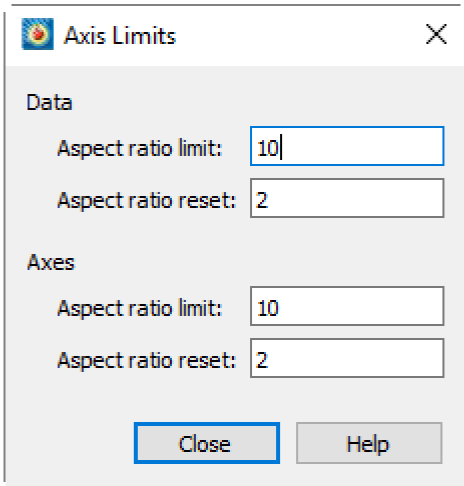
Three-dimensional View Details
: Set the specifications for parameters affecting the 3D display of your plot, including the perspective, field of view, angular orientation of the plot, and view distance.

Reset 3D Axes

By default, the 3D axes are calculated to fit the data. If you alter your data to expand or contract the overall data size, the axes will not automatically adjust to the new size. Use the Reset 3D Axes option (accessed via the **Plot** menu) to reset the axes to fit the data.

> The Reset 3D Axes option also resets the 3D origin. If you have modified your 3D origin using the **3D Rotate** dialog (see The Rotate Dialog for details), the **Reset 3D Axes** option will reset it to approximately the centroid of the data.

Three-dimensional Axis Limits



In a 3D plot, whenever you read a data file, manipulate the values of variables assigned to the axes, or change variables assigned to the axes, Tecplot 360 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.

Because there are many valid forms in which the data could be plotted, by using the **3D Axis Limits** dialog (accessed via the **Plot** menu), you can input information to help Tecplot 360 determine how to automatically configure the plot the way you want.

Aspect Ratio

Ratio of the range of the variable assigned to one axis (multiplied by the axis size factor) and the range of the variable assigned to another axis (multiplied by the axis size factor).

Data Aspect Ratio Limit

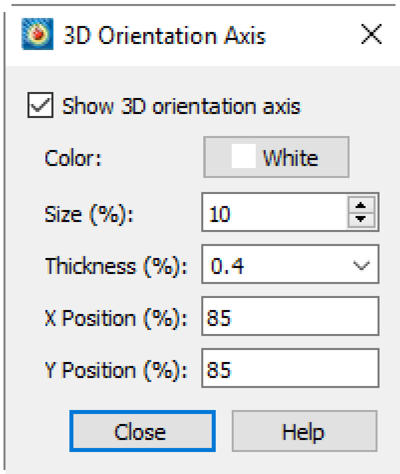
When the data aspect ratio of any two axes exceeds the Data Aspect Ratio Limit, Tecplot 360 automatically rescales the longer axis so that the new data aspect ratio is equal to the Data Aspect Ratio Reset value.

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. Use a smaller number for evenly scaled axes.

Axes Aspect Ratio Limit

Works similarly to the Data Aspect Ratio Limit, except Axes Aspect Ratio Limit attends to the shape and size of the axes box.

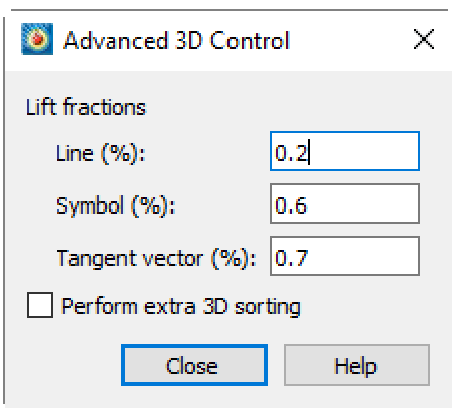
Three-dimensional Orientation Axis



The 3D orientation axis is a representation of your axes that immediately shows you the orientation. By default, all 3D plots show the 3D orientation axis in the upper right of the frame.

Using the **3D Orientation Axis** dialog under the **Plot** menu, you can control whether the 3D 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 3D orientation axis by simply clicking on it and dragging the axis to the desired location in the frame.

Advanced 3D Control



Lift Fractions

The lift fraction is the fraction of the distance from the 3D origin of the object to your eye. If you specify lift fractions for 3D lines, tangent vectors, or scatter symbols, plotted objects of the appropriate type are lifted slightly towards you so that they lie on top of surface elements.

Perform Extra 3D Sorting

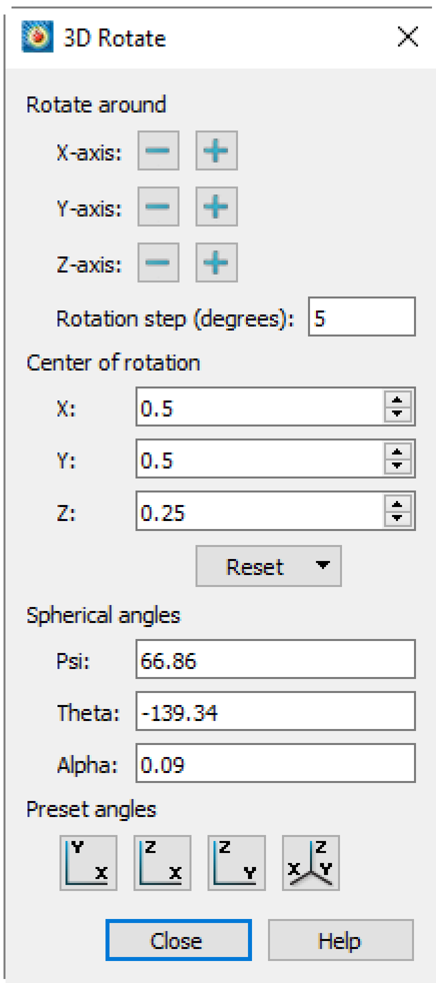
For some 3D plots (i.e. plots with translucency), Tecplot 360 uses a painter's algorithm. A quick sorting algorithm is used by default. The data objects are divided into smaller objects. The smallest object is usually a cell, finite element, vector, or scatter symbol. These objects are sorted based upon the distance from viewer, starting with the objects farthest from the viewer and working forward. This does not detect problems such as intersecting objects. If the "Perform Extra 3D Sorting" check box is selected, a slower, more accurate approach is used to detect problems for you.

There are instances when Tecplot 360 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 360 draws only whole elements, one of these elements will be drawn last and each will cover (incorrectly) a portion of another element. If this occurs while printing or exporting, choosing an image format will often resolve the problem

All of the settings in the **Advanced 3D Control** dialog are specific to the current frame.

The Rotate Dialog

You may rotate your 3D plots using the **3D Rotate** dialog accessible via **View**→**Rotate**. The 3D rotation tools located in the toolbar are discussed in Three-dimensional Rotation.



The **3D Rotate** dialog has the following options:

Rotate around

- Click the + or - buttons next to an axis to rotate the plot incrementally around that axis.
- The Rotation Step field specifies the increment by which each click of the + or - button rotates the plot.

Center of Rotation

X

the X-axis. Enter a value in the text field, or use the increase or decrease arrows at the right to specify a value.

Y

Rotation of the eye/origin ray about the Y-axis. Enter a value in the text field, or use the increase or decrease arrows at the right to specify a value.

Z

Rotation of the eye/origin ray about the Z-axis. Enter a value in the text field, or use the increase or decrease arrows at the right to specify a value.

Reset

Use this drop-down to set the center of rotation to be the Center of Data (the center of the bounding box of the data) or Center of View (the point hit by a probe at frame coordinates 50%, 50%).

> Center of View can result in an error if there is no data in the center of the frame. In this situation, the center of rotation will not move.

Spherical Angles

Eye origin view. The angular orientation of the plot is defined by three spherical rotation angles:

ψ (Psi)

Tilt of eye origin ray away from Z-axis. (Range –720 to 720.)

θ (Theta)

Rotation of the eye origin ray about the Z-axis. (Range –720 to 720.)

α (Alpha)

Twist about the eye origin ray. (Range –720 to 720.)

The eye origin ray is a line from the origin of the 3D 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 26.

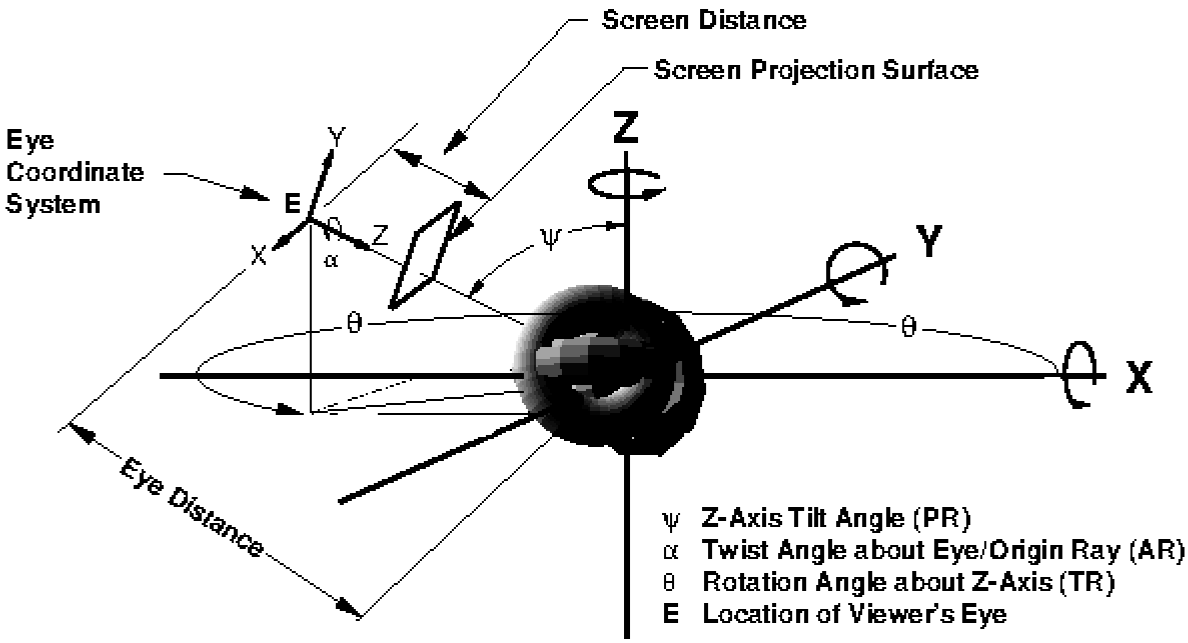


Figure 26. 3D angles and 3D projection.

Preset Angles

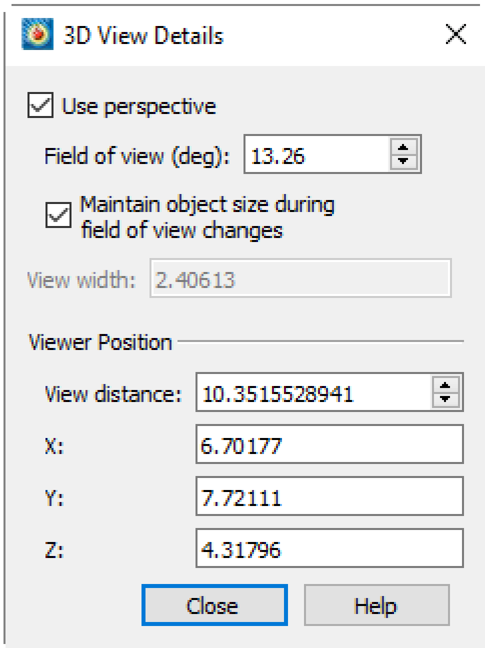
Specify one of four pre-defined orientations: the XY-Plane, the YZ-Plane, the XZ-Plane; or a default orientation with a Psi=60, Theta=225, and Alpha=0).

Rotate About the Viewer Position

In addition to the rotation capabilities described above, you may also 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, then turn your head using the `Alt` key and left mouse button.

Three-dimensional View Details

Use the **3D View Details** dialog (accessed via the **View** menu) to control a variety of parameters affecting the display of 3D plots.



Use Perspective

Sets Tecplot 360's projection type. If selected, Tecplot 360 draws the current frame with perspective projection. If not selected, Tecplot 360 draws the current frame with orthographic[7] projection. (Range is 0.1 to 179.9.)

Field of View (deg)

Sets the amount of the plot (in terms of spherical arc) in front of the viewer that may be seen. Zooming in or out of a 3D perspective plot changes this number and the viewer's position.

Maintain Object Size During Field of View Changes

If selected, Field of View changes result in the viewer's position being moved so that approximately the same amount of the plane is visible after the change.

If not selected, Field of View changes do not change the viewer's position and result in the entire plot appearing to grow or shrink.

View Width

Sets the amount of the plot (in X-axis units) in front of the viewer that may be seen. Zooming in or out of a 3D orthographic plot changes this number, but not the viewer's position. Not available when using perspective.

Viewer Position

Change the viewer's relation to the image by resetting the X, Y, or Z-location, or by changing the view distance.

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 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 by dragging with the middle mouse button magnifies the plot. When the plot projection is perspective, zooming by dragging 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 or while using the 3D mouse.

> Working with very large datasets may result in slow zooming, rotating, and translating. See Performance Dialog for further information on plot approximation if zoom, rotate, or translate performance is poor.

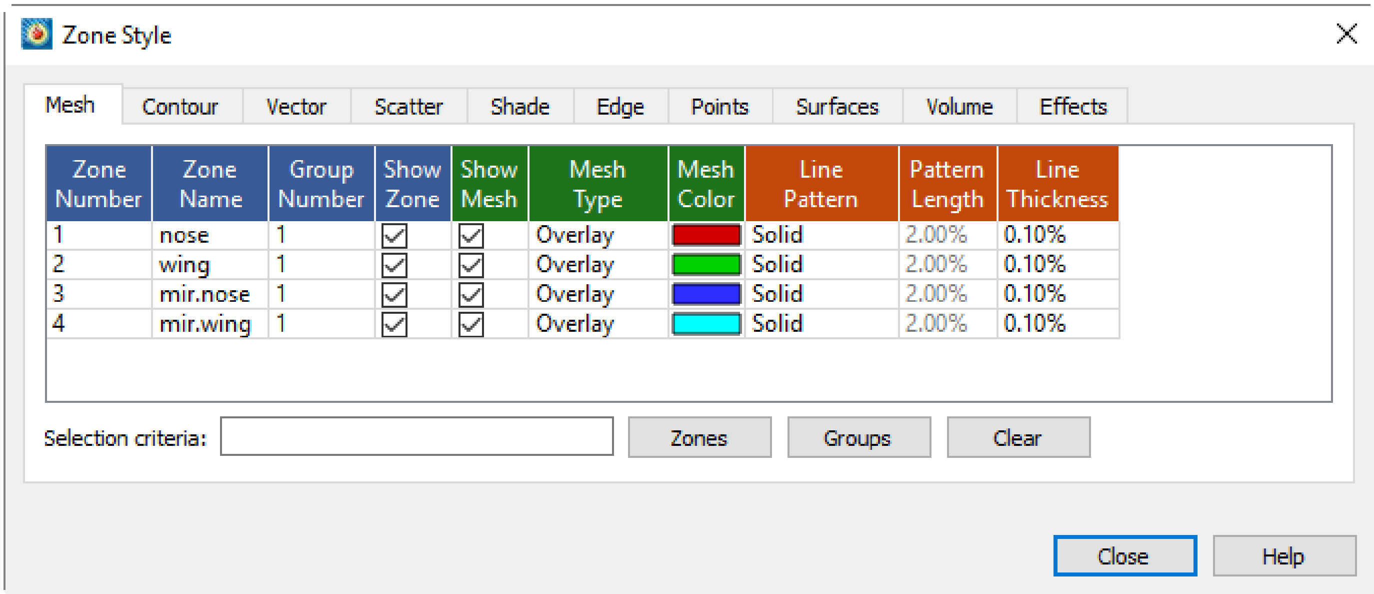
Mesh Layer and Edge Layer

When working with two or three-dimensional field plots, Tecplot 360 allows you to interactively add or subtract any combination of plot layers. These layers can be applied to any set of zones in the active data set. This chapter discusses the Mesh Layer and the Edge Layer.

Mesh Layer

Toggle-on "Mesh" in the Plot sidebar to add a mesh layer to your plot. The mesh plot layer displays 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. For finite element zones, the mesh is a plot of every edge of all of the elements that are defined by the connectivity list for the node points. See Data Structure for an in-depth description of ordered (IJK) and finite element data structures.

Mesh Layer Modification



Once you have loaded your data, you can modify your mesh plot attributes using the Mesh page of the **Zone Style** dialog (accessed via the Plot sidebar or **Plot**→**Zone Style**). As discussed in Field Plot Modification and the Zone Style Dialog, the changes made using the first four (blue) columns apply to the entire plot, while changes in the other columns apply to the active plot layer.

> In order for the changes made on the Mesh page to be visible in your plot, the Mesh layer must be turned on. You can turn on the Mesh layer using the checkbox in the Plot sidebar.

Show Mesh

Checkbox that determines whether the mesh is visible for each active zone.

Mesh Type

Right-click to choose wire frame, overlay, or hidden line. See Mesh Types below.

Mesh Color

Right-click to choose the mesh color using the Color Chooser.

Line Pattern

Right-click to choose the line pattern for the mesh.

Pattern Length

Right-click to choose the pattern length in percentage of frame height. You may choose a preset pattern length or enter your own.

Line Thickness

Right-click to choose the mesh line thickness. You may choose a predefined setting or enter your own.

> For information on using the controls at the bottom of the Zone Style dialog to select

zones by name, see the description of these at the end of Field Plot Modification and the Zone Style Dialog.

Mesh Types

Tecplot 360 has three distinct mesh types:

Wire Frame

Wire frame meshes are drawn below any other zone layers on the same zone. In 3D Cartesian plots, no hidden lines are removed. For 3D volume zones (finite element volume or IJK-ordered), the full 3D 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 Surfaces for further details. By default, only the mesh on exposed cell faces is shown.

Overlay

Similar to Wire Frame, mesh lines are drawn over all other zone layers except for vectors and scatter symbols. In 3D Cartesian plots, the area behind the cells of the plot is still visible (unless another plot type such as contour flooding prevents this). As with Wire Frame, the visibility of the mesh is dependent upon your choice of "Surfaces to Plot" on the Surfaces page of the **Zone Style** dialog. See Surfaces for further details.

Hidden Line

Similar to Overlay, 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 3D volume zones, using this plot type obscures everything inside the zone. If you choose this option for 3D volume zones, then choosing to plot every surface (using the Surfaces page of the **Zone Style** dialog) has the same effect as plotting only exposed cell faces, but is much slower.

The opaque surfaces created by Hidden Line are not affected by the Lighting Zone effect (there is no light source shading). However, it is affected by translucency.

Figure 27 shows the available mesh plot types, along with the effects of choosing Overlay and Wire Frame in combination with contour flooding.

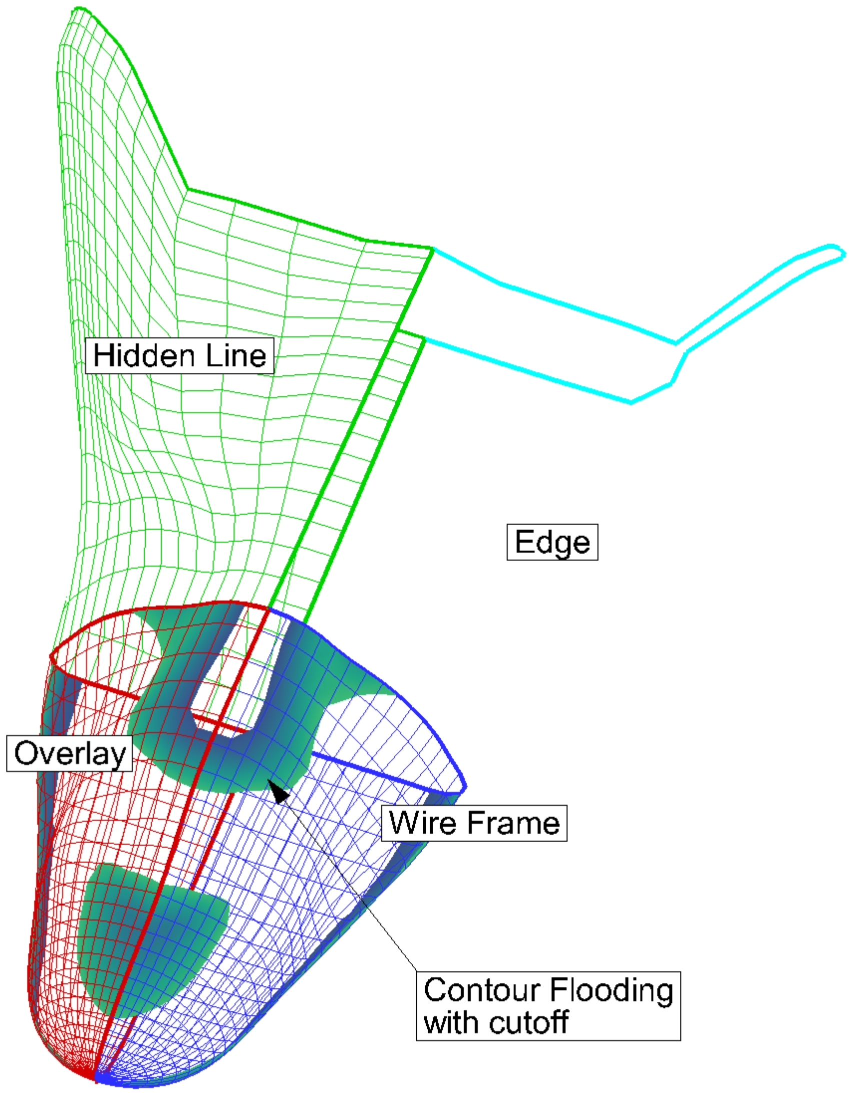


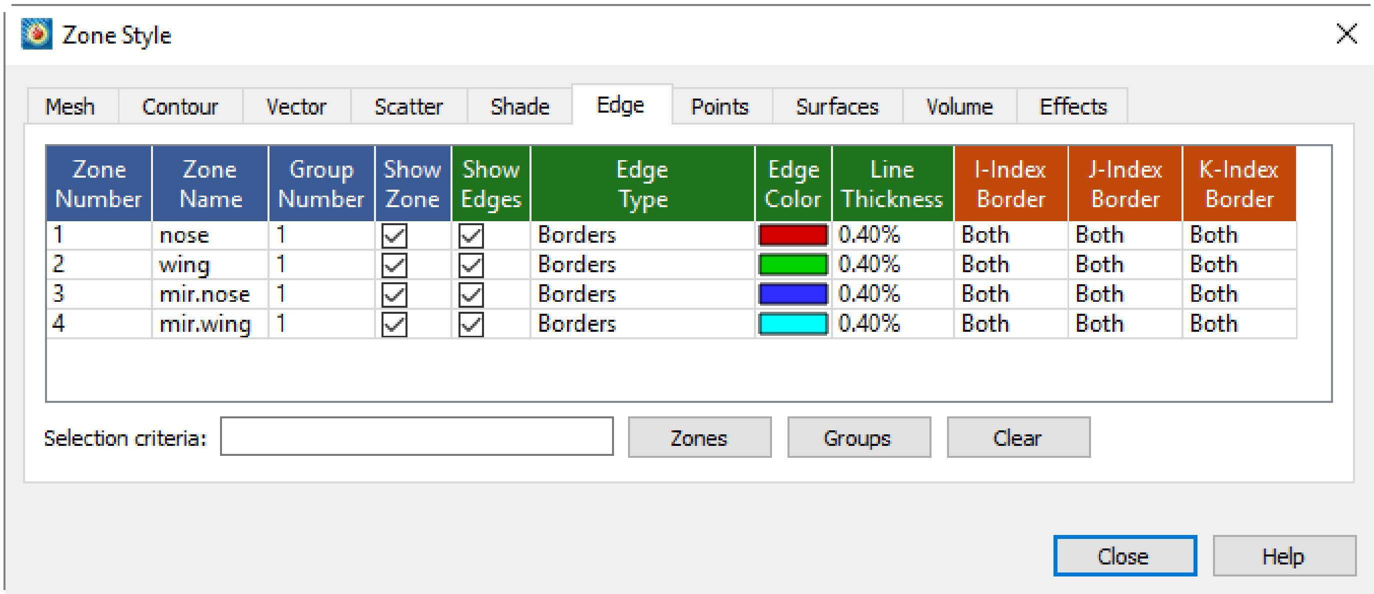
Figure 27. Mesh plot types.

Edge Layer

An edge plot layer displays the connections of the outer lines (IJ-ordered zones), finite element surface zones, or planes (IJK-ordered zones). The Edge layer allows you to display the edges (creases and borders) of your data. Zone edges exist only for ordered zones or 2D finite element zones. Three-dimensional finite element zones do not have boundaries.

Edge Layer Modification

You can control the following attributes from the Edge page of the **Zone Style** dialog:



In order for the changes made on the Edge page to be visible in your plot, the Edge layer must be turned on. You can turn on the Edge layer using the checkbox in the Plot sidebar.

Show Edges

Checkbox that determines whether the edges are visible for each active zone.

Edge Type

Right-click to choose borders and/or creases. See Edge Types below.

Edge Color

Right-click to choose the edge color using the Color Chooser.

Line Thickness

Right-click to choose the edge line thickness. You may choose a predefined setting or enter your own.

Index Borders

Right-click to choose whether to show the corresponding index border: None, Min Only, Max Only, or Both (Min and Max).

For information on using the controls at the bottom of the Zone Style dialog to select zones by name, see the description of these at the end of Field Plot Modification and the Zone Style Dialog.

Edge Types

There are two types of edges in Tecplot 360: creases and borders. An edge border is the boundary of a zone. An edge crease appears when the inside angle between two cells is less than a user-defined limit.

The inside angle can range from 0-180 degrees (where 180 degrees indicates coplanar surfaces). The default inside angle for determining an edge crease is 135 degrees. You can change the crease angle by going to **Plot**→**Edge Details**.

For 2D plots, only edge borders are available, and for FE-volume zones, only edge creases are available.

You can change the Edge Type using the Edge Type column on the Edge page of the **Zone Style** dialog.

Edge Display

For IJ-ordered zones, the available edges 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 edges of the surface areas form 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 on the Surfaces page of the **Zone Style** dialog is set to one of the planes options, such as I, J, or K-planes, for IJK-ordered zones the edges are the edges of each plane (I, J, or K-plane). By default, all available edges are drawn when the Edge layer is active. You can specify which of the available edges are plotted using the **Zone Style** dialog.

Contour Layer

Contour plots can be used to show the variation of one variable across the data field. To add a contour layer to your plot, toggle-on "Contour" in the Plot sidebar.

Contour plots can only be plotted with organized data, such as IJ-ordered, IJK-ordered, or FE-data. Refer to Working with Unorganized Datasets for information on organizing your dataset.

Additional options can be set on the Contour & Multi-Coloring Details (accessed via  to the right of Contour in the Plot sidebar or **Plot**→**Contour/Multi-Coloring**) and the Contour page of the **Zone Style** dialog.

An example of each contour plot type is shown in Figure 28.

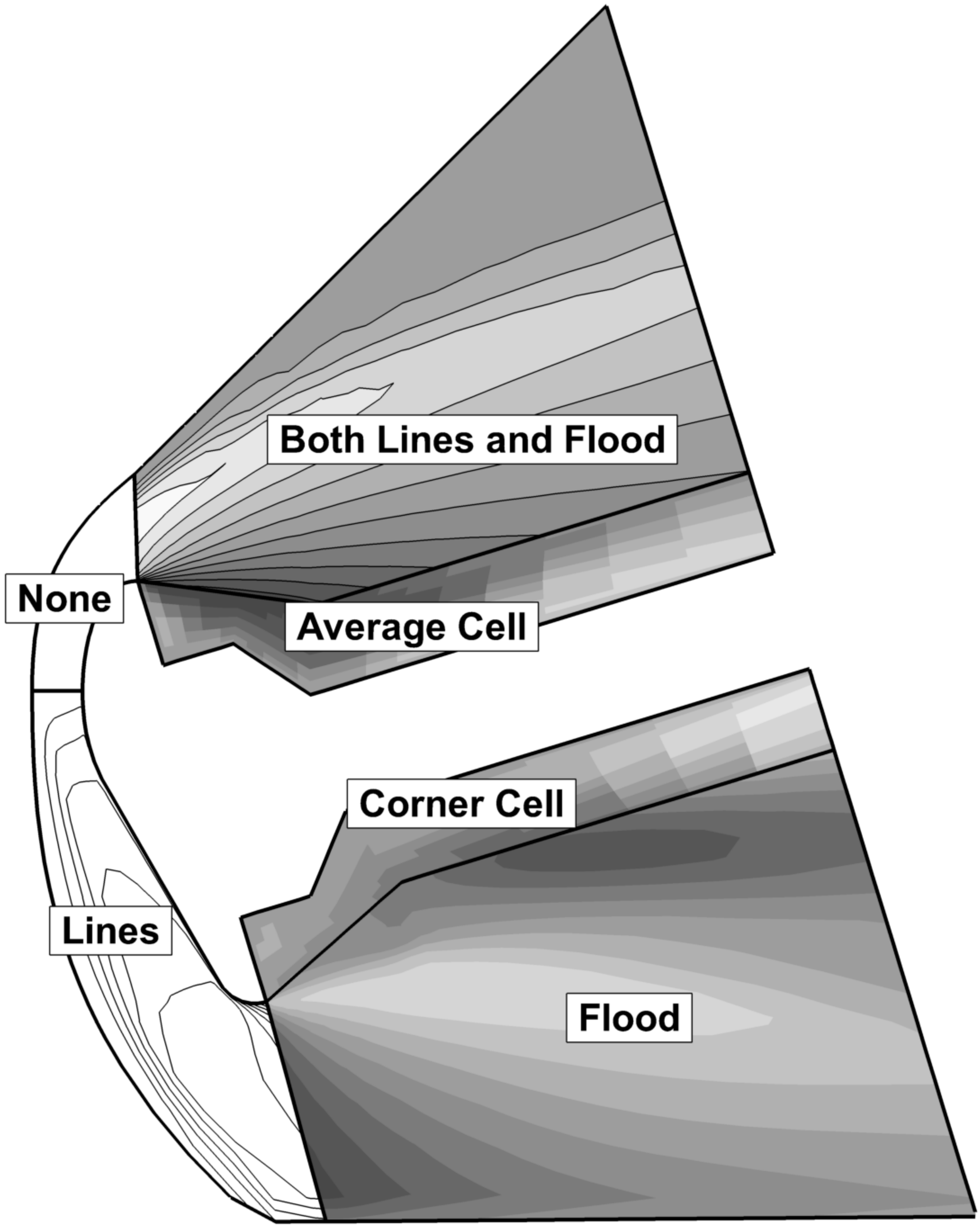


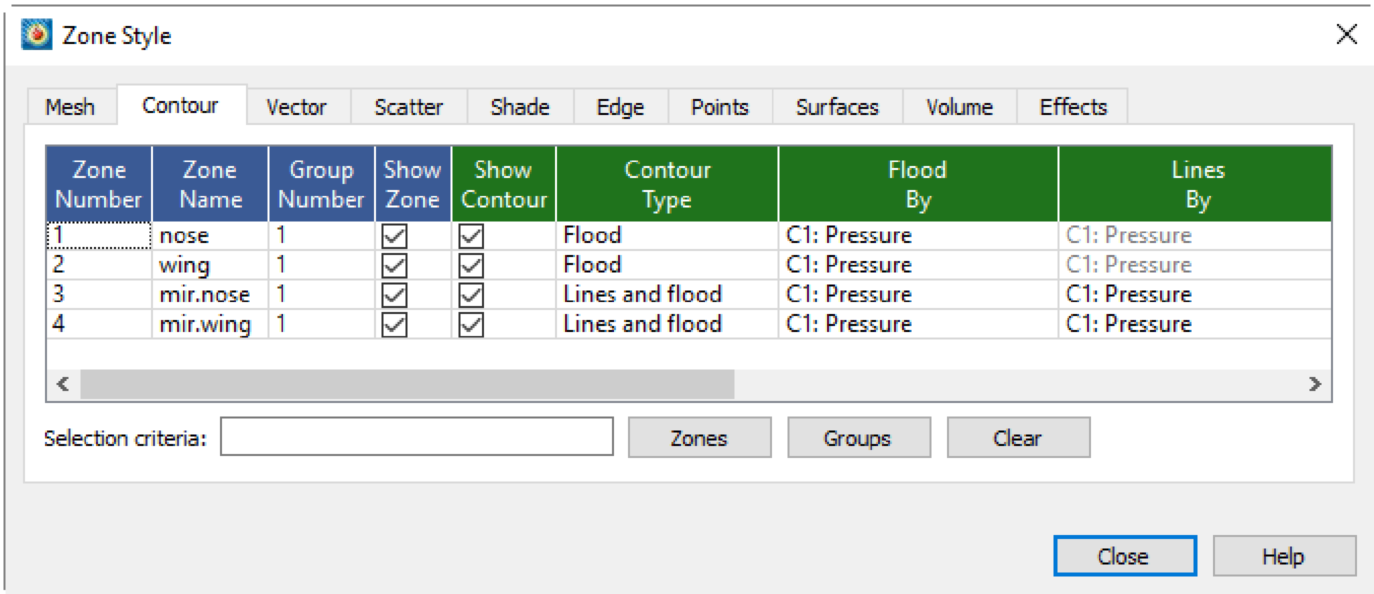
Figure 28. Contour plot types.

Contour plots for streamtraces, iso-surfaces, and slices are controlled by their respective details dialogs and are not discussed here. (Refer to Rod/Ribbon Page, Iso-Surface Contour and Shade and Contour Page, respectively.)

Contour Layer Modification

You can modify the following attributes of your contour plot using the Contour page of the **Zone Style** dialog.

In order for the changes made on the Contour page to be visible in your plot, the Contour layer must be active. You can turn on the Contour layer in the Plot sidebar.



Show Contour

Toggle-on to show the contour for the selected zone(s).

Contour Type

Right-click to choose the contour plot type:

Lines

Draws lines of constant value of the specified contour variable.

Flood

Floods regions between contour lines with colors from a color map.

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 Contour & Multi-Coloring Details for details.

Both Lines and Flood

Combines the above two options.

Average Cell Flood

Floods cells or finite elements with colors from a color map according to the average value of the contour variable over the data points bounding the cell. If the variables are located at the nodes, the values at the nodes are averaged. If the variables are cell-centered, the cell-centered values are averaged to the nodes and the nodes are then averaged.

Go to **Data→Data Set Info** to determine whether the variables are nodal or cell-centered.

Primary Value Flood

Floods cells or finite elements with colors from a 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.

If the variables are located at the nodes, the value of the lowest indexed node in the cell is used. When plotting IJK-ordered, FE-brick or FE-tetra cells, each face is considered independently of the other faces. You may get different colors on the different faces of the same cell.

If the variables are cell-centered, the cell-centered value is used directly. When plotting I, J, or K-planes in 3D, 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.

Go to **Data → Data Set Info** to determine whether the variables are nodal or cell-centered.

Flood By

Right-click to select either a contour group (C1, C2, C3, C4, C5, C6, C7, or C8) or assign variables to the RGB color map. For contour groups, the associated variable is shown in this column and the menu next to the group number. See Contour Groups and RGB Coloring for more information. Applicable only when the contour type includes a flood.

Lines By

Right-click to select which contour group identifies the contour lines (the variable associated with each contour group is also shown). Applicable only when the contour type includes lines.

Line Color

Right-click to choose the contour line color using the Color Chooser.

Line Pattern

Right-click to choose the line pattern for the mesh.

Pattern Length

Right-click to choose the line pattern length as a percentage of frame height. You may choose a preset pattern length or enter your own.

Line Thickness

Right-click to choose the contour line thickness. You may choose a predefined setting or enter your own.

Line Color

Coose the contour line color.

Use Lighting (3D only)

Turn on or off the lighting effects. See Translucency for more information on lighting effects.

Options such as contour labels, contour legends, and special settings for contour bands or contour lines are set by the selected contour group (see Contour Groups).

> For information on using the controls at the bottom of the Zone Style dialog to select zones by name, see the description of these at the end of Field Plot Modification and the Zone Style Dialog.

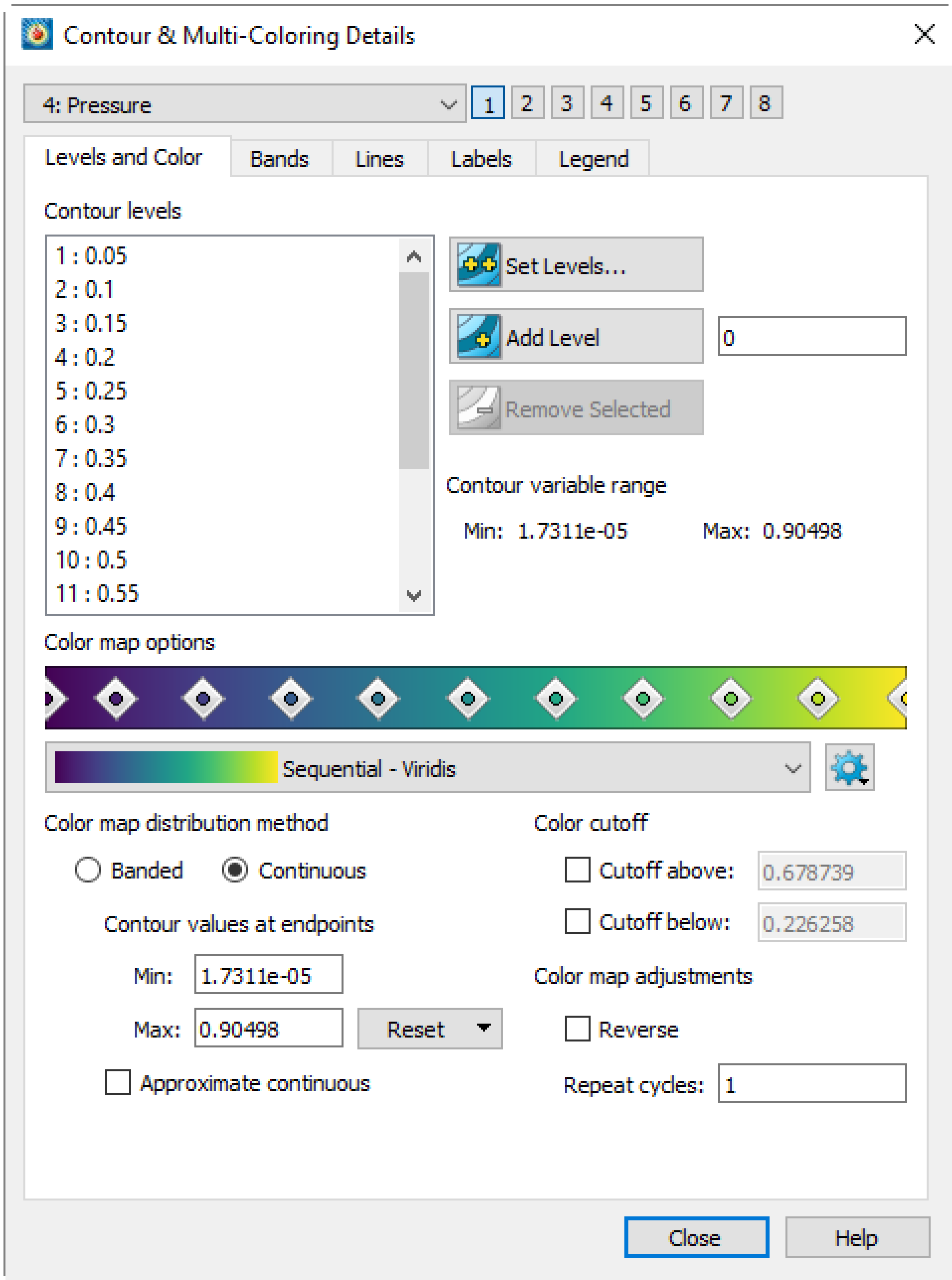
Contour & Multi-Coloring Details

Opening the **Contour & Multi-Coloring Details** dialog from **Plot**→**Contour/Multi-Coloring** or from the Plot sidebar lets you specify:

- Contour Groups
- Contour Levels and Color
- Contour Bands
- Contour Lines
- Contour Labels
- Contour Legend

Contour Groups

The **Contour & Multi-Coloring Details** dialog is shown below. This dialog is divided into five pages. The controls at the top for choosing a contour group and variable are available regardless of what page of the dialog is displayed.



Variable

Assign a variable from your dataset to the active Contour Group (1, 2, 3, 4, 5, 6, 7, or 8). The variable selected here will be contoured subject to the controls in the dialog.

1, 2, 3, 4, 5, 6, 7, 8

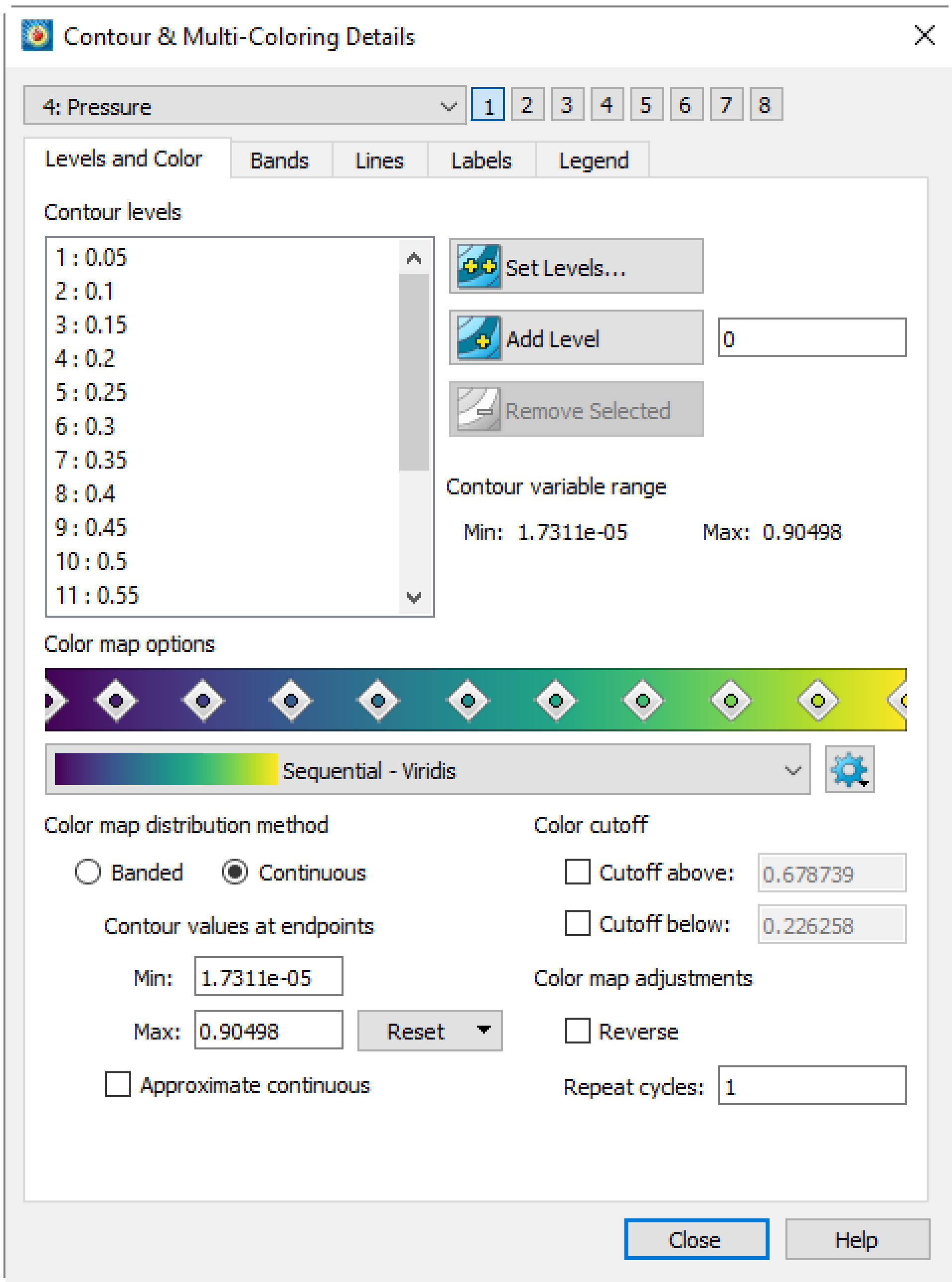
Use the 1, 2, 3, 4, 5, 6, 7, and 8 buttons to specify attributes for a specific contour group. Each contour group has its own settings for the contour attributes established in the dialog.

The Contour Group Variables (1-8) can be used to color contour, mesh, scatter, or vector zone layers, as specified in the Color Chooser dialog and the Flood By and Lines By columns on the Contour page of the **Zone Style** dialog.

Contour Levels and Color

A contour level is a value at which contour lines are drawn, or for banded contour flooding, the border between different colors of flooding. Adjust contour levels using the Levels and Color page of the **Contour & Multi-Coloring Details** dialog (accessed via the Plot sidebar or the **Plot** menu, or from the Contour Page of the Slice Details dialog).

From the Levels and Color page of the dialog, you can add, subtract, and rearrange contour levels using the top part of the dialog. The bottom part of the dialog allows you to choose or modify the color map to be used for coloring the contour variable.



The minimum and maximum value of the contour value is also displayed here. When the contour variable is calculated from subzone data, one or more of the values displayed may be estimates, which is indicated by (est) next to the affected values.

Contour Level Addition

You can add new levels in one of three ways:

- Enter a value in the text field in the Levels and Color page of the **Contour & Multi-Coloring Details** dialog and click **Add Level** to propagate it to the Levels list on the left.
- Add a new range of contour levels to the existing set by clicking **Set Levels**, then using the **Enter Contour Levels** dialog as described in New Contour Level Specification.
- Choose  from the Toolbar, then click at any location in the contour plot where you would like to add a new contour level. Tecplot 360 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.

Contour Level Removal

You can remove contour levels by:

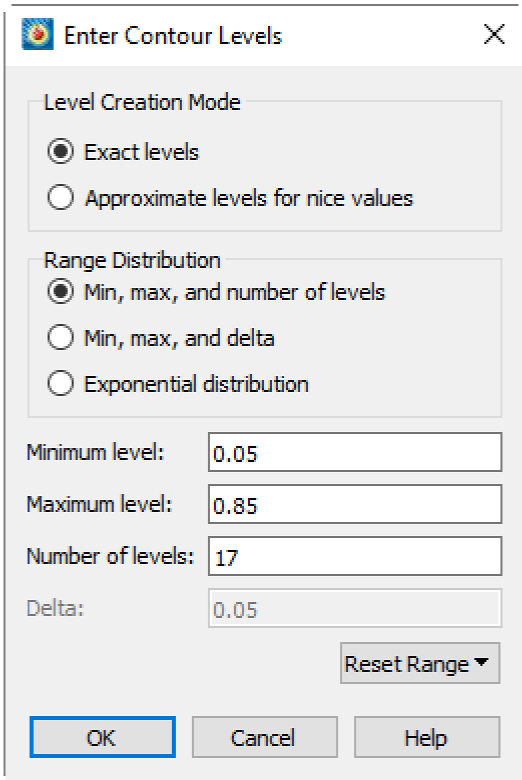
- Selecting one or more contour levels on the Levels and Color page of the **Contour & Multi-Coloring Details** dialog, then clicking **Remove Selected Levels**.
- Selecting  from the Toolbar, then clicking any contour line in your contour plot. Tecplot 360 deletes the specified contour level, or the nearest contour level to the specified point.

Contour Level Adjustment

You can interactively adjust a contour level with the  tool from the Toolbar. Hold down the Control 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 new value of the contour level can be viewed on the Levels and Color page of the **Contour Details** dialog.

New Contour Level Specification

You may specify a new set of contour levels by clicking the **Set Levels** button on the Levels and Color page of the **Contour Details** dialog. This button calls up the **Enter Contour Levels** dialog.



You can create contour levels in two ways:

Exact levels

Specify the levels exactly using one of three methods.

Min, Max, and Number of Levels (default)

Enter a minimum and maximum level value, together with the number of levels to be distributed equally throughout the range.

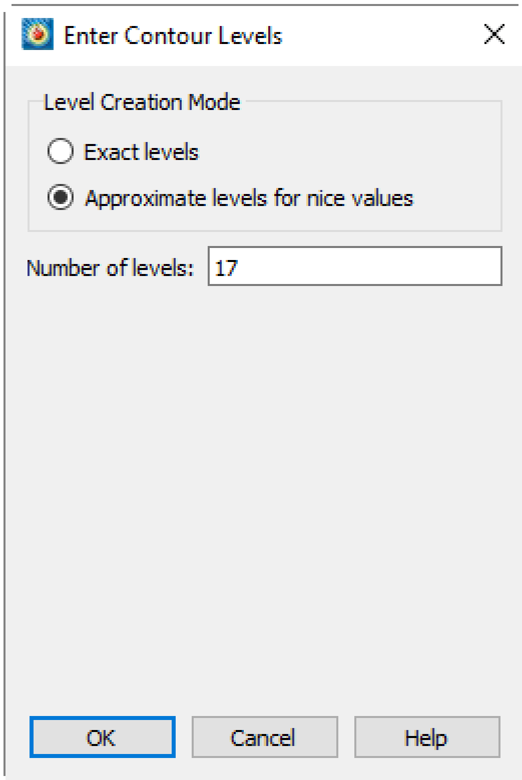
The Reset Range drop-down menu, available with this method, allows you to reset the Min and Max values to the contour variable’s minimum and maximum, optionally excluding the effects of blanking.

Min, Max, and Delta

Enter a minimum and maximum level value together with a delta (step-size between levels).

Exponential Distribution

Enter a minimum and maximum level value together with the number of levels to be distributed exponentially throughout the range.



Approximate levels

Enter the number of contour levels you want and let Tecplot 360 choose their values. The values chosen will be "nice" levels based on the values of the current contour variable.

Contour Coloring

You can choose a built-in color map, edit an existing color map, or create a new color map in the Levels and Contour page.

The built-in color maps cannot be changed. When you begin to modify a built-in color map by moving, adding, or removing control points, the built-in color map is automatically and immediately copied to a custom color map. The copy is given a name ending with the word "modified" and a number, and it is this copy to which further changes apply. Custom color maps can be edited, renamed, or deleted using the gear icon next to the color map menu.

You can create a new color map, then, by first choosing a built-in color map that is close to the color map you want to create (if none is particularly close, any will do) and beginning to modify it. You can then give it a proper name by clicking the gear icon and choosing Rename Custom Color Map.

Custom color maps are automatically saved with layouts. You can also use them in other plots by using the export and import functions on the gear menu.

Color Map Control Points

You can control how colors are distributed across the color map here.

Adjust a control point

Drag the diamond-shaped control points horizontally to control the intervals between them. The color gradations in the color map will be stretched or compressed as necessary as you move the control points.

Create a new control point

Right-click a part of the color map where there is not yet a control point, then choose Add Control Point to create a control point there. The control point's left and right colors will initially be set to the colors in effect at the clicked location.

Change the control point colors

Right-click a control point to change its left (primary) or right (secondary) color by choosing Set Control Point Color, then choosing the color to be set from the submenu. The Color Chooser appears to let you choose the color to be used. The colormap will consist of a gradient of color between one control point's right color and the next control point's left color. (The first and last control points have only right and left colors, respectively.)

Delete a control point

Right-click a control point and choose Remove Control Point to delete it.

Color Map

Select the color map to use for contour coloring. The default is "Sequential - Viridis", a perceptually linear color map from blue to yellow.

Click the gear menu to the right of the Color Map menu to rename or delete a color map, or to import or export custom color maps.

Exporting Custom Color Maps

Saves all custom color maps as a .map file. These can be imported to other projects to maintain consistency.

Import Color Maps

Opens a browser window to import a `.map` file.

Custom color maps may also be created by using the `$!CREATECOLORMAP` command in a macro, configuration, layout, map, or stylesheet file.

> When exporting, all the color maps that aren't one of Tecplot's original set of 45 build-in, immutable, color maps are saved into one .map file.

A short example for loading custom color maps is shown below. Custom color maps may also be loaded via the `$!INCLUDEMACRO` command. The `$!GLOBALCONTOUR` command in the script below assigns a color map to one of the eight contour layers. For more information see the Scripting Guide

```
$!INCLUDEMACRO "|MACROFILEPATH|/Macaw.map"
```

```
$!GLOBALCONTOUR 1 COLORMAPNAME = 'Sequential - Macaw'
$!GLOBALCONTOUR 2 COLORMAPNAME = 'Sequential - Macaw'
$!GLOBALCONTOUR 3 COLORMAPNAME = 'Sequential - Macaw'
$!GLOBALCONTOUR 4 COLORMAPNAME = 'Sequential - Macaw'
$!GLOBALCONTOUR 5 COLORMAPNAME = 'Sequential - Macaw'
$!GLOBALCONTOUR 6 COLORMAPNAME = 'Sequential - Macaw'
$!GLOBALCONTOUR 7 COLORMAPNAME = 'Sequential - Macaw'
$!GLOBALCONTOUR 8 COLORMAPNAME = 'Sequential - Macaw'
```

For Tecplot 360 2017 R3 and later, the `$!CREATECOLORMAP` macro command is allowed in the configuration file. This allows users to define their own color maps and select the default for each contour group. See the Scripting Guide for more information.

Color Distribution Methods

Banded

A solid color is assigned for all values within the band between two levels. (See Contour Bands.)

Continuous

The 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. You may reset the endpoints to the contour variable's min/max (considering the effects of blanking), the contour variable's min/max excluding the effects of blanking, or to the contour level's min/max using the **Reset** menu to the right of these fields.

Approximate Continuous

Causes each cell to be flooded using interpolation between the RGB values at each node. 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.

Color Cutoff

Lets you specify a range within which contour flooding and multi-colored objects (such as scatter symbols) are displayed.

Color Map Adjustments

The following optional adjustments are applied to the color map generated by the above settings.

Reversed Color Map

You can reverse the color map by toggling on "Reverse". Two plots, one with the color map going in the default direction and one with the color map reversed, are shown in Figure 29.

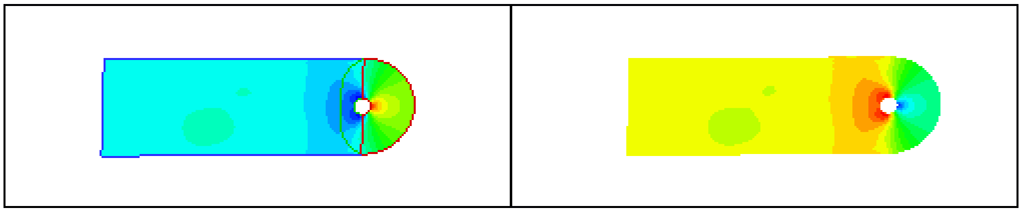


Figure 29. Left: Flooded contour plot with default settings. Right: Flooded contour plot with a reversed colormap.

Repeat 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 twice is shown in Figure 30.

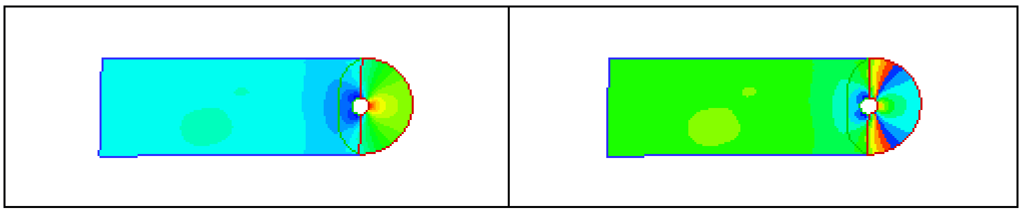
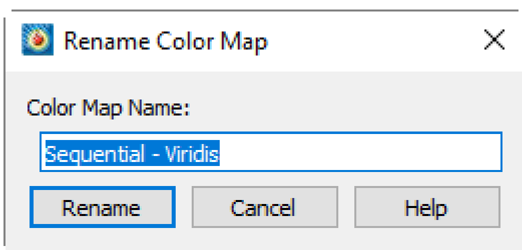


Figure 30. Left: Flooded contour plot with default settings. Right: Flooded contour plot with the color map cycled two times.

Rename Color Map

The Rename Color Map dialog allows you to enter a new name for a user-defined color map. This name will appear throughout Tecplot 360.

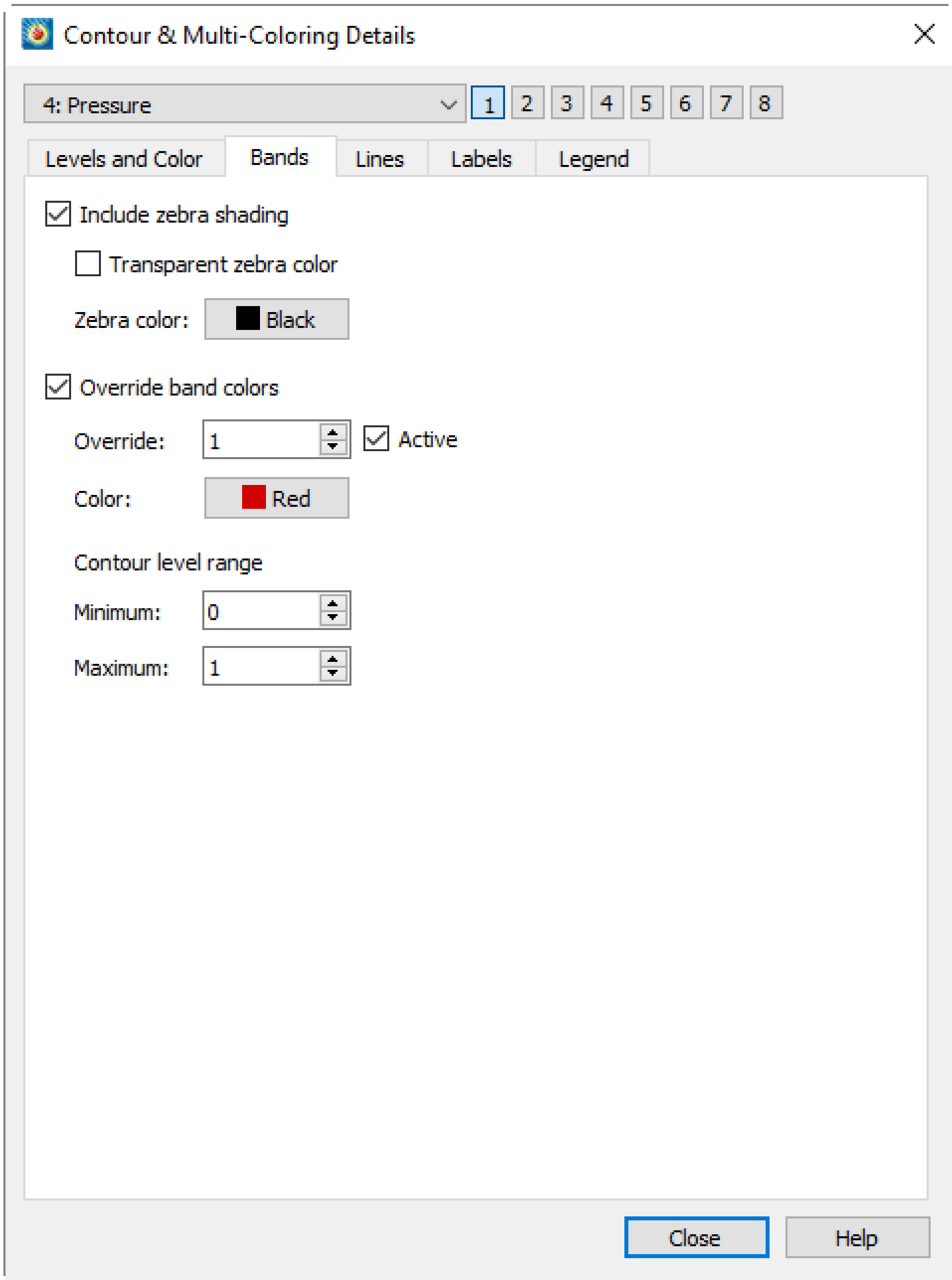


You may not give a custom color map the same name as a built-in color map.

When renaming a color map, spaces will be removed from the beginning and end of the new name.

Contour Bands

When Coloring Distribution for a group is set to "Banded" (via the Levels and Color page of the **Contour & Multi-Coloring Details** dialog), you may customize the color bands on the Bands page of the dialog.



The Bands page of the **Contour Details** dialog has the following options:

Include Zebra Shading

This effect colors every other band with a specific color (or no color at all).

Override Band Colors

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.

Override

Specifies which override is being edited.

Active

Toggle this checkbox to turn the override on or off.

Color

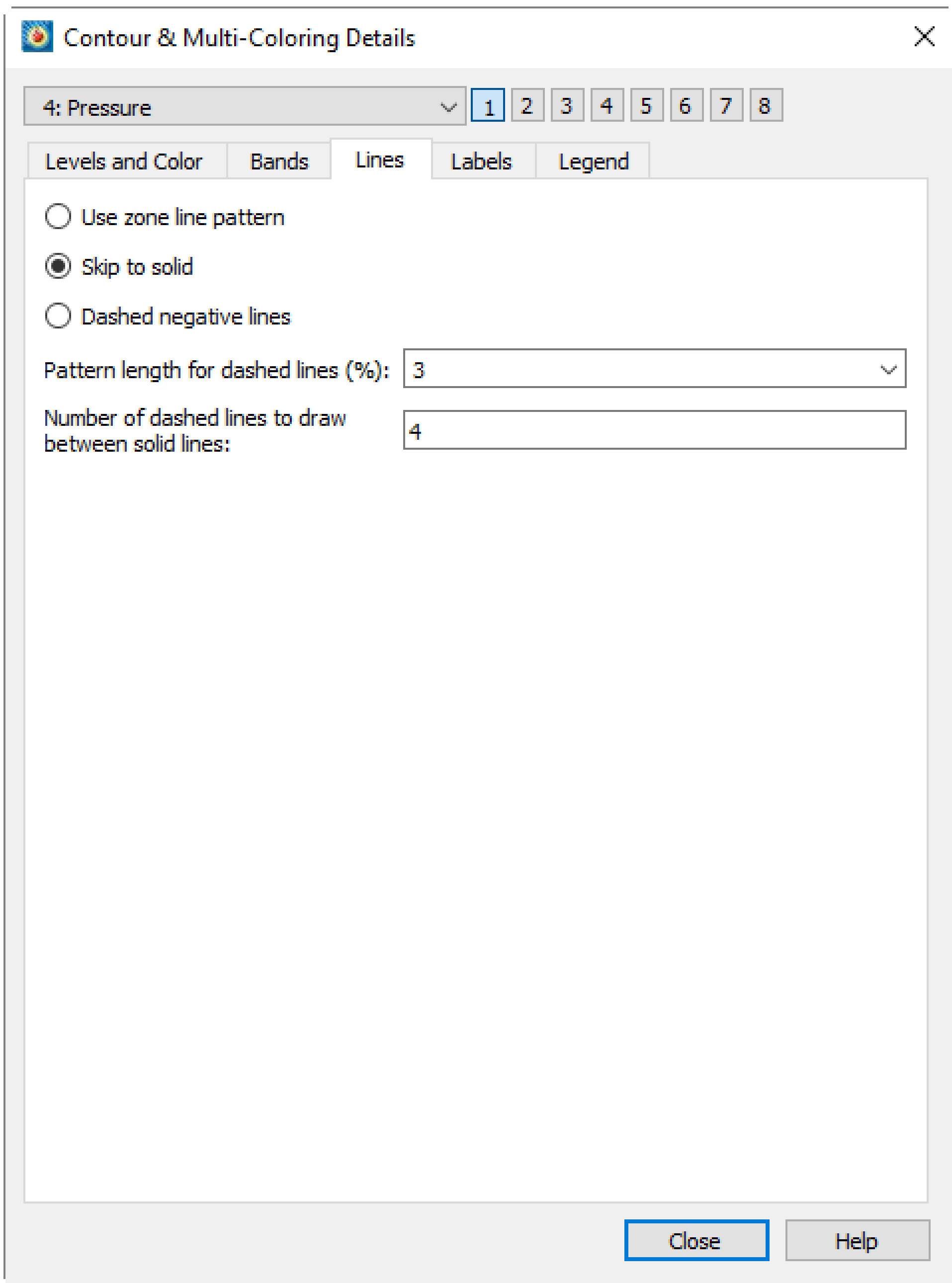
Select the color for the override using the Color Chooser.

Minimum, Maximum

Choose the range which will be overridden to the specified color.

Contour Lines

The contour line settings determine how contour lines are drawn for all zones in the active frame's dataset. The settings are established on the Lines page of the **Contour & Multi-Coloring Details** dialog.



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.

> If you are adding contour lines to polyhedral zones, the patterns will not be continuous from one cell to the next. The pattern will restart at every cell boundary.

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

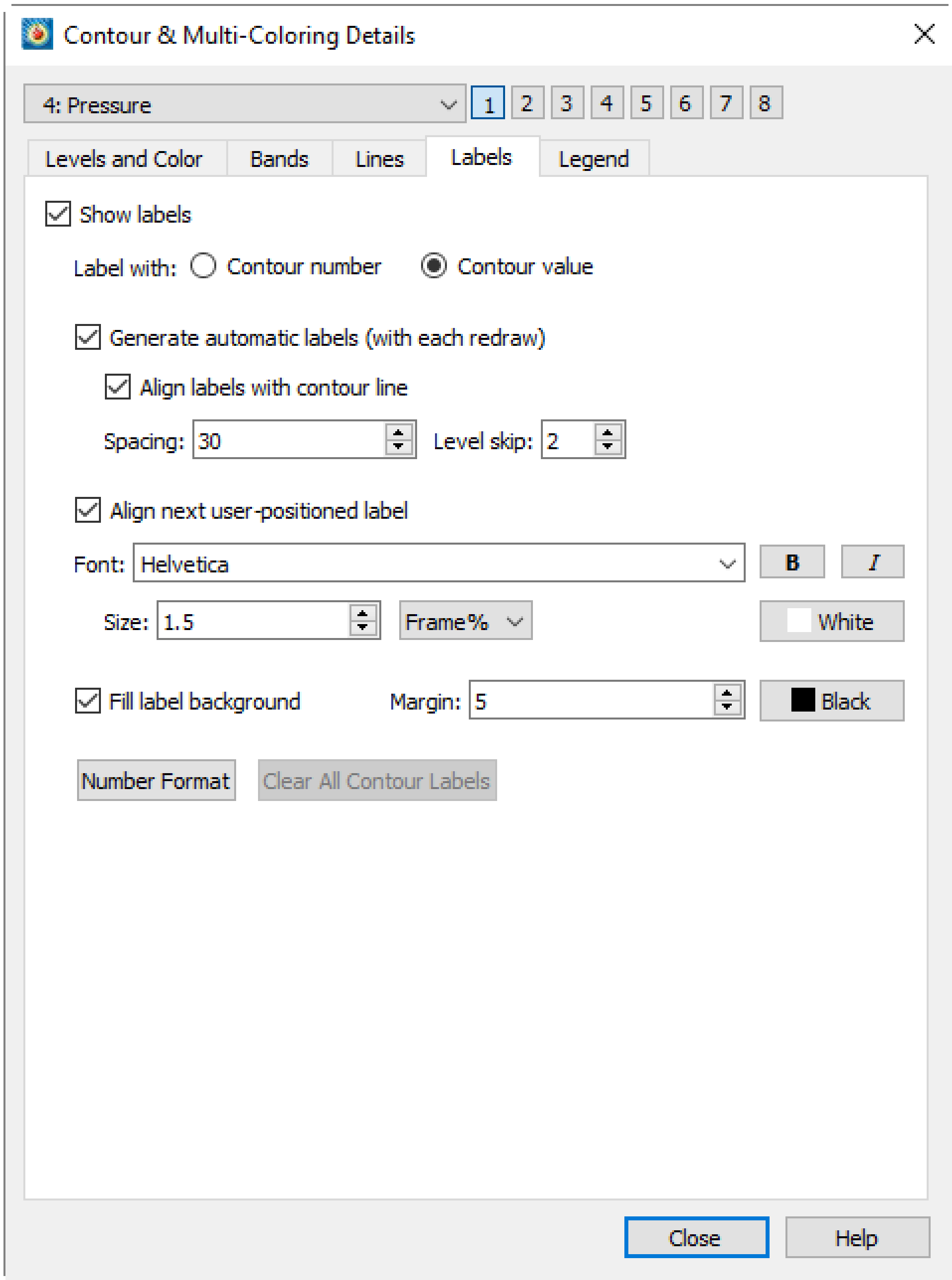
Draw lines of positive contour variable value as solid lines and lines of negative contour variable value as dashed lines.

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 360 create them for you automatically. You can also have Tecplot 360 create and save a set of contour labels automatically, then interactively add contour labels to this saved set.

> The contour plot type must be lines or lines and flood in order to use Contour labels.

Contour labels may be configured in the Labels page of the **Contour & Multi-Coloring Details** dialog or with the Add Contour Label mouse mode tool from the Toolbar. You can modify the following options using the Labels page of the **Contour Details** dialog.



Show Labels

Toggle-on "Show Labels" to include contour labels in your plot. You can then label the contour levels by selecting either **Use Contour Number** or **Use Contour Value**.

Label Format

Use the midsection of the dialog to customize label color, font, and fill settings. (See Font Folders and Fallback for more information on how fonts work with Tecplot 360.)

Generate Automatic Labels (with each Redraw)

At each Redraw, Tecplot 360 creates a new set of contour labels. At any time, you can deselect the "Generate Automatic Labels" (with each Redraw) check box, and Tecplot 360 retains the last set of labels generated.

Align Labels with Contour Line

Use the Spacing field to specify the spacing of the contour labels along the contour line, as a percentage of the frame. Use the Level Skip field to specify a skip value between the contour levels to be labeled.

Align Next User-Positioned Label

If the "Align Next User-Positioned Label" is selected, the next label is aligned with the contour line. Otherwise, the label is written with normal, upright text.

Number Format

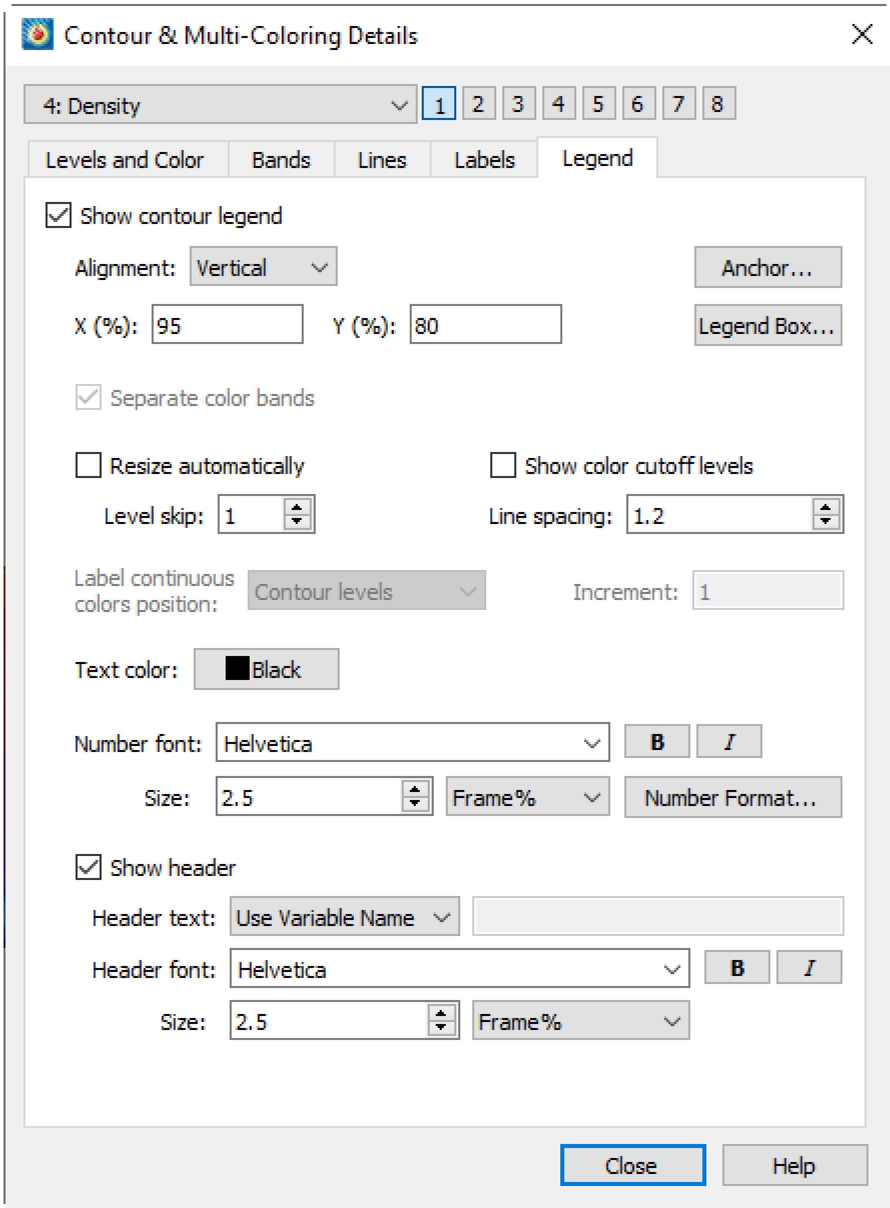
Use the **Number Format** button to specify the number formatting of the Contour labels. See Specify Number Format for more details.

Clear All Contour Labels

When "Generate Automatic Labels" is deselected, you can click this button to erase the current set of contour labels.

Contour Legend

To include a contour legend, select the Legend page of the **Contour & Multi-Coloring Details** dialog for the appropriate contour group.



The following options are available:

Show Contour Legend

Select this option to show the contour legend on your plot.

Alignment

Select Vertical or Horizontal.

Anchor

Specify which part of the legend is anchored in the selected position using the Anchor Alignment dialog.

Position

X (%) and Y (%) as percentages of the frame width and height. (You can also move the legend interactively.)

Legend Box

Displays the Legend Box dialog to adjust the appearance of the box around the legend, or turn it off entirely.

Separate Color Bands

Select this check box to separate the color bands in the legend with black lines. Use this option to visually separate similar colors. If this box is not selected, similar adjacent colors may tend to blur together.

Resize Automatically

Automatically skips some levels to create a reasonably sized legend.

Include Color Cutoff Levels

Color bands and labels for levels affected by Color Cutoff are shown in the legend.

Label Continuous Colors Position

If you have selected "Continuous Color Distribution" on the Levels and Color page of the **Contour Details** dialog, you have three options for placement of labels on the legend:

Contour Levels

This option places one label for each contour level. See Contour Levels and Color.

Specified Increment

Enter a value in the Increment text field.

Color Map Divisions

Places one label for each control point on the color map.

Level Skip

Enter the number of levels between numbers on the legend. This also affects the number of levels between contour labels on the plot. Skipping levels on the contour legend compresses the color bar (if one appears); it does not change the spacing between text entries on the legend.

Line Spacing

Enter the spacing between contour legend numbers. This does not change the number of entries in the legend, so a large value here creates a large legend. Use Level Skip to reduce the number of entries in the legend.

Text Color

Affects the color of all text in the legend.

Number Format

Adjust the format of numbers in the legend. See Specify Number Format for more details on numeric formatting

Number Font

Choose the font used for numbers in the legend, including size and variety (boldface or italic). See Font Folders and Fallback for more information on how fonts work with Tecplot 360.

Show Header

Toggle-on to display the contour legend header text.

Header Text

Allows selection of content displayed in the legend header.

Use Variable Name

The legend header will display the variable name assigned to the contour group.

Use Text

The legend header will display text entered into the text field next to the options widget. The text can contain dynamic text (See Dynamic Text for information on dynamic text) and formatted using tags (See Text Details for information text formatting tags).

Use LaTeX

The legend header will display a processed LaTeX expression. See LaTeX Expressions for information on LaTeX formatting.

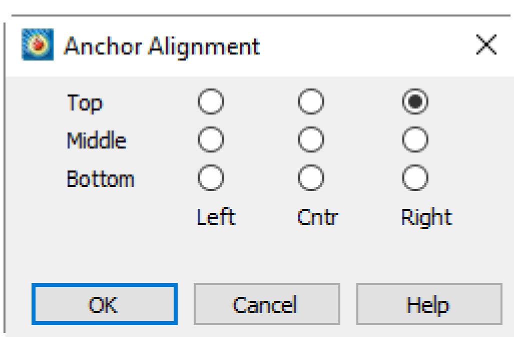
Header Format

Adjust the font and height for the legend header or the legend labels, including font size and variety (boldface or italic). (See Font Folders and Fallback for more information on how fonts work with Tecplot 360.)

Not all fonts have Bold and/or Italic variants. For fonts that do not have these styles, the **B** and/or **I** buttons may have no effect.

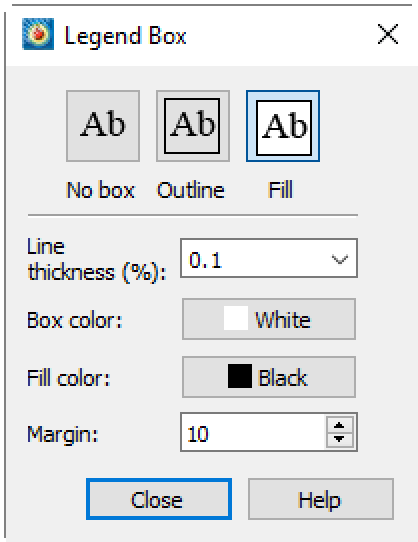
Anchor Alignment

The **Anchor Alignment** dialog allows you to specify the anchor point, or fixed point, of the object. As the box grows or shrinks, the anchor location is fixed while the rest of the box adjusts to accommodate the new size. There are nine possible anchor points, corresponding to the left, right, and center positions on the headline, midline, and baseline of the box.



Legend Box

The Legend Box dialog appears when you click the Legend Box button in the Legend page of the Contour Details dialog.



Box Type

Choose (left to right) No Box, Outline, or Fill. If you choose Outline or Fill, the following options control the appearance of the box.

Line Thickness

Specify the line thickness as a percentage of frame height.

Box Color

Choose a color for the legend box outline.

Fill Color

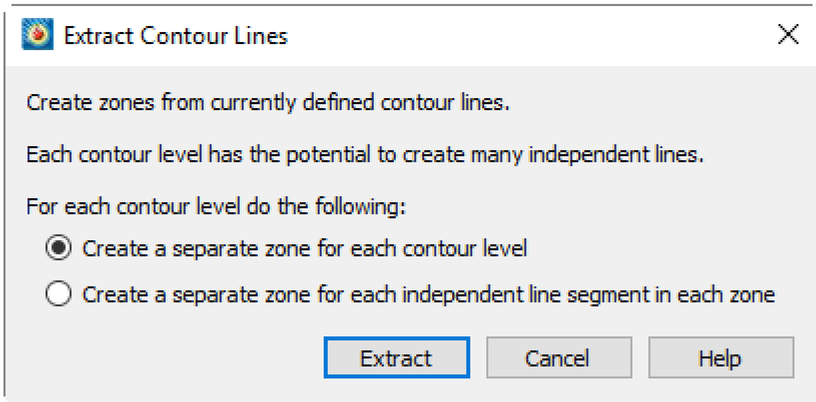
Choose a color for the legend box fill (Fill mode only).

Margin

Specify the margin between the legend text and legend box as a percentage of the text height.

Extract Contour Lines

Normally, contour lines are derived from the dataset "on the fly" and do not add any data to the dataset. To extract existing contour lines to Tecplot zones, allowing you to retain them even if the contour details are changed, go to **Data→Extract→Contour Lines**. Your data will be altered by the creation and naming of new zones.



Using the **Extract Contour Lines** dialog, you have the following options:

Create a separate zone for each contour level

A new zone will be created for each contour line plotted. The number of new zones will equal the number of contour levels. The created zones are FE-line segment type zones.

Create a separate zone for each independent line segment in each zone

With this option you may create many more zones than there are contour levels. New I-ordered line segment zones are created in each source zone for each topologically independent contour line.

After generating the zones, we recommend you activate the Mesh Layer when plotting the new zones.

Vector Layer

You can create vector plots by activating the **Vector** layer in the **Plot sidebar**, and specifying the vector component variables. **Vector** plot attributes can be modified using the **Vector** page of the **Zone Style** dialog.

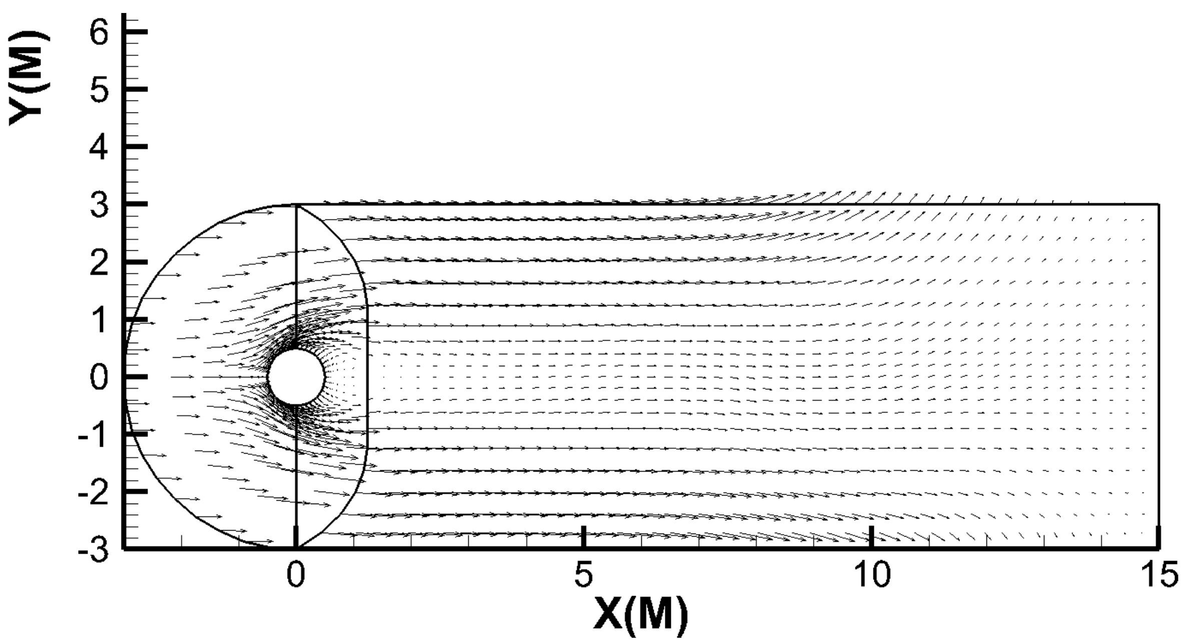
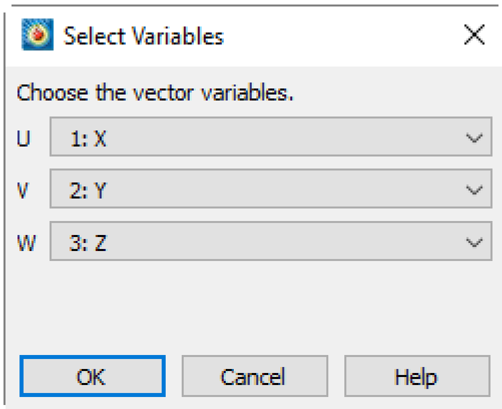


Figure 31. A vector plot of the cylinder data (with the edge layer also active)

Vector Variables

When you activate the **Vector zone** layer (via the **Plot sidebar**), Tecplot 360 checks to see whether vector components have been assigned for the current dataset in the current plot type. If you have not assigned vector components, the **Select Variables** dialog, shown here, will be launched.

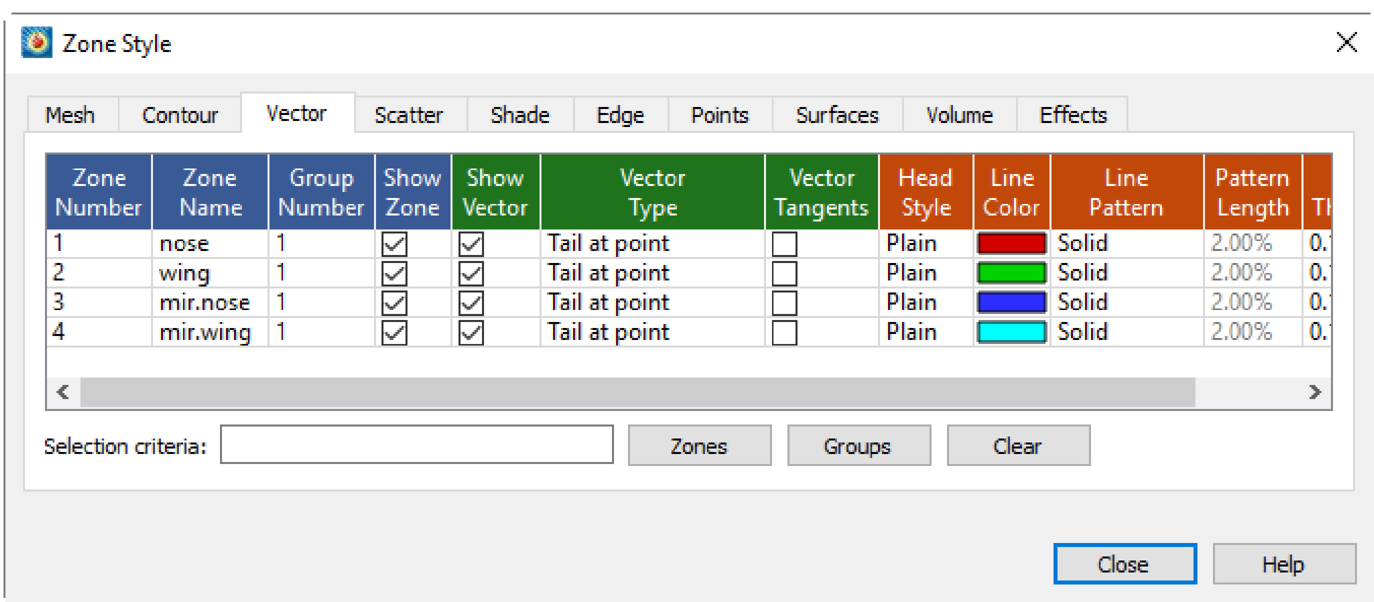


Choose variables by selecting the desired U, V, and W (3D only) variables from their respective drop-downs. You may select any of the current dataset's variables as any component. You can change the component variables at any time by choosing **Vector→Variables** from the **Plot** menu.

Once you have selected the **Vector check box** in the **Plot sidebar** and have chosen your vector components, your vector plot will appear. If vectors are not visible, refer to Definition Page.

Vector Plot Modification

You can modify your vector plot attributes using the **Vector page** of the **Zone Style** dialog. You can control any of the following attributes from the **Vector page** of the **Zone Style** dialog.



> In order for the changes made on the **Vector page** to be visible in your plot, the **Vector layer** must be active (toggled on in the **Plot sidebar**).

Show Vector

Toggle-on to show the vector for a particular zone.

Vector Type

Select from the following options:

Tail at Point (default)

Draws the tail of the vector at the data point.

Head at Point

Draws the head of the vector at the data point.

Anchor at Midpoint

Positions the midpoint of the vector at the data point.

Head Only

Draws the head of the vector at the data point and does not draw a tail.

Figure 32 shows examples of each of the vector plot types.

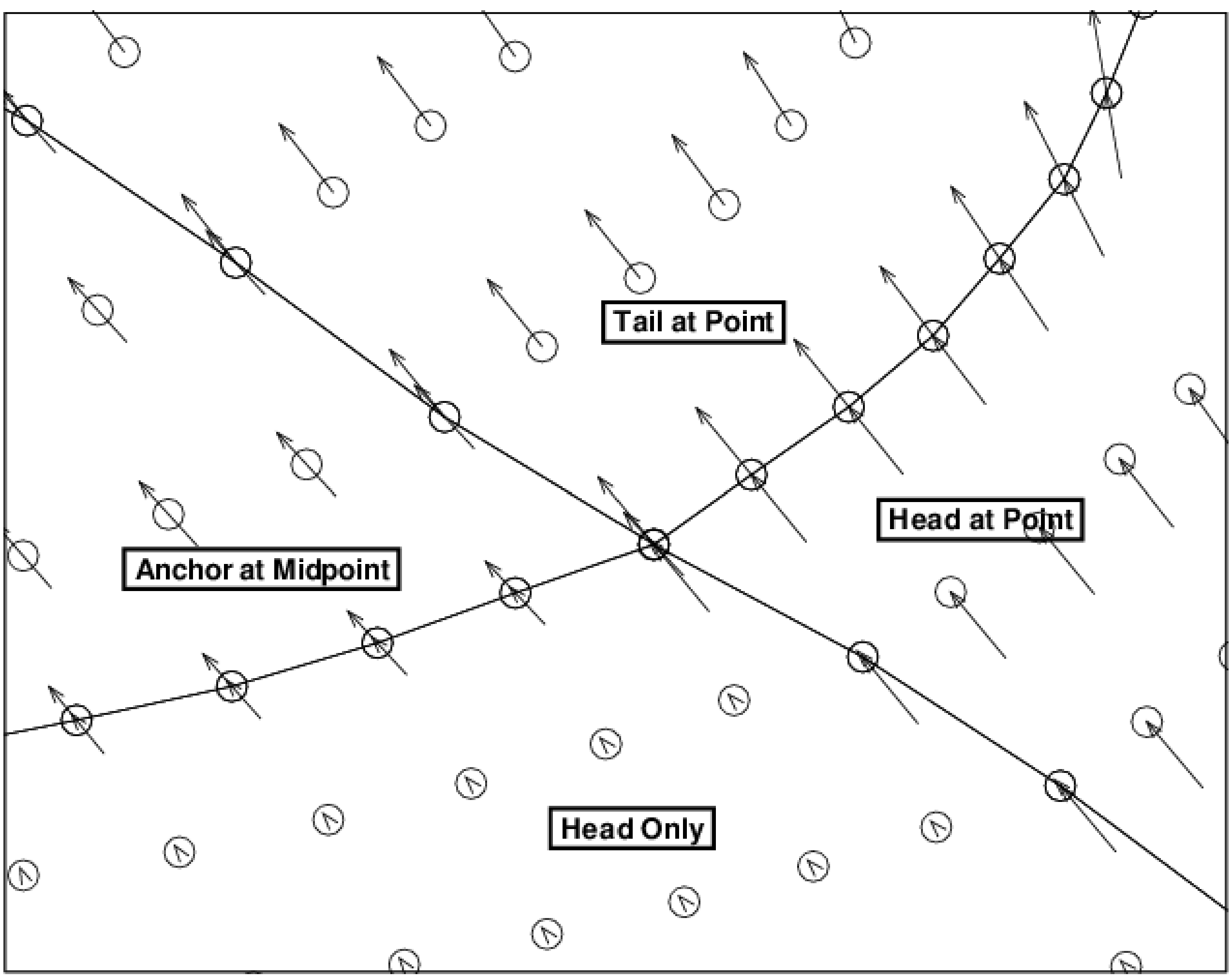


Figure 32. The Vector plot types: tail at point, head at point, anchor at midpoint, and head only.

Vector Tangents

Toggle-on to display only the tangent component of vectors, not both tangent and normal components. Tangent vectors are drawn on 3D 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.

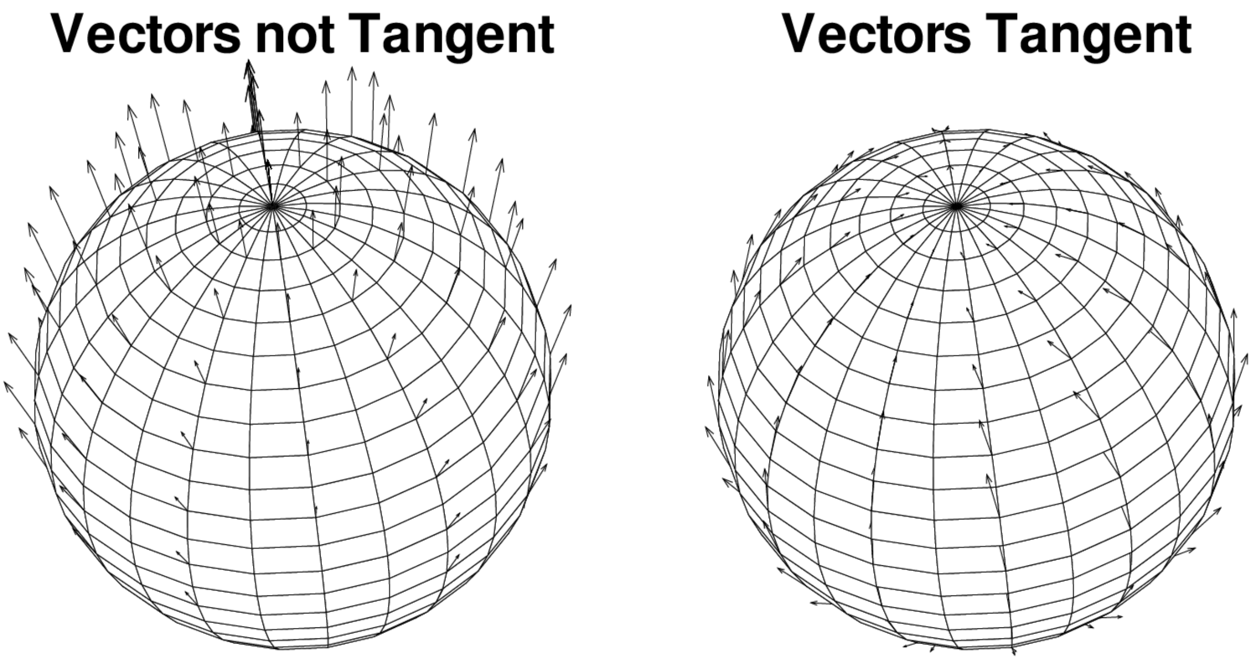


Figure 33. Comparison of the Vector Tangent options. A) vectors are drawn with both the normal and tangent components. B) vectors are drawn with only the tangent components.

Head Style

`Right-click` to choose an arrowhead style for the vectors. Figure 34 displays the available styles.

Plain (default)

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.

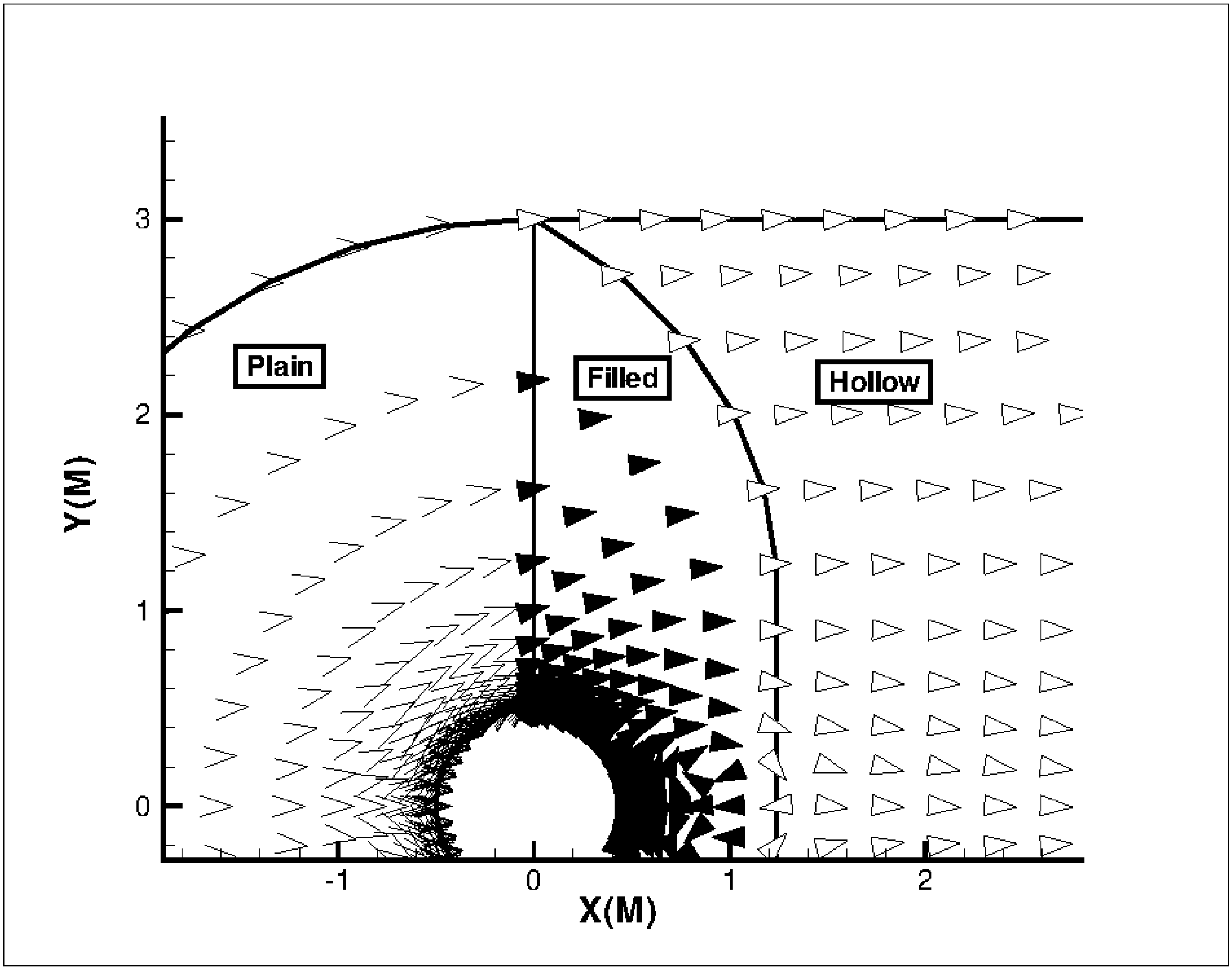


Figure 34. Arrowhead types for vector plots (plain, filled and hollow).

Line Color

`Right-click` to choose the vector color in the Color Chooser dialog

Line Pattern

`Right-click` to choose the vector line pattern.

Pattern Length

`Right-click` to choose the vector line pattern length as a percentage of frame height. You may choose a preset length or enter an exact number.

Line Thickness

`Right-click` to choose the vector line thickness as a percentage of frame height. You may choose a preset thickness or enter an exact number.

If your data consists of a dense mesh of points, a **vector plot** may be too crowded to be of much use. You can "thin" the plot by plotting only a certain subset of the data points with the **Index Skip** attribute from the Points page of the **Zone Style** dialog.

For information on using the controls at the bottom of the **Zone Style** dialog to select zones by name, see the description of these at the end of Field Plot Modification and the Zone Style Dialog.

Use the **Vector Details** dialog to adjust other **Vector layer** settings, including length scaling, arrowhead appearance, and reference vector. See Vector Details.

Vector Details

The **Vector Details** dialog (**Plot**→**Vector**→**Details**) lets you specify the length of vectors and the appearance of arrowheads. You can also display a reference vector. on your plot. These settings are all global to the current frame.

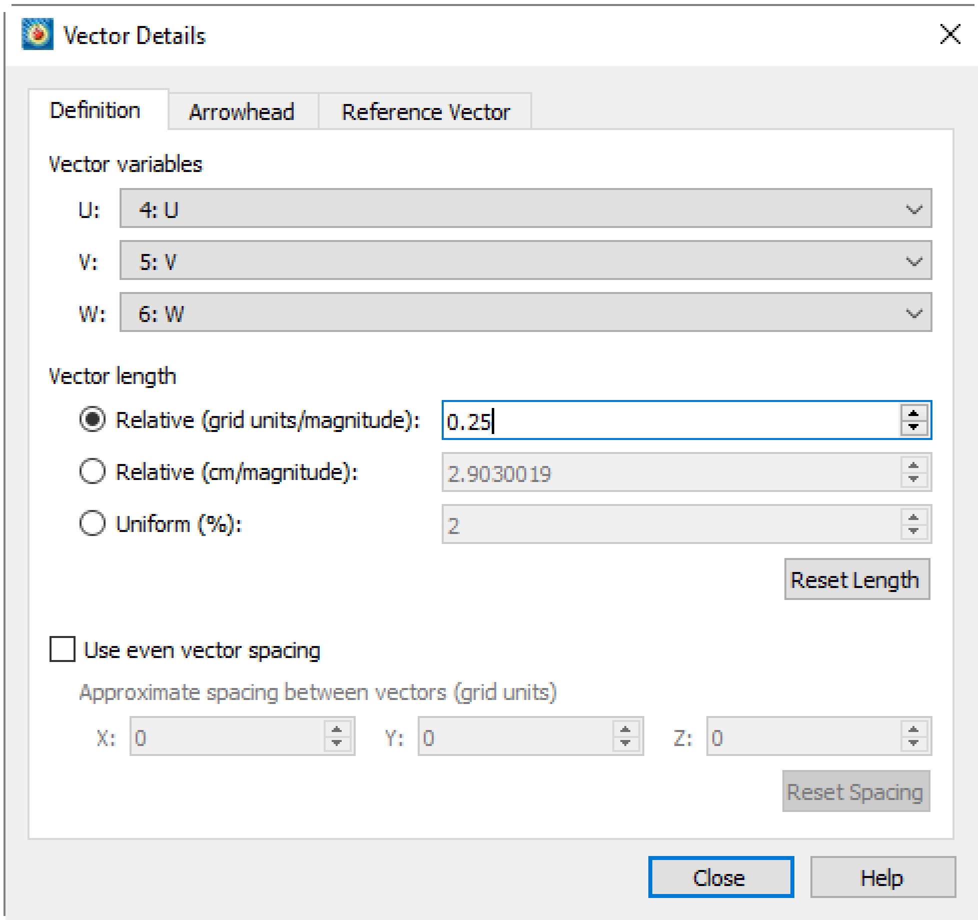
In order for most changes made on the **Vector Details** dialog to be visible in your plot, you must have the **Vector layer** toggled on in the Plot sidebar, and the vectors of at least one zone must be visible (see Vector Plot Modification).

The **Vector Details** dialog has three pages:

- Definition Page
- Arrowhead Page
- Reference Vector Page

Definition Page

The **Definition** page of the **Vector Details** dialog allows you to change the vector variables, adjust the length of displayed vectors, and use even vector spacing.



The **Definition** page of the **Vector Details** dialog has the following options:

Vector variables

Select the variables for U, V, and W (3D only) vector variables. Selecting new vector variables will update the plot if any vectors are displayed.

Vector length

The following option are available for the length of the vector:

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.

Reset Length

For relative vector length, the default is based on the size of the longest vector. `Click` **Reset Length** to change the vector length to a relative vector length with the scale factor expressed in grid units per unit of vector magnitude. Not available when **Uniform** is selected.

For either of the "Relative" options for vector length, the value you specify is a scale factor that is multiplied by the vector magnitude to determine the length of the vector.

> Since 3D vectors are plotted in the plane of the screen, a 3D vector’s length will depend on both the vector length settings and the orientation of the vector. The length may be distorted more if the length setting is **Relative** and the 3D projection is **Perspective**.

Use even vector spacing

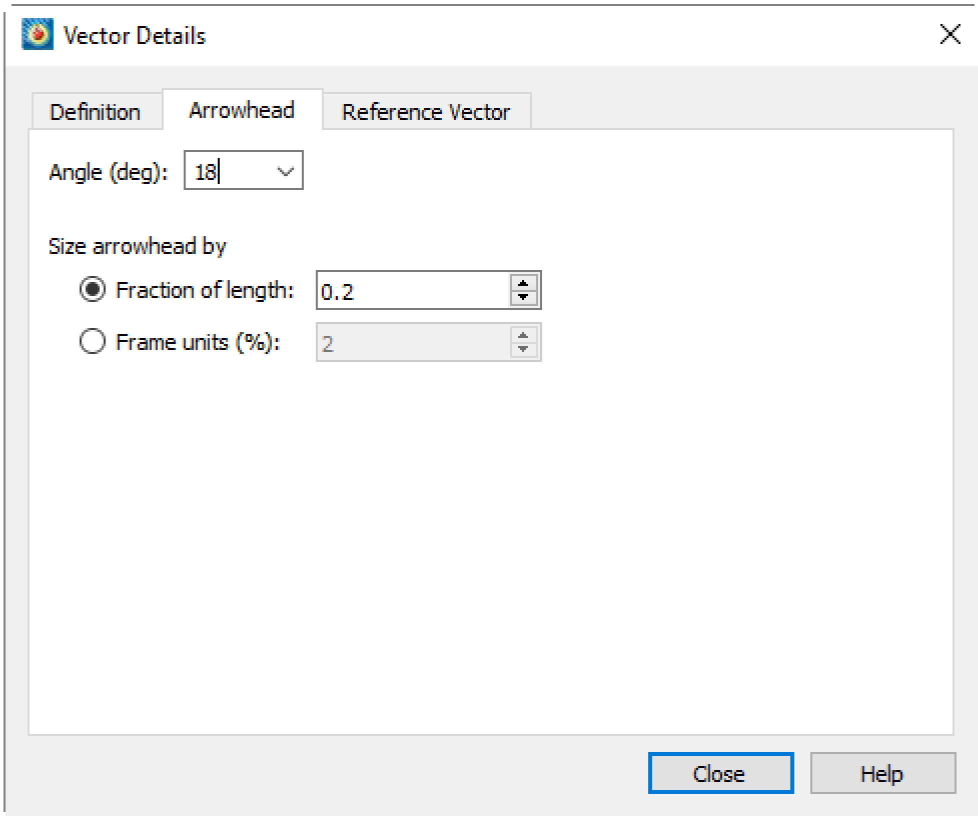
Select to turn on even spacing for vectors based on grid units of X, Y, and Z (3D only). Turning on even vector spacing (with vectors displayed) will selectively remove some of the vectors in the plot to approximately enforce the display intervals specified in the spacing controls. This is useful for clarifying areas of the plot that are cluttered with an excess number of vectors. When turned on, the initial values of spacing, specified in grid units, are set to attempt to display a maximum of 40 vectors in each direction across the frame based on current view. This default number of vectors can be changed in Tecplot’s configuration file, `tecplot.cfg`; refer to the `VECTORDEFAULTSPACINGCOUNT` option of the `$!FRAMESETUP` macro command.

Reset Spacing

Resets the spacing intervals to these default values. Selecting new axis variables will also reset the vector spacing values.

Arrowhead Page

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). By default, Tecplot 360 specifies size as a fraction of the vector length.



The **Arrowhead** page of the **Vector Details** dialog has the following options:

Angle (deg)

The arrowhead angle is the angle that one side of the arrowhead makes with the vector, i.e. the apex angle is twice the arrowhead angle. To specify the arrowhead angle, enter a value from 1 to 90, or choose a value from the drop-down, indicated by the down-arrow button.

Size arrowhead by

Fraction of Length

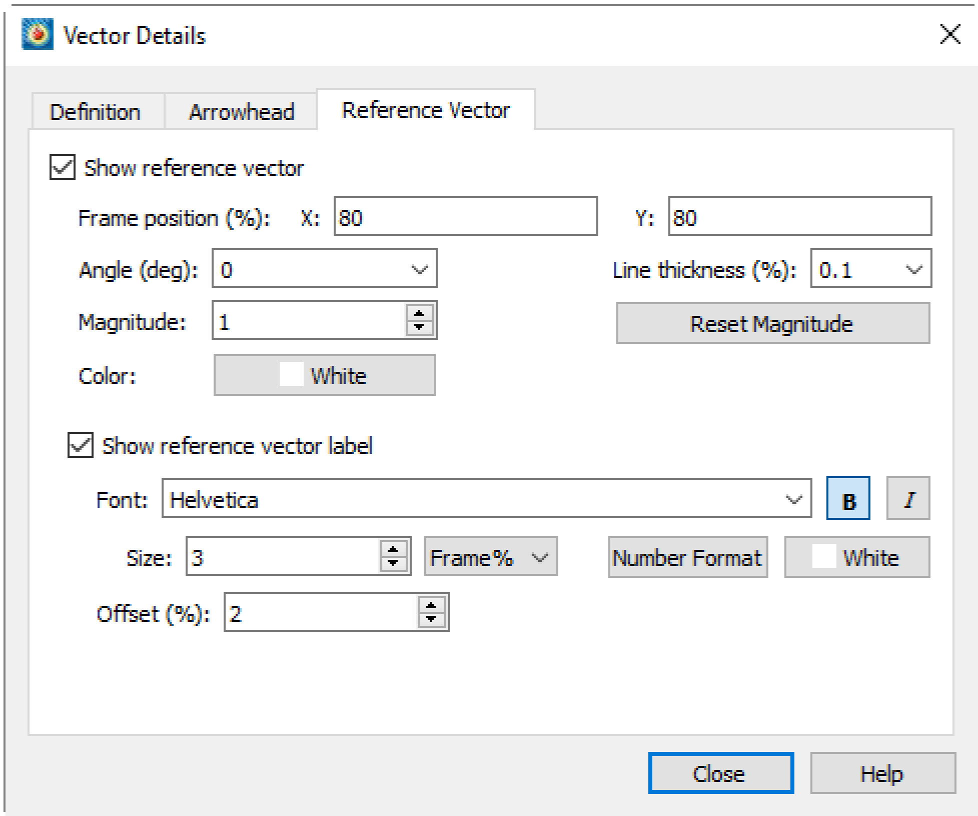
Enter a decimal value from zero to ten.

Frame Units (%)

Enter a percentage value from zero to 100.

Reference Vector Page

A reference vector is a vector of specified magnitude placed on the plot as a visual measure of vector length.



The **Reference Vector** page of the **Vector Details** dialog has the following options:

Show Reference Vector

Toggle-on to include a reference vector in your plot.

Frame Position

Enter the coordinates of the starting point of the reference vector, as a percentage of the frame width (X) and frame height (Y).

Angle (deg)

Enter the orientation of the vector in degrees from horizontal, 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.

Color

Choose a color with the Color Chooser. Multi-color and RGB coloring are not available.

Line Thickness (%)

Enter the desired line thickness, or choose a value from the drop-down.

Reset Magnitude

Resets the magnitude of the reference vector based on the average vector magnitude.

Show Reference Vector Label

Toggle-on to include the magnitude of the reference vector in the label. Select and modify any of the following options:

Font

Choose or enter the name of the font to be used for the label. Click the `B` or `I` buttons to select the bold and/or italic variety of the font. (See Font Folders and Fallback for more information on how fonts work with Tecplot 360.)

Size

Enter the size and choose units either of frame height percentage or points.

Number Format

`Click` to specify how the number will be formatted using the Specify Number Format dialog. See Specify Number Format for a discussion of this dialog.

Color

Choose a color with the Color Chooser. Multi-color and RGB coloring are not available.

Offset (%)

Choose the spacing between the label and the reference vector as a percentage of frame height.

A plot with a reference vector is shown here. The units for the reference vector were added using a text object (see Text).

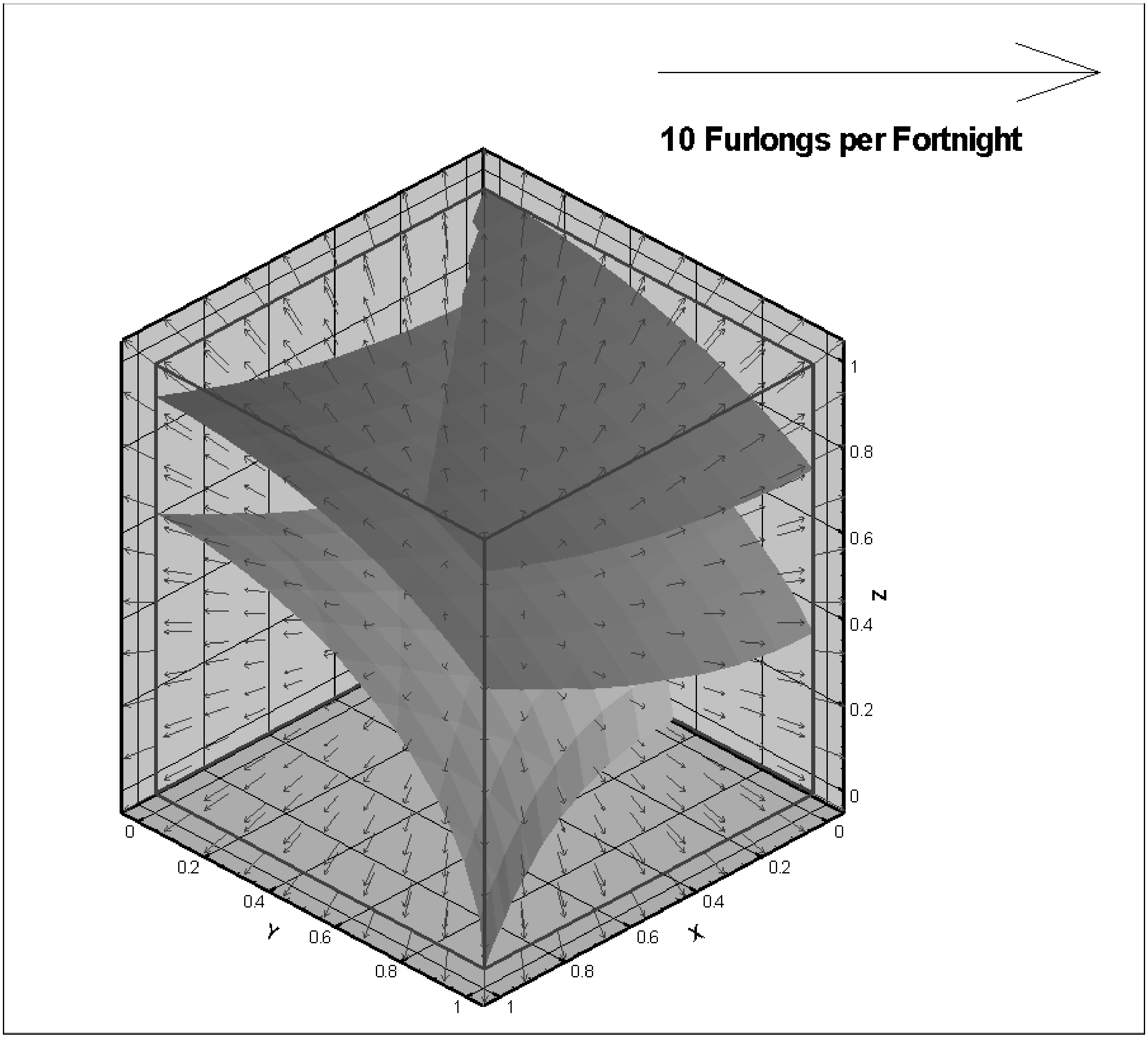


Figure 35. Reference Vector with Units Added

Scatter Layer

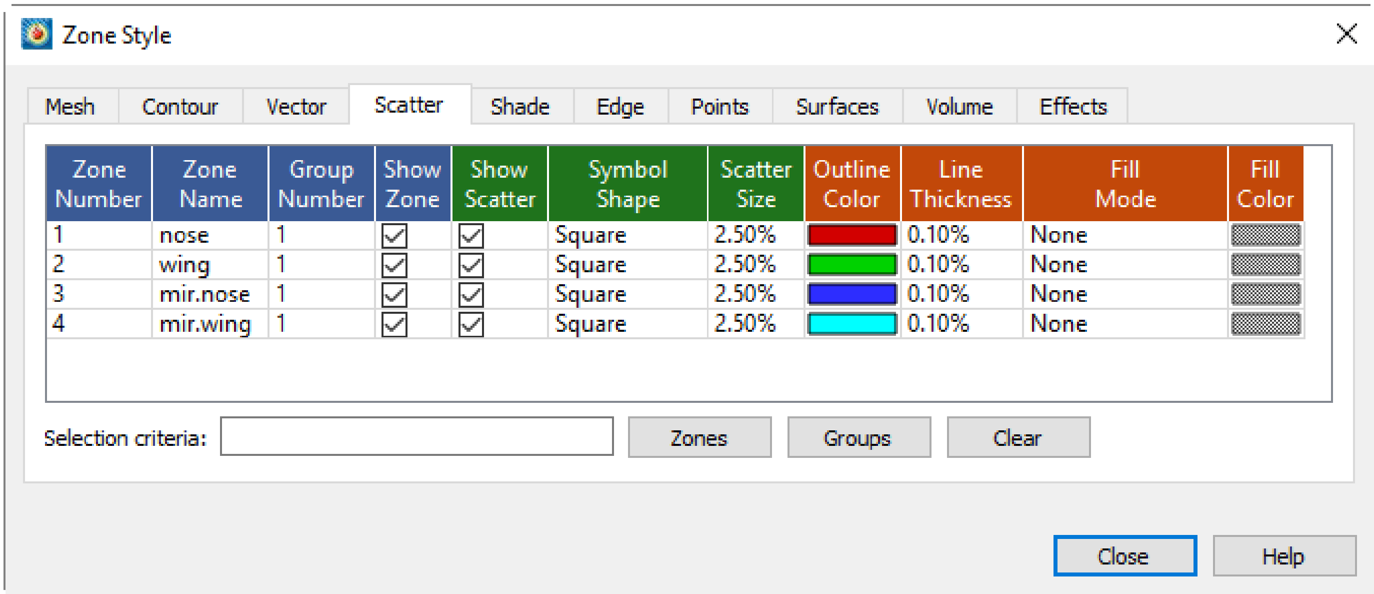
Scatter plots are plots of symbols at the data points in a field. The symbols may be sized according to the values of a specified variable, colored by the values of the contour variable, or may be uniformly sized or colored. Unlike contour plots, scatter plots do not require any mesh structure connecting the points, this allows you to make scatter plots of irregular data.

To add a scatter layer to your plot, activate the "Scatter" toggle in the Plot sidebar. You can modify your Scatter plot using the Scatter page of the **Zone Style** dialog and the **Scatter** submenu of the **Plot** menu.

Scatter Plot Modification

Once you have loaded your data, you can modify your scatter plot attributes using the Scatter page of the **Zone Style** dialog. You can control any of the following attributes for a zone or group of zones from the Scatter page of the **Zone Style** dialog.

> For information on using the controls at the bottom of the Zone Style dialog to select zones by name, see the description of these at the end of [Field Plot Modification and the Zone Style Dialog](#).



> In order for the changes made on the Scatter page to be visible in your plot, the Scatter layer must be turned on in the Plot sidebar.

Show Scatter

Right-click to choose whether or not to show the scatter layer for the highlighted zone(s).

Symbol Shape

Right-click to choose one of the following symbol shapes:

- Square (default)
- Delta
- Gradient
- Left Triangle
- Right Triangle
- Diamond
- Circle
- Cube (rendered as a square in 2D)
- Sphere (rendered as a circle in 2D)
- Octahedron (rendered as a diamond in 2D)

- Point
- Character - Use a specified ASCII character from a selected font (as specified in the Enter ASCII Character dialog). See also Text.

> 3D scatter symbols should only be used if your dataset is on the order of thousands of points. If your dataset is large (e.g. millions of points), try using 2D scatter symbols instead for better interactive performance.

Scatter Size

Right-click to select the symbol size either by a constant percentage of the frame height or from a variable in the dataset. For constant size, you may choose a preset size or enter one of your own. (See Scatter Size/Font for complete instructions for sizing scatter symbols by variable.)

Outline Color

Right-click to choose a color from the Color Chooser. Besides a constant color, you can also choose:

Multi

Each symbol is colored according to the value of the selected contour variable at that data point. Choose a contour group at the bottom of the Color Chooser dialog.

RGB

Each symbol is colored according to the values at that data point for the variables assigned to RGB. Click the **RGB** button at the bottom of the Color Chooser dialog.

Line Thickness

Right-click to choose the thickness of the scatter outlines for each highlighted zone(s), either a preset from the menu or by entering your own value.

Fill Mode

The 3D symbol shapes, Cube, Sphere, and Octahedron are filled with the line color, but the other shapes have other fill modes available. Right-click to choose:

None (default)

Unfilled symbols.

Use Specific Color

Uses the color shown in the Fill Color column.

Use Line Color

Matches outline color.

Use Background Color

Matches frame color.

Fill Color

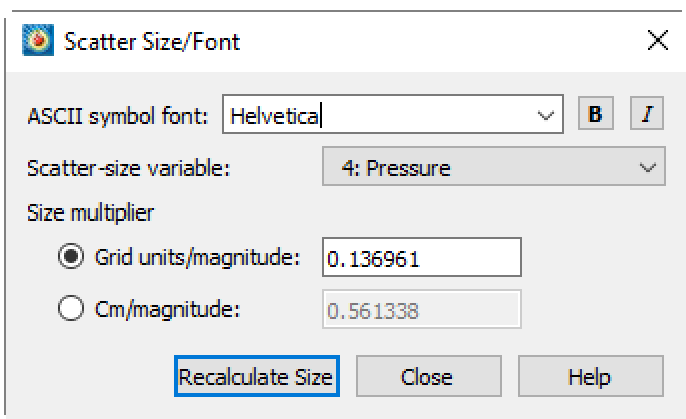
Right-click to select a fill color using the Color Chooser.

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 3D shapes, adjust the Light Source to use Specular Highlighting. Scatter Size and Line Thickness are not available for the point symbol. Points are always one pixel in size.

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 with the Index Skip attribute from the Points page of the **Zone Style** dialog. The Point scatter symbol allows for quick viewing and panning in 3D plots. It is also a useful tool for identifying features in volume zones.

Scatter Size/Font

Use the **Scatter Size/Fonts** dialog (accessed via **Plot**→**Scatter**→**Size/Font** in 2D/3D or **Plot**→**Symbol Font** in line plot modes) to choose the base font used for ASCII character symbols and the scatter-size variabjle used to scale scatter symbols. This dialog is also available by right-clicking a scatter size in the Scatter page of the Zone Style dialog and choosing **Select Variable**. The **Scatter Size/Font** dialog is shown below:



The following options are available:

ASCII Symbol Font

Select a font from the drop-down and optionally a bold and/or italic style. (See Font Folders and Fallback for more information on how fonts work with Tecplot 360.) For XY and Polar line plots, this is the only option available.

Not all fonts have Bold and/or Italic variants. For fonts that do not have these styles, the `B` and/or I buttons may have no effect.

Scatter-size Variable

Select a variable from the drop-down of the dataset's variables. If the Scatter Size field is set to "Size by Variable" on the Scatter page of the **Zone Style** dialog, this variable is used to calculate the scatter symbol size at each data point. The actual size of each symbol is determined by multiplying the value of the variable at each point by the Size Multiplier. If the Scatter Size field is not set to "Size by Variable", this field has no effect.

Size Multiplier

Enter the scale factor that multiplies the values of the Scatter-size Variable to size the scatter symbols. If the Scatter Size field on the **Zone Style** dialog is not set to "Size by Variable", this field has no effect. The Size Multiplier multiplied by the scatter variable value gives the size of the scatter symbol at a point, in units, specified by the following option buttons:

Grid Units/Magnitude

Select this to express the Size Multiplier in terms of grid units per unit of variable magnitude.

Cm/Magnitude

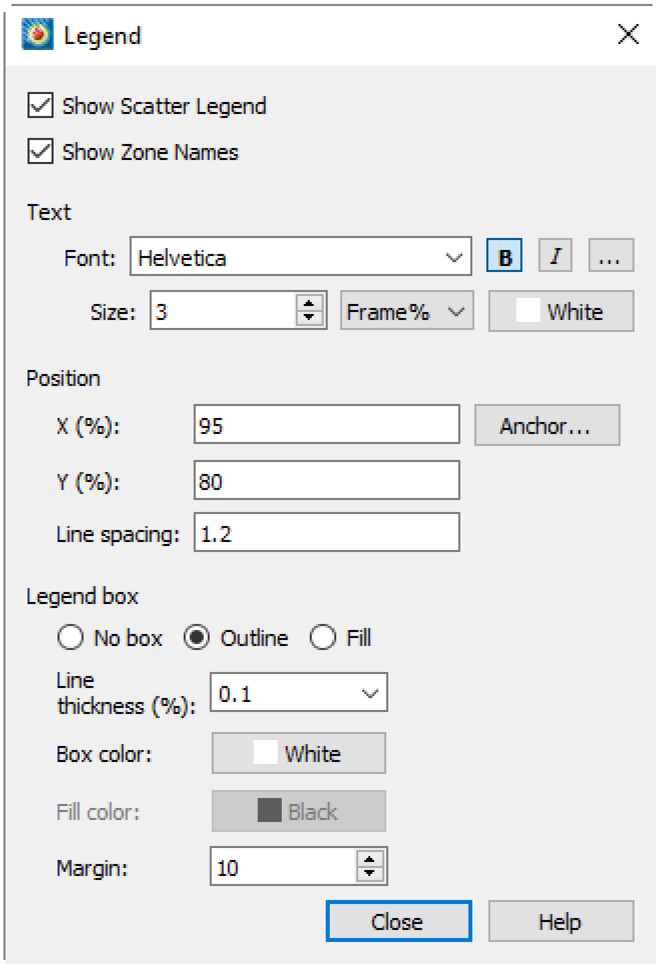
Select this to express the Size Multiplier in terms of screen centimeters per unit of variable magnitude.

Recalculate Size

Select to reset the Size Multiplier to Tecplot 360's initial value.

Scatter Legends

To include the scatter legend, select "Scatter Legend" from the **Scatter** sub-menu of the **Plot** menu. Select the following options in the **Scatter Legend** dialog.



Show Scatter Legend

Toggle-on to include a scatter legend in the plot.

Show Zone Names

Toggle-on to include zone names in the legend.

Text

Format the text for the legend by choosing a color and font, and specifying the text height. (See Font Folders and Fallback for more information on how fonts work with Tecplot 360.) Click the (...) button to display the Select Font dialog.

Position

Specify the location of the anchor point 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.

Legend Box

Select the type of box to draw around the legend (No Box, Outline, or Fill). If you choose Outline or Fill, 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

Choose a color for the legend box fill (Fill mode only).

Margin

Specify the margin between the legend text and legend box as a percentage of the text height.

Shade Layer

Although most commonly used with 3D surfaces, shade plots can also be used to flood 2D plots with solid colors, or to light source shade the exterior of 3D volume plots. In 3D plots, zone effects (translucency and lighting) cause color variation (shading) throughout the zone(s). Shading can also help you discern the shape of the plot.

Toggle-on "Shade" in the Plot sidebar to add shading to your plot. Use the Shade page of the **Zone Style** dialog to customize shading. Refer to Translucency for information on translucency and lighting zone effects.

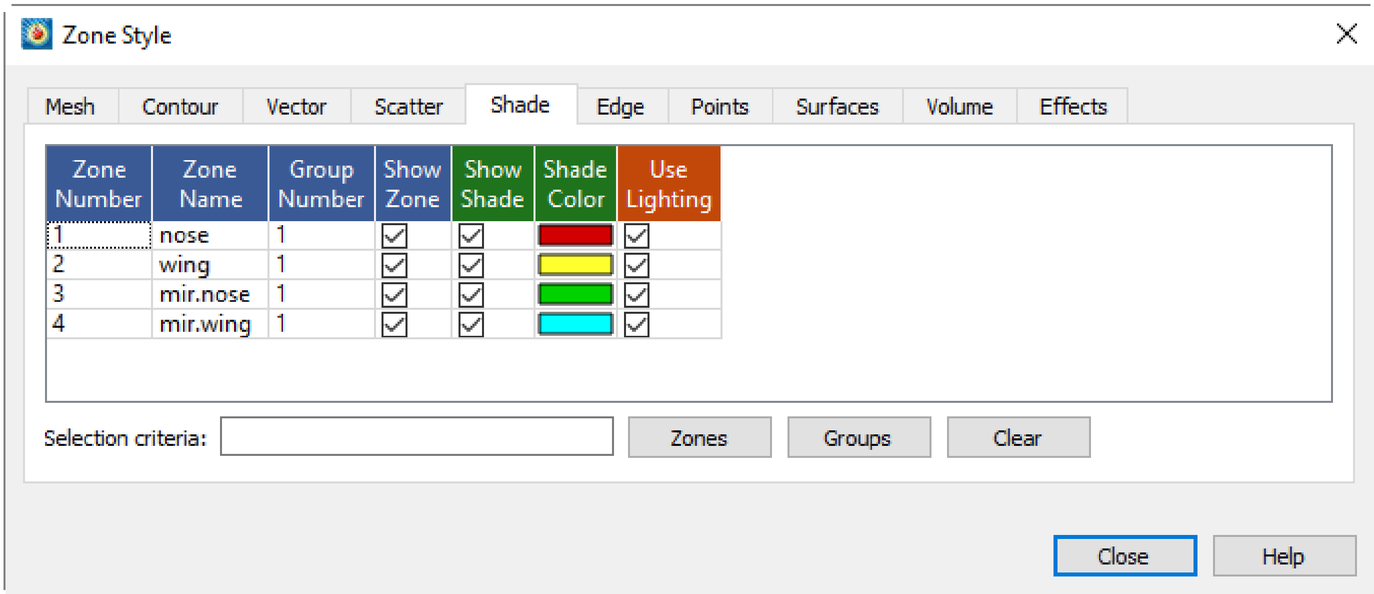
Shade plots require IJ or IJK-ordered, or finite element data. I-ordered, or irregular data cannot be used to create shade plots.

Shade Layer Modifiction

You can modify your shading attributes using the Shade page of the **Zone Style** dialog (accessed via the Plot sidebar or **Plot**→**Zone Style**).

In order for the changes made on the Shade page to be visible in your plot, the Shade layer must be turned on in the Plot sidebar.

You can control any of the following attributes from the Shade page of the **Zone Style** dialog:



Show Shade

Turns the shade layer on or off for each active zone.

Shade Color

Right-click to select the shade color using the Color Chooser. In 2D Cartesian plots, only solid zone flooding is available (i.e. no lighting effects).

Use Lighting

(3D only) Turns the lighting zone effect off or on. When "no" is selected, the shade color is used to uniformly color the zone. Refer to Translucency for information on translucency and lighting zone effects.

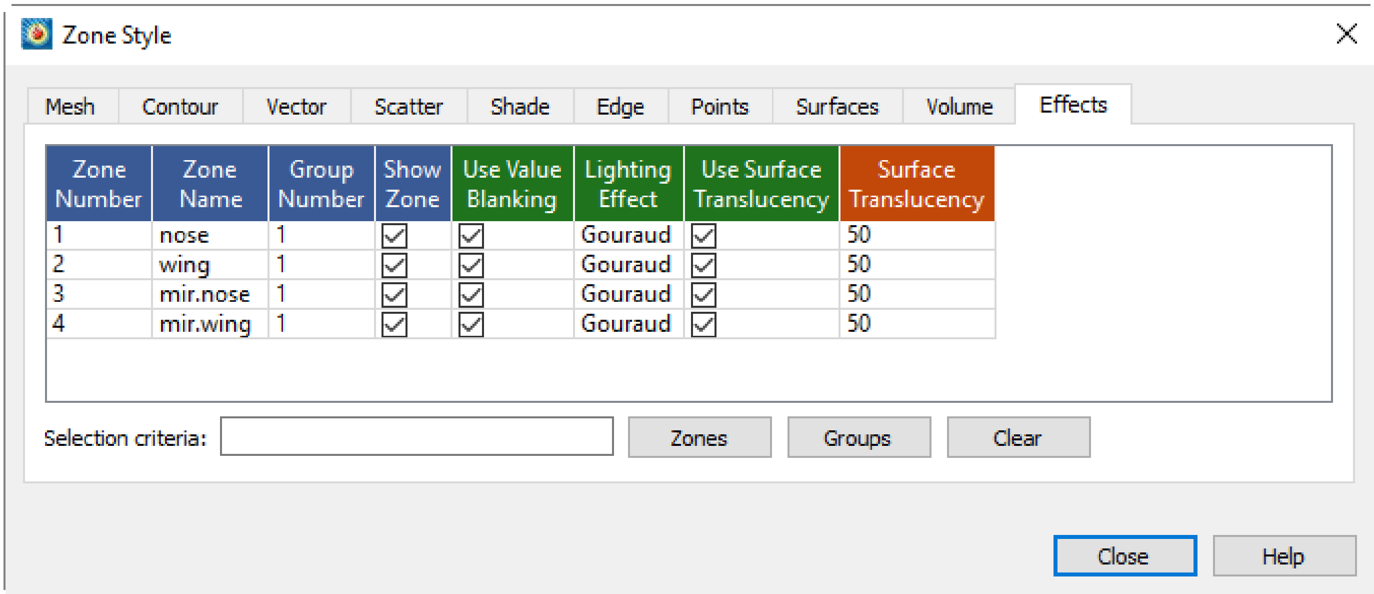
> For information on using the controls at the bottom of the Zone Style dialog to select zones by name, see the description of these at the end of Field Plot Modification and the Zone Style Dialog.

Translucency and Lighting

You can enhance the shade and contour zone layers in 3D plots using Translucency and Lighting Effects (referred to collectively as the "3D zone effects"). The 3D zone effects for streamtraces, slices, and iso-surfaces can be activated using their respective dialogs (accessed via the **Plot** menu or the Plot sidebar).

> For changes related to lighting or translucency to be visible, the desired effect must be toggled-on in the Show Effects section of the Plot sidebar.

The Effects page of the **Zone Style** dialog, which controls translucency and lighting effects, is shown below.



For information on using the controls at the bottom of the Zone Style dialog to select zones by name, see the description of these at the end of Field Plot Modification and the Zone Style Dialog.

Translucency

Turn on the translucency zone effect by toggling-on "translucency" in the Show Effects region of the Plot sidebar. When a zone is translucent, you may view objects inside or behind the zone. You can control the translucency of a zone using the Surface Translucency attribute in the Effects page of the **Zone Style** dialog. The level of translucency may be set to a value between 1 (nearly solid) and 99 (nearly invisible). There are nine pre-set percentages ranging from 10 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 36.

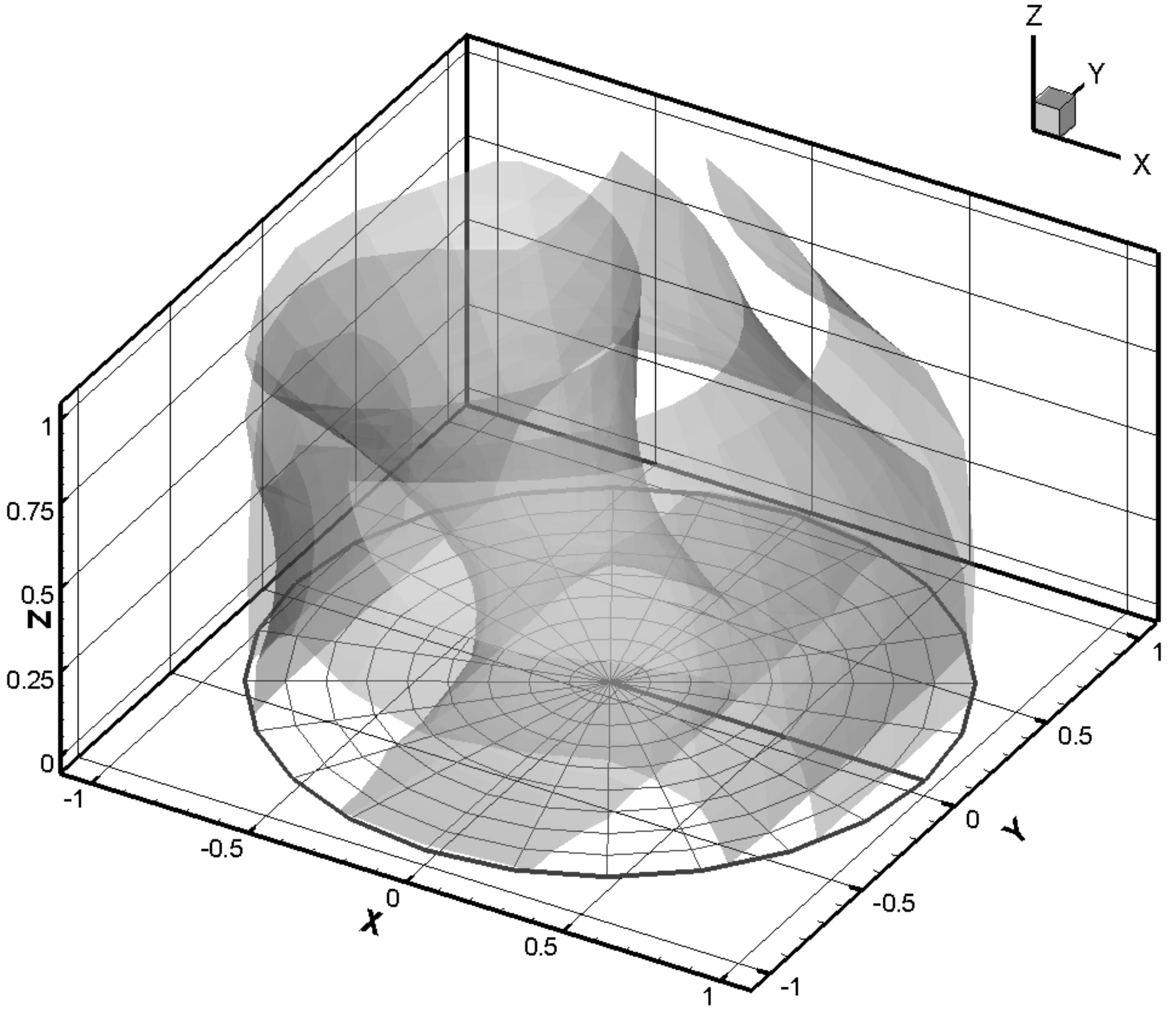


Figure 36. An example of a plot using translucency.

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. Use the Effects page of the **Zone Style** dialog to change translucency settings for zones.

The Translucency toggle-switch on the Plot sidebar applies only to zones, not to slices, streamtraces, or iso-surfaces. Translucency for those objects is controlled through their respective dialogs.

Translucency will only appear on your screen or in exported bitmap images. Translucent objects appear opaque in printouts and in exported vector images. See Hard Limits for more details.

Blanking

The Use Value Blanking column on the Effects page of the Zone Style dialog enables you to constrain the display of each zone - that is, instruct whether each zone should obey or ignore any blanking settings present in the plot. To constrain a zone to obey value blanking, select the zone, toggle-on the Use Value Blanking checkbox. To learn more about blanking, see Blanking.

Lighting Effects

There are two types of lighting effects: Paneled and Gouraud. Right-click in the Lighting Effect column of the Effects page to choose either Paneled or Gouraud shading for the selected zone(s).

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 3D light source.

Gouraud

This plot type offers smoother, more continuous shading than Paneled shading, but it also results in slower plotting and larger print files. Gouraud shading is not continuous across zone boundaries unless face neighbors are specified in the data.[8] Gouraud shading is not available for finite element volume zones when blanking is active; the zone's lighting effect reverts to Paneled shading in this case.

If IJK-ordered data with Surfaces to Plot is set to Exposed Cell Faces, faces exposed by blanking will revert to Paneled shading.

Figure 37 shows two shade plots. The one on the left uses a Paneled lighting effect and the one on the right uses a Gouraud lighting effect.

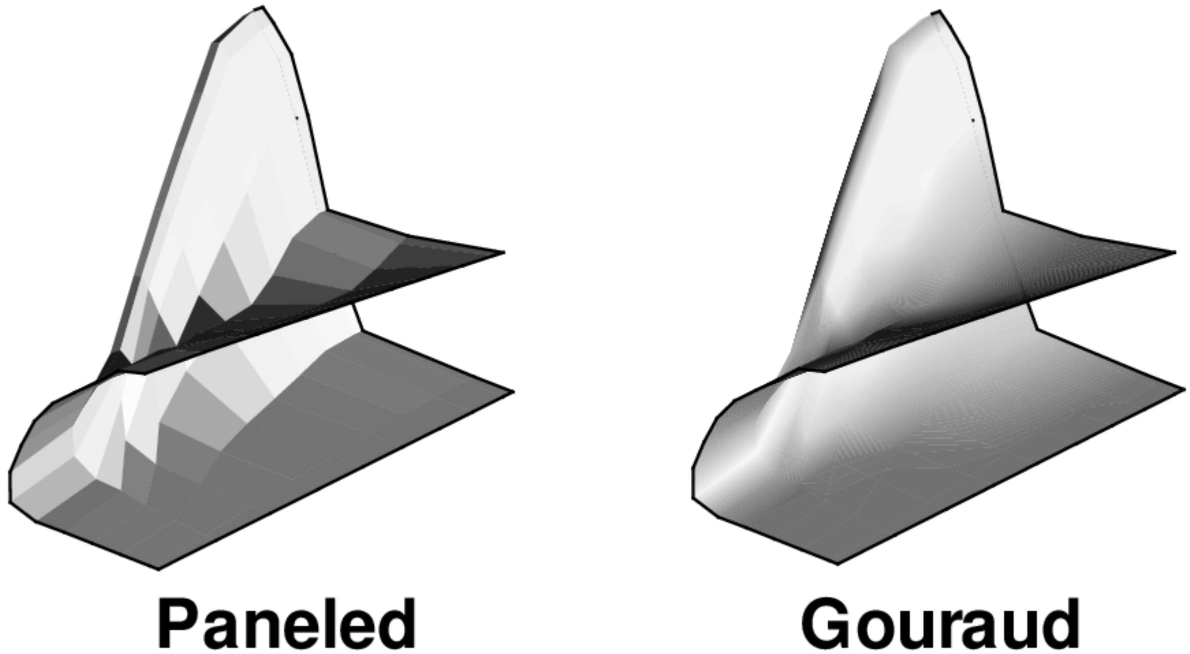
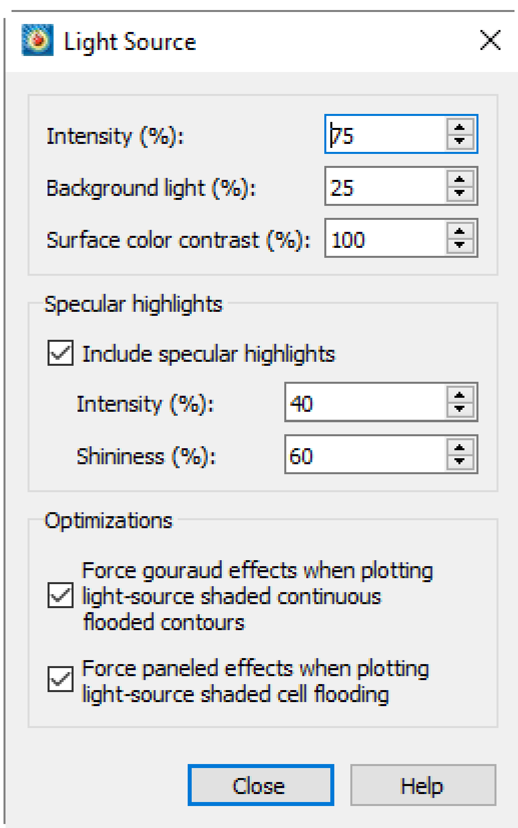


Figure 37. A comparison of the paneled (left) and Gouraud (right) lighting effects.

Three-dimensional Light Source

The light source is a point of light infinitely far from the drawing area. You can open the **Light Source** dialog (shown below) by selecting the button ( ) next to Lighting effect toggle the Plot sidebar, or by selecting "Light Source" from the **Plot** menu.



The **Light Source** dialog has the following options:

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

> 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 the 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).

Include Specular Highlights

Turns on/off specular highlight for all light-source shaded objects in the plot, adding the semblance of reflected light to 3D shaded or flooded objects.

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).

Lighting Optimizations

Some combinations of lighting type and plot style may result in very slow redrawing of plots.

Tecplot 360 provides lighting optimizations to avoid such conditions and instead draws a similar, but less computationally-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 Graphics Cache for information on the graphics cache.)

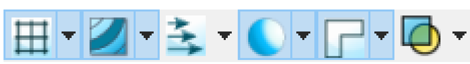
Moving the Light Source

You may move the light source interactively by clicking the sun button next to the lighting effect toggle in the Plot sidebar. With the light source tool active, click or drag in the workspace to position the light source in 3D space. Choose another tool from the toolbar or sidebar to exit this mode.

Slices

You can add slices to volume and surface zones in your plot in order to view X, Y, or Z planes within your data. With IJK-ordered data, you can also add slices on the I, J, or K planes. It is also possible to create slices with arbitrary orientations.

Slices can include lighting effects, contours, meshes, and more. To customize these and other attributes of slices, use the **Slice Details** dialog, accessible in the **Plot sidebar** or the **Plot menu**, or use the context menu and context toolbar.



The context toolbar appears above the context menu when you `right-click` a slice in your plot. This toolbar allows you to turn on or off the grid, contour, vector, shade, edge, and translucency layers for the selected slice(s). Additionally, you may adjust frequently-used style settings for each layer using the drop-down menu to the right of each, for example selecting a color for the grid (or choosing a variable by which to color it).

Tecplot 360 includes a simple interactive method for creating slices. Select the **Slice tool** in the toolbar or in the **Plot sidebar** to activate a crosshair cursor, then click in a volume or surface zone to "drop" a slice into that zone. You can then drag the slice to change its position. Refer to Slice Tool for more information on working with the **Slice** tool.

For additional control, additional slices, or another way to insert slices derived from the dataset, use the **Definition** page of the **Slice Details** dialog. You can open the **Slice Details** dialog by clicking the button next to the **Slices toggle** in the **Plot sidebar** or by choosing **Slices** from the **Plot menu**.

Interactively created slices are derived from the dataset and are defined by a constant X, Y, or Z location (or constant I, J, or K indexes, for IJK ordered zones). Tecplot 360 considers this type of slice as part of your plot’s style and does not add it to the dataset unless you extract it to a zone (using **Data**→**Extract**→**Slices**).

Interactively Created Slices

Use the **Slice Details** dialog to customize interactively created slices (that is, slices derived from your dataset). `Click` the  button next to the **Slices checkbox** in the **Plot sidebar**, or select **Slices** from the **Plot menu**, to launch the **Slice Details** dialog. To add slices to your plot in the dialog, toggle-on **Show Group** *n* in the **Slice Details** dialog.

The **Slice Details** dialog includes the following pages: Slice Definition Page, Contour Page, Vector Page, Other Page, and Animate Page.

> You must toggle-on `Show Group` *n* in order for the changes made in the **Slice Details** dialog to be visible in your plot.

Slice Groups

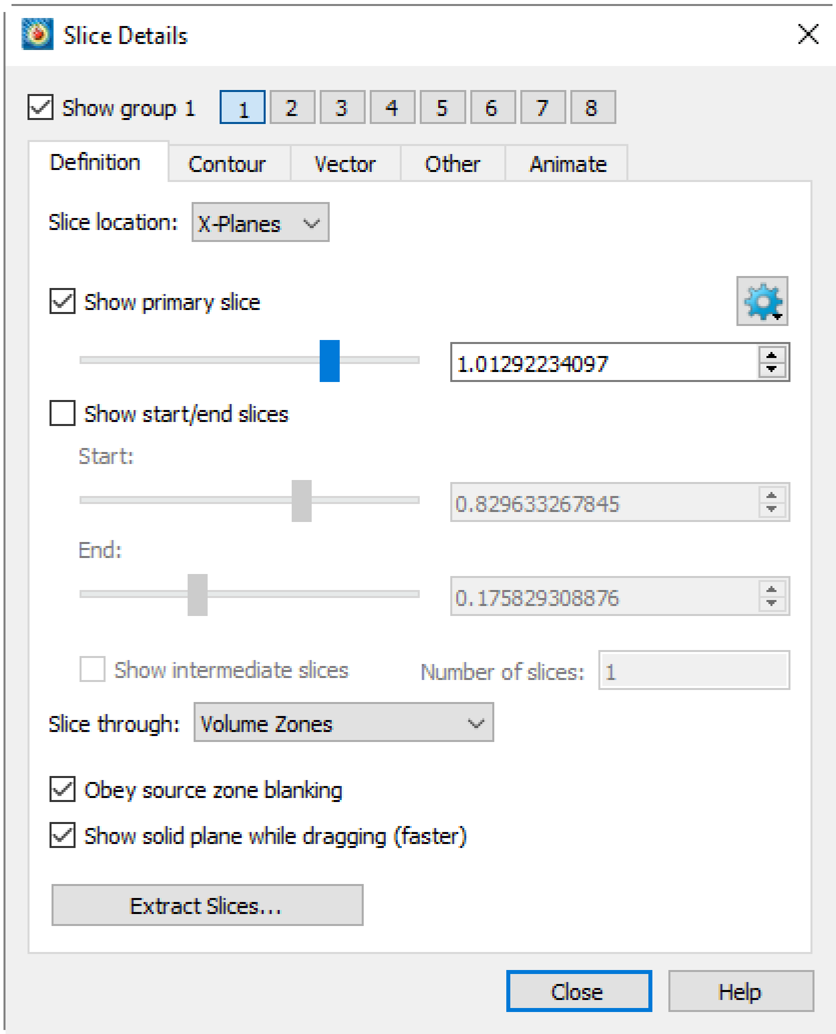
Up to eight different slice groups can be set. Each slice group can use different slice planes or different ranges for the same slice plane. Changing the settings in the **Slice Details** dialog allows you to make the appearance of each slice group unique. The slice group is specified using the numbers at the top of the **Slice Details** dialog

> You must toggle-on `Show Group` *n* (where $n = 1–8$) in order to include the **Slice group** in your plot.

Slice Definition Page

Use the **Definition** page of the **Slice Details** dialog to customize the position of the active slice group (accessed via the **Plot sidebar** or **Plot**→**Slices**).

You can toggle-on **Show Start/End Slices** and use the **Start slider** to move the first (start) slice, or type in the slice position. Activate the last (end) slice and move it with the **End slider**.



The following options are available:

Slice Location

Select which plane the slice is drawn on (X, Y, Z, I, J, or K). You may also choose Arbitrary to place the slice on an arbitrary plane; see Arbitrary Slice Orientation.

Show Primary Slice

Toggle-on to include the primary slice (first slice placed) in your plot. Use the slider or the text field to position the primary slice.

Show Start/End Slices

Toggle-on to include start and end slices in your plot. Use the corresponding sliders or text fields to position the slices.

Show Intermediate Slices

Toggle-on to show intermediate slices evenly distributed between the start and end slices.

Num Slices

Enter the number of intermediate slicing planes in the text field. (Range 1-5000.)

Range for all Sliders

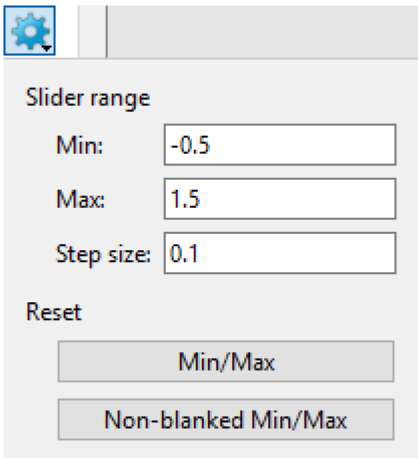
Click the gear icon next to **Show Primary Slice** to reveal a flyout dialog that sets the range of the sliders.

Min, Max, Step Size

Specify the start, end, and step for the slider range.

Reset Slider Range

Automatically selects the min/max of values based on Slice location or the min/max of the non-blanked values based on Slice location. **Non-blanked Min/Max** is disabled for I-, J-, and K-Planes.



Slice Through

Choose to slice through volume zones, surface zones, or the surfaces of volume zones.

Obey Source Blanking

When active, slices are subject to any blanking used for the data. When inactive, slices are generated for blanked and unblanked regions. See also Blanking.

Show solid plane while dragging

Toggle-on to show a solid-colored plane in place of the slice while dragging. Since the data within the slice does not need to be drawn to the screen, this can make dragging noticeably faster and smoother, particularly with large data sets.

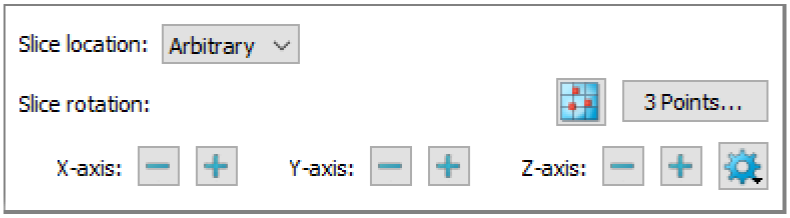
> The **Show solid plane while dragging** checkbox is a global setting that applies to all slice groups, not just the selected group. It is not available if the slice plane is I, J, or K.

Extract Slices

Select the Extract Slices button to open the Extract Slices dialog. See Extracting Slices to Zones for more information.

Arbitrary Slice Orientation

To orient slices in an arbitrary direction, choose **Arbitrary** from the **Slice Location** menu. As with other slices, you may specify origin points for a primary slice and/or for start and end slices. Slices pass through the indicated origin point(s), so you can easily align the edge of a slice or group of slices along some other feature of the plot, such as an axis. If intermediate slices are activated, they are drawn equally spaced between the slices defined by the start and end origins.



An arbitrarily-oriented slice may be manipulated interactively. When the slice tool is active, a slice-normal interactor appears in the plot.

- Drag the far end of the interactor to change where the normal points and thus the orientation of the slice(s).
- Drag the near end of the slice normal to move the origin point of the slice.
- If multiple slices are active, you may click a slice to move the normal to that slice.

> Interactive orientation of arbitrary slices is not available with scaled axes or perspective view. In these cases, the interactor does not appear.

You may adjust the orientation of the slice in small increments using the + and - buttons for the X, Y, and Z axes to rotate the slice a step at a time around an axis.

With an arbitrary slice, you may also:

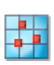
Orient a Slice by Specifying a Normal

Click the gear icon to enter a normal vector in a flyout dialog.

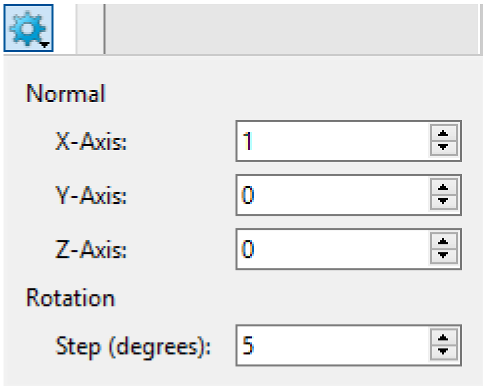
Orient a Slice by Entering Three Points

Click the Three Points button to specify three points on the cutting plane using the Enter 3 Points dialog.

Orient a Slice by Probing Three Points

Click the  three-point probe icon on the Definition page of the Slice Details dialog to specify three points on the cutting plane by clicking the plot.

Orient a Slice by Specifying a Normal



Click the gear icon to reveal a flyout dialog that lets you choose the orientation of the slice by numerically specifying its normal. A normal vector is established between the slice origin and the specified coordinates, and the slice is oriented perpendicular to this vector, as shown below.

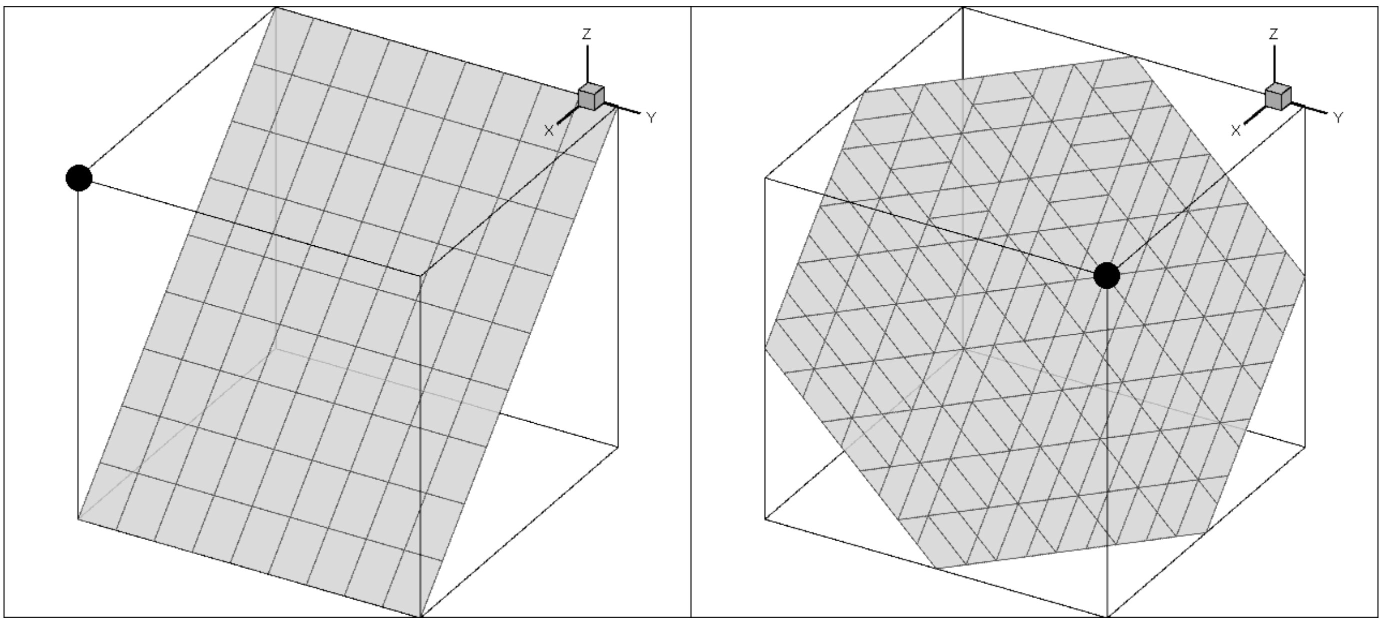
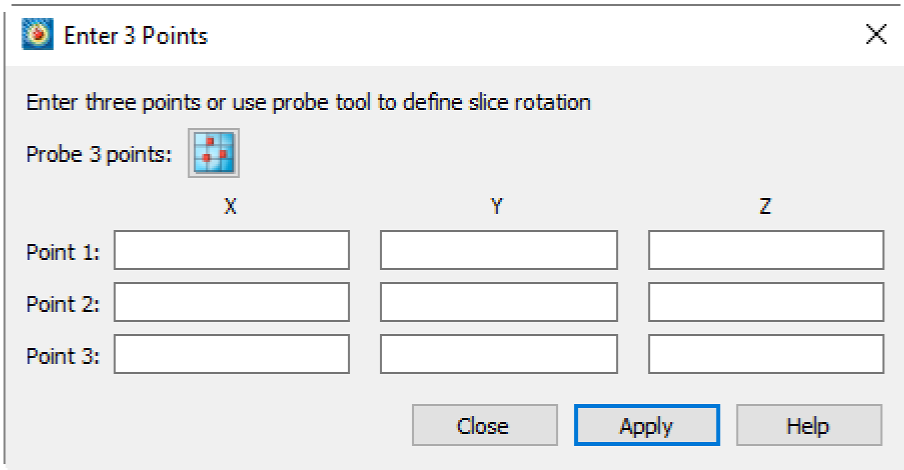


Figure 38. Arbitrarily-oriented slices in a unit cube. 0, 0, 0 is at back corner; the black dot is the specified normal. Left: normal at 0, 1, 1. Right: normal at 1, 1, 1.

You can also adjust the step value by which the + and - buttons rotate the slice in this flyout.

Orient a Slice by Entering Three Points



You can orient a slice by specifying the coordinates of three points on a plane. These points must form a triangle; they cannot be coincident or collinear.

Click the **3 Points** button in the **Slice Details** dialog to open the **Enter 3 Points** dialog, shown here, then enter the X, Y, and Z coordinates of the three points. When you click **Apply**, the origin and normal vector of the slice are recalculated so that the slice plane passes through all three specified points. (The third entered point is used as the slice origin.)

> The values in the **Enter 3 Points** dialog are not updated when the slice orientation is adjusted using the other available methods.

In this dialog, you may also click the three-point probe button to Orient a Slice by Probing Three Points.

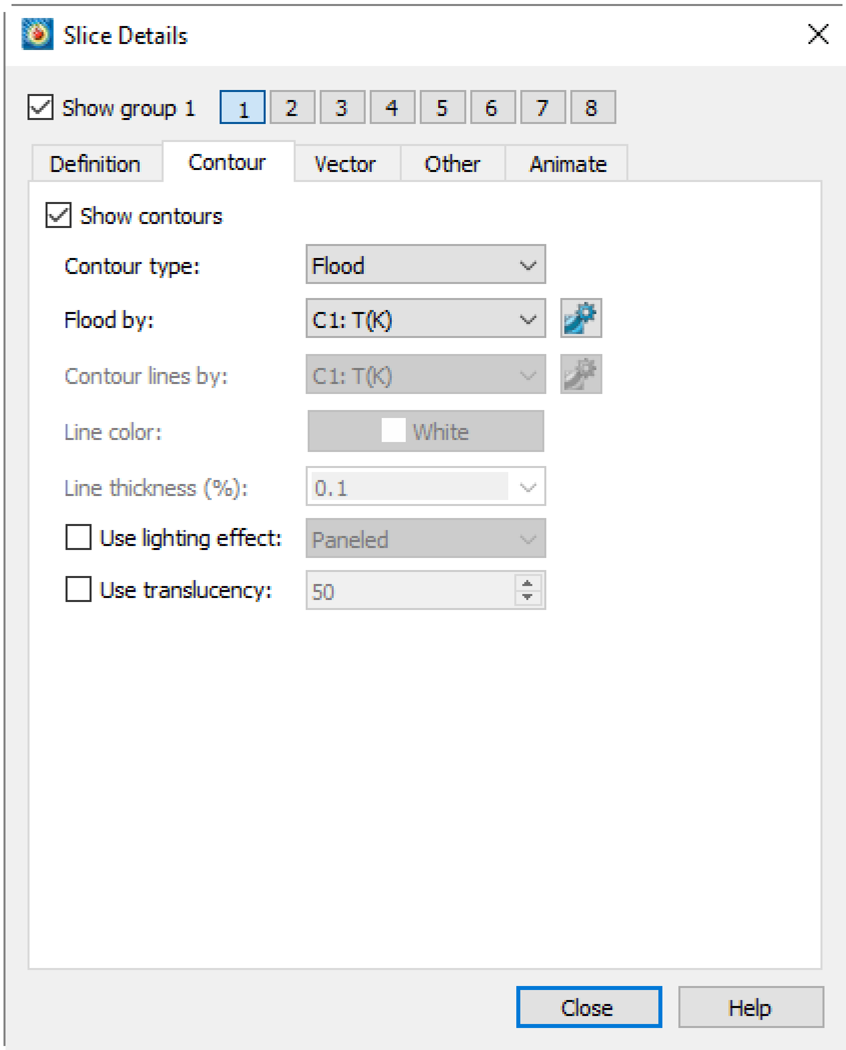
Orient a Slice by Probing Three Points

In the **Slice Details** dialog or the **Enter 3 Points** dialog, click the three-point probe button, then click three points in your plot. The X, Y, and Z coordinates of the three points are collected in the **Enter 3 Points** dialog. After the third point is clicked, the slice's origin and normal are recalculated so that the cutting plane passes through all three clicked points. The third point is used as the slice's origin.

> The probe tool keyboard shortcuts may be used when clicking to change the points selected, for example choosing the nearest data point by holding the `Control` key while clicking. See Probe Tool for other shortcuts.

Contour Page

Use the Contour page to control the contour attributes of the active slice group (determined by the number buttons on the top of the page).



The following options are available:

Show Contours

Select this check box to show contours.

Contour Type

Select the contour type 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.

 **Icon**

Use this button to bring up the Contour & Multi-Coloring Details dialog.

Contour Lines by

If you chose contour lines or lines and flood, select the contour group by which to draw the lines.

 **Icon**

Use this button to bring up the **Contour Details** dialog.

Line Color

Choose the line color in the Color Chooser. 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

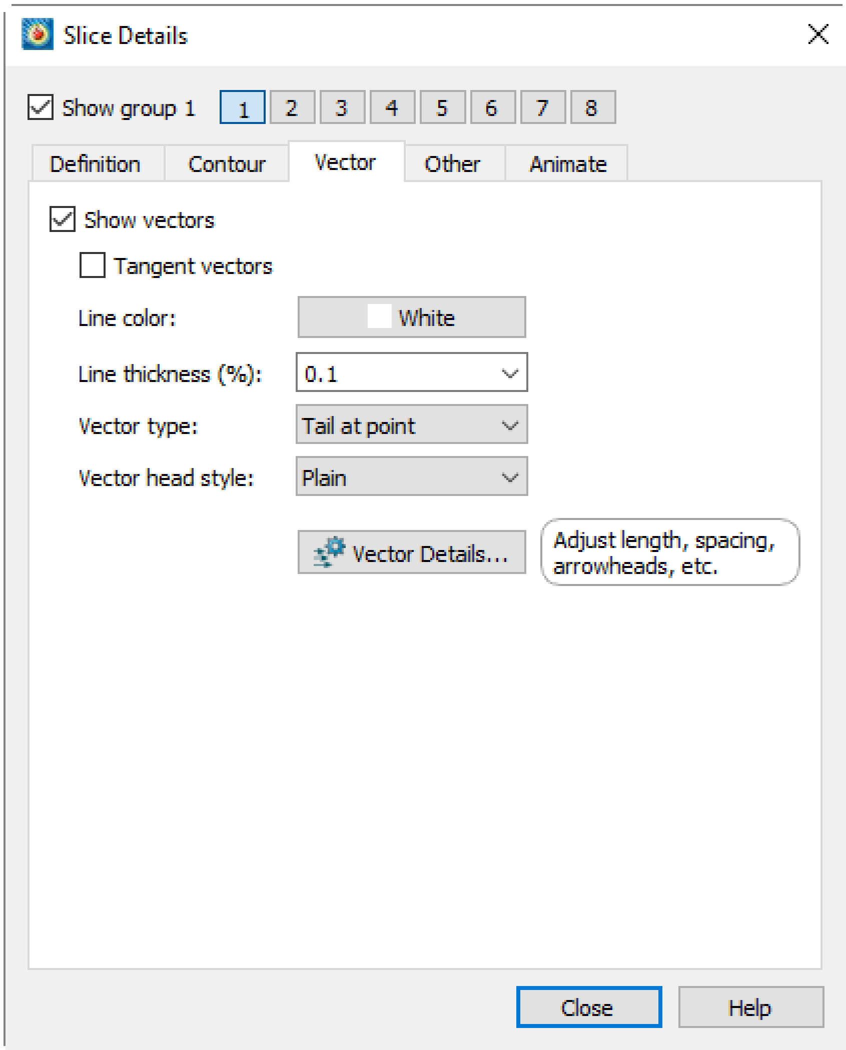
Select this check box to enable the lighting effect drop-down menu where you may choose "Paneled" or "Gouraud" shading. See Lighting Effects for more details on lighting effects.

Use 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).

Vector Page

Use the Vector page of the **Slice Details** dialog to control the vector attributes of the active slice group (determined by the group number buttons on the top of the page).



The following options are available:

Show Vectors

Select this check box to show vectors.

Tangent Vectors

Select to use tangent vectors for your slices. See Vector Plot Modification for more information.

Line Color

Choose the line color from the Color Chooser. 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 height. 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

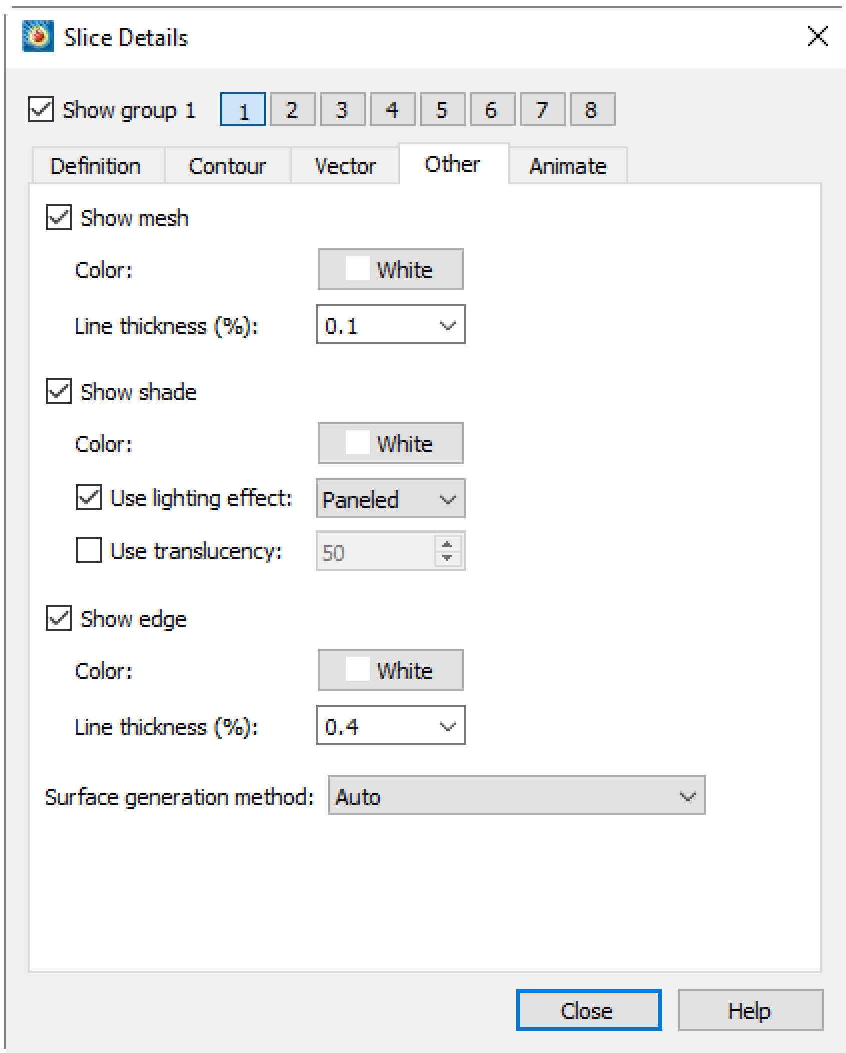
Use this drop-down to set the vector head style for your slices. Choose from Plain, Filled, and Hollow.

Vector Details...

Opens the Vector Details dialog to adjust Vector length, spacing, arrowheads, etc.

Other Page

The Other page of the **Slice Details** dialog controls the mesh, shade, and edge attributes of the active slice group. (The active slice group is determined by the group number buttons on the top of the page.)



The following options are available:

Show Mesh

Select this check box to show mesh lines.

Color

Choose the line color in the Color Chooser. 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 on the Contour page of this dialog.

Color

Choose the shade color from the Color Chooser. 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). By default, slice translucency is toggled-on at 10 percent when your plot is loaded.

Show Edge

Select this check box to show selected edge lines on all slices.

Color

Choose the edge color from the Color Chooser. Multi-color and RGB coloring are not available.

Line Thickness

Specify the edge 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.

Surface Generation Method

Determines how the surface is generated.

Auto

Auto selects one of the surface generation algorithms best suited for the zones participating in the slice generation. "All Polygons" is used if one or more of the participating zones is polytope, otherwise slices use "Allow Quads".

Allow Quads

Allow Quads can produce quads or triangles, and the resulting surface more closely resembles the shape of the volume cells from the source zone. Since the quads are not arbitrarily divided into triangles, no biases are introduced, and the resulting surface may appear smoother. This method is preferred when the source zone is FE-Brick or IJK-Ordered and the surface is aligned with the source cells.

All Triangles

All Triangles is an advanced algorithm that can handle complex saddle issues and guarantees that there will be no holes in the final surface. As the surface is composed entirely of triangles, it can be delivered more efficiently to the graphics hardware.

All Polygons

All Polygons is similar to the "All triangles" method except that all interior faces generated as a result of triangulation that are not part of the original mesh are eliminated. This preserves the original mesh of the source zones on the resulting slice.

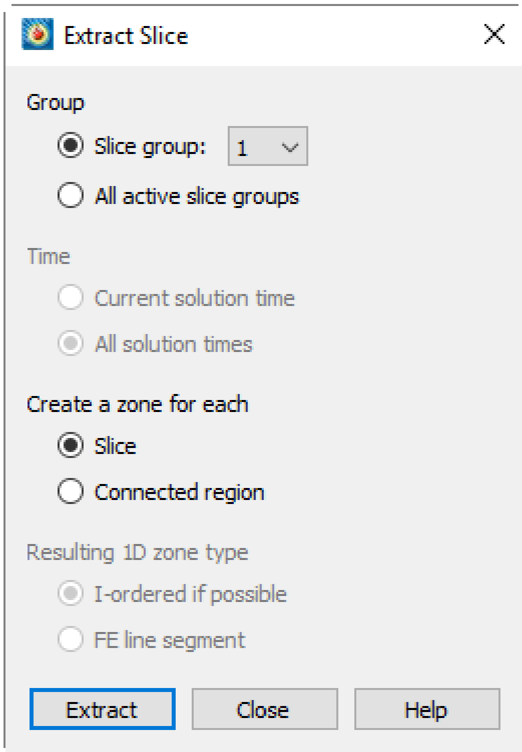
Animate Page

See Slice Animation.

Extracting Slices to Zones

Normally, slices are derived from the dataset "on the fly" and do not add any data to the dataset. To extract slices to Tecplot zones, allowing you to retain them even if the slice details are changed, use **Data→Extract→Slices**.

In most cases it is not necessary to extract slices to zones, as it is possible to apply most style options directly to a slice. When you need to display multiple sets of slices in various directions, or simply want to "freeze" a slice for some other reason, you may wish to extract at least some of the slices to zones.



Extracting slices has the following options:

Group

Select between an individual group extraction or all active groups extraction, where active means slices visible on the screen.

Slice Group

Select from the dropdown menu which group is extracted to a zone.

All Active Slice Groups

Extracts slice groups that are set to active.

Time

Specify between extracting a slice for all solution times or the current solution time.

Current Solution Time

Extract a slice for the active solution time.

All Solution Times

Extract a slice for each solution time.

Create a zone for each

Specify how each slice is extracted into single or multiple zones.

Slice

Extracts each slice to a single zone.

Connected Region

Extracts each slice to its connected regions. Connected regions are defined by zone boundaries and areas of disconnected meshes.

Resulting 1D Zone Type

Only active when the slice source is not a volume.

I-ordered If Possible

Creates an I-ordered zone for each slice zone that is a single connected region.

FE Line Segment

Creates all extracted linear zones as FE-LineSeg zones.

Values in the extracted slice are nodal unless Contour Type is set to Primary Value flood on the Contour page of the Slice Details dialog. In this case, the value location for the extracted slice is the same as for the source zones.

Streamtraces

A streamtrace is the path traced by a massless particle placed at an arbitrary location in a steady-state vector field. Streamtraces may be used to illustrate the nature of the vector field flow in a particular region of the plot. See Calculating Particle Paths and Streaklines for information on adding streaklines and particle paths to your plot.

Because streamtraces are dependent upon a vector field, you must define vector components before creating streamtraces in Tecplot 360. However, it is not necessary to activate the Vector zone layer to use streamtraces.

To add streamtraces to your plot, toggle-on "Show Streamtraces" in the **Streamtrace Details** dialog or in the Plot sidebar and use either the Add Streamtrace tool  in the toolbar or the Plot sidebar, or the **Create Streams** button on the Position page of the **Streamtrace Details** dialog, to specify the location of your streamtraces.

When working with the Add Streamtrace tool, click to add individual streamtraces, or drag to seed a rake (group) of streamtraces.

To create streamtraces with a format other than Surface Line, select a format from the "Create Streamtraces with Format" drop-down menu on the Position page of the **Streamtrace Details** dialog.

If you are drawing a rake on concave 3D volume surfaces using the Add Streamtrace tool, hold down the Shift key to draw the rake outside of the data.

There are two main categories of streamtraces:

Surface line streamtraces (or streamlines)

Surface streamtraces are confined to the surface on which they are placed. They can be placed in zones displayed as a 2D or 3D surface, or on a displayed boundary of a 3D zone, such as the K=1 face of an IJK-ordered zone. If you try to place surface streamtraces in the interior of a zone displayed as a 3D volume, an error message appears, and no streamtraces are drawn. See Lines Page. When surface streamtraces are placed on a no-slip boundary surface, they will propagate according to the flow field very near the surface (see Surface Streamtraces on No-slip Boundaries for more information).

Volume streamtraces

Volume streamtraces can be created in 3D volume zones only (IJK-ordered or FE-volume zones). See Rod/Ribbon Page. Volume streamtraces are subdivided into three categories:

- Volume Lines, or volume streamlines.
- Volume Ribbons, or streamribbons.
- Volume Rods, or streamrods.

If you have added streamtraces to your plot, but cannot see them, go to the Volume page of the **Zone Style** dialog and verify that Show Streamtraces is set to "Yes". Refer to Derived Volume Object Plotting for details.

Streamtrace Details Dialog

You can control the style of your streamtraces using the **Streamtrace Details** dialog (accessed via **Plot**→**Streamtraces** or the button to the right of the Streamtraces checkbox in the Plot sidebar). These style attributes affect all streamtraces in the active frame, including those already placed. They do not affect extracted streamtrace zones, discussed in Streamtrace Extraction as Zones, because these are ordinary ordered zones, and not streamtraces.

In order for the changes made on the Streamtrace Details dialog to be visible in your plot, you must have Show Streamtraces toggled on in the dialog, or Streamtraces toggled on in the Plot sidebar.

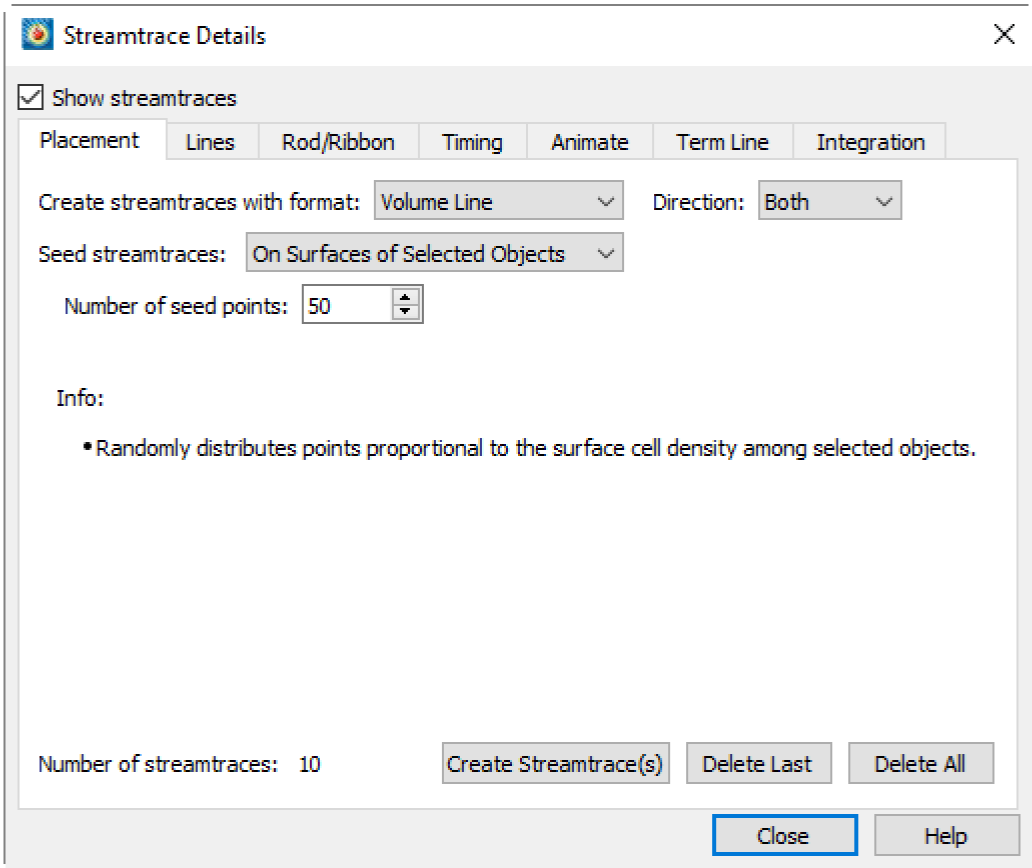
The Streamtrace Details dialog has seven pages:

- Placement Page
- Lines Page
- Rod/Ribbon Page
- Animate Page
- Term Line Page
- Integration Page

The dialog includes a checkbox at the top to globally turn the display of streamtraces on or off. This is available on all pages of the dialog and is the same as the corresponding checkbox in the Plot sidebar.

Placement Page

Use the Placement page of the **Streamtrace Details** dialog (accessed via the Plot sidebar or **Plot→Streamtraces**) to control the next streamtrace or streamtrace rake to be placed.



Alternatively, you can add streamtraces using the Add Streamtrace tool. See also Add Streamtrace.

The following options are available:

Create Streamtraces with Format

Choose the format for the next streamtrace from the drop-down. The options are as follows:

Surface Line

Two-dimensional and 3D surface streamlines. Surface lines are confined to the surface upon which they are placed, which may be surface zones or displayed boundaries of volume zones. If placed in the interior of a 3D volume zone (such as on a slice through the zone), these streamtraces are not plotted.

Volume Line

Three-dimensional volume streamline plotted through 3D space. The streamline path is integrated in three dimensions within the 3D volume field.

Volume Ribbon

Three-dimensional volume streamtrace with a defined thickness that twists in accordance with the local stream-wise vorticity of the vector field: a streamribbon. When you select this option, you should also check the ribbon width on the Rod/Ribbon page of the **Streamtrace Details** dialog. The width affects all streamtraces, including those already placed. The default width is automatically calculated based on the extent of your data, but it may still be too large. The center of the streamribbon is a 3D volume streamline. The streamribbon rotates about this streamline in accordance with the local vector field. Streamribbons have an orientation at each step.

Volume Rod

Three-dimensional volume streamtrace with a defined thickness and a polygonal cross-section: a streamrod. The cross-section of a streamrod rotates around a volume streamline in accordance with the local stream-wise vorticity. The center of the streamrod is a regular 3D volume streamline. Streamrods have an orientation at each step. As with streamribbons, you should check the rod width on the Rod/Ribbon page of the **Streamtrace Details** dialog, as well as the number of rod points (three, by default). The number of points indicates the cross-sectional shape of the rod. Three is an equilateral triangle; four, a square; five, a regular pentagon; and so forth. Like the width parameter, the number of points applies to all streamrods, including those already placed.

Seed Streamtraces

Choose the method for seeding the streamtraces. This will determine the options available in the central portion of the dialog.

Using Streamtrace Placement Tool

Click on the plot to seed a stream at the clicked point. Click and drag to seed a rake (a series of streamtraces) of the specified number of equally-spaced points along the path.

On Surfaces of Active Zones

Tecplot 360 randomly distributes the specified number of seed points on the active zones.

On Surfaces of Selected Objects

Tecplot 360 randomly distributes the specified number of seed points on the surfaces of the selected objects.

By Entering XYZ Positions

Specify the placements of the seed point or rake numerically by entering XYZ coordinates in the position fields, which appear when you choose method of placing streamtraces.

By Entering IJK Positions

Same as above, except you enter IJK coordinates in the position fields. In this case you must also choose a zone.

Direction

Select the stream integration direction from the following options:

Forward

Select for forward integration from the starting point.

Backward

Select for backward integration from the starting point. When the streamlines are calculated backwards, the arrowheads still point in the forward direction.

Both

Select for both forward and backward integration from the starting point. (For streamribbons and streamrods, you should avoid this option.)

Number of Seed Points

Enter the number of seed points here. When placing streamtraces using XYZ or IJK positions, this field is available only when creating a rake; the specified number of streamtraces will be created at equal intervals between the start and end points of the rake.

Position fields

(Only for XYZ or IJK Positions) - Specify the position of the start point, or the range of the rake (series of streamtraces), in IJK or XYZ coordinates.

Zone (Only for IJK)

Select from the drop-down the zone for which the I, J, (and K) indices are being specified.

Create Rake

Select to identify the starting position as the start of a rake and to activate the Rake Ending Position fields.

Streamtrace Start Position

Specify the starting position for a single streamtrace, or (if "Create Rake" is selected) the beginning of a rake of streamtraces. There are two or three fields, labeled either X, Y, (and Z) or I, J, (and K). Enter the desired value in each field, or use the up and down arrows to increase or decrease the values.

Rake End Position

(Only if Create Rake is selected) - Specify the end position for a rake of streamtraces. There are two or three fields, labeled either X, Y, (and Z) or I, J, (and K). Enter the desired value in each field, or use the up and down arrows to increase or decrease the values.

Create Streamtraces

Click to seed the streamtrace or rake of streamtraces. Not available when using the streamtrace placement tool; click on the plot instead.

Number of Streamtraces

(Information only) - The number of streamtraces currently placed.

Delete Last

Select to delete the last streamtrace or rake placed.

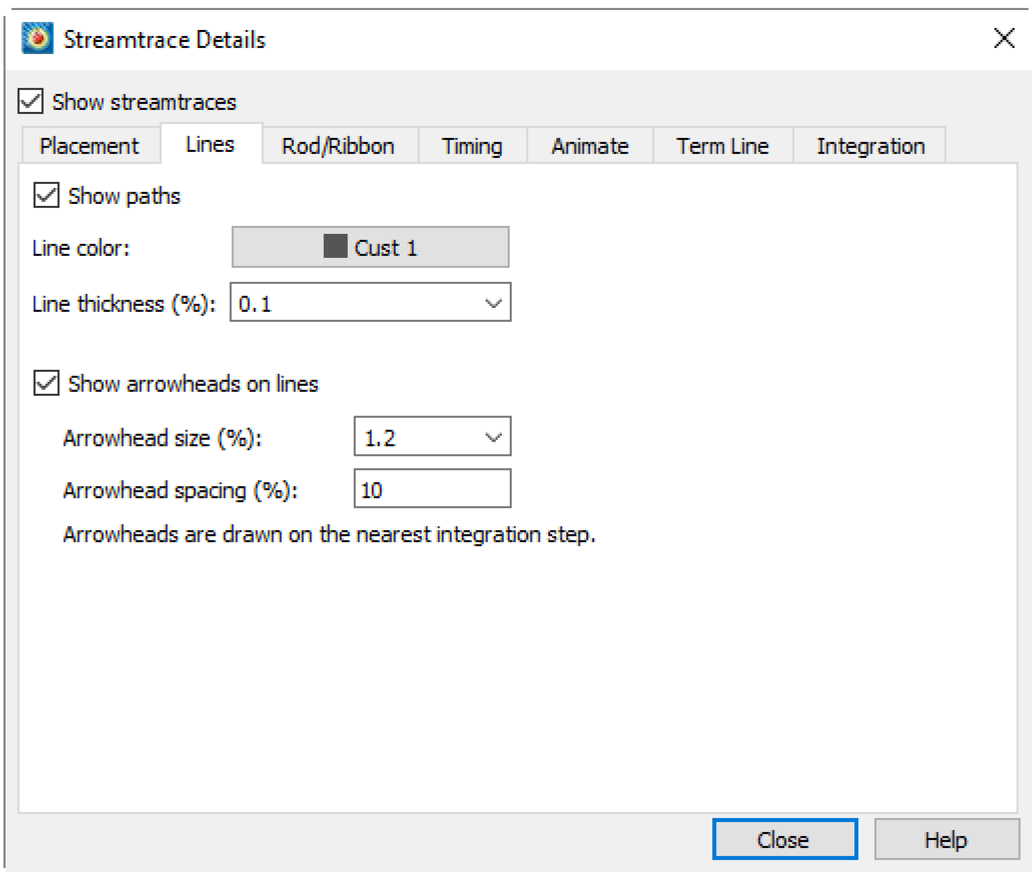
Delete All

Select to delete all streamtraces in the current plot.

> 1, 2, 3, 4, 5, 6, 7, 8, 9 - When the Streamtrace tool is selected, press 1-9 to change the number of streamtraces to add when placing a rake of streamtraces.

Lines Page

Surface streamtraces or streamlines are confined to the surface on which they are placed. They can only be placed in zones displayed as a 2D or 3D surface. If you try to place streamlines in a zone displayed as a 3D volume, an error dialog appears, and no streamlines are drawn. The following attributes may be set with the Lines page of the **Streamtrace Details** dialog. You cannot customize streamtraces using the Lines page until after at least one streamtrace has been drawn.



Show Paths

Toggle-on to include streamtraces in your plot.

Line Color

Select the color for all streamtraces in the Color Chooser. 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:

Line Thickness

Enter a value, or choose a pre-set value for the streamline thickness (as a percentage of the frame height for 2D lines and as a percentage of the median axis length for 3D surface lines and volume lines), or choose a pre-set value from the drop-down menu.

Arrows

Toggle-on **Show Arrowheads on Lines** to display arrowheads along all streamlines (surface and volume) in the active frame. Arrows are not shown on volume ribbons or volume rods. You can also control the following attributes of the displayed arrows:

Arrowhead Size

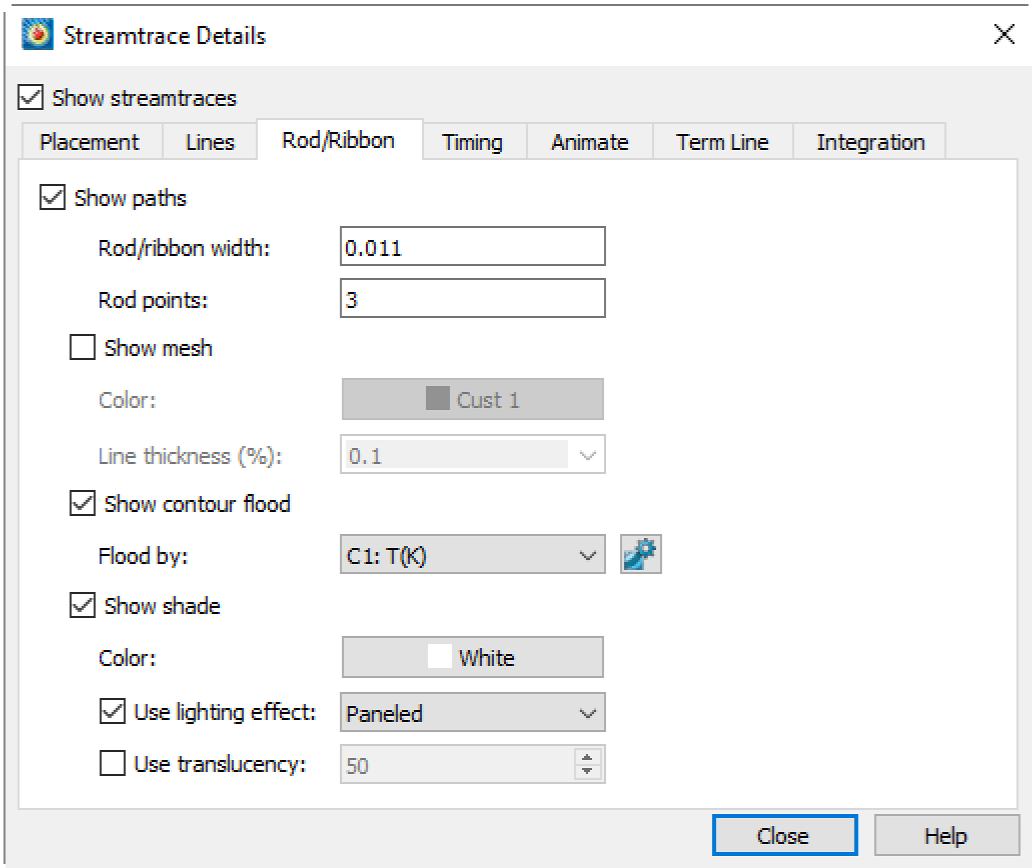
Either enter a value for the arrowhead size (as a percentage of the frame height), or choose a pre-set value from the drop-down menu.

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.

Rod/Ribbon Page

The following attributes may be set with the Rod/Ribbon page of the **Streamtrace Details** dialog. They affect volume ribbons and volume rods only. You cannot customize streamtraces using the Rod/Ribbon page until after at least one streamtrace has been drawn.



Show Paths

Toggle-on to include streamtraces in your plot.

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 them individually by creating a set of streamtraces with a specific width, extracting the set as a zone, and then configuring a new set of streamtraces with the second width. See Streamtrace Extraction as Zones.

Rod Points

Volume rods have a polygonal cross-section; this parameter tells Tecplot 360 what that cross-section should be. (Three is an equilateral triangle, four is a square, five is 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 the set as a zone, then configure a new set of streamtraces with the second cross-section. See Streamtrace Extraction as Zones.

Show Mesh

Toggle-on to display a mesh.

Color

Select a mesh color in the Color Chooser, or choose a custom color or multi-color.

Line Thickness

Select a line thickness from the drop-down menu, or enter your own number in the text field.

Show Contour Flood

Toggle-on to display contour flooding.

Flood by

Select which contour group to flood by. Use this button to display the Select Contour Variables dialog.

Show Shade

Toggle-on to display shading.

Color

Select a shade color in the Color Chooser. Multi-color and RGB coloring are not available (use contour flooding instead).

Use Lighting Effect

Toggle-on to enable the lighting effect drop-down menu, from which you can select "Paneled" or "Gouraud" shading.

Use Surface Translucency

Toggle-on to enable the surface translucency text field, where you can set the surface translucency from one (nearly opaque) to 99 (nearly transparent).

Timing Page

Use the Timing page of the **Streamtrace Details** dialog (accessed via the Plot sidebar or **Plot→Streamtraces**) to control timed markers for streamlines, and timed dashes for all types of streamtraces. Stream markers are drawn at time locations along streamlines. The spacing between stream markers is proportional to the magnitude of the local vector field.

Stream markers are symbols plotted along streamtrace paths to identify the positions of particles at certain times. Figure 39 shows a plot with both streamtrace markers and dashes.

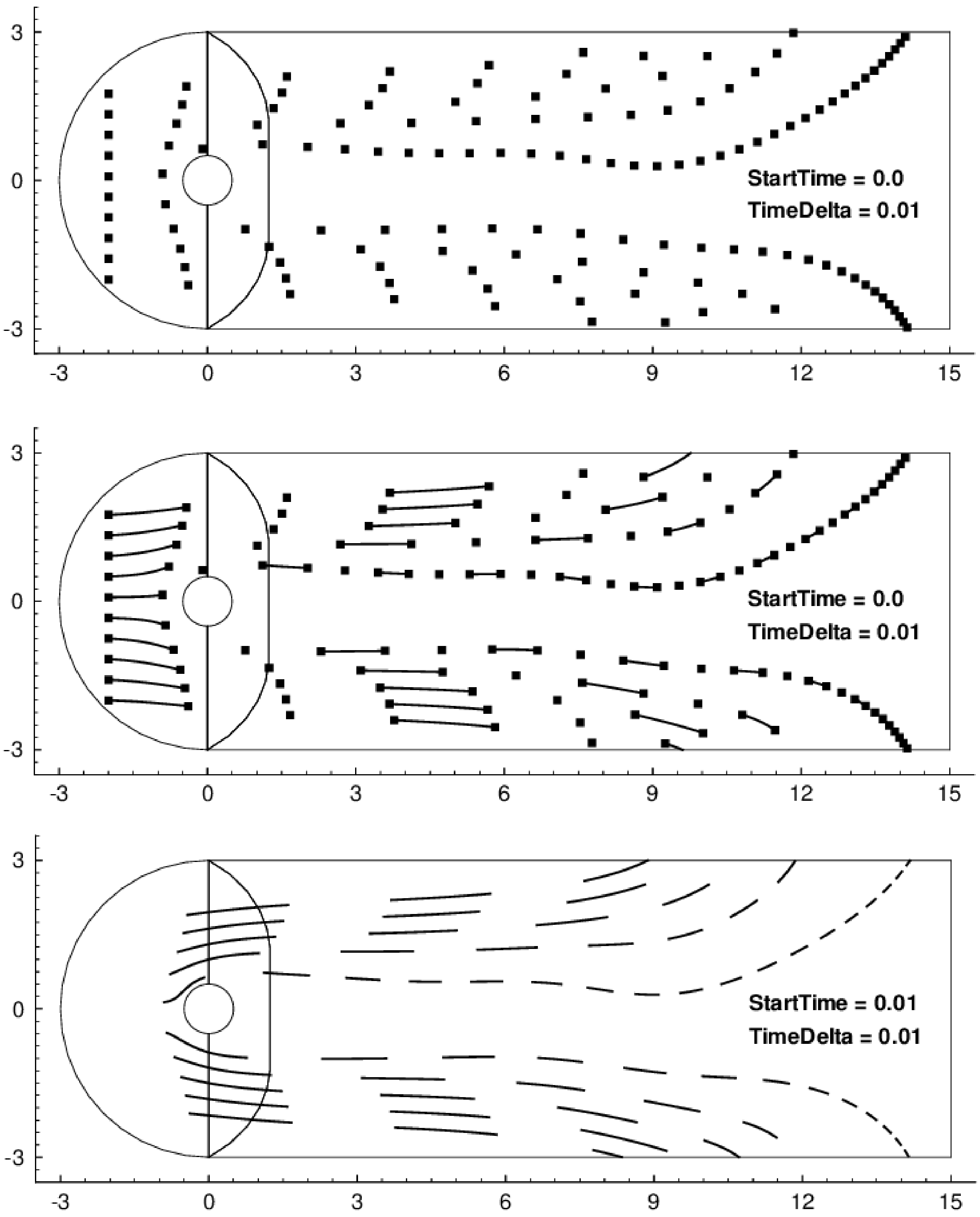
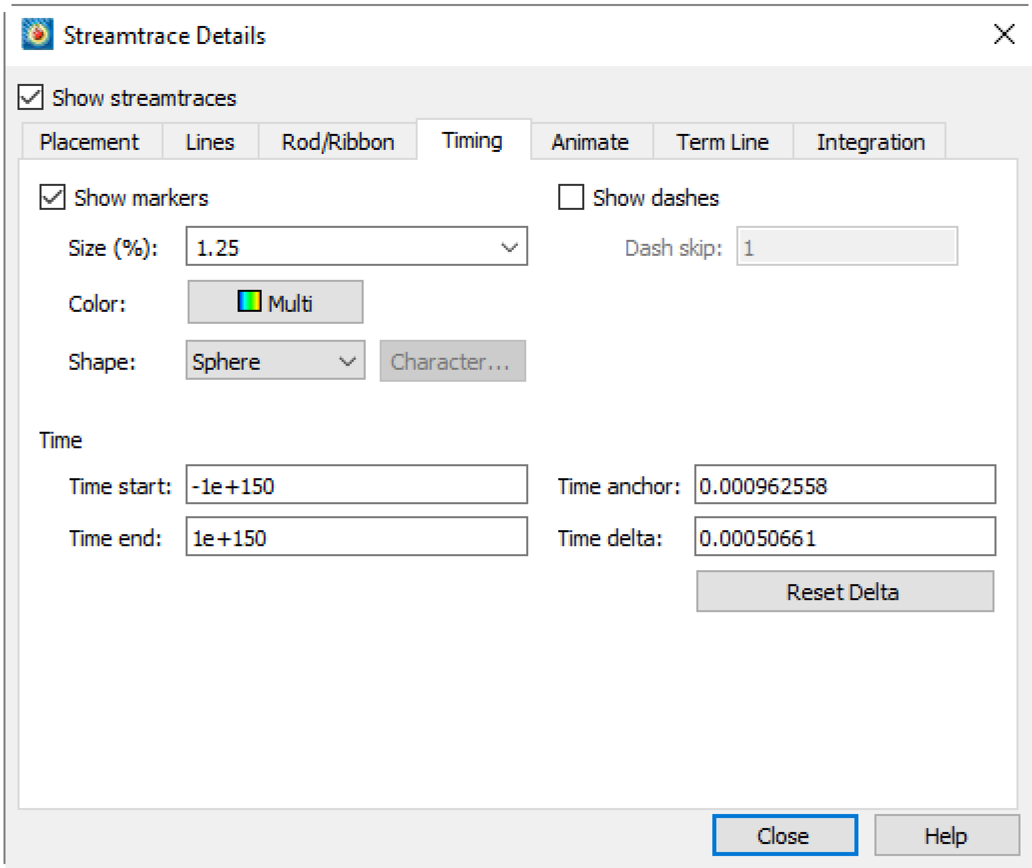


Figure 39. Streamtrace markers (top), dashes (bottom), and both (middle).

The spacing between stream markers is proportional to the magnitude of the local vector field. 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 menu on the Timing page of the **Streamtrace Details** dialog. Selecting "Other" from the list activates the Enter ASCII Character option, where you may enter an ASCII character to be used as your stream marker.

To place stream markers or dashes along your streamtraces, open the Timing page of the **Streamtrace Details** dialog (accessed via the Plot sidebar or the **Plot** menu).



The Timing page has the following options:

Show Markers

Toggle-on to include stream markers. Stream markers are only available for streamlines (surface and volume). Specify the size, color, and shape of the markers in the fields provided. The default marker shape is a sphere (in 3D plots) or a circle (2D).

Show Dashes

Toggle-on to include stream dashes. The lengths of the dashes and the spaces between the dashes are controlled by the value of Delta. Enter a value into the Dash Skip field to control the number of time deltas used for the "off" sections of the streamtraces.

Time Start

Enter the time at which the first marker should be 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.

Time End

Enter the time after which no more stream markers are drawn.

Time Anchor

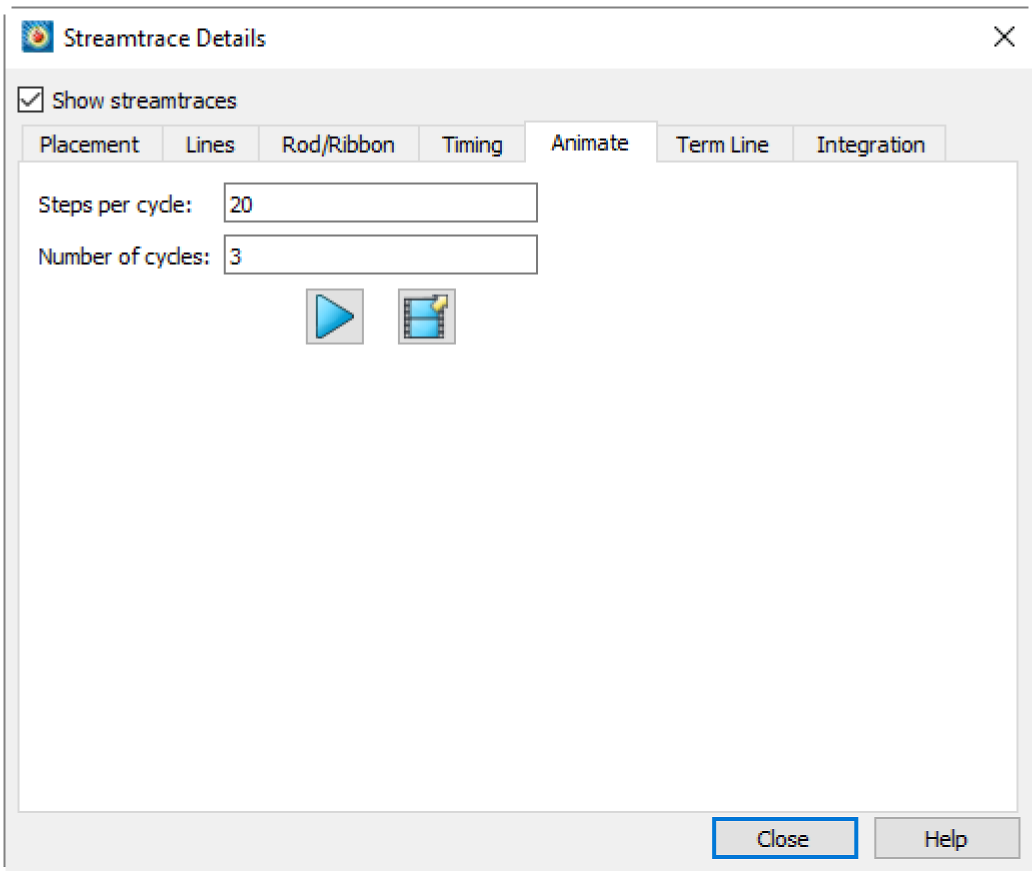
Enter the time that a dash is guaranteed to start, provided the start and end time surround the dash.

Time Delta

Enter the time interval that measures the time between stream markers. The actual distance between markers is the product of this number and the local vector magnitude. Click **Reset Delta** to reset this to the default value.

Animate Page

The Animate page controls animation of streamtraces.



You may specify the number of steps per cycle, the number of cycles, and the destination for animation (the screen or a file). Click the **Animate** button to begin animation and see Animation for further details.

Term Line Page

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. Figure 40 shows the cylinder data with some streamtraces terminated by a 2D streamtrace termination line.

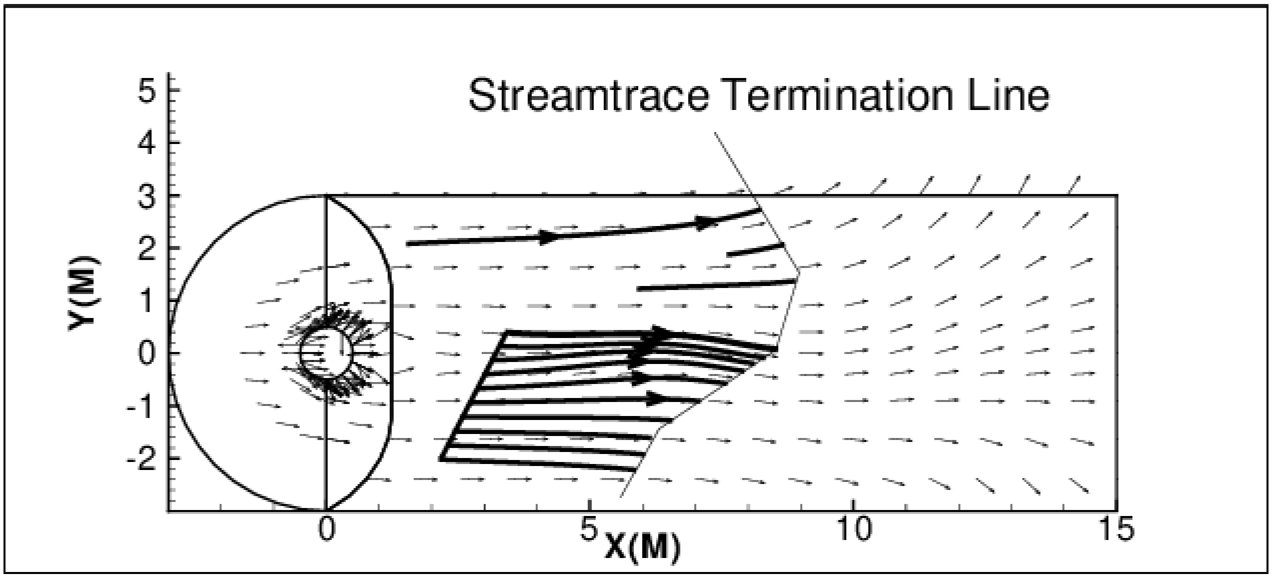
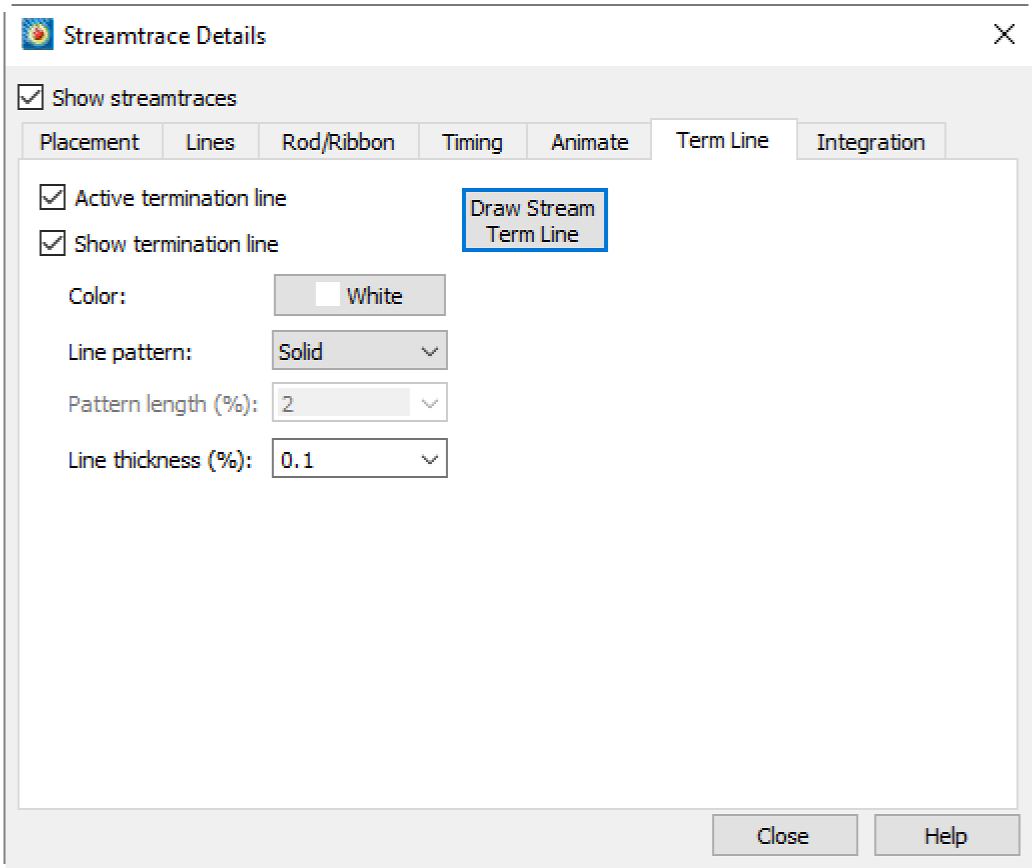


Figure 40. A streamtrace termination line drawn through surface streamlines.

Streamtraces are terminated whenever any of the following occur:

- The maximum number of integration steps is reached.
- Any point where the streamtrace passes outside the available data.
- The streamtrace reaches a point where the velocity magnitude is zero.

You control the streamtrace termination line from the Term Line page of the **Streamtrace Details** dialog.



From the **Term Line** page, you can control the following attributes of the termination line:

Active Termination Line

Toggle-on to activate the termination line and terminate any streamtraces that cross it. Toggle-off this option and redraw the plot to view unterminated streamtraces.

Draw Stream Term Line

Click to draw the termination line on the plot. This is the same as choosing the  button on the toolbar. Click the individual points on the plot that make up the termination line, then double-click to end the line construction.

Show Termination Line

Toggle-on to display the termination line. Toggle-off this option and redraw the plot to display terminated streamlines (if the termination line is active), but not the termination line itself.

Color

Choose the color of the termination line in the Color Chooser.

Line Pattern

Choose the pattern for the termination line.

Pattern Length

The length of the pattern as a percentage of frame height.

Line Thickness

The thickness of the termination line as a percentage of frame height.

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).

Only one termination line can exist at any one time in a given frame. If you draw a second termination line, the first one is automatically deleted.

Termination Lines in the Eye Coordinate System

The streamtrace termination line is drawn in the grid coordinate system, which in 2D Cartesian plots moves with the data as you zoom and translate. In 3D Cartesian plots, the termination line is drawn in what is known as the *eye coordinate system*. Grid coordinates align with the eye coordinate system, therefore the termination line moves with the data as you zoom and translate, but remains fixed when you rotate the plot.

When you rotate a 3D dataset 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 41 shows a 3D volume plot with streamribbons and a streamtrace termination line. This figure illustrates how the termination points vary as the plot is rotated. Notice that the termination line, being rendered in the eye coordinate system, remains in place on the screen as the plot is rotated.

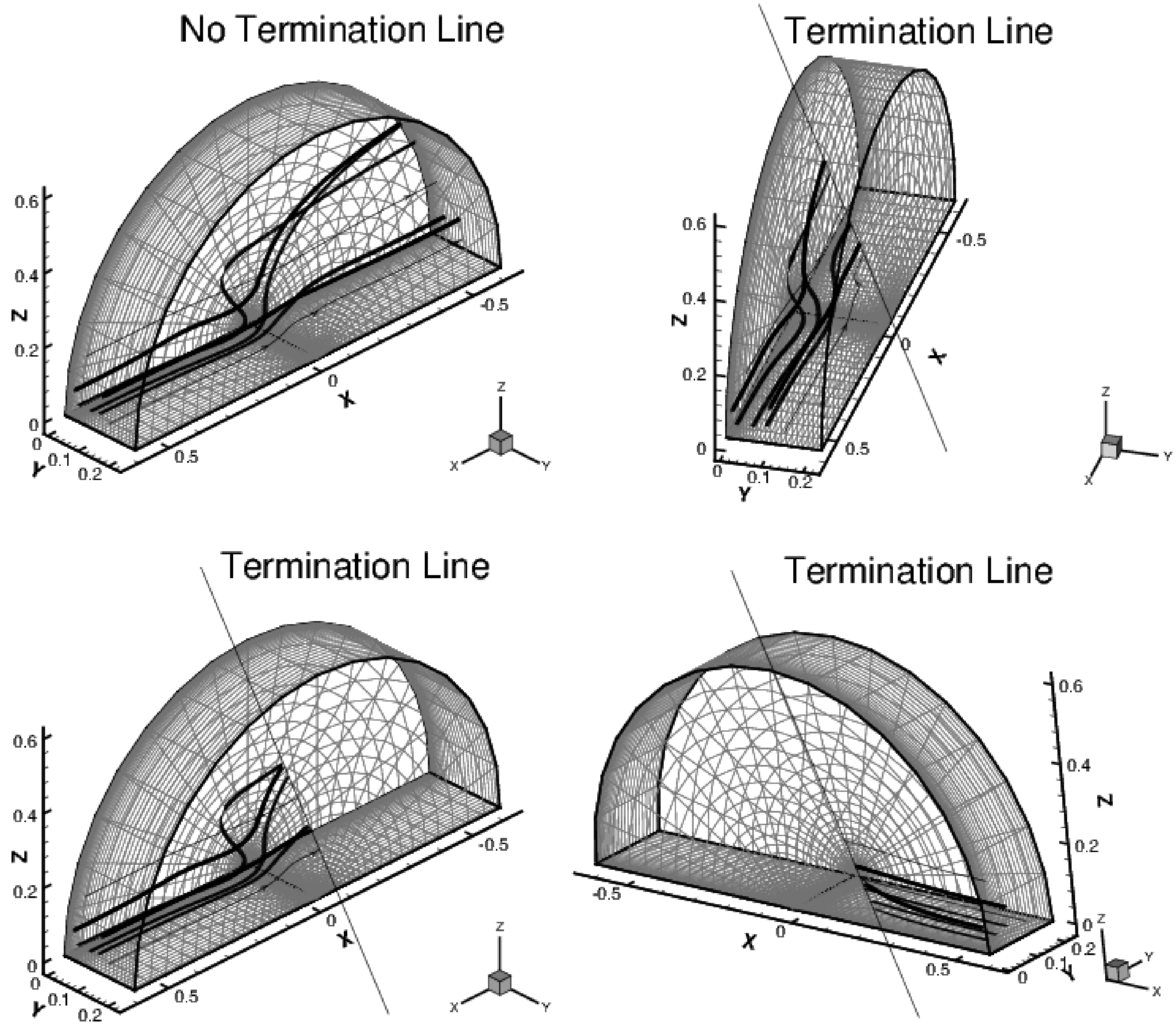
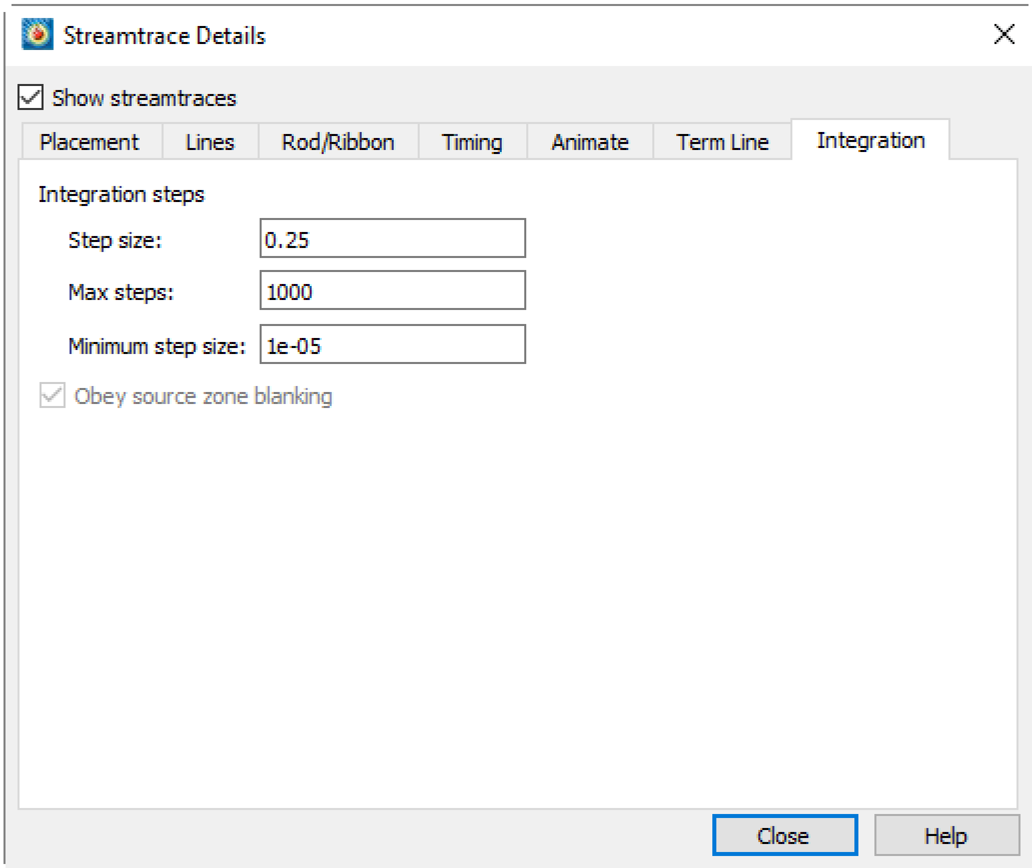


Figure 41. Rotating Volume streamtraces with a termination line in the eye coordinate system.

Integration Page

Tecplot 360 uses an adaptive step-size, trapezoidal integration algorithm to calculate streamtraces. This creates the streamtrace by moving it 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 360 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:



Step Size

Enter the initial and maximum step size Tecplot 360 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 truncation errors and 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 into 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 360 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.

Obey Source Zone Blanking

When active, streamtraces are generated for non-blanked regions only. When inactive, streamtraces are generated for both blanked and unblanked regions.

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 the streamtrace passes outside the available data.
- The streamtrace reaches a point where the velocity magnitude is zero.
- The streamtrace crosses the stream termination line.

When there is a small gap between the zones, streamtraces will terminate at a zone boundary even if there is an adjacent zone into which the streamtraces should proceed. Specifying face neighbors in the data file to connect the zones can reduce this issue. Increasing the minimum integration step size can also ameliorate this problem.

Surface Streamtraces on No-slip Boundaries

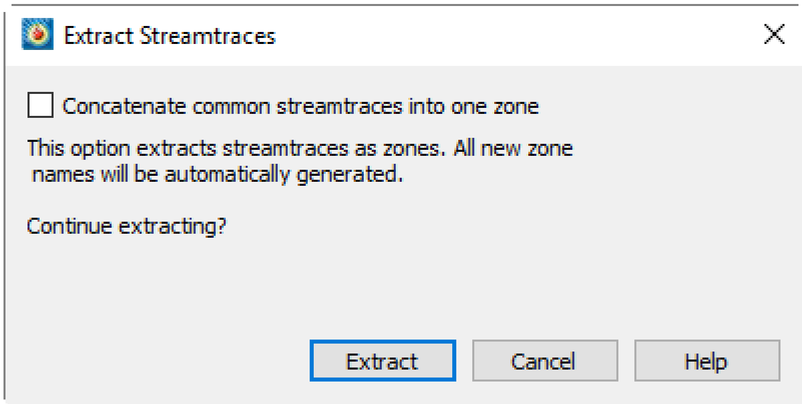
A no-slip boundary is the boundary of a volume zone whose vector variables are either passive or are zero throughout the boundary. The boundary may be either a displayed boundary of a volume zone (included in the zone's Surfaces to Plot setting in the Zone Style dialog), or a surface zone that resides on the boundary of a volume zone. In the latter case, the surface zone's nodes must coincide with nodes on the exterior of one or more volume zones. For either case, blanked regions of the surface are not considered when determining whether a boundary is a no-slip boundary. That is, if value blanking or IJK blanking is enabled, then only the non-blanked portion of the surface is examined to determine if it is a no-slip boundary.

When surface streamtraces are placed on a no-slip boundary, they propagate according to the normal gradient of tangential velocity (proportional to shear stress) at the surface, calculated using vector variable values from the nearby interior of the volume zone. If the solution is unsteady, only volume zones in the currently displayed solution time will be used. If the boundary is a surface zone, then corresponding volume zones needn't be displayed in the current plot (they may be turned off in the Zone Style dialog), but preference will be given to volume zones that have non-zero velocity, and that are active (not turned off in the Zone Style dialog).

If you extract a surface streamtrace, the velocity component variables in the resulting zone (which would normally be set to zero because the velocity on the surface is zero) are replaced with the gradient of the tangential velocity that was used to calculate the streamtrace.

Streamtrace Extraction as Zones

Normally, streamtraces are derived from the dataset "on the fly" and do not add any data to the dataset. To extract existing streamtraces to zones, allowing you to retain them even if streamtrace settings are changed, choose **Extract→Streamtraces** from the **Data** menu.



If you want all streamtraces of a given format extracted to a single zone, select the toggle-on "Concatenate common streamtraces into one zone" in the **Extract Streamtraces** dialog. If you select this option, Tecplot 360 extracts all surface lines into one zone, all volume lines into another, all volume ribbons into a third, and all volume rods into a fourth. Tecplot 360 uses FE zones (FE-Line segment for streamlines and FE-Quad for rods and ribbons). If you do not select this option, and timing is not turned on to show dashes, each streamtrace is extracted into its own ordered zone (I-Ordered for streamlines and IJ-Ordered for rods and ribbons).

After you have extracted your streamtraces, you will still see the original streamtraces, which may obscure the plotted streamtrace zones. Once you have extracted the zones, you can delete the original streamtraces by selecting **Delete All** on the Position page of the **Streamtrace Details** dialog. 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.

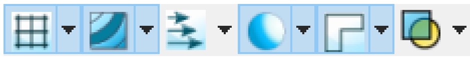
Streamtrace Errors

Streamtraces will not appear under the following conditions:

- Unorganized data (I-ordered zones). Refer to Working with Unorganized Datasets for information on organizing unorganized data.
- Zero-valued vectors. Refer to Vector Layer for information on working with vectors.
- The streamtrace was placed outside of the data. If you are drawing a rake on a concave 3D volume surfaces, hold down the Shift key to draw the rake outside of the data.
- Inappropriate integration step size. Refer to the Integration Page for information on integrating streamtraces.

Iso-Surfaces

An iso-surface displays a constant value of the contour variable as a surface on your plot. Iso-surfaces require that the data set contains volume zones (IJK-ordered, brick, tetrahedral, or polyhedral zones). You can modify iso-surfaces in the **Iso-Surface Details** dialog, which you can access by selecting the  button next to the Iso-surfaces toggle in the Plot sidebar. Or you can open the **Iso-Surface Details** dialog by selecting "Iso-Surfaces" from the **Plot** menu. Finally, you may right-click an iso-surface in your plot and use the context menu and toolbar to make quick changes to its settings.



The context toolbar appears above the context menu when you right-click an iso-surface. This toolbar allows you to turn on or off the grid, contour, shade, and translucency layers for the selected iso-surface(s). Additionally, you may adjust frequently-used style settings for each layer using the drop-down menu to the right of each, for example selecting a color for the grid (or choosing a variable by which to color it).

To view changes made in the **Iso-surface Details** dialog in your plot, you must have "Iso-surfaces" toggled on in the Plot sidebar.

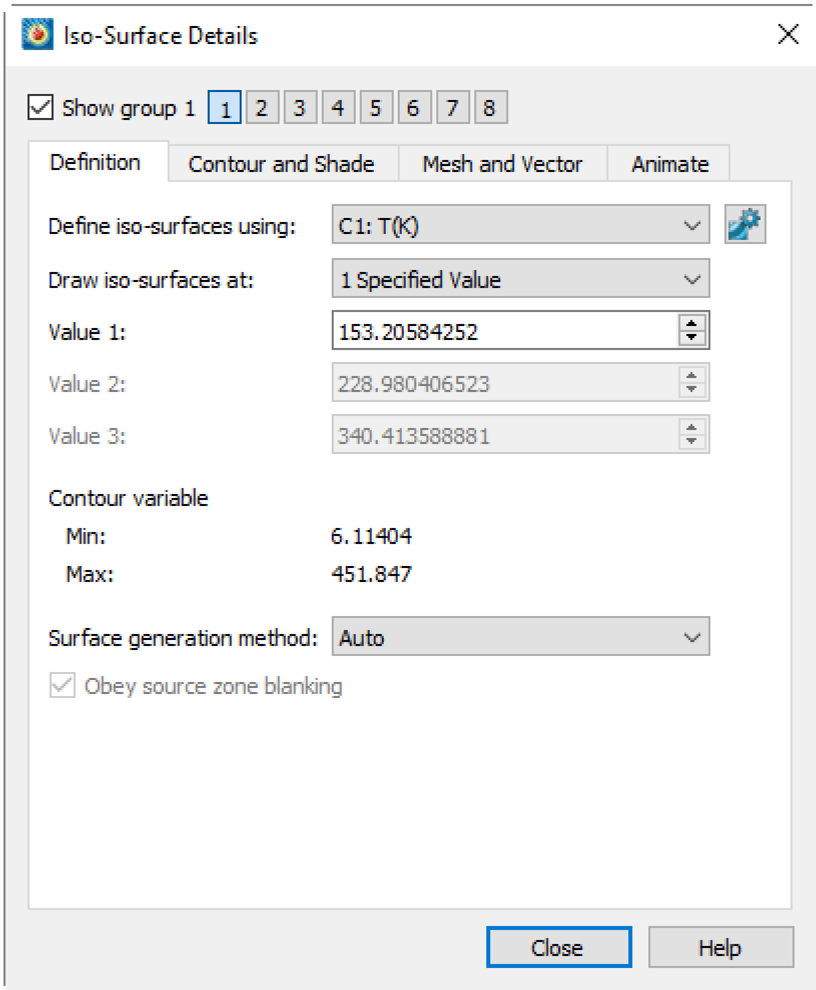
Iso-Surface Groups

You can work with up to eight different iso-surface groups in Tecplot 360. Zone layers or other objects that reference the same group for an attribute show the same plot style for that attribute. Each iso-surface group has its own settings for the attributes set in the **Iso-surface Details** dialog. Refer to the following sections for details on each attribute. Choose the desired group number from 1-8 at the top of the dialog, then toggle-on the "Show Group *n*" checkbox to include the corresponding iso-surface group in your plot.

If you have added iso-surfaces to your plot, but cannot see them, go to the Volume page of the **Zone Style** dialog and verify that Show Iso-surfaces is set to "Yes". Refer to Derived Volume Object Plotting for details. ls.

Iso-Surface Definition

Use the Definition page of the **Iso-Surface Details** to control Tecplot 360’s rendering of iso-surfaces. Tecplot 360 applies the attributes set on this page (and on every page of the dialog) to the Iso-Surface group selected at the top of the dialog next to the Show Group toggle.



The Definition page of the dialog includes the following controls:

Show Group n

Select this check box to display iso-surfaces, where *n* can be 1-8.

Define Iso-Surfaces using

Use this drop-down menu to select the desired contour group. Use the nearby button to display the **Contour & Multi-Coloring Details** dialog, where you can define contour groups. (See Contour & Multi-Coloring Details.)

Draw Iso-Surfaces at

Use this drop-down menu to have Tecplot 360 draw iso-surfaces at:

Contour Group Levels

Go to the **Contour & Multi-Coloring Details** dialog (accessed with the button) to alter the Contour Levels. Refer to Contour Levels and Color for details.

At Specified Value(s)

Specify up to three values of the contour variable at which to draw iso-surfaces.

Contour Variable Min/Max

Indicates the minimum and maximum values of the contour variable. When the contour variable is calculated from subzone data, one or more of the values displayed may be estimates, which is indicated by **(est)** next to the affected values.

Surface Generation Method

Determines how the surface is generated.

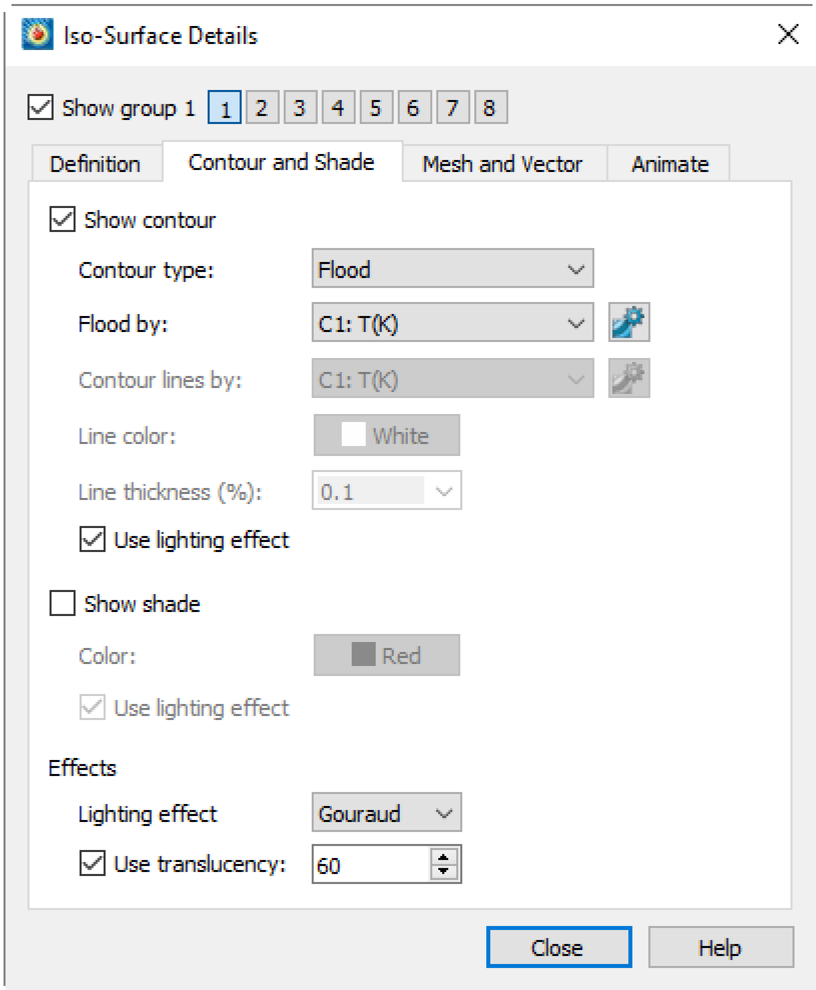
- *Auto* selects one of the surface generation algorithms best suited for the zones participating in the iso-surface generation. "All Polygons" is used if one or more of the participating zones is polytope, otherwise iso-surfaces use "All Triangles" unless the iso-surface is defined by a coordinate variable in which case "Allow Quads" is used.
- *Allow Quads* can produce quads or triangles, and the resulting surface more closely resembles the shape of the volume cells from the source zone. Since the quads are not arbitrarily divided into triangles, no biases are introduced, and the resulting surface may appear smoother. This method is preferred when the source zone is FE-Brick or IJK-Ordered and the surface is aligned with the source cells.
- *All Triangles* is an advanced algorithm that can handle complex saddle issues and guarantees that there will be no holes in the final surface. As the surface is composed entirely of triangles, it can be delivered more efficiently to the graphics hardware.
- *All Polygons* is similar to the "All triangles" method except that all interior faces generated as a result of triangulation that are not part of the original mesh are eliminated. This preserves the original mesh of the source zones on the resulting iso-surface.

Obey Source Zone Blanking

When active, iso-surfaces are generated for non-blanked regions only. When inactive, iso-surfaces are generated for blanked and unblanked regions. unblanked regions.

Iso-Surface Contour and Shade

The Contour and Shade page of the **Iso-Surface Details** dialog controls the contour and shade style settings for all iso-surfaces. (These styles act independently of the style assigned to zones by the **Zone Style** dialog.) Tecplot 360 applies the attributes set on this page (and every page of the dialog) to the Iso-Surface group selected at the top of the dialog next to the Show Group toggle.



The Contour and Shade page of the dialog includes the following controls:

Show Contour

Toggle-on "Show Contour" to display contours on iso-surfaces.

Contour Type

Select the contour type 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 the contours, or select RGB flooding. Use the nearby button to bring up the **Contour & Multi-Coloring Details** dialog, where you can define contour groups. (See Contour & Multi-Coloring Details.)

Contour Lines by

If you chose contour lines, select the contour group by which to draw lines. Use the nearby button to display the **Contour & Multi-Coloring Details** dialog to define contour groups.

Line Color

If you chose contour lines, click Line Color to select a line color in the Color Chooser.

Line Thickness

If you chose contour lines, select a contour line thickness from the drop-down menu, or enter your own number in the text field.

Use Lighting Effect

Toggle-on to enable the lighting effect selected in the Light effect drop-down menu.

Show Shade

Toggle-on to display shading on iso-surfaces.

Color

Select a shade color using the Color Chooser.

Use Lighting Effect

Toggle-on to enable the lighting effect selected in the Light effect drop-down menu.

Effects

Lighting Effect

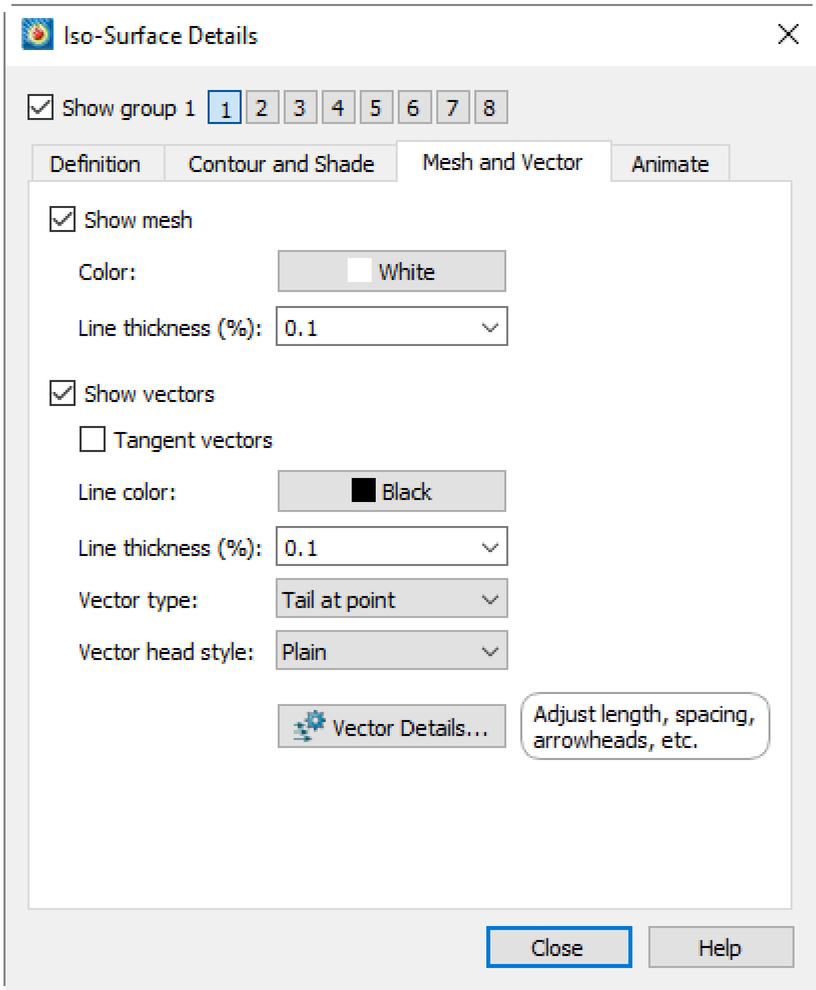
Select between Paneled or Gouraud shading.

Use Translucency

Toggle-on to enable the surface translucency text field, where you may set the surface translucency from 1 (nearly opaque) to 99 (nearly transparent). ly transparent).

Iso-Surface Mesh and Vector

The Mesh and Vector page of the **Iso-Surface Details** dialog controls the mesh and vector style settings for all iso-surfaces. (These styles act independently of the style assigned to zones by the **Zone Style** dialog.) Tecplot 360 applies the attributes set on this page (and every page of the dialog) to the Iso-Surface group selected at the top of the dialog next to the Show Group toggle.



The Mesh and Vector page of the dialog includes the following controls:

Show Mesh

Toggle-on "Show Mesh" to display the mesh on iso-surfaces.

Color

Select a mesh color in the Color Chooser.

Line Thickness

Select a mesh line thickness from the drop-down menu, or enter a number in the text field.

Show Vectors

Toggle-on "Show vectors" to display the vectors on iso-surfaces.

Tangent Vectors

Select to use tangent vectors for your iso-surfaces. See Vector Plot Modification for more information.

Line Color

Choose the line color from the Color Chooser. 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 height. 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 iso-surfaces. 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 iso-surfaces. Choose from Plain, Filled, and Hollow.

Vector Details...

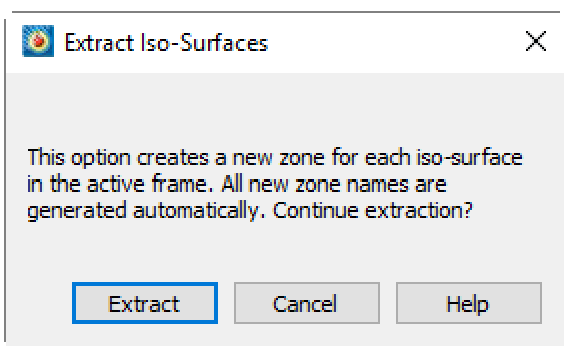
Opens the Vector Details dialog to adjust Vector length, spacing, arrowheads, etc. c.

Iso-Surface Animation

Refer to Iso-surfaces Animation.

Iso-Surface Extraction

Normally, iso-surfaces are derived from the dataset "on the fly" and do not add any data to the dataset. To extract existing iso-surfaces to Tecplot zones, allowing you to retain them even if the contour variable is changed, select **Extract**→**Iso-Surfaces** from the **Data** menu.



When you click Extract, one new iso-surface zone is created for each iso-surface visible in your plot. All of the variables in the dataset are interpolated from the 3D volume zones to the data points of the iso-surfaces.

Iso-surface zones are FE-surface quadrilateral element-type zones, regardless of the original 3D 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 original iso-surfaces.

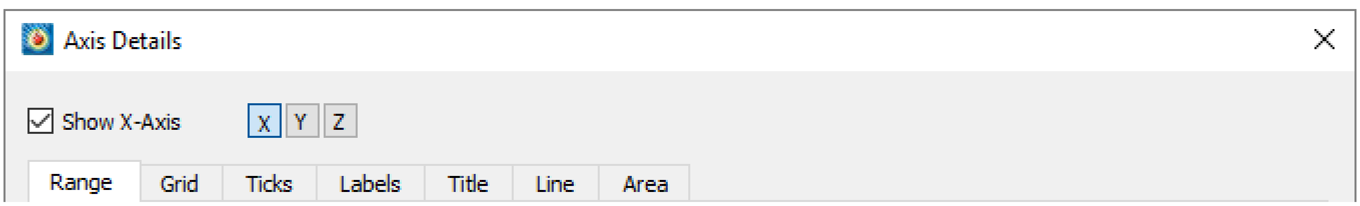
Axes

Tecplot 360 automatically enables the axes for 3D, 2D, XY, and Polar plot types. There are five distinct sets of axes, one for each plot type. When the axes are generated, the axis labels, position, spacing, and tick mark labels are created. You can adjust any of these settings by using the **Axis Details** dialog (accessed via the **Plot** menu). Each page of the **Axis Details** dialog controls a different aspect of the axis, and each page is available for each axis.

Axis Display

Use the "Show Axis" toggle in the **Axis Details** dialog to turn on an axis display. By default, displaying an axis 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.

To edit an axis from the **Axis Details** dialog, use the axis buttons ([X], [Y], [R], etc.) at the top of the dialog to indicate which axis you are working with. To edit a different axis, select a different axis button.



Axis Variable Assignment

For 2D and 3D Cartesian plots, Tecplot 360 initially assigns the first and second variables in the dataset to the X- and Y-axes, respectively. For 3D axes, the third variable in the dataset is assigned to the Z-axis.

To change variable assignments for 2D and 3D axes, select "Assign XY" or "Assign XYZ" from the **Plot** menu. For line plots, assigning axis variables is part of defining the mappings. See XY and Polar Line Plots for more information.

The axis range may be modified using the Range page of the **Axis Details** dialog, accessed via the **Plot** menu or by right-clicking an axis in a plot. See Axis Range Options for XY and 2D/3D Plots and Axis Range Options for Polar Plots

When working with axis ranges, please keep the following definitions in mind:

Axis Range

Specifies the minimum and maximum data values displayed along the axis. The range for an axis fits the value of the first variable assigned to that axis. If you deactivate the current layer and activate another layer, it may be necessary to reset the axis range.

Axis Length

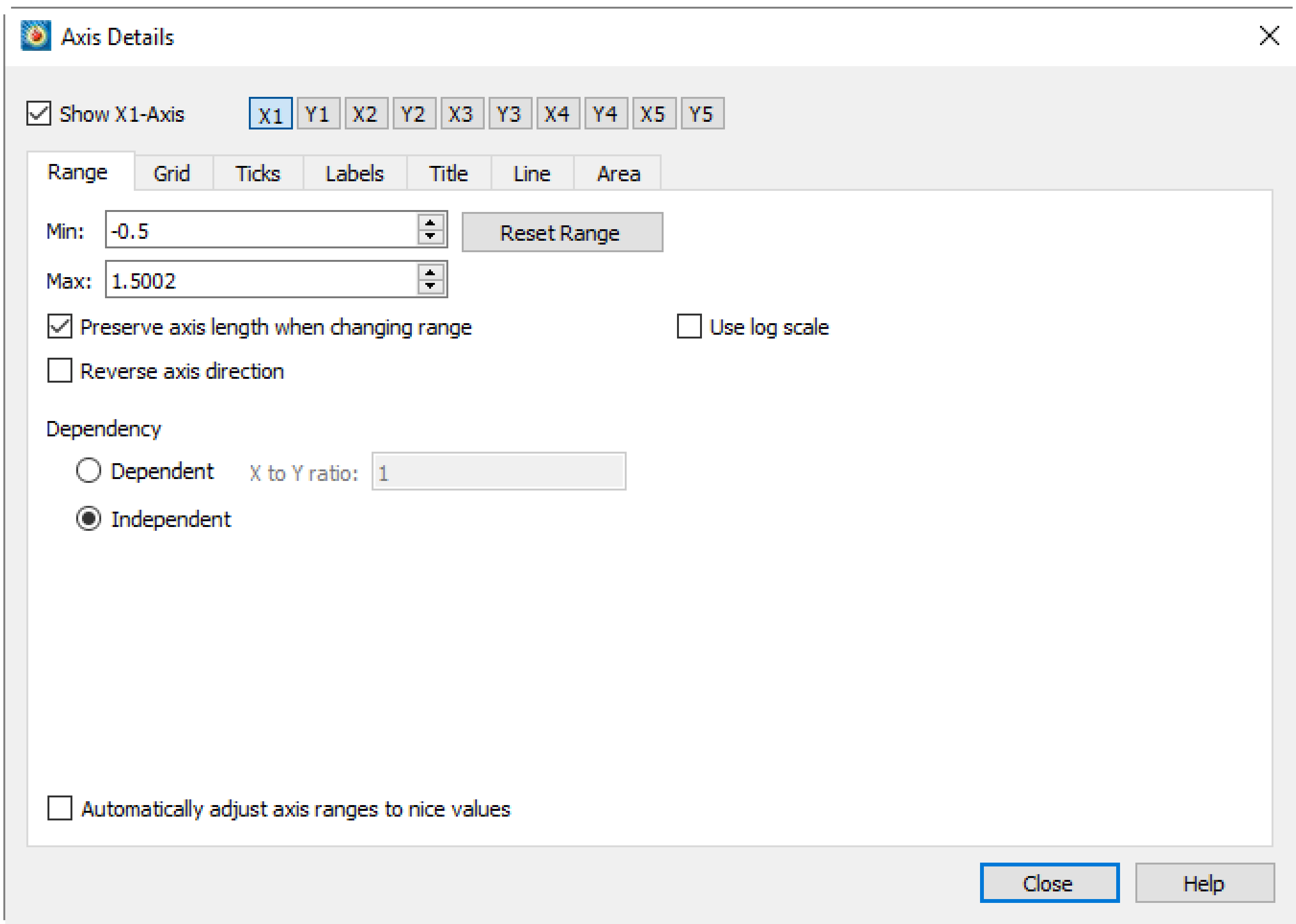
Specifies the physical length of the axis on the screen or paper.

Axis Scale

Specifies the ratio of the axis length to the axis range.

Axis Range Options for XY and 2D/3D Plots

This section discusses the options for XY Line, 2D Cartesian, and 3D Cartesian plots that can be found on the Range page of the **Axis Details** dialog.



The Range page has the following options:

Show Axis

Toggle-on this checkbox to show the selected axis (*n*) on the plot. Use the buttons [X], [Y], etc. to the right of this checkbox to select the axis to show.

Min

Enter the minimum value of the axis range.

Max

Enter the maximum value of the axis range.

Reset Range

Reset the **Max** and **Min** fields by selecting one of three options from the drop-down menu:

Reset to Nice Values

Sets the range to slightly larger than the current axis variable range in order to begin and end the axis at major axis increments.

Set to Variable Min/Max

Sets the range to the minimum and maximum variable values, considering the effects of blanking.

Make Current Values Nice

Rounds the axis range to the nearest major axis increment.

Preserve axis length when changing range

If toggled-on, and your axes are Independent, changes to the X to Y Ratio will affect the axes' range, but not their scale. Toggle-off to change both the axis range and axis scale simultaneously. See Figure 42.

Use log scale

The X and Y axes of 2D or XY Line plots can have a linear scale (default) or logarithmic scale. When "Auto Spacing" is selected with logarithmic scale, large numbers are displayed in scientific notation (i.e., $3.48x10^5$). It is strongly recommended that you use "Auto Spacing" with log axes. Navigate to the Ticks or Labels page of the dialog, and toggle-on "Auto Spacing" to use this option.

Reverse axis direction

Toggle on to display the axis from high to low rather than from low to high. Not available for 3D Cartesian plots.

Dependency

Select whether to set the axes as dependent upon or independent of each other. For XY Line or 2D Cartesian, select "Independent" or "Dependent".

> When a logarithmic scale is being used in an XY Line plot, the axes must be independent.

For 3D Cartesian, select from one of the following options:

Independent

All axes are independent.

XY Dependent

The X and Y axes are dependent upon each other. 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.

Ratios (For 2D and 3D plots only)

2D Plots

If "Dependent" is selected, enter the X to Y Ratio.

3D Plots

If "XY Dependent" is selected, enter the X to Y Ratio.

3D Plots

If "XYZ Dependent" is selected, enter the X to Y Ratio and the X to Z Ratio.

Automatically Adjust Axis Range to Nice Values

Automatically adjusts the axis ranges to the nearest major axis increments.

Size Factors (For 2D and 3D plots only)

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.

Reset Size/Dependency

Resets the Dependency controls to their defaults.

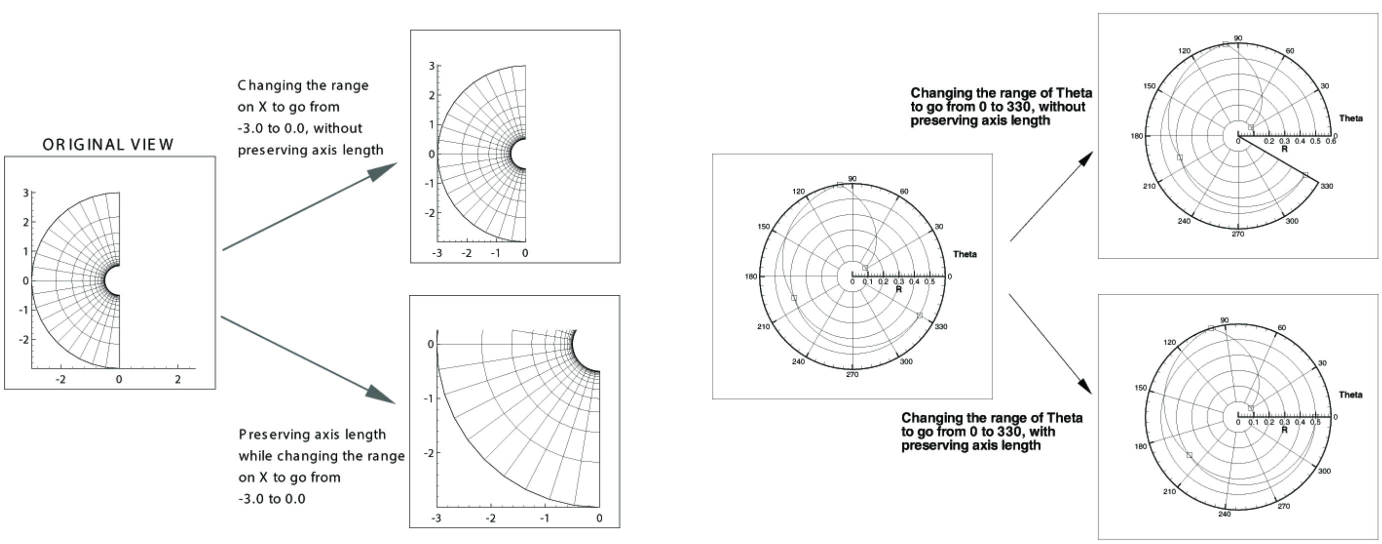
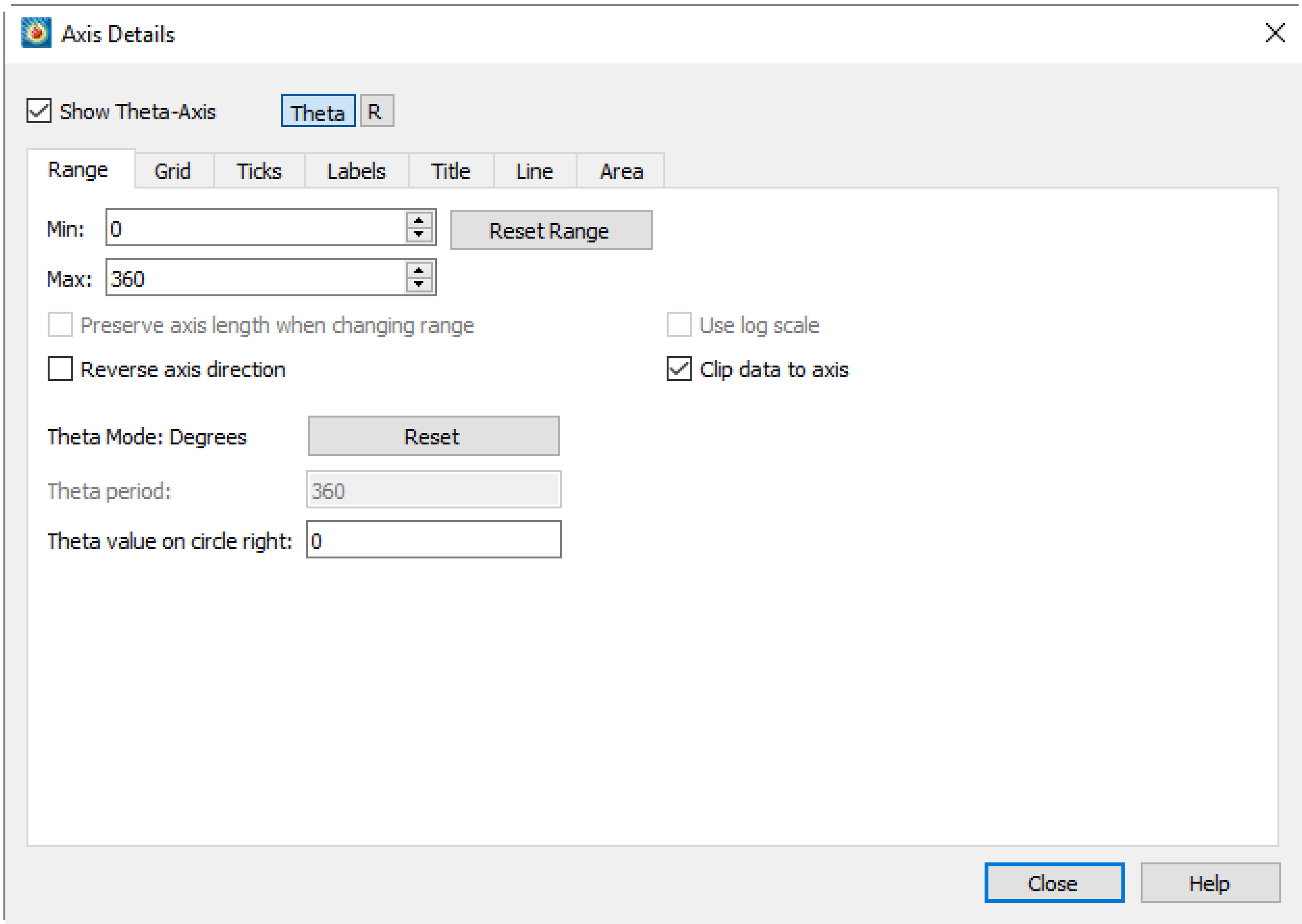


Figure 42. Preserving length versus preserving scale while changing range (left); preserving length versus preserving scale while changing range in a Polar Line plot (right).

Axis Range Options for Polar Plots

Polar axes are different from any other axis type due to their cyclical nature. Each polar axis (Theta and R) has very different settings, unlike XY or XYZ axes. For the Theta-axis you can change the Theta Mode, Theta Period, and Theta Value on Circle Right; for the R-axis you can change the origin; and for both axes you can clip the data to the axes.



Show Axis

Toggle-on this checkbox to show the selected axis on the plot. Use the buttons Theta, R, etc. to the right of this checkbox to select an axis.

Min

Enter the minimum value of the axis range.

Max

Enter the maximum value of the axis range.

Reset Range

Reset the **Max** and **Min** fields by selecting one of three options from the drop-down menu.

Reset to Entire Circle

Sets the range of Theta to encompass an entire circle.

Reset to Nice Values

Sets the range to slightly larger than the current axis variable range in order to begin and end the axis at major axis increments.

Set to Var Min/Max

Sets the range to the minimum and maximum variable values.

Make Current Values Nice

Rounds the axis range to the nearest major axis increment.

Preserve length when changing range

If toggled-on, changes to the Theta to R Ratio will affect the axes' range, but not their scale. Toggle-off to change both the axis range and axis scale simultaneously. See Figure 42 for an illustration of the difference.

Use log scale

The R axis can have a linear scale (default) or a logarithmic scale. When "Auto Spacing" is selected with logarithmic scale, large numbers are displayed in scientific notation (i.e., $3.48x10^5$). It is strongly recommended that you use "Auto Spacing" with log axes. Navigate to the Ticks or Labels page of the dialog, and toggle-on "Auto Spacing" to use this option.

Reverse axis direction

Toggle on to display the axis from high to low rather than from low to high. Not available for 3D Cartesian plots.

Clip Data to axis

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 each axis. To activate or deactivate clipping, toggle "Clip Data to Axis" on or off. This feature is illustrated in Figure 43.

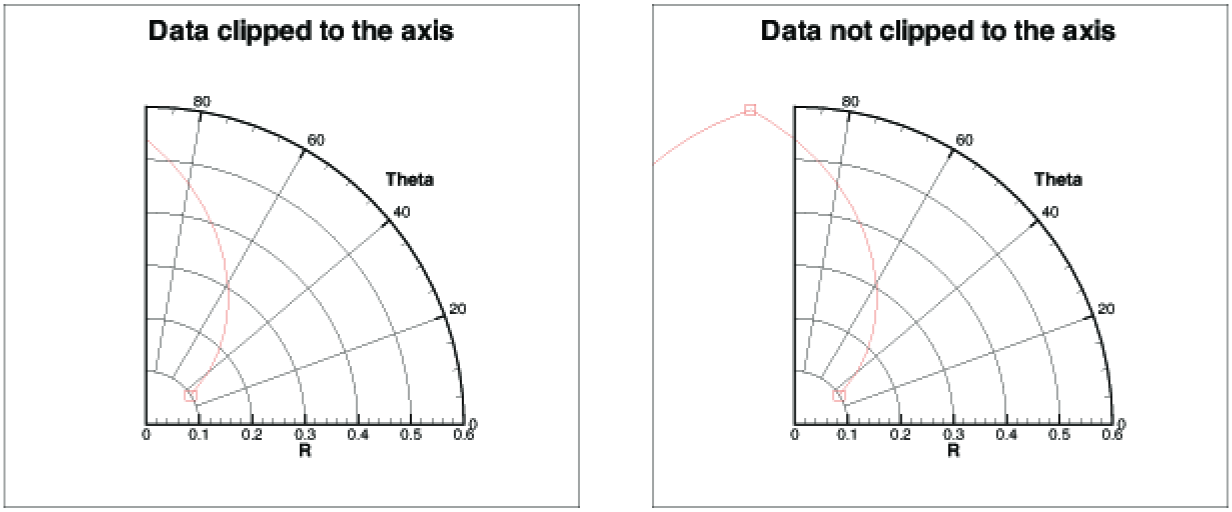


Figure 43. An example of clipping polar data to an axis.

Theta Mode

By default, the Theta-axis is expressed in degrees mode with a range of 0 to 360. For the Theta axis, you can plot the angles in units of Radians, Degrees, or Arbitrary (where arbitrary sets the Theta range to the maximum and minimum values of the Theta-axis variable).

To set the Theta Mode, choose from the following options:

- 0 - 360 Degrees
- -180 – 180 Degrees
- 0 – 2Pi Radians
- Pi-Pi Radians
- Fit to Var Min/Max

Selecting any of these options changes the Theta Mode, resets the Theta-axis range, and resets the Theta Period. When the Theta Mode is Radians, Tecplot 360 draws Theta labels as fractional units of Pi.

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 "Fit to Var Min/Max", the period can be set to any value.

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.

R-origin

The R-origin specifies what value of R is represented at the center of the axis. The effect of changing the R-origin is displayed in Figure 44.

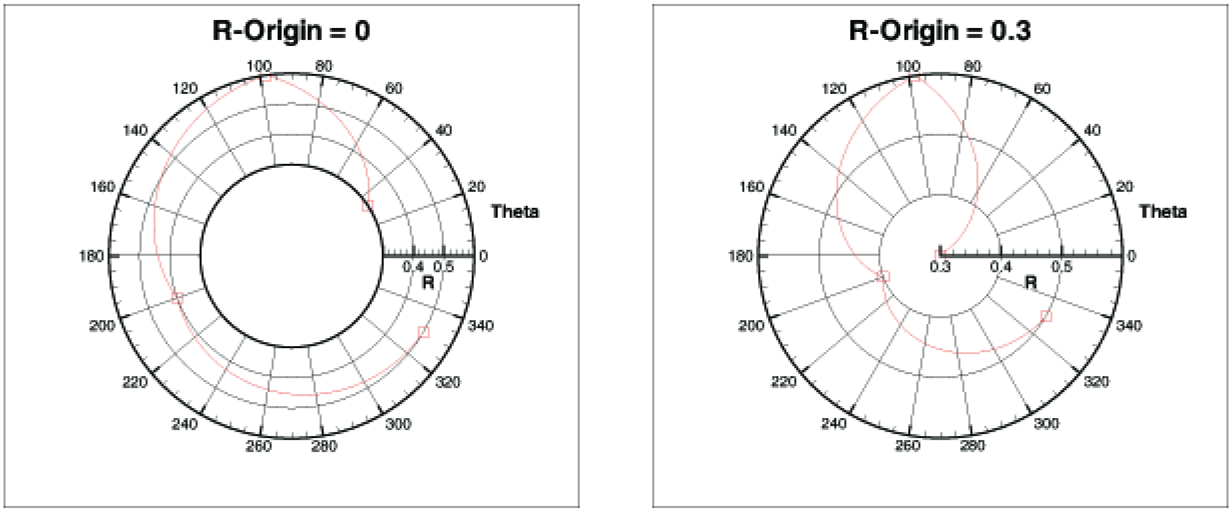


Figure 44. An example of changing the R-origin from a range of 0.3 to 0.6 on a polar plot.

Axis Grid Options

You control the gridlines and precise dot grid from the Grid page of the **Axis Details** dialog, as shown below. On this page, you can customize the line pattern, pattern length, line thickness, and gridline color for Gridlines and Minor Gridlines.

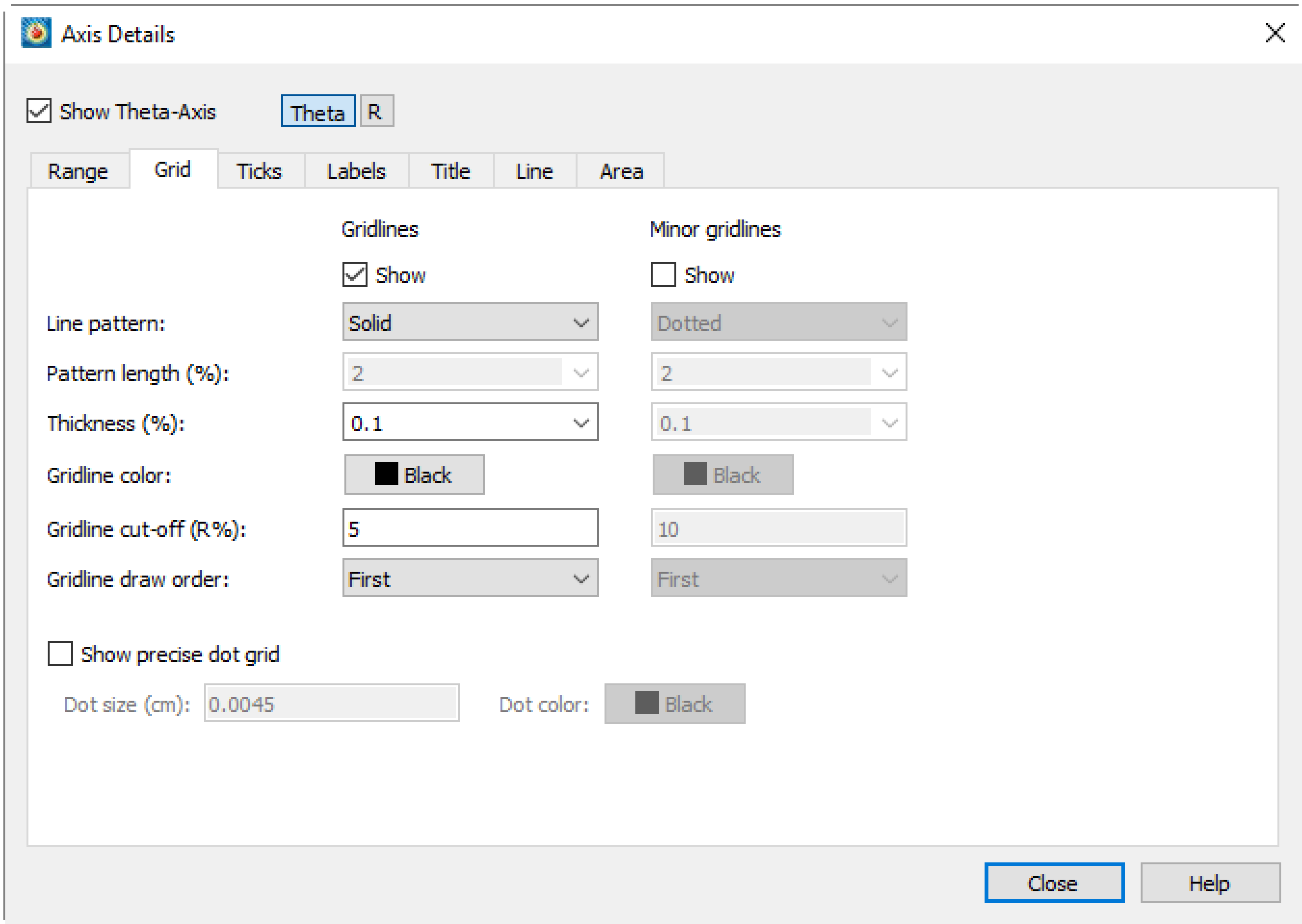
Gridlines

Match the spacing of tick marks.

Minor Gridlines

Subdivisions of Gridlines.

The spacing of Gridlines is controlled by the tick mark spacing; see Tick Mark Options for more information.



Gridlines, Minor Gridlines

To activate Gridlines or Minor Gridlines, toggle-on "Show" under these headings on the Grid page. Both kinds of gridlines have the following options:

Line Pattern

From the Line Pattern drop-down menu, choose one of the following line patterns: Solid, Dashed, DashDot, Dotted, LongDash, or DashDotDot.

Pattern Length (%)

(Theta axis only) Select a pattern length from the drop-down menu, or enter in your own.

Thickness (%)

Select a line thickness from the drop-down menu, or enter your own as a percentage of frame height.

Gridline Color

Select the color of your gridlines using the Color Chooser.

Gridline Draw Order

For all axes except 3D, 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.

Gridline Cutoff (R%)

(Theta axis only) Select the point along the R-axis where you want to stop drawing Theta lines.

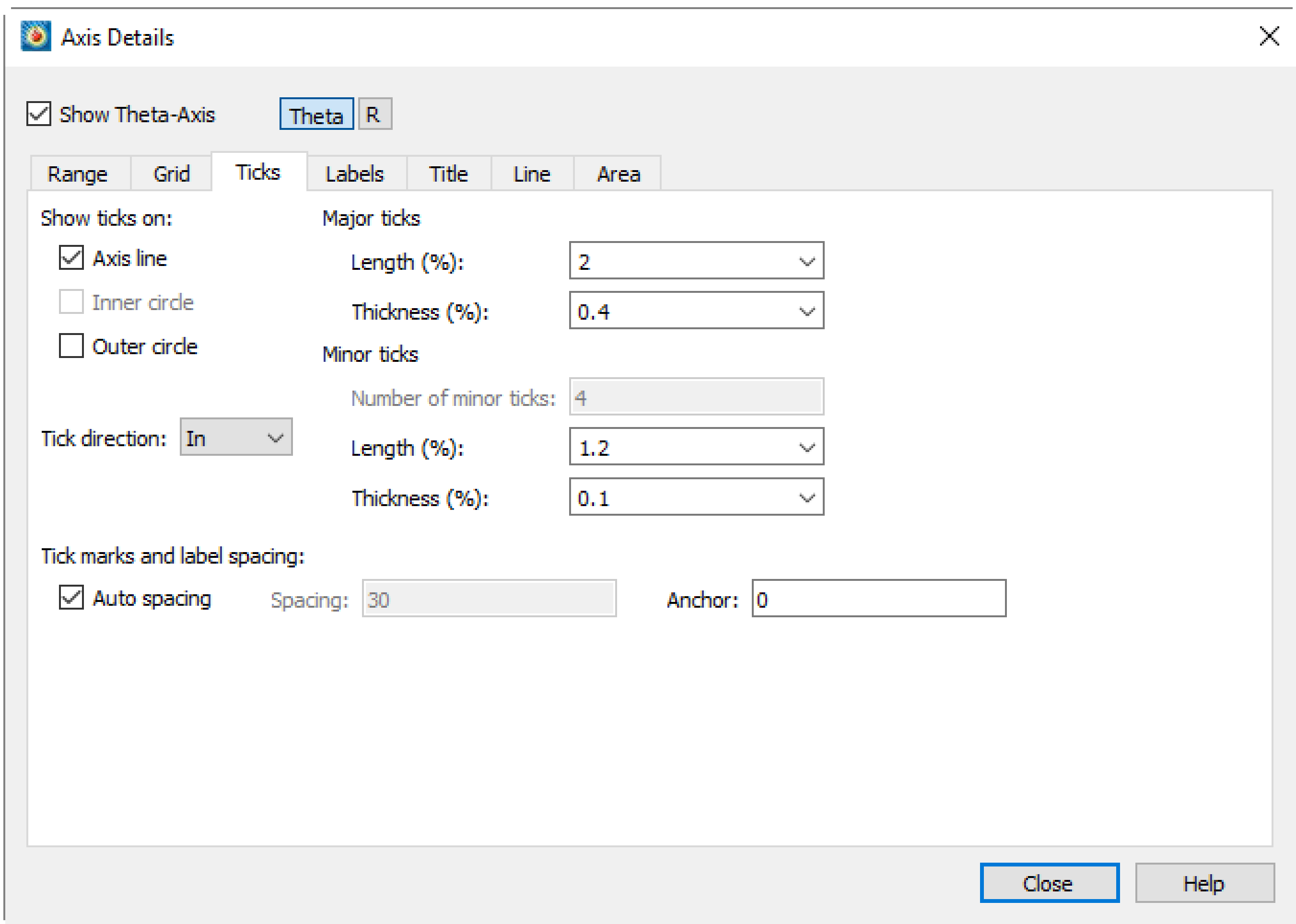
In a Polar Line plot, the abundance of gridlines at the center may obscure data. You can specify a gridline cutoff along the R-axis of Polar plots on the Grid page of the **Axis Details** dialog for the Theta-axis.

Show Precise Dot Grid

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 plots and Line plots when either axis for the first active line mapping uses a log scale.

Tick Mark Options

Tick marks can be placed on each axis and labeled with either numbers or text strings. You can define your tick marks and their placement using the Ticks page of the **Axis Details** dialog. You can define the tick mark labels using the Label page of the **Axis Details** dialog.



Show Ticks On

For each plot type, you can display tick marks at different sections of the axis.(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

The line that represents the specified axis.

Grid Border Bottom

By default, the axis line and grid border left/bottom are in the same position. Grid Border Bottom is the lower left-most position of the grid as defined by the viewport settings on the Area page of the **Axis Details** dialog.

Grid Border Top

Grid Border Top is the top right-most position of the grid as defined by the viewport settings on the Area page of the **Axis Details** dialog.

3D Cartesian axis tick marks can be displayed in the following areas:

Axis Line

The line that represents the specified axis.

Opposite Edge

The complimentary line that is opposite the axis line.

Polar R-axis tick marks can be displayed in the following areas:

Axis Line

The line that represents the R-axis.

All R-axes

Only available if "Draw Axis in Both Directions" or "Draw Perpendicular Axis" is toggled-on for the R-axis. The All R-axes setting will draw tick marks on the additional axes that are drawn.

Grid Border Start

Start point of the polar grid area.

Grid Border End

End point of the polar grid area.

> Grid Border Start and Grid Border End are only available if the polar plot does not form a complete circle. If the data forms a complete circle, there is no start or end point on which to draw the ticks.

Polar Theta-axis tick marks can be displayed in the following areas:

Axis Line

The line that represents the Theta-axis.

Inner Circle

Only available if the minimum (Min) value on the R-axis is greater than the R-origin (Max) value. The Min and Max values are located on the Range page of the **Axis Details** dialog (see Axis Range Options for Polar Plots). 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

The outer edge of the polar grid area.

Major Tick Mark Length and Thickness

Tick mark length and thickness can be set independently for major and minor tick marks using the Length and Thickness fields o.

Number of Minor Ticks Length and Thickness

To specify the number of minor tick marks, you must first toggle-off "Auto Spacing" at the bottom of the page. The number of minor tick marks can be set in the "Number of Minor Ticks" text field in the Minor Ticks section of the dialog. You may also set their length and thickness.

> There is not a separate control for showing minor tick marks. To hide minor tick marks, enter zero in the "Number of Minor Ticks" text field.

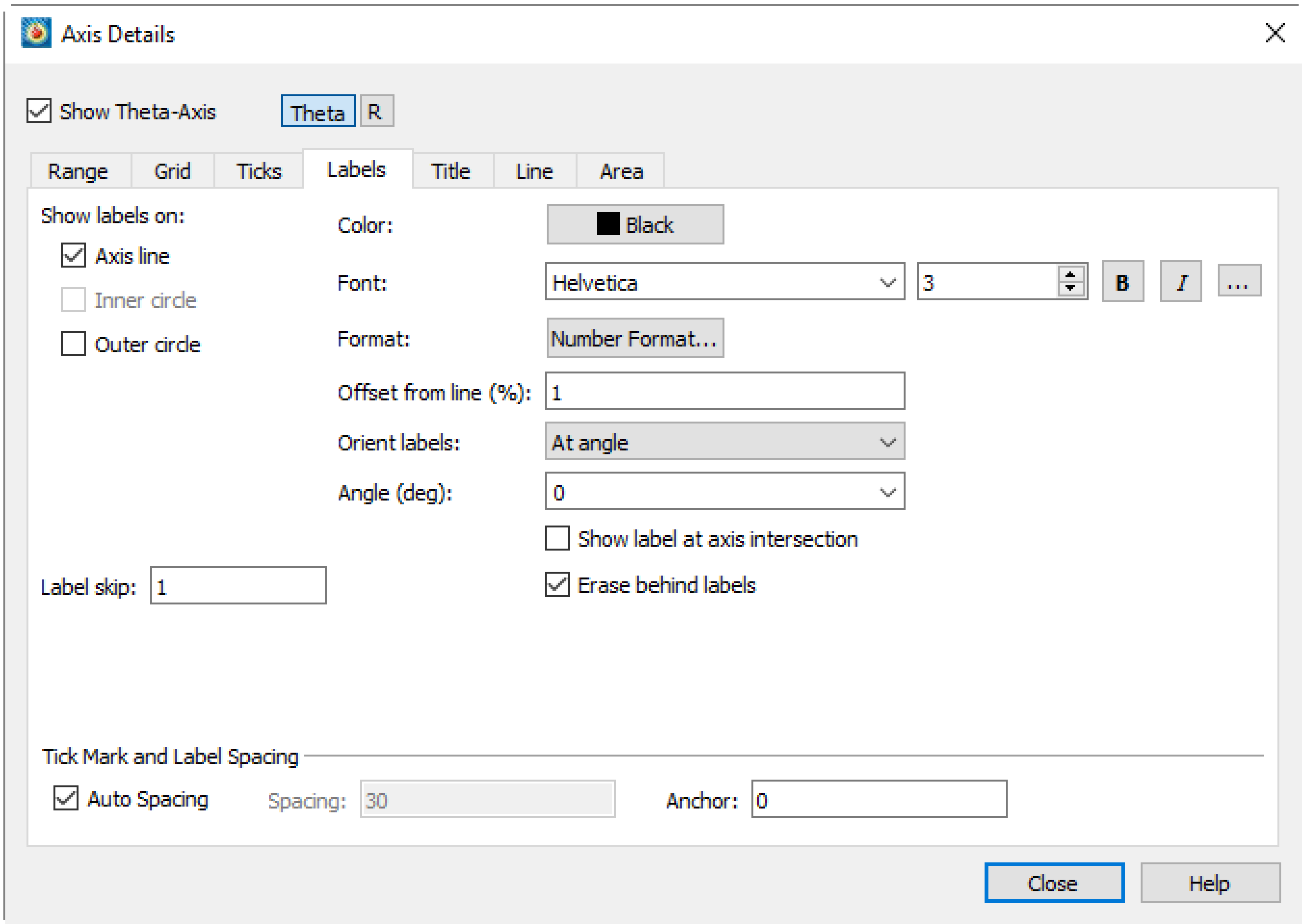
Tick Mark and Label Spacing

You can control tick mark and 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 360 recalculates the spacing. With "Auto Spacing" selected, Tecplot 360 also calculates the number of minor tick marks for you.

Spacing values are shared between tick marks and tick labels. You can change the spacing by adjusting the Auto Spacing, Spacing, and Anchor controls at the bottom of the page under "Tick Mark and Label Spacing".

Tick Mark Label Options

From the Labels page of the **Axis Details** dialog, you can specify the label attributes for the tick marks of each axis.



Show Labels On

Toggle-on the appropriate options for label display. The available options are dependent on plot type.

For Sketch, XY line, and 2D Cartesian line plots, choose from:

Axis Line

When toggled-on, axis labels will display on the selected axis line.

Grid Border Bottom

When toggled-on, axis labels will display on the bottom of the grid.

Grid Border Top

When toggled-on, axis labels will display on the top of the grid.

For 3D Cartesian line plots, choose from:

Axis Line

When toggled-on, axis labels will display on the selected axis line.

Opposite Edge

When toggled-on, axis labels will display on the opposite edge of the plot.

For Polar Line plots, choose from:

Axis Line

When toggled-on, axis labels will display on the selected axis line.

Inner Circle

When toggled-on, axis labels will display on the inner edge of the polar grid area. (Only available if the minimum value on the R-axis is greater than the R-Origin value.)

Outer Circle

When toggled-on, axis labels will display on the outer edge of the polar grid area.

Color and Font

Select the color and font in which you want your labels to appear. (See Font Folders and Fallback for more information on how fonts work with Tecplot 360.)

Number Format

Choose how the numbers should be formatted for axis labels using the Specify Number Format dialog.

Offset from Line (%)

Enter the offset of the tick mark labels from the axis.

Orient Labels

Select from the following additional options for label display:

At Angle

Labels oriented at the angle specified in the **Angle** drop-down menu.

Parallel

Labels are parallel to the axis.

Perpendicular

Labels are perpendicular to the axis.

Angle (deg)

If Orient Labels is set to "At Angle", specify the orientation of the tick mark labels relative to the axis. The angle is measured in degrees counter-clockwise from the axis.

Show Label as Axis Intersection

[2D, XY, and Polar Only] - Toggle-on to draw a label at the point where two axes intersect. Use this toggle if you have axis labels that are colliding, or are stacked on top of one another at the intersection of two axes.

Erase Behind Labels

Toggle-on to include a rectangle (with the color of the frame background) behind the label to increase the visibility of the label.

Label Skip

Specify the interval between tick mark labels.

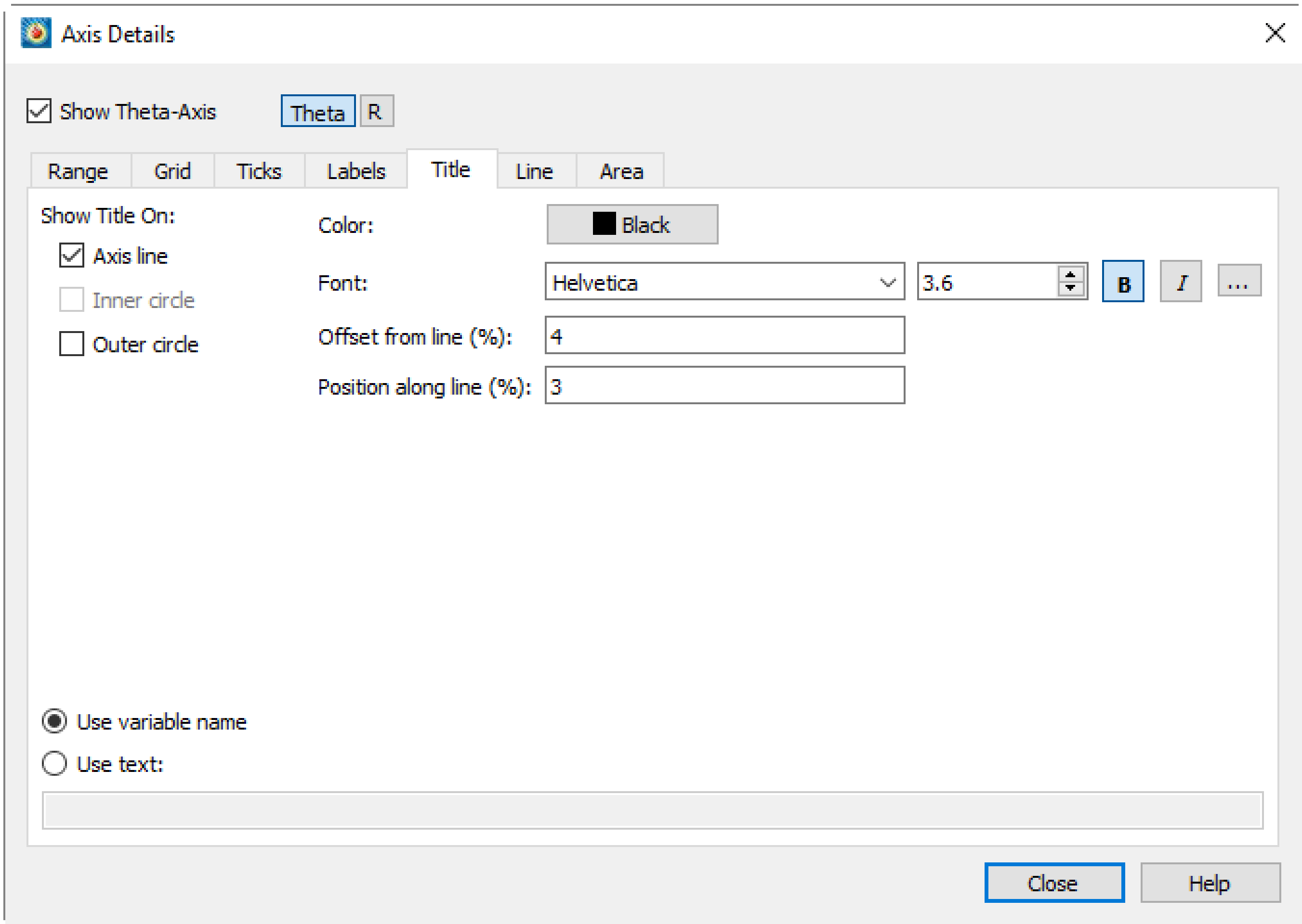
Tick Mark and Label Spacing

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 360 recalculates the spacing. With Auto Spacing selected, Tecplot 360 also calculates the number of minor tick marks for you.

Spacing values are shared between the tick marks and tick labels. You can change the spacing by adjusting the Auto Spacing, Spacing, and Anchor controls at the bottom of the Ticks or Label pages of the **Axis Details** dialog.

Axis Title Options

An axis title is a text label that identifies the axis. By default, Tecplot 360 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:

Show Title On

For any plot type, you can specify where to show the axis title. Toggle-on "Axis Line" to show the axis title directly on the corresponding axis. The remaining available options are dependent upon plot type.

For 3D

Opposite Edge.

For 2D, XY Line or Sketch

Grid Border Bottom or Grid Border Top.

For Polar Line plots

Inner Circle or Outer Circle.

Color and Font

Select the color and font in which you would like your axis to appear. (See Font Folders and Fallback for more information on how fonts work with Tecplot 360.)

Offset from Line

Prevents Tecplot 360 from printing your axis title directly on top of the axis. You may specify a positive or negative offset from one side or the other of the axis. An offset of zero prints the edge of the axis title on the axis.

You may also adjust the axis title offset using the Adjustor tool from the Toolbar (not available for 3D Cartesian plots).

Position along Line

Specify the start position of the axis title as a percentage of axis length.

Title

Choose one of the following display options:

Use Variable Name

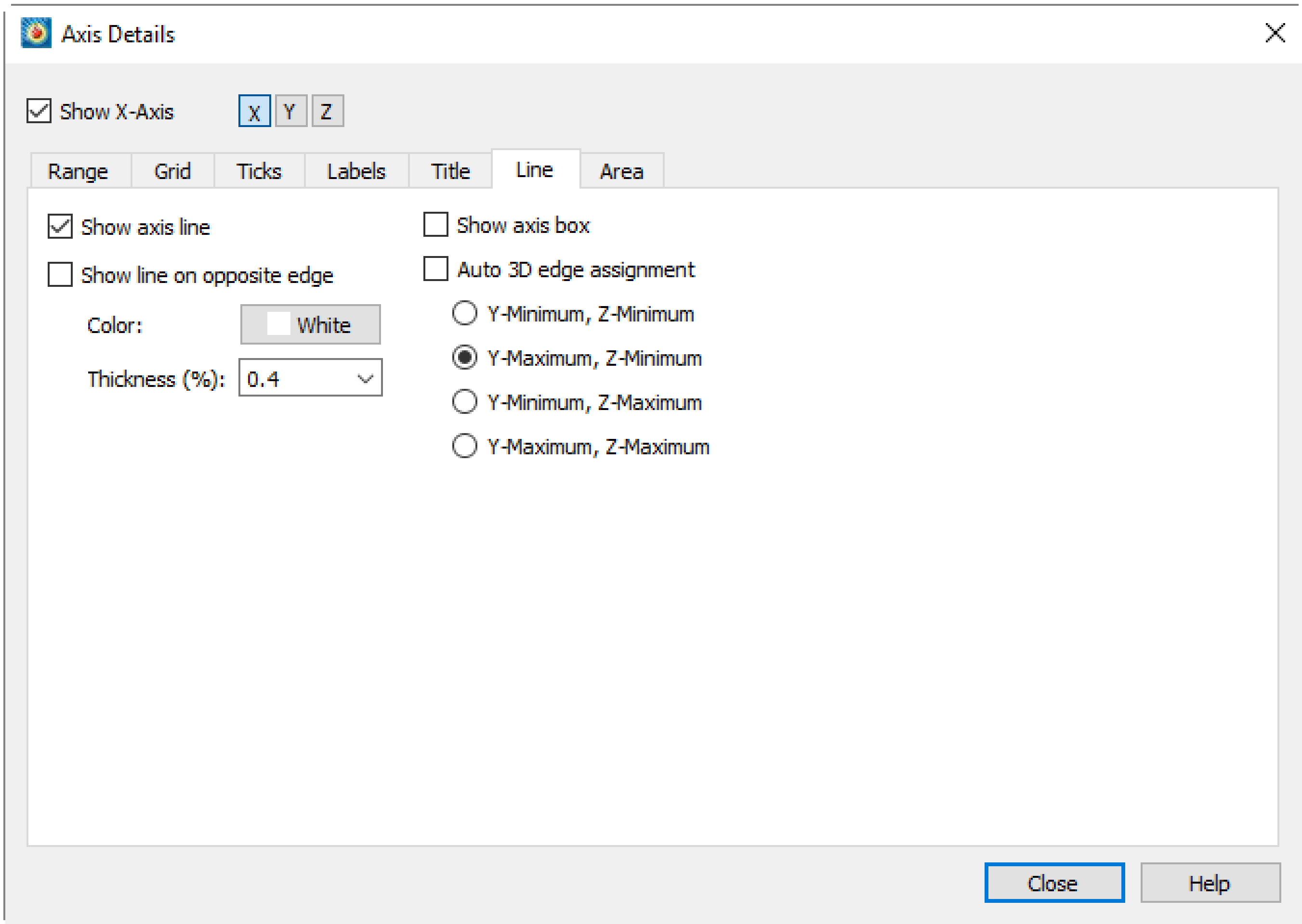
The legend header will display the variable name assigned to the contour group.

Use Text

The legend header will display text entered into the text field next to the options widget. The text can contain dynamic text (See Dynamic Text for information on dynamic text) and formatted using tags (See Text Details for information text formatting tags).

Axis Line Options

The actual axis line is shown by default whenever the axis is shown. However, you can hide the axis line without turning off the axis as a whole. To show or hide the axis line, select the Lines page of **Axis Details** dialog (accessed via the **Plot** menu).



The options in this dialog vary depending on the plot type and, for some plot types, the axis selected. All of the available options are described below, but not all will be available at any one time.

Show Axis Line

Toggle-on or off to display or hide an axis line.

Color

Select the color of your axis line using the Color Chooser.

Thickness

Select the thickness of your axis line.

Show Line on Opposite Edge

(3D Cartesian only) - Toggle-on or off to draw a line on the opposite edge from the selected axis line.

Show Grid Border

Turning on the grid border draws a border around your grid. Toggle-on or off to display or hide the borders of your grid. Not available for 3D Cartesian plots.

Color

Select the color of your grid border.

Thickness

Select the thickness of your grid border.

Align Axis With

Select what part of your plot with which you would like to align your axis. Not available for 3D Cartesian plots.

Sketch, XY line, and 2D Cartesian plots, choose from **Axis Value**, **Bottom**, **Top**, or **Viewport**. If you choose Axis Value (e.g. Y Value), enter the desired value in the text field. For Bottom or Top, enter the offset from the bottom or top. For Viewport, enter the position within the viewport.

Polar plots (Theta-axis), choose from **R Value**, **Inner Circle**, or **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.

Polar plots (R-axis),choose from **Theta Value**, **Inner Circle**, or **Outer Circle**. When aligning with a Theta Value, you may enter a Theta 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.

Theta Value

Align the R-axis with a specific Theta B=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.

Theta-Axis Value

Specify a Theta-axis value to specify the position of the axis line.

Offset (%)

Enter the offset of the line from the axis.

Angle

Enter the angle of the line from the axis.

Draw Axis in Both Directions

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.

Toggle-on "Draw Axis in Both Directions" to extend the axis line so that it spans the width of the grid area.

Draw Perpendicular Axis

Toggle-on "Draw Perpendicular Axis" to draw an axis line perpendicular to the main axis line.

Show Viewport Border

(Polar line plots only) Toggle-on "Show Viewport Border" to show the viewport border. The viewport border is defined on the Area page of the **Axis Details** dialog.

Color

Select the color of your viewport border.

Thickness

Select the thickness of your viewport border.

Show Line on Opposite Edge

(3D Cartesian only) - When toggled-on, axis lines will display on the opposite edge of the plot.

Color

Select the color of your grid border.

Thickness

Select the thickness of your grid border.

Show Axis Box

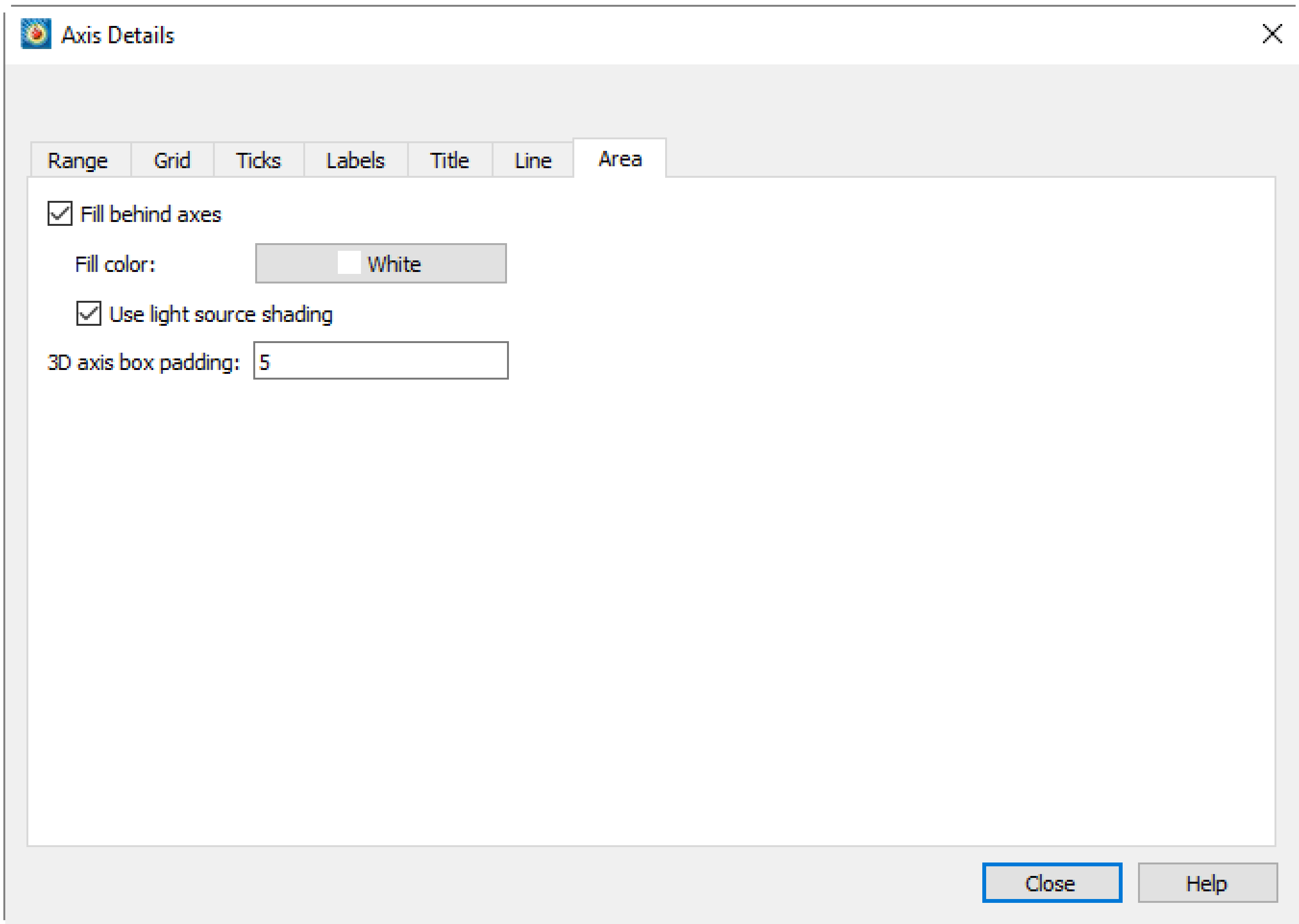
(3D Cartesian only) Toggle-on "Show Axis Box" to display all edges of all axes.

Auto 3D Edge Assignment

(3D Cartesian only) - Toggle-on "Auto 3D Edge Assignment" to place the three axis lines so they will not interfere with the drawing of the plot. If toggled-off, you have the option to place the line at any pair of edges, such as Y-Minimum & Z-Minimum, Y-Maximum & Z-Minimum, Y-Minimum and Z-Maximum, or Y-Maximum & Z-Maximum. The available pairs depend on the axis selected to edit.

Grid Area Options

The grid area of your plot is the area defined by the axes. From the Area page of **Axis Details**, you control whether the grid area or viewport are color-filled. The Area page is shown below.




```
 The Area page has the following options, some of which may not be
available depending on plot type and, for Polar line plots, the selected
axis:
```

Fill Grid Area

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.) Click the **Color** button to select the color using the Color Chooser.

For 3D Cartesian and Polar Line plots, the grid area is altered by changes to the axis ranges.

3D Cartesian options:

Fill Behind Axes

Select this option to fill the area behind the axes (your grid area) with a specific color.

Fill Color

Select the color with which to fill your grid area in the Color Chooser.

Use Light Source Shading

Select this option to light source shade the axis planes.

3D Axis Box Padding

Enter the minimum distance from the data to the axis box. Lowering this value does not immediately affect the appearance of the plot, since the existing axes still meet the new constraint; you can use Reset Range on the Range page see Axis Range Options for XY and 2D/3D Plots to force the plot to redraw.

Polar Line options:

Fill Grid Area

Select this option to fill the grid area with a specified color.

Fill Viewport

Select this option to fill the viewport area with a specified color.

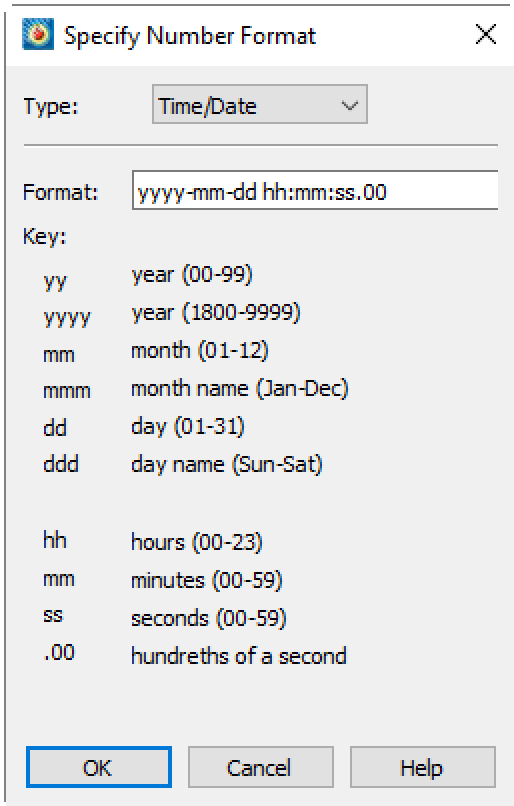
Viewport Position (%)

Select the position of the Viewport. The Viewport is the percentage of the entire plot area occupied by the plot grid. Select the location (as a percentage of the entire plot area) in which to place the Left, Right, Top, and Bottom borders of your grid area.

Time/Date Format Options

The **Specify Number Format** dialog is available for all plot types, including Sketch. You can use this dialog to display your axis labels in a number of different formats, including Time/Date format. The **Specify Number Format** dialog can be accessed by going to the **Plot→Axis→Label** or the **Plot→Axis→Range** page, and selecting the [Number Format] button. See Specify Number Format for more information on the **Specify Number Format** dialog.

You can also display elapsed time, instead of absolute time, on any axis. To do this, the original time/date data in your data file must indicate the elapsed time.)



To specify your axis label using Time/Date format, select "Time/Date" from the **Type** drop-down menu. Data is read forward from December 30, 1899. Tecplot 360 also accepts negative values to support dates back to January 1, 1800.

You can format your labels by entering the available Time/Date codes in the Format field. When entering a format, any combination or subset of the Time/Date formula may be used.

If you use "m" immediately after the "h" or "hh" code or immediately before the "ss" code, Tecplot 360 displays minutes instead of the month.

Use the following formula and table to enter your Time/Date codes:

Time Date Formula:

```
years-months-days hours:minutes:seconds
```

Time/Date Codes:

	Time/Date Code	Display Format
Years	yy	00-99
	yyyy	1800-9999

	Time/Date Code	Display Format
Months [1]	mmmmm	first letter of the month
	m	1-12
	mm	01-12
	mmm	Jan-Dec
	mmmm	January-December
Days	[d] [2]	total number of elapsed days
	d	1-31
	dd	01-31
	ddd	Sun-Sat
	dddd	Sunday-Saturday
	ddddd	S,M,T,W,T,F,S
Hours [3]	[h]	total number of elapsed hours
	h	0-23 or 1-12
	hh	00-23 or 1-12
	AM/PM	AM or PM
	A/P	AM or PM as "A" or "P"
Minutes	[m]	total number of elapsed minutes
	m	0-59
	mm	00-59
Seconds	s	0-59
	ss	00-59
	.0	Tenths
	.00	Hundredths
	.000	Thousandths

[1] Codes can be entered as upper or lower case letters; however, letters will be displayed as shown in the display format. Numbers that cannot be formatted as a time or date will be displayed as asterisks.

[2] Total number of elapsed days, hours, and minutes are valid for time values greater than or equal to zero, and equal to or less than 1,000,000 days.

[3] If you enter "AM/PM" or "A/P" in your Time/Date format, the "h" and "hh" hour codes are expressed using a 12-hour clock. Otherwise, hours are expressed in military time (24 hour clock).

Placing a backslash in front of a y, m, d, or s in the Time/Date formula will keep it from being processed as part of the formula. All characters not part of the Time/Date formula will appear as entered.

For example, "\year yyyy" will appear as "year 2008", as the backslash keeps the first y from being processed as part of the formula.

Examples

To display the time and date on your plot as a "Sat-Jan-05-2008", enter the following code:

```
ddd-mmm-dd-yyyy
```

To display the time and date on your plot as a "1-3-08", enter the following code:

```
m-d-yy
```

To display the time and date on your plot as a "9:30:05 AM", enter the following code:

```
h:mm:ss AM
```

To display an elapsed time, such as "3:10:15", enter the following code:

```
[d]:hh:mm
```

Microsoft Excel Support

Tecplot 360 supports Microsoft Excel (except for Mac Excel "1904" format) Time/Date number strings, with the exception of AM/PM time specifications, long day names, and long month names. This support allows you to create number formats in Excel and import them for use with your Tecplot 360 plots, or vice versa.

Time/Date number strings can be transferred from Excel to Tecplot 360 from Mar 1, 1900 forward.

Loading Time/Date Data

You can load time/date data into Tecplot 360 in the same way as any other data point. The following methods load time/date data as a floating-point number. (After loading, use **Plot**→**Axis**→**Labels**→**Number Format** to change the axis format to a time/date display.) The following methods can be used to load time/date data into Tecplot 360.

Excel Macro (add-on)

This add-on offers more options to load data from Excel. Although it requires installation into Excel, after installing, this method enables quick and simple data loading from Excel into Tecplot 360. Refer to Excel Add-In for details.

Text Spreadsheet Loader

For loading delimited files, use this loader. Refer to Text Spreadsheet Loader for instructions. Time/date data included in this format must be represented by the floating point number used by Excel and Tecplot 360. (See the Data Format Guide for more information on this formatting.)[xxxxxxxxxxxxxxxxxx placeholder for data format guide]

Text, Geometries and Images

You can enhance any plot, or create a drawing from scratch, using Tecplot 360's text and drawing tools. Tecplot 360 provides tools for creating polylines, circles, ellipses, squares, rectangles, and text. You can also insert BMP, JPEG, or PNG images to enhance your plot.

A plot that contains only these add-on elements and no plotted data is referred to as a *sketch* and can be created with the **Sketch** plot type. Figure 45 shows a sketch created with Tecplot 360 drawing tools.

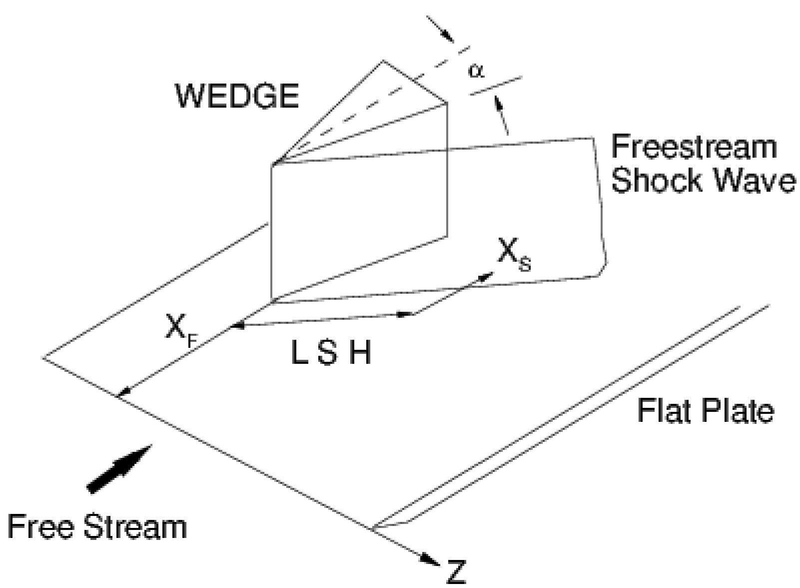


Figure 45. A sketch created with Tecplot 360.

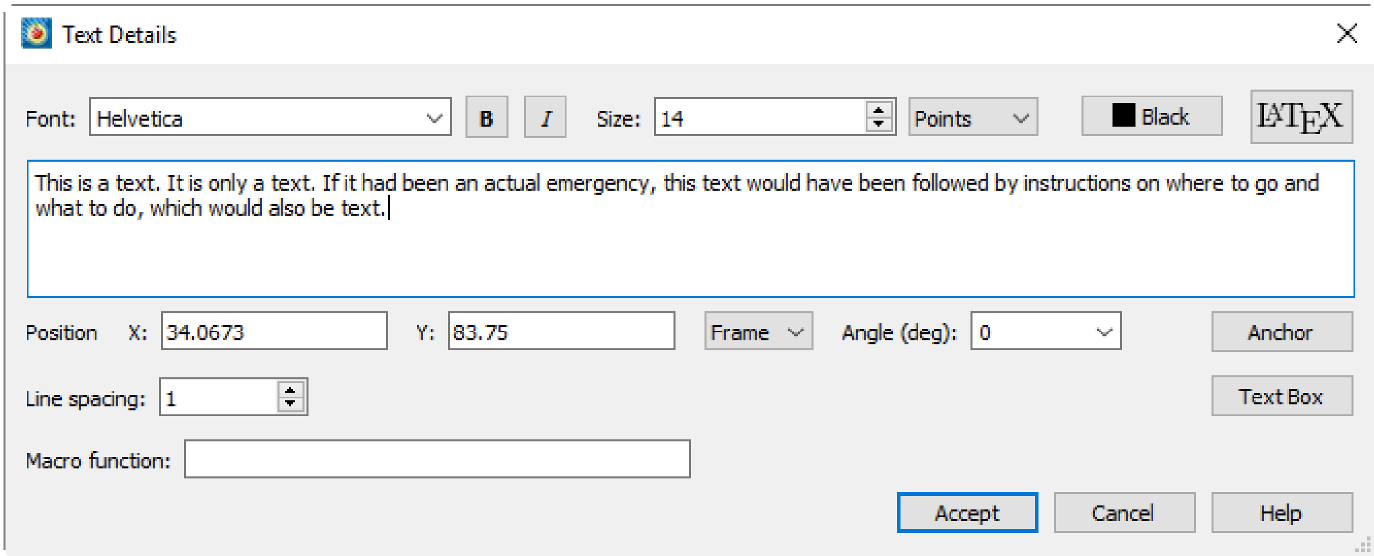
Text

To add text to your plot or sketch, either select the **Text** tool from the Toolbar or **Text** from the **Insert menu**. Click anywhere in a frame to indicate the location of the text. Use the **Text Details** dialog to enter and modify text and its formatting. To create additional text elements, you may click in the Tecplot 360 workspace at the desired location of the next text element while the dialog is open.

You can edit existing text by first choosing the Selector tool, then double-clicking the text element to open the **Text Details** dialog. Alternatively, right-click the text element and many of the text attributes can be changed in the **context menu** as well as selecting **Text Details**.

Text Details

The **Text Details** dialog has the following options:



Text Entry Field

Type the desired text in this box. Text will appear in your plot as you type.

You can embed **Greek**, **Math**, and **User-defined** characters into English-font strings by enclosing them with text formatting tags, together with the keyboard characters. See Custom Character and Symbol Definition for more information on defining your own characters.

The text formatting tags and their effects are as follows (format tags are not case sensitive and may be either upper or lower case):

...

Boldface

<i>...</i>

Italic

<verbatim>...</verbatim>

Verbatim

_{...}

Subscripts

^{...}

Superscripts

<greek>...</greek>

Greek font.

$...$

Math font.

<userdef>...</userdef>

User-defined font.

<helvetica>...</helvetica>

Helvetica font.

<times>...</times>

Times font.

<courier>...</courier>

Courier font.

> Not all fonts have Bold and/or Italic variants. For fonts that do not have these styles, the <b> and/or <i> tags may have no effect.

Embedding and escaping special characters work only in English-font text; they have no effect in text created in **Greek**, **Math**, or **User-defined** character sets.

You can produce subscripts or superscripts by enclosing any characters with _{...} or ^{...}, respectively. Tecplot 360 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 360 text strings. If you alternate subscripts and superscripts, Tecplot 360 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.

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.

> To produce more elaborate math expressions use latex expressions instead. See LaTeX Expressions.

Font

Select a font for the text. (See Font Folders and Fallback for more information on how fonts work with Tecplot 360.) You may click the **B** and *I* buttons as shortcuts for selecting a bold, italic, or bold italic variation of the chosen font.

> Not all fonts have Bold and/or Italic variants. For fonts that do not have these styles, the **B** and/or *I* buttons may have no effect.

Size

Select a character height unit from the drop-down menu, then enter the size in the associated field:

Points

Specify character height in points. One point is 1/72nd of an inch.

Frame%

Specify character height as a percentage of frame height.

Grid

Specify character height in grid units. (Available only when Position By is set to Grid.)

X/Y Position

Choose the position of the text element's anchor point (see Anchor button).

Position by

Choose to position the text using the frame or grid coordinate system.

Macro function

Enter the name of the macro function to be linked to this text object. See Linking Text and Geometries to Macros for more information. To run the linked macro function, hold down Control while right-clicking the text object in the workspace. (On Mac, hold down Command while right-clicking.)

Color

Select a color for the text using the Color Chooser.

Angle (deg)

Specify the orientation of the text relative to the axis. The angle is measured in degrees counter-clockwise from horizontal. Horizontal text is at zero degrees; vertical text is at 90 degrees. You can either enter an angle in degrees, or select from one of the preset angles in the drop-down.

Line Spacing

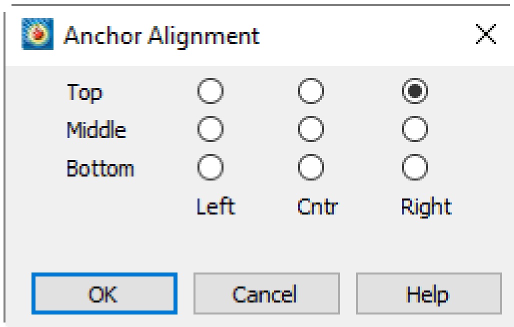
Enter the line spacing when multiple lines of text are entered: 1 for single-spacing, 2 for double-spacing, etc. Fractional values are permitted.

Text Box

Opens the Text Box dialog to specify the appearance of the box around the text (if any). See Text Box for more information.

Anchor

Choose how the text is aligned with the anchor point. As new text is added, the text expands away from the anchor point chosen in the Anchor Alignment dialog.



Font Folders and Fallback

Tecplot 360 supports TrueType (`.ttf`) and OpenType (`.otf`) fonts. Fonts installed in the standard folder for fonts on your operating system (for example, `C:\Windows\Fonts` on a Windows computer) appear in the font selection menu in the **Select Font** dialog and in other places where you choose a font.

If the fonts you want to use with Tecplot 360 are installed in other folders, or if your operating system does not have a default system location for fonts, you can edit the `tecplot.cfg` file in your Tecplot 360 installation directory, using Notepad or another text editor, to indicate the directories where the fonts you want to use are installed. Add a line like one of the following:

```
$!Interface FontPath = ' "/font/path" '
$!Interface FontPath = ' "C:\Windows\Fonts" "C:\More Fonts" '
$!Interface FontPath = ' "/first/path/" "/second/path" "/third/path" '
```

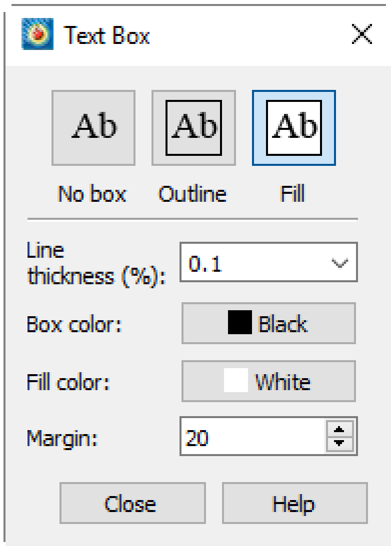
Replace the paths with the paths to the folders that contain the fonts you wish to use, taking special note of the single quote marks that wrap the entire list of folders and the double quotes around each folder's path. You can use just one or several font folders. As shown in the second example, if you wish to use your computer's standard fonts as well as fonts from other folders, you must manually specify the standard font folder along with the others.

If you open a layout that specifies a font that Tecplot 360 does not have access to on your system, the text "falls back" to Times. If the layout specifies a style you do not have (for example, a bold version of a font you have but which does not include a bold version on your system), the correct font is used, but the style (boldface, in this case) is not displayed.

It is possible to choose a font by typing its name directly into the font selection field in the **Text Details** dialog and other places where a font can be chosen, instead of choosing the desired font from the provided menu. If the font you specify in this way is not available to Tecplot 360 on your system, the text will appear as Times on your computer. However, the name you typed is recorded when you save a layout, and the text appears in the correct font if the layout is later opened on a computer on which the font is available.

Text Box

Use the **Text Box** dialog (accessed via the Text Box button in the **Text Details** dialog) to control the appearance of a box around the text. The following options are available:



None

Select this option to specify that no box is drawn around the text.

Outline

Select this option to specify an outline box around the text. The box is not filled, so any underlying Tecplot 360 object can still be seen.

Filled

Select this option to specify a filled box around the text. A filled box is opaque; if you place it over another Tecplot 360 object, the underlying object cannot be seen.

Line Thickness (%)

Specifies the thickness of the text box as a percentage of the frame width.

Box Color

Select the box outline color from the **Color Chooser** dialog.

Fill Color

Select the box fill color from the **Color Chooser** dialog.

Margin

Specify the margin as a percentage of the text character height.

Special Characters

Tecplot 360 supports Unicode text encoding. Your operating system should provide ways to enter Unicode text in non-English character sets, and any Unicode characters you can enter using these methods will display and print correctly in Tecplot 360.

If your system does not have the ability to enter Unicode text, characters in the ordinal range 160-255 provide support for most of the major European languages. The following table shows the characters in this range supported by Tecplot 360. Note that the two right-hand columns represent the extended European characters. Text formatting tags for Greek, Math, or User-defined characters work only with characters in the range 32-126 and is not available for the extended European characters. If your system is configured for European text entry, the European characters should appear and print automatically with no additional setup.

Character Index	English Text	Greek	Math	User Defined
32	(space)			
33	!	!	ϒ	
34	"	∀	′	
35	#	#	≤	
36	$	∃	⁄	
37	%	%	∞	
38	&	&	ƒ	
39	'	∍	♣	
40	(	(	♦	
41	)	)	♥	
42	*	∗	♠	
43	+	+	↔	
44	,	,	←	
45	-	−	↑	
46	.	.	→	
47	/	/	↓	
48	0	0	°	
49	1	1	±	
50	2	2	″	
51	3	3	≥	
52	4	4	×	
53	5	5	∝	
54	6	6	∂	
55	7	7	•	
56	8	8	÷	
57	9	9	≠	
58	:	:	≡	
59	;	;	≈	
60	<	<	…	
61	=	=	⏐	
62	>	>	⎯	
63	?	?	↵	
64	@	≅	ℵ	
65	A	Α	ℑ	
66	B	Β	ℜ	+
67	C	Χ	℘	×
68	D	Δ	⊗	✳
69	E	Ε	⊕	△
70	F	Φ	∅	▽
71	G	Γ	∩	⊡
72	H	Η	∪	◈
73	I	Ι	⊃	⊙
74	J	ϑ	⊇	⊙
75	K	Κ	⊄	★
76	L	Λ	⊂	∘
77	M	Μ	⊆	+
78	N	Ν	∈	○
79	O	Ο	∉	∅

Character Index	English Text	Greek	Math	User Defined
80	P	Π	∠	∅
81	Q	Θ	∇	☼
82	R	Ρ	®	☼
83	S	Σ	©	☼
84	T	Τ	™	☼
85	U	Υ	∏	☼
86	V	ς	√	☼
87	W	Ω	⋅	●
88	X	Ξ	¬	●
89	Y	Ψ	∧	●
90	Z	Ζ	∨	●
91	[	[	⇔	
92	\	∴	⇐	
93	]	]	⇑	
94	^	⊥	⇒	
95	_	_	⇓	
96	`		◊	
97	a	α	〈	●
98	b	β	®	●
99	c	χ	©	●
100	d	δ	™	●
101	e	ε	∑	●
102	f	φ	⎛	⎅
103	g	γ	⎜	
104	h	η	⎝	
105	i	ι	⎡	
106	j	ϕ	⎢	
107	k	κ	⎣	
108	l	λ	⎧	
109	m	μ	⎨	
110	n	ν	⎩	
111	o	ο	⎪	
112	p	π		
113	q	θ	〉	
114	r	ρ	∫	
115	s	σ	⌠	
116	t	τ	⎮	
117	u	υ	⌡	
118	v	ϖ	⎞	
119	w	ω	⎟	
120	x	ξ	⎠	
121	y	ψ	⎤	
122	z	ζ	⎥	
123	{	{	⎦	
124	|	|	⎫	
125	}	}	⎬	
126	~	∼	⎭	
127				

Character Index	Extended Character
160	
161	¡
162	¢
163	£
164	¤
165	¥
166	¦
167	§
168	¨
169	©
170	ª
171	«
172	¬
173	-
174	®
175	¯
176	°
177	±
178	²
179	³
180	´
181	µ
182	¶
183	·
184	¸
185	¹
186	º
187	»
188	¼
189	½
190	¾
191	¿
192	À
193	Á
194	Â
195	Ã
196	Ä
197	Å
198	Æ
199	Ç
200	È
201	É
202	Ê
203	Ë
204	Ì
205	Í
206	Î
207	Ï

Character Index	Extended Character
208	Ð
209	Ñ
210	Ò
211	Ó
212	Ô
213	Õ
214	Ö
215	×
216	Ø
217	Ù
218	Ú
219	Û
220	Ü
221	Ý
222	Þ
223	ß
224	à
225	á
226	â
227	ã
228	ä
229	å
230	æ
231	ç
232	è
233	é
234	ê
235	ë
236	ì
237	í
238	î
239	ï
240	ð
241	ñ
242	ò
243	ó
244	ô
245	õ
246	ö
247	÷
248	ø
249	ù
250	ú
251	û
252	ü
253	ý
254	þ
255	ÿ

Figure 46. Character Indices in Tecplot 360.

If your keyboard is not configured to produce a specific character, you can generate it by including the sequence `nnn` in your text, where *nnn* is from the character index table found in the table. For example, if your keyboard will not generate the é and you want to show the word "latté," enter:

```
latt\233
```

Custom Characters

You can create symbols, characters, and even custom fonts for use in Tecplot 360. See Custom Character and Symbol Definition for further instructions.

Dynamic Text

You can add special placeholders, called dynamic text, that change with the data or the display environment. For example, you can add a date placeholder that Tecplot 360 will replace with the current date at each redraw. Similarly, you can add a zone name or variable name placeholder.

For placeholders that represent an attribute of a Tecplot 360 object of which Tecplot 360 supports multiple instances (such as a frame, a dataset, or a zone), the placeholder by default refers to the current or active instance: for example, to the active frame if the placeholder refers to an attribute of frames. If there is no obvious default or active instance, the placeholder by itself usually refers to the first instance.

For such placeholders, you may specify the desired instance of the referenced type of object by enclosing an index in square brackets immediately following the placeholder name. For example, `&(AUXZONE[3]:BC)` refers to the zone auxiliary data named `BC` in the *third* zone.

In most cases, the index may also be written as `ACTIVEOFFSET=n` to specify the n^{th} active instance of an object type. For example, `&(ENDSLICEPOS[ACTIVEOFFSET=2])` refers to the end position of the *second* active slice group.

For placeholders that represent the minimum or maximum value of a variable, the text **(est)** is appended to the displayed value if it is an estimate calculated from variables in a subzone data file.

The complete list of placeholders is as follows:

Variables	Notes
&(AUXDATASET:*name*)	The value of the named auxiliary data attached to the dataset.
&(AUXFRAME:*name*)	The value of the named auxiliary data attached to the frame.
&(AUXPAGE:*name*)	The value of the named auxiliary data attached to the page.

Variables	Notes
&(AUXVAR[*nnn*]:*name*)	The value of the named auxiliary data attached to variable *nnn*.
&(AUXLINEMAP[Q]:*name*)	The value of the named auxiliary data attached to linemap Q, where Q, can be a simple integer *n*, in which case it represents the n^{th} linemap or Q can be the expression `ACTIVEOFFSET=n` in which case it represents the n^{th} active linemap.
&(AUXZONE[Q]:*name*)	The value of the named auxiliary data attached to linemap Q, where Q, can be a simple integer *n*, in which case it represents the n^{th} fieldmap or Q can be the expression `ACTIVEOFFSET=n` in which case it represents the n^{th} active fieldmap.
&(AXISMAX*n*)	Maximum value of the current *n*-axis range, where *n* is one of: A[9], R, X, Y, or Z.
&(AXISMIN*n*)	Minimum value of the current *n*-axis range, where *n* is one of: A[9], R, X, Y, or Z.
&(BYTEORDERING)	Displays the platform's byte ordering (INTEL or MOTOROLA).
&(DATE)	The current date, in the format *dd Mmm yyyy*.
&(DATASETFNAME[*nnn*])	Filename of the nnn^{th} file associated with the current dataset. If *nnn* is omitted, then all dataset filenames are shown, separated by new lines.
&(DATASETTITLE)	The current dataset title.
&(ENDSLICEPOS[*Q*])	The position of the ending slice plane. If *Q* is an integer then that represents the n^{th} slice group. If *Q* is `ACTIVEOFFSET=n` then that represents the n^{th} active slice group
&(EXPORTISRECORDING)	Returns "YES" if recording is active, otherwise returns "NO".
&(FRAMENAME)	The frame name.
&(INBATCHMODE)	Returns a value of 1 if the software is in batch mode, 0 if interactive.
&(ISDATASETAVAILABLE)	Returns a value of 1 if a dataset exists for the current frame, 0 if nonexistent.
&(ISOSURFACELEVEL[*nnn*])	The value of the contour variable on the nnn^{th} iso-surface for the first iso-surface group.

Variables	**Notes**
&(ISOSURFACELEVEL[*Q*][*nnn*])	The value of the contour variable on the *nnn*th iso-surface for the iso-surface group represented by *Q*. If *Q* is an integer then that represents the *n*th isosurface group. If *Q* is `ACTIVEOFFSET=n` then that represents the *n*th active isosurface group
&(LAYOUTFNAME)	The name of the current layout file.
&(LOOP)	Innermost loop counter.
&(MACROFILEPATH)	Path to the folder containing the most recently opened macro file.
&(MAX*n*)	Maximum value of the *n* variable, where *n* is one of: A[9], R, X, Y, or Z. For 2D or 3D Cartesian plots, the value is calculated from all active zones. For line plots, the value is calculated from the zone assigned to the first active linemap.
&(MAXB)	Maximum value of the blanking variable for the first active constraint. For 2D or 3D Cartesian plots, the value is calculated from the active zones. For line plots, the value is calculated from the zone assigned to the first active linemap.
&(MAXC)	Maximum value of the contour variable for contour group 1. For 2D or 3D Cartesian plots, the value is calculated from the active zones. For line plots, the value is calculated from the zone assigned to the first active linemap.
&(MAXI), &(MAXJ), &(MAXK)	[I, J, K]-dimension of the first active zone. For finite-element zones, MAXI is the number of nodes, MAXJ is the number of elements, and MAXK is the number of nodes per element (cell-based) or total number of faces (face-based) of the first active finite-element zone.
&(MAXS)	Maximum value of the scatter sizing variable of the active zones.
&(MAXU), &(MAXV), &(MAXW)	Maximum value of the variable assigned to the [X, Y, Z]-vector component of the active zones.
&(MAXVAR[*nnn*])	Maximum value of variable *nnn*.

Variables	Notes
&(MIN*n*)	Minimum value of the *n* variable, where *n* is one of: A[9], R, X, Y, or Z. For 2D or 3D Cartesian plots, the value is calculated from all active zones. For line plots, the value is calculated from the zone assigned to the first active linemap.
&(MINB)	Minimum value of the blanking variable of the first active blanking constraint. For 2D or 3D Cartesian plots, the value is calculated from all active zones. For line plots, the value is calculated from the zone assigned to the first active linemap.
&(MINC)	Minimum value of the contour variable of contour group 1. For 2D or 3D Cartesian plots, the value is calculated from all active zones. For line plots, the value is calculated from the zone assigned to the first active linemap.
&(MINS)	Minimum value of the scatter sizing variable for the active zones.
&(MINU), &(MINV), &(MINW)	Minimum value of the variable assigned to the [X, Y, Z]-vector component for the active zones.
&(MINVAR[*nnn*])	Minimum value of variable *nnn*.
&(NUMFRAMES)	Number of frames.
&(NUMPAGES)	Number of pages.
&(NUMPROCESSORSUSED)	Number of processors used. This may be different than the total number on the machine because of the $!Limits MaxAvailableProcessors configuration file command, or because of a product limitation.
&(NUMVARS)	Number of variables in the current dataset.
&(NUMXYMAPS)	Number of XY-linemaps assigned to the current frame.
&(NUMZONES)	Number of zones in current dataset.
&(OPSYS)	Displays the current operating system. 1=Linux/Macintosh, 2=Windows.
&(PAPERHEIGHT)	The paper height (in inches).
&(PAPERWIDTH)	The paper width (in inches).
&(PLATFORM)	The platform type (e.g. LINUX or WINDOWS).

Variables	Notes
&(PLOTTYPE)	Plot type of the current frame: 0 for Sketch, 1 for XY Line, 2 for Cartesian 2D, 3 for Cartesian 3D, and 4 for Polar Line.
&(PRIMARYSLICEPOS [*Q*])	The primary slice position. If *Q* is an integer then that represents the n^{th} slice group. If *Q* is `ACTIVEOFFSET=n` then that represents the n^{th} active slice group
&(PRINTFNAME)	The name of the current print file.
&(SLICEPLANETYPE[*Q*])	The type of the slice plane (X, Y, Z, I, J or K-planes). If Q is an integer then that represents the n^{th} slice group. If Q is `ACTIVEOFFSET=n` then that represents the n^{th} active slice group
&(SOLUTIONTIME)	The current solution time.
&(SOLUTIONTIME[*Q*])	Solution time of *Q*. If *Q* is an integer *nnn* then this represents the solution time for zone *nnn*. If *Q* is `ACTIVEOFFSET=n` then this represents the solution time for the first zone associated with the nnn^{th} active fieldmap. &(SOLUTIONTIME[5]) displays the solution time of the 5th zone. &(SOLUTIONTIME[ACTIVEOFFSET=3]) displays the solution time of the first zone in the 3rd active fieldmap.
&(STARTSLICEPOS[*Q*])	The position of the starting slice plane. If *Q* is an integer then that represents the n^{th} slice group. If *Q* is `ACTIVEOFFSET=n` then that represents the n^{th} active slice group
&(STRANDID[*n*])	The strandID of zone *n*.
&(STREAMSTARTPOS[*nnn*])	Starting position (X, Y, Z) of the nnn^{th} streamtrace.
&(STREAMTYPE[*nnn*])	Type (Surface Line, Volume Line, Volume Ribbon, Volume Rod) of the nnn^{th} streamtrace.
&($*string*)	The value of the system environment variable *string*.
&(TECHOME)	Path to the home directory.
&(TECPLOTVERSION)	Displays the version number.
&(TIME)	The current time, in the format *hh:mm:ss*.
&(VARNAME[*nnn*])	The variable name of variable *nnn*.

Variables	Notes
&(ZONEMESHCOLOR[*Q*])	Color of the mesh for zone represented by *Q*. If *Q* is an integer *nnn* then this represents the mesh color for zone *nnn*. If *Q* is `ACTIVEOFFSET=n` then this represents the mesh color for the first zone associated with the *nnn*th active fieldmap.
&(ZONENAME[Q])	The name of the zone represented by *Q*. If *Q* is an integer *nnn* then this represents the name of zone *nnn*. If *Q* is `ACTIVEOFFSET=n` and this is a field plot then this represents the name of the first zone associated with the *nnn*th active fieldmap. If *Q* is `ACTIVEOFFSET=n` and this is a line plot then this represents the name the zone associated with the *nnn*th active linemap.

The placeholders should be typed exactly as shown, although the index or offset parameters shown must be replaced with the actual index or offset you wish to use.

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)"
```

Environment Variables

System environment variables can be accessed directly from Tecplot 360 by using `&($string)`, where *string* is the name of the desired environment variable. Using environment variables within Tecplot 360 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. Note that all environment variables are treated as text strings.

Formatting Numbers and Strings

If you want a dynamic text string to be formatted in a specific way, you may include C-style formatting strings in the macro variable specification.

The syntax for including a format string is:

&(*DynamicTextString*%*formatstring*)

The *formatstring* should be in the following format. Note that the flags, the width, and the precision are all optional; only the specifier is required. The square brackets [] are not a part of the format string.

```
[flags][width][.precision]specifier
```

The following are available as flags:

-	Left-justify (default is right justification)
+	Precede positive numbers with + sign (default is sign only for negative numbers)
space	A blank space will be written in place of the sign if the number is positive
0	Left-pads numbers with zeroes to fill the specified width (rather than spaces)

The *width* specifies the minimum number of characters to be printed. If the dynamic text string is shorter than this length, it is padded with spaces. The string is not truncated if it is longer than this length.

The *precision* specifies the following:

character strings	The maximum number of characters to be printed.
integer values	The minimum number of digits printed; the number will be padded with leading zeroes to fulfill this requirement.
floating-point values	The number of digits to be printed after the decimal point.

When no *precision* is specified, the default is 1. When printing a character string value in a fixed-width field, then, you will want to specify the same value for both the *width* and the *precision* to achieve the desired result (otherwise you will print a maximum of 1 character due to the default *precision* of 1).

The following letters may be used as the *specifier*:

s	string of characters
d	signed integer
e	scientific notation with a lowercase "e"
E	scientific notation with an uppercase "E"
f	floating point
g	use %e or %f, whichever is shorter
G	use %E or %f, whichever is shorter
u	unsigned integer, written out in decimal format
o	unsigned integer, written out in octal format
x	unsigned integer, written out in hexadecimal (where a - f are lowercase)
X	unsigned integer, written out in hexadecimal (where A - F are uppercase)

Example 1

To display the message `Maximum contour value is: xxxxxx` where `xxxxxx` only has two digits to the right of the decimal place. You would use:

```
Maximum contour value is: &(MAXC%.2f)
```

If `|MAXC|` currently has a value of 356.84206 then the resulting string would show:

```
Maximum contour value is: 356.84
```

Example 2

If, in the above example, you wanted to use exponential format you could use:

```
Maximum contour value is: &(MAXC%12.6e)
```

Here the result would be:

```
Maximum contour value is: 3.568421e+02
```

Formatting Dates and Times

For dynamic text strings that represent a date and/or time, such as `&(SOLUTIONTIME)`, the following may be used:

yy	Two-digit year ("09")
yyyy	Four-digit year ("2009")
m	Month number ("3", "12"), or minute if preceded by hours token
mm	Zero-padded two-digit month ("09") or minute if preceded by hour token
mmm	Abbreviated month ("Oct" for October)
mmmm	Full month ("October")
mmmmm	Single-letter month ("O")
d	Day of month ("9", "10")
dd	Zero-padded two-digit day of month ("09")
ddd	Abbreviated day of week ("Sun" for Sunday)

dddd	Spelled-out day of week ("Sunday")
ddddd	Single-letter day of week ("S")
h	Hour ("7", "11")
hh	Zero-padded two-digit hour ("07")
s	Seconds ("5", "22")
ss	Zero-padded two-digit seconds ("04")
.0, .00, .000	Milliseconds to 1, 2, or 3 digits
am/pm, a/p	AM/PM indicator. All times are 24-hour unless an AM/PM indicator is included in the format string, in which case 12-hour formatting is used.
[d], [h], [m]	Elapsed days, hours, or minutes.
\-	Escapes (removes the special meaning of) the following character. May be used if your format string includes static text containing any of the above tokens.

Example:

Here is a date and time format string for a solution time:

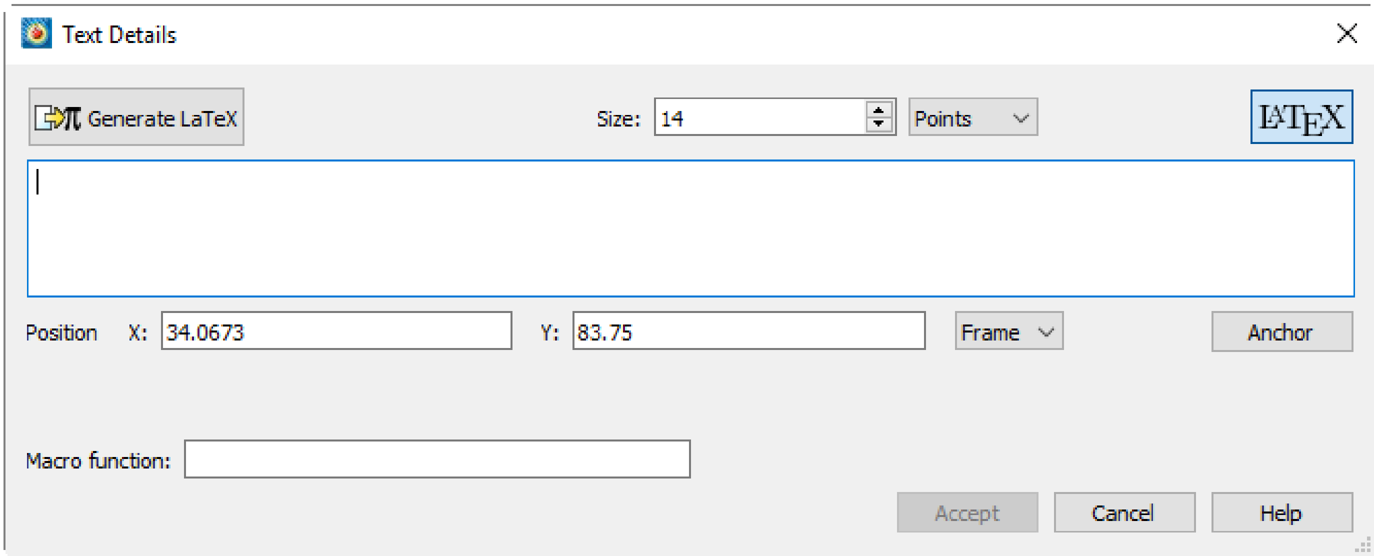
```
"Recorded on &(SOLUTIONTIME%dddd dd-mmm-yyyy at hh:mm:ss am/pm)"
```

The result would be similar to:

```
Recorded on Monday 14-Sep-2009 at 04:31:42 PM"
```

LaTeX Expressions

LaTeX is a computer language designed for typesetting. The most popular use of LaTeX is math and Greek fonts for technical purposes. Use the **LaTeX** option of the **Text Details** dialog (accessed via the LaTeX Box button in the **Text Details** dialog) to switch between normal and LaTeX text.



The following options are available:

Generate LaTeX

Renders the LaTeX output based on the current text field.

Size

Select a character height unit from the drop-down menu, then enter the size in the associated field. LaTeX size commands are still valid and will resize relative to the Size value:

Points

Specify character height in points. One point is 1/72nd of an inch.

Frame%

Specify character height as a percentage of frame height.

Grid

Specify character height in grid units. (Available only when Position By is set to Grid.)

LaTeX

Switches between regular and LaTeX output in the text field.

X/Y Position

Choose the position of the text element's anchor point (see Anchor button).

Position by

Choose to position the text using the frame or grid coordinate system.

Anchor

Choose how the text is aligned with the anchor point. As new text is added, the text expands away from the anchor point chosen in the Anchor Alignment dialog (See Text Details for a complete image of the Anchor Alignment dialog).

Macro function

Enter the name of the macro function to be linked to this text object. See Linking Text and Geometries to Macros for more information. To run the linked macro function, hold down Control while right-clicking the text object in the workspace. (On Mac, hold down Command while right-clicking.).

A few requirements are needed before LaTeX outputs can be generated. In order to generate LaTeX outputs, see LaTeX Setup for more information on how to install recommended LaTeX engines.

LaTeX Examples

Example 1: Mathematical Symbols

LaTeX allows for easy configuration symbol production. The example below shows how to create simple vector and square root symbols:

```
$$\vec a =\sqrt 2$$
```

The dollar signs denote the start of an equation. The output is shown below:

$$\vec a = \sqrt 2$$

Example 2: Mathematical Formula

In LaTeX, syntax is crucial to produce a correct output. Each symbol has its own syntax. The formula for Kirchhoff's current law, for example, is used like so:

```
Kirchhoff's current law\ldots
\begin{equation}
\sum_{k=1}^{n} I_k = 0 \; .
\end{equation}
```

Notice in Figure 47 below how the begin and end equations separate the equation from regular text. LaTeX changes paragraph spacing as well as symbols.

Kirchhoff's current law...

$$\sum_{k=1}^{n} I_k = 0 \; . \qquad (1)$$

Figure 47. Kirchhoff's Current Law in LaTeX

To prevent equation numbering, use the `\begin{equation*}⋯\end{equation*}` syntax.

Example 3: How to add summations in inline text versus display style

Note the difference in typsetting style between inline text style and display style equations.

```
\noindent
This is the inline text style:
$\lim_{n \to \infty} \sum_{k=1}^n \frac{1}{k^2} = \frac{\pi^2}{6}$.
\newline\noindent
And this is display style:
$$lim_{n \to \infty} \sum_{k=1}^n \frac{1}{k^2} = \frac{\pi^2}{6}$$
or
\begin{equation*} \lim_{n \to \infty} \sum_{k=1}^n \frac{1}{k^2} = \frac{\pi^2}{6}
\end{equation*}
```

This is the inline text style: $\lim_{n \to \infty} \sum_{k=1}^n \frac{1}{k^2} = \frac{\pi^2}{6}$.
And this is display style:

$$lim_{n \to \infty} \sum_{k=1}^n \frac{1}{k^2} = \frac{\pi^2}{6}$$

or

$$\lim_{n \to \infty} \sum_{k=1}^n \frac{1}{k^2} = \frac{\pi^2}{6}$$

Figure 48. Inline text versus display style

Example 4: Sizing and Accented Phrase

LaTeX allows for easy alphabetical accents. The following example shows the variety of accented phrases that can be accomplished by LaTeX typesetting. The phrase:

```
\huge ?`D\'onde est\'a el avi\'on? \\
```

Is easily converted to non-English symbols. In LaTeX, these symbols are converted inline with other text allowing fluid transitions between both.

Figure 49. Multi-lingual phrases in LaTeX

Example 4: Color, Sizing, and Text Boxes

LaTeX allows for easy customizability with certain keywords. Some keywords and formatting are dependent on packages that do not come preinstalled with your LaTeX engine. The following example uses the xcolor package to show custom formatting. To initialize xcolor in the preamble, add the following to the end of the preamble in the `tecplot_latex.mcr` file:

```
\usepackage{xcolor}
```

Now your Tecplot 360 installation will be ready to use the xcolor package. Use the following phrases as an example for color, size, and text box formatting:

```
\fcolorbox{red}{gray!20}{\color{red} Colors are fun} but \ldots \\
\emph{\small sometimes they are not ideal} \\
{\color{blue}\framebox{This might be better.} \\
But \colorbox{orange}{whatever you do,} \\
\color{black}don't make a {\huge Big Deal} about it.
```

This example shows three different text boxes each with different parameters. `\fcolorbox` allows for a colored frame as well as a background. `\framebox` has a transparent background and takes the frame color of whichever color is currently being used. `\colorbox` creates a highlight in the color specified. The output is shown below:

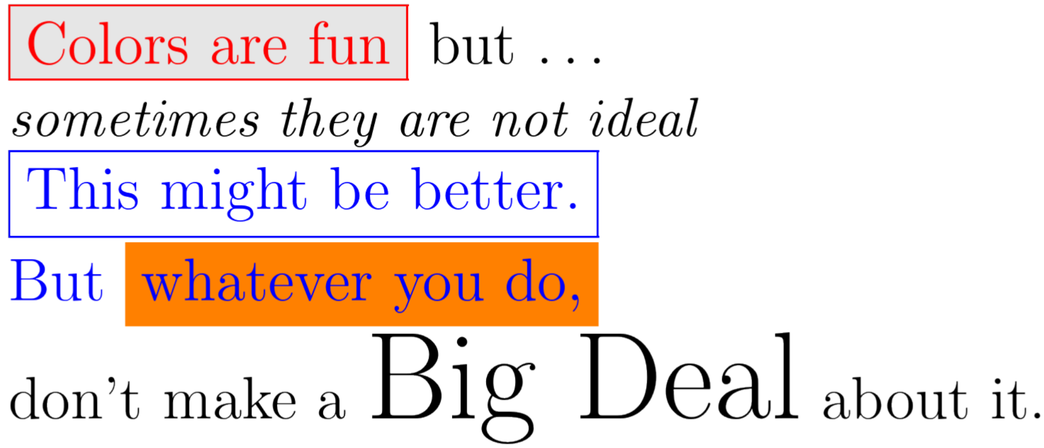


Figure 50. LaTeX Color, Sizing, and Text Boxes

Additional Examples

Example 5: Color with Equations

Example of LaTeX color with equations. See Example 4: Color, Sizing, and Text Boxes for information about xcolor package.

```
$\color{white}\overline{u^\prime v^\prime}$
```

$$\overline{u'v'}$$

Figure 51. Overline LaTeX with color

Example 7: Complex Fractions

```
$\frac{T - T_\infty}{T_f - T_\infty}$
```

$$\frac{T - T_\infty}{T_f - T_\infty}$$

Figure 52. Complex LaTeX fraction

Example 8: Simple Inline Equation

```
\textbf{Normalized Mean Velocity, $\overline{W}/U_\infty$}
```

Normalized Mean Velocity, $\overline{W}/U_\infty$

Figure 53. Inline LaTeX Equation

Additional LaTeX Resources

See this page for greek letter and math symbol syntax: www.overleaf.com/learn/latex/List_of_Greek_letters_and_math_symbols

For more examples on LaTeX use, syntax, or concepts check Tobias Oetiker's PDF, The Not So Short Introduction to LaTeX.

Many frequently asked questions can also be found on the TeX StackExchange website: tex.stackexchange.com/

Geometries

Geometries in Tecplot 360 are simply lines and shapes, including polylines (a set of line segments), circles, ellipses, rectangles, and squares. Images are also considered geometries. Character Indices in Tecplot 360 shows some examples of geometries.

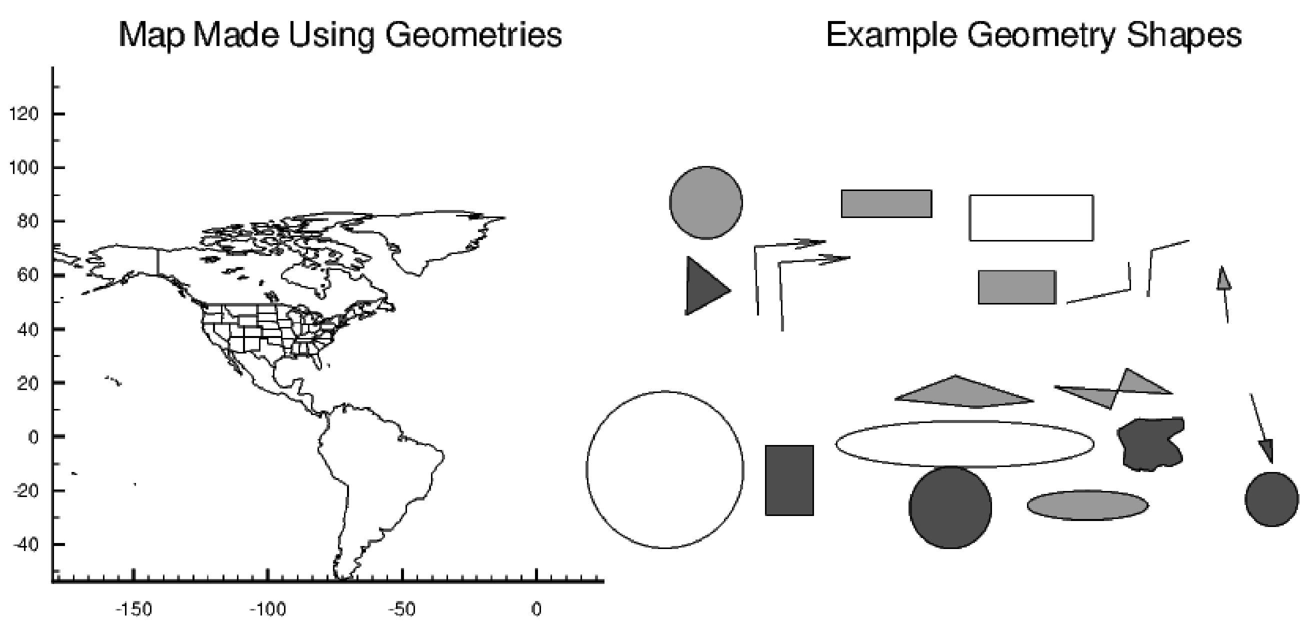


Figure 54. Sample Geometries

Geometry Creation

Geometries are created by drawing them in a frame using tools from the Toolbar or the **Insert** menu.

(There is no tool for inserting an image; to insert an image, choose **Insert→Image/Georeferenced Image**).

Polyline

A polyline is a single geometry consisting of one or more line segments. Add a polyline to your plot by using the button from the Toolbar or by selecting **Insert→Polyline**. To draw the polyline, 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, right-click, or press Escape 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.

Table 5. Table Keyboard Shortcuts for Polylines

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.

Circle

Add a circle to your plot by using the button from the Toolbar or by selecting **Insert→Circle**. To draw the circles, click at the desired center point of the circle; drag the mouse until the circle is the desired radius, then release.

Ellipse

Add an ellipse to your plot by using the button from the Toolbar or by selecting **Insert→Ellipse**. To draw the ellipse, click at the desired center point of the ellipse; drag the mouse until the ellipse is the desired size and shape, then release.

Square

Add a square to your plot, using the button from the Toolbar or by selecting **Insert→Square**. 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.

Rectangle

Add a rectangle to your plot, using the button from the Toolbar or by selecting **Insert→Rectangle**.

To draw the rectangle, drag the mouse until the rectangle is the desired size and shape. In contrast to squares, rectangles can propagate in any direction.

Images

Tecplot 360 can import images from `JPEG`, `BMP`, and `PNG` files. These images can be used as logos or as a backdrop to your plot. To add an image to your plot, choose **Insert→Image/Georeferenced Image** and browse to the desired image file. When you insert an image, the image is initially centered in the frame at a preset size. It can be resized by dragging its handles (at the corners and at the midpoints of its edges) when it is selected.

Images cannot be included in data files. When you save a data file, even if you indicate that you wish 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 Layout Files, Layout Package Files, Stylesheets for details.

Images *are* included in layout package files.

Georeferenced Images

Georeferenced images can be added if they are paired with world files. Valid world files include `.bmpw`, `.bpw`, `.jpgw`, `.jgw`, `.pgw`, `.pngw`, `.wld`. When a world file is selected in the Insert Image File loader, it will automatically look for a corresponding image file of the same name.

World files supply information about the image's coordinate system including the location, scale, and rotation of the image on a defined coordinate system. For more information about how each line in a world file represents the coordinate system, see this Wikipedia World File article.

Georeferenced Image Dialog

Displays information about the georeferenced image and allows the georeferenced image to be moved.

File Name
: Displays the path of the original georeferenced image file.

Resolution
: Displays the original resolution of the georeferenced image in pixels.

Z Position
: Enter the Z-coordinate of the anchor position of the georeferenced image.

Z min
: Displays the minimum value of the Z-coordinate.

Z max

Displays the maximum value of the Z-coordinate.

Show in all Frames with the Same Data

Toggle-on this option to display the geometry in all frames sharing the active frame’s dataset.

Geometry Details

Use the **Geometry Details** dialog to specify attributes of polylines, circles, ellipses, squares, rectangles, and images. To access the **Geometry Details** dialog, first choose the **Selector tool**, then double-click the geometry or image, or right-click it and choose Geometry Details or Image Details from the context menu. From the **context menu** the line and fill attributes can be changed easier than selecting **Geometry Details**.

The following options are available for most types of geometries (the line-related options are not, however, available for images):

Line Color

Select a color for the geometry from the Color Chooser.

Line Pattern

Select the desired pattern (Solid, Dashed, Dotted, LongDash, or DashDotDot).

Pattern Length (%)

Specify the length of the line pattern as a percentage of the frame width.

Line Thickness (%)

Specify the thickness of the line as a percentage of the frame width.

Fill with Color

Toggle-on to fill a circle, ellipse, square, rectangle or line segment polygon. Then select a color for the fill using the Color Chooser.

Clipping

Clipping displays only the portion of an object that falls within a specified 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. For this reason, the clipping options are not available for geometries using frame coordinates.

If you have specified the Grid coordinate system, you can choose to clip your geometry to the frame or 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. This option is therefore unavailable in 3D Cartesian plots.

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.

Draw Order

Geometries can be drawn 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.

> You can place text and geometries in any order you like. Tecplot 360 draws all geometries first, in the order in which they were placed, then all text. Use the **Send to Back** and **Bring To Front** commands on the **Edit** menu to reorder objects.

Attach to Zone/Map

Toggle-on **Attach to Zone/Map** to attach the geometry to a particular zone or mapping by entering the number of the zone or mapping. Geometries that are attached to an inactive or non-existent zone are not displayed.

Show in all Frames with the Same Data

Toggle-on this option to display the geometry in all frames sharing the active frame's dataset.

Macro Function

Specify the name of the macro function to be linked to the geometry. See Linking Text and Geometries to Macros for more information. To run the linked macro function, hold down Control while right-clicking the geometry in the workspace. (On Mac, hold down Command while right-clicking.)

Coordinate System

Specify the coordinate system for the geometry (**Frame** or **Grid**). When created, screen objects default to the grid coordinate system. Changing the coordinate system automatically converts position and size fields so that the geometry will remain in the same place at the same size in the current view.

Frame

The geometry is always displayed at constant size and position when you zoom in or out of the plot.

Grid

The geometry resizes and moves with the data grid. However, the geometry remains fixed when you rotate the plot. Changing the center of rotation may cause the geometry to move.

X, Y

Enter the X and Y-coordinates of the anchor position of the geometry (in frame units if the coordinate system is frame; in grid units if the coordinate system is grid). This point is the center for a circle, the upper left corner for a rectangle, etc.

The following fields are specific to a single geometry type:

Polyline

The Arrowhead Style options control the appearance of an arrowhead on the polyline.

Attachment

Choose the end or ends of the polyline by selecting the appropriate check boxes.

Style

Plain, filled, or hollow.

Size(%)

Specify the size of the arrowhead as a percentage of frame height.

Angle

Specify the angle the arrowhead makes with the polyline.

Circle

Controls the radius and precision of approximation of the circle:

Radius

Set the radius of the circle (in coordinate system units, **Frame** or **Grid**).

Number of Sides

Enter the number of polylines used to approximate the circle.

Ellipse

Controls the shape and precision of approximation of the ellipse:

Horizontal Axis

Set the horizontal axis of the ellipse (in coordinate system units, **Frame** or **Grid**).

Vertical Axis

Set the vertical axis of the ellipse (in coordinate system units- **Frame** or **Grid**).

Number of Sides

Enter the number of polylines used to approximate the ellipse.

Square

Controls the size of the square:

Size

Set the size of the square (in coordinate system units, **Frame** or **Grid**).

Rectangle

Controls the size and shape of the rectangle:

Width

Set the width of the rectangle (in coordinate system units, **Frame** or **Grid**).

Height

Set the height of the rectangle (in coordinate system units, **Frame** or **Grid**).

Image

Displays information about the image and allows the resizing method to be set.

File Name

Displays the path of the original image file.

Resolution

Displays the original resolution of the image in pixels.

Filter

The Resize filter determines how the image is resized to fit the screen. The following filters are available:

Fast (textures) (default)

Tecplot 360 uses OpenGL textures to resize the image. This is the fastest option (given sufficient graphics space). However, the accuracy of the image may suffer, especially when reducing an image to a size much smaller than it was before.

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 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.

Three-dimensional Line Geometries

Three-dimensional line geometries cannot be created interactively; they must be created in a data file or using an add-on. In order to display 3D geometries, you must either include at least one zone in the data file with the 3D geometries or read the 3D geometries in, using the **Add to Current Data Set** option, after having first read a dataset into the frame.

Linking Text and Geometries to Macros

Each text or geometry you create, or image you insert, can be linked to a macro function. This macro function is called when you hold down the Control key (Command key on Mac) and click the right mouse button on the text or geometry or image. Macro functions are specified with the Macro Function field in the **Geometry Details** dialog or in the **Text Details** dialog (accessed by selecting your object and right-clicking.

In order to be attached to a text or geometry object, the macro function must be a "retained" macro function. A macro function is "retained" via either of the following scenarios:

- running a macro file that contains the required macro functions
- including it in your `tecplot.mcr` file (which is run at startup, kmaking it a special case of the preceding scenario)

In both cases, the macro function is defined using the `$!MACROFUNCTION` macro command. Refer to the Scripting Guide for additional information.

Part 4: Data Manipulation

Blanking

Blanking allows you to exclude specific portions of zones from being plotted (in other words, selectively display certain cells or data points) based on variable values. In 3D, the result is analogous to a cutaway view.

Blanking settings apply only to the active frame, but Value Blanking settings for multiple frames may be synchronized using frame linking. Refer to Frame Linking for more information on linking. Blanking results for volume zones depend upon the Surfaces to Plot setting on the Surfaces page of the **Zone Style** dialog (See Surfaces for more details).

In the following discussions, the term "cell" is used. In I-ordered datasets, a cell is the connection between two adjacent points. In IJ-ordered datasets, a cell is the quadrilateral area bounded by four neighboring data points. In IJK-ordered datasets, a cell is the six-faced (hexahedral) volume bounded by eight neighboring data points. For finite element datasets, a cell is equivalent to an element.

The forms of blanking are as follows:

Value Blanking

Cells (or portions of cells) of selected zones or line plot mappings are excluded based on the value of the value blanking variable at the data point of each cell or at the point where each cell intersects a constraint boundary.

IJK Blanking

Cells of one IJK-ordered zone are included or excluded based on the index values. (IJK-ordered zones only)

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 selected zones of all types, while IJK Blanking affects a single IJK-ordered zone.

In general, all types of blanking affect all field layers, zones, and all other plot attributes except for edge layers. with the following exceptions:

Table 6. Plot attributes not affected by blanking.

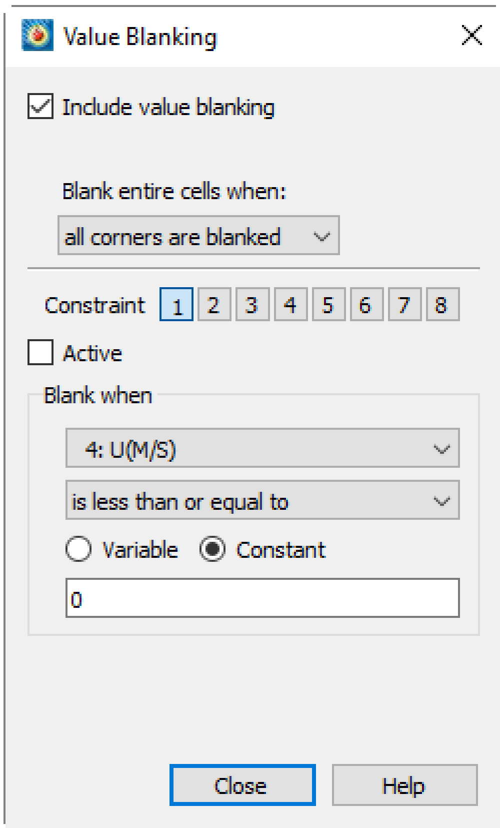
Type of Blanking	Attribute Not Blanked
Value Blanking	Edge Layer
IJK Blanking	• Derived Objects[10] (slices, streamtraces, or iso-surfaces) • Finite-element zones • Unstructured/Unorganized zones

Value Blanking

Value blanking allows you to selectively eliminate or trim cells (only) and elements from Line and 3D field plots. For each active constraint, you specify a value blanking variable, a constant value or another variable, and a conditional statement telling Tecplot 360 to blank that region in relation to the specified variable or constant.

Value Blanking for Field Plots

To include value blanking in your plot, go to the **Plot** menu and select **Blanking**→**Value Blanking**. The **Value Blanking** dialog has the following options:



Include Value Blanking

Toggle-on to include value blanking.

Note that it is possible to edit the settings in the Value Blanking dialog without actually activating blanking. This can make it faster to set up blanking since the plot won't need to redraw after every change of settings.

Blank entire cells when

Select one of the following blanking styles:

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 value is blanked

Cells are removed from the plot based on the primary value for a cell. The primary value for the cell is dependent upon the zone type and the variable value location, as outlined in the following table.

Zone Type	Value Location	Source of Primary Value
Ordered	Cell Centered	Cell value
Finite element	Cell Centered	Cell value
Ordered	Nodal	Lowest indexed corner in the cell
Finite element	Nodal	First node in the connectivity list for the cell

Refer to the Zone/Variable Info Page of the **Data Set Information** dialog to determine the Zone Type and Value Location.

Constraint

You can establish up to eight value blanking constraints. Click a numbered button here to choose the constraint to be added or changed.

Active

Toggle-on to activate the chosen constraint.

Blank when

For each constraint, set the following parameters:

- Select the variable to use for value blanking.

> It is often convenient to create a new variable for use as the value blanking variable. This allows you to manipulate its values without changing any other part of the plot. You can create a new variable using the **Specify Equations** dialog (accessed via **Data**→**Alter**). See Equation Syntax.

- Specify an operation to describe how the blanking variable will be compared to the constant or variable following it.
- Choose to compare the blanking variable to either another variable or to a constant. If you are comparing to a variable, select the variable; if comparing to a constant, enter the constant.

Show Constraint Boundary

Toggle-on to display a line that separates the region of your data that is blanked from the region which is not blanked. Set the appearance of this line using the following controls:

Color

Select the color of the boundary in the Color Chooser.

Line Pattern

Select the line pattern for the boundary.

Pattern length

Select the length of the pattern as a percentage of frame height.

Line thickness

Select the width of the boundary line as a percentage of frame height.

> Value Blanking has no effect on edges of an ordered zone. If the edge is turned on, the edge of the entire zone (without value blanking) is plotted.

The following figure illustrates the various value blanking modes for plots.

Figure 55. The effects of the different value blanking options in field plots for a constraint where a variable is less than or equal to zero. The dark shading indicates the areas that are not blanked. Clockwise from upper left: Blank cell when primary value is blanked. Blank cell when all corners are blanked. Trim cells along mathematical constraint boundary. Blank cell when any corner is blanked.

Value Blanking Settings for Individual Zones

Using the Effects page of the **Zone Style** dialog (accessed via the **Plot** menu or the Plot sidebar), you can turn value blanking on and off for individual zones. Simply highlight the zone(s), and toggle the checkbox in the Use Value Blanking column.

Value Blanking for Line Plots

For line plots, value blanking excludes data points from consideration in the resulting plot. The settings are the same as for Value Blanking for Field Plots, except that you cannot choose the behavior of blanking relative to cells, as this does not apply to line plots.

As alternatives or adjuncts to value blanking, 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 of the independent variable for individual mappings.

Figure 56 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.

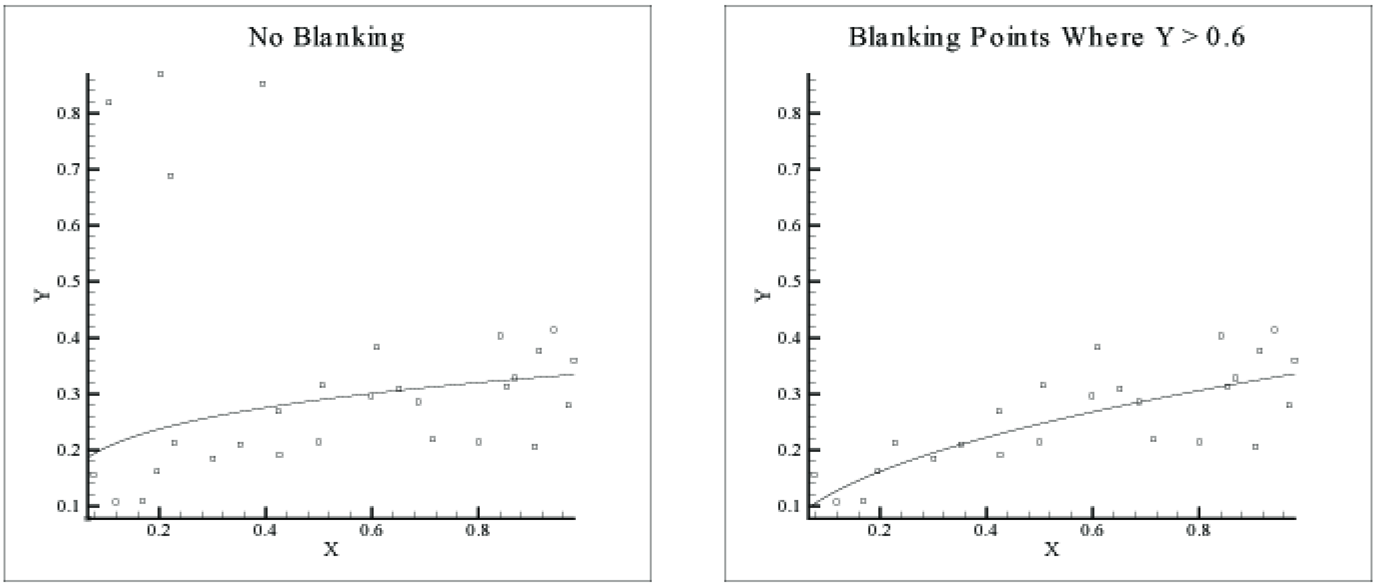


Figure 56. XY Line plots showing the effect of value blanking.

IJK Blanking

IJK Blanking is available for 2D and 3D ordered zones, in the 2D and 3D Cartesian plot types. 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 dataset with a section "cut away" to show the interior, such as the plot shown in Figure 57.

> To use IJK-blanking, you must have a 2D or 3D 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.

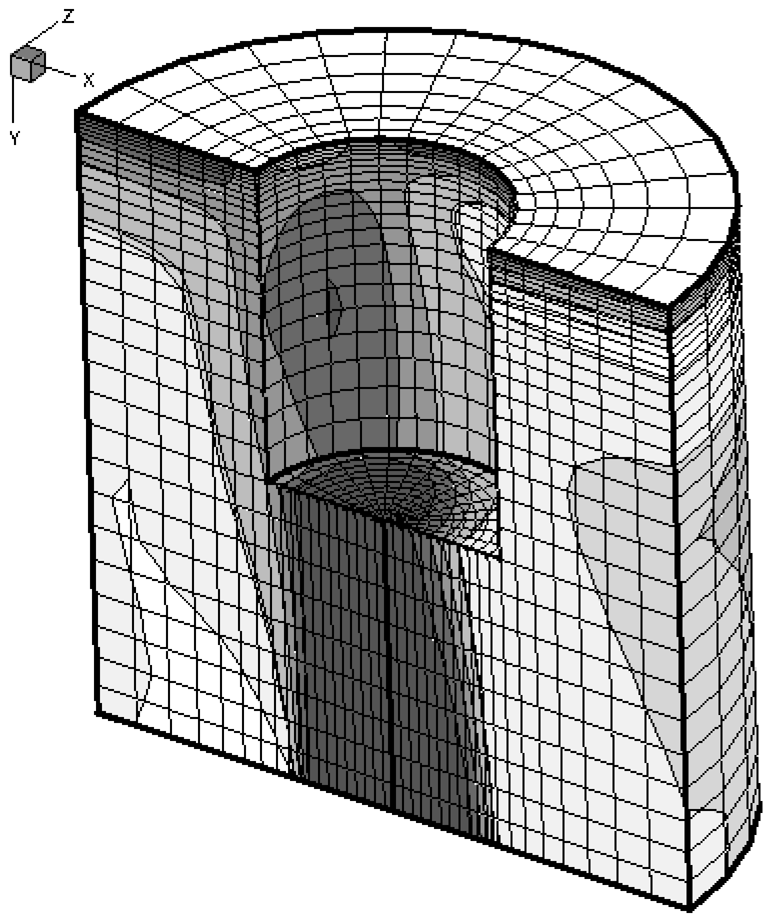
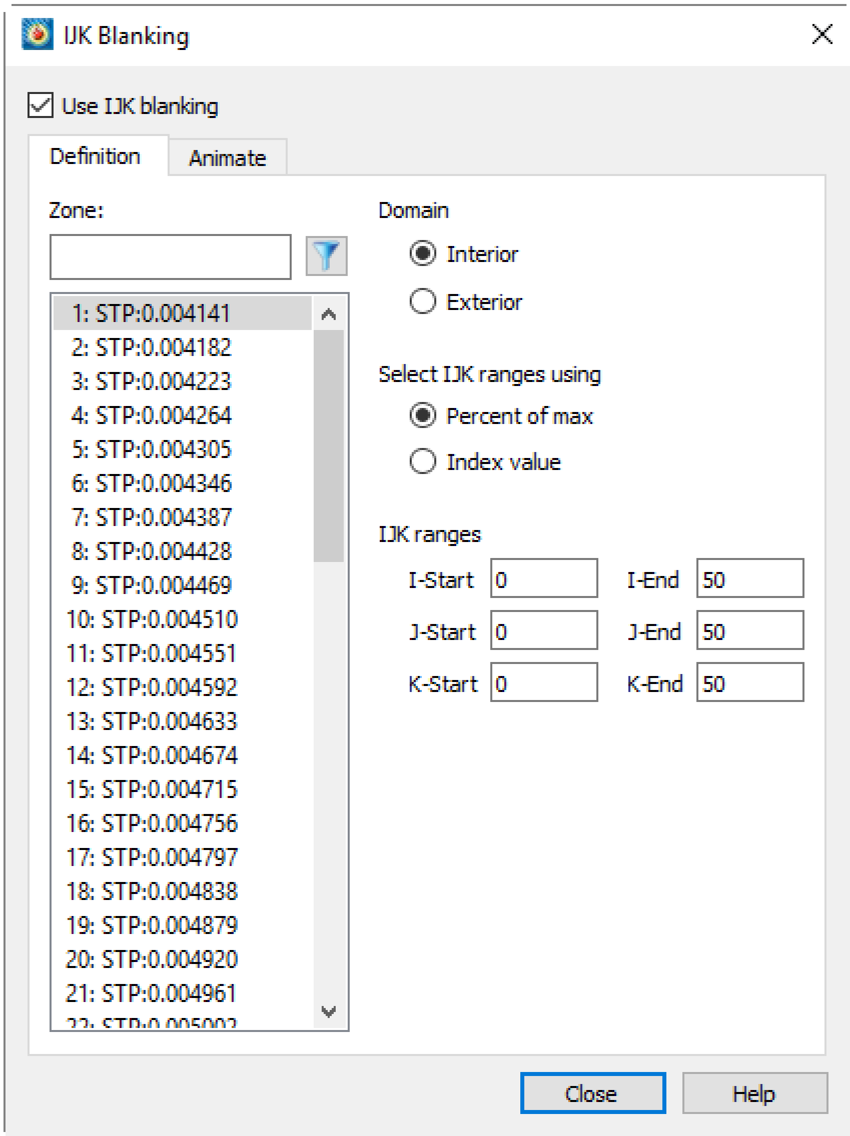


Figure 57. A cutaway plot created with IJK Blanking.

To use IJK-blanking, select **Blanking→IJK Blanking** from the **Plot** menu.This option is available only when your data set has at least one IJK ordered zone.



The **Use IJK Blanking** checkbox at the top of this dialog turns IJK blanking on or off. The Definition page has the following options.

Use IJK Blanking

Toggle-on to include IJK Blanking in your plot.

Zone

Select the zone to which to apply IJK Blanking by clicking in the displayed list of IJK ordered zones. You may select only one zone.

If your data set has many zones, it is useful to filter the zone list. Type part of a zone name and press Enter (or click the filter button next to the zone name field) to display only the zones having a name containing the entered text.

Domain

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 58 shows an ordered zone with IJK Blanking with Interior domain.

Exterior

Cells outside the specified index ranges are blanked. Those inside are plotted. Exterior plots a sub-zone of the zone. The right side of Figure 58 shows an ordered zone with IJK Blanking with Exterior domain.

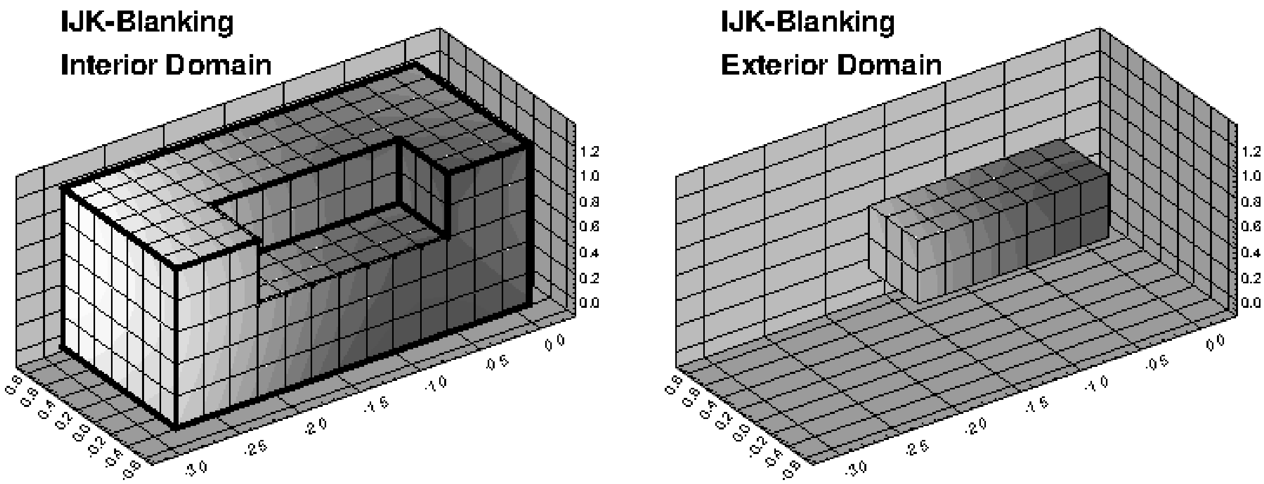


Figure 58. IJK Blanking with Interior domain (left) and Exterior domain (right).

Select IJK ranges using

Specify the format in which you will specify the index ranges by selecting one of the following option buttons:

Index Value

Specify the I, J, and K-index ranges using absolute index values.

Percent of Max

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 dataset by setting the start percentage to 33.3 and the end percentage to 66.6.

> 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 layout can easily be used for data sets containing zones of various sizes.

For information on using the Animation page, see IJK Blanking Animation.

Blanking Settings for Derived Objects

You can opt to turn blanking on or off for derived objects (iso-surfaces, streamtraces, or slices) in their respective **Details** dialogs.

Iso-surfaces

The option is located on the Style page of the **Iso-surface Details** dialog (click the button to the right of Iso-Surfaces in the Plot sidebar). Refer to Iso-Surface Contour and Shade for details.

Streamtraces

The option is located on the Integration page of the **Streamtrace Details** dialog (click the button to the right of Streamtraces in the Plot sidebar). Refer to Integration Page for details.

Slices

The option is located on the Definition page of the **Slice Details** dialog. Refer to Definition Page for details.

Data Operations

Plots in Tecplot 360 rely on the datasets attached to each frame. You can modify, create, transform, interpolate, duplicate, and delete the data in the current dataset using the **Data** menu. You can also use the data operation capabilities of Tecplot 360 to create plots of analytical functions. By using layout files, macros, and/or equation files, you can create complex data operations that can be repeated on different datasets.

Changes to the dataset within Tecplot 360 do not affect the original data file(s). You can save the modified data by choosing Write Data 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 overwritten with new data, the operations are applied to the new data. Alternatively, any datasets that have been modified are also saved to data files (see Layout Files and Layout Package Files for details).

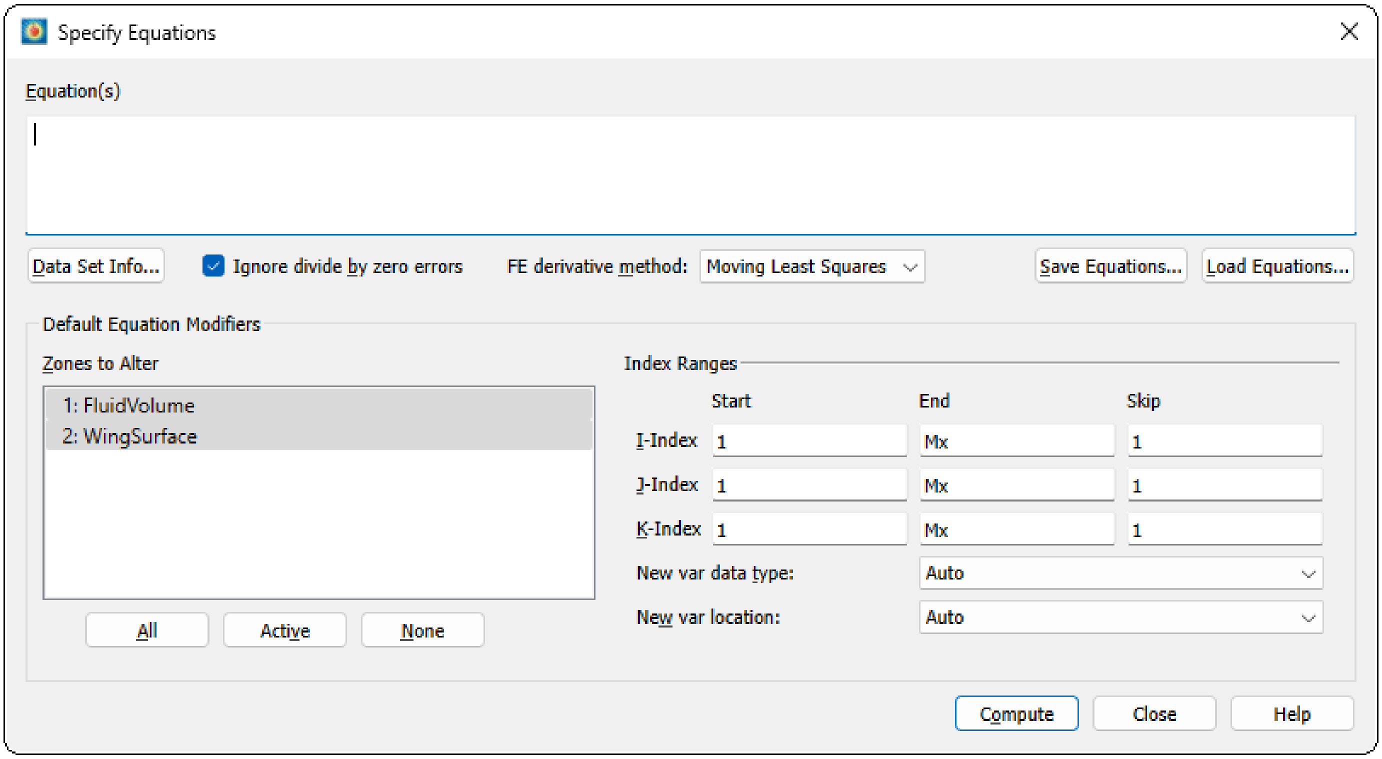
Data Alteration through Equations

Use **Data**→**Alter**→**Specify Equations** (or click the $f(x)$ toolbar button) to alter data in existing zones. The dialog allows you to change the values of entire variables or specific data points. You can also use this dialog to create new variables.

Changes made to the dataset in the **Specify Equations** dialog are not made to the original data file. You can save the changes by saving a layout file or writing the new data to a file. Saving a layout file will keep your data file in its original state, but use journaled commands to reapply the equations.

> Variables defined in your data file, or variables you create yourself, are referenced in equations by surrounding them with curly braces, like `{RPM}`. This syntax must be used even when first defining a variable, for example `{RPM} = {RPS} * 60` to define a new

RPM variable from an existing RPS variable. Variable and function names provided by Tecplot 360 do not require the braces.



The **Specify Equations** dialog has the following options and fields:

Equation(s)

Enter the equation(s) using the syntax described in Equation Syntax.

Data Set Info

Opens the Data Set Info dialog (see Data Set Information).

Ignore divide by zero errors

When activated, suppresses the error dialog and returns the largest or smallest possible value depending on the sign of the dividend when a divide by zero condition occurs. 0/0 will return 0.

FE derivative method

Select the preferred method to use for computing finite-element derivatives. The default is Moving Least Squares. If Green-Gauss is selected, an information icon will appear to the right of the selector. Hovering the mouse over this icon will display the following information about when the calculation will fall back to the Moving Least Squares method.

> The Green-Gauss method will be preferred for calculating derivatives of finite elements. The calculation will fall back to Moving Least Squares for polygonal or polyhedral elements or higher order elements. Moving Least Squares will also be used for second or higher order derivatives.

Save Equations

Save all equations in the Equation(s) field to a text file with an `.eqn` format.

Load Equations

Load an equation file previously saved.

Zones to Alter

Select which zones to alter. You can manually select the zones in the list (holding Shift when clicking to create a range, or Control to toggle individual zones on or off). You may use the buttons below the list to select all zones, all active zones, or no zones.

> When creating a new variable, if all zones are not selected, Tecplot will create a passive variable as a placeholder for any zones that are not included in the data alteration since all zones in a dataset must have the same number of variables. Passive variables effectively return zero for every point or cell value, do not consume any memory, and can be replaced at any time with other data alterations.

Index Ranges

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. 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.

For Ordered Data, the I-Index, J-Index, and K-index options correspond to the I, J, and K values in the dataset. For finite element data, the I-Index corresponds to the range of nodes and the J-Index corresponds to the cell-centered values. The K-index has no bearing on finite element data.

The Skip field indicates the increment between indices. 1 means apply the calculation to every data point, 2 means to every other data point, 3 to every third point, and so on. When creating a new variable, the new variable's value is set to zero at any index value that is skipped.

New Var Data Type

(if applicable) - Select the data type of the new variable. The following data types are available:

Auto

Tecplot 360 assigns the data type based upon the variables used in the right-hand side of the equation.

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 unsigned integer values (can contain values from zero to 255).

Bit

Either zero or one.

New Var Location

(if applicable) - Select the location of the new variable. The options are:

Auto

(default) - "Auto" is set to node unless all variables in the equation are located at the cell center.

Node

Stores variable with nodal information. (see Indexing Nodal Ordered Data).

Cell-Center

Stores variable with nodal information. (see Indexing Cell-centered Ordered Data).

Compute

Click the Compute button to alter the data. If an error occurs during the alteration (because of division by zero, overflow, underflow, etc.), an error message is displayed, all of the zones are restored to their previous state as of the beginning of processing that equation, and processing stops. The results of equations that have been successfully processed, however, are retained.

For example, if you enter three equations, and the second contains an error, the final state is the result of processing only the first equation. The second equation is rolled back due to the error, and the third is not processed at all.

> Every time you hit the [Compute] button, the equations are calculated. Be sure to remove previously computed equations before computing new ones.

Equation Syntax

You can enter multiple equations in the Equation(s) text field of the **Specify Equations** dialog. Each equation occupies one line of the text field as demarcated by the Enter key (not necessarily by where they wrap in the text field). The equations are applied in order to all specified zones and data points.

Tecplot 360 equations have the following form:

LValue = *F*(*RValue1*, *RValue2*, *RValue3*,...)

Where:

F() - A mathematical expression.

LValue - An existing or new variable.

RValueN - A value (such as a constant, variable value, or index value).

If *LValue* already exists in the dataset of the active frame, the equation is used to modify that variable. (The same variable may be used on the right side of the equation to refer to the pre-calculation value and thereby modify it based on its original value.) If the variable does not already exist, the equation is used to create a new variable as a function of existing variables.

There may be any number of spaces within the equation. However, there cannot be any spaces between the letters of intrinsic-function names nor in variables referred to by name. (See Equation Operators and Functions.)

Equation Variables and Values

A variable is specified in one of the following ways:

Its order in the data file

A variable may be referenced according to its order in the data file, where `V1` is the first variable in the data file, `V2` is the second, and so forth.

To create a new variable using this specification, you must specify the number of the next available variable (i.e. if there are 5 variables in the dataset), the new variable must be called `V6`. You will receive an error message, if you attempt to assign an invalid variable number.

You can confirm the number and order of variables in the data file in the **Data Set Information** dialog (choose **Data**→**Data Set Info**). The variables in the dataset are listed on the right-hand side of the page.

By its name

To reference a variable by its name, enclose the name with curly braces (`{` and `}`). For example, to set `V3` equal to the value of the variable named R/RFR, you can enter:

```
V3 = {R/RFR}
```

Variable names are not case sensitive. Leading and trailing spaces are also not considered. However, spaces within the variable name are significant.

If two or more variables have the same name, the first variable is used when the variable is referred to by name. 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 for all zones.

By a letter code

Variables and index values may be referenced by the following letter codes:

<table>
<tr><td>I</td><td>Depends on the data type and the new variable’s ‘New Var Location’ value:

Ordered data:

If ‘New Var Location’ is set to ‘Node’, I is the I index number of a node.
If ‘New Var Location’ is set to ‘Cell center’, I is the lowest I index associated with the cell.

FE data:

If ‘New Var Location’ is set to ‘Node’, I is the node index number.
If ‘New Var Location’ is set to ‘Cell center’, I is 1.

</td></tr>
<tr><td>J</td><td>Depends on the data type and the new variable’s ‘New Var Location’ value:

Ordered data:

If ‘New Var Location’ is set to ‘Node’, J is the J index number of a node.
If ‘New Var Location’ is set to ‘Cell center’, J is the lowest J index associated with the cell.

FE data:

If ‘New Var Location’ is set to ‘Node’, J is 1.
If ‘New Var Location’ is set to ‘Cell center’, J is the element number.

</td></tr>
<tr><td>K</td><td>Depends on the data type and the new variable’s ‘New Var Location’ value:

Ordered data:

If ‘New Var Location’ is set to ‘Node’, K is the K index number of a node.
If ‘New Var Location’ is set to ‘Cell center’, K is the lowest K index associated with the cell.

FE data:

If ‘New Var Location’ is set to ‘Node’, K is 1.
If ‘New Var Location’ is set to ‘Cell center’, K is 1.

</td></tr>
<tr><td>X</td><td>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.</td></tr>
<tr><td>Y</td><td>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.</td></tr>
</table>

Z	The variable assigned to the Z-axis (if in 3D Cartesian).
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 defined).
V	The Y-component of vectors (if defined).
W	The Z-component of vectors (if defined).
B	The value-blanking variable for the first active constraint (if applicable).
C	The contour variable for contour group 1 (if defined in the **Contour Details** dialog).
S	The scatter-sizing variable (if defined in the **Scatter Size/Font** dialog).
SOLUTIONTIME	The current solution time.
ZONENUM	The current zone number.

Letter codes may be used anywhere on the right-hand side of the equation. Do not enclose them in curly braces.

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.

The variables referenced by the letter codes are for the current frame.

Equation Operators and Functions

Binary Operators

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

Operators are evaluated from left to right within a precedence level.

Functions

The following functions are available. Arguments may be variables, constants, expressions, other functions.

SIN(X)	Sine of X (must be specified in radians)
COS(X)	Cosine of X (must be specified in radians)
TAN(X)	Tangent of X (must be specified in radians)
ABS(X)	Absolute value of X
ASIN(X)	Arcsine of X (result given in radians)
ACOS(X)	Arccosine of X (result given in radians)
ATAN(X)	Arctangent of X (result given in radians)
ATAN2(A,B)	Arctangent of A/B (result given in radians)
SQRT(X)	Returns the positive square root of X
LOG(X), ALOG(X)	Natural logarithm (base e) of X. Both forms are equivalent.
LOG10(X), ALOG10(X)	Logarithm base 10 of X. Both forms are equivalent.
EXP(X)	Exponentiation (base e); EXP(X)=e^X
MIN(A, B)	Minimum of A or B
MAX(A, B)	Maximum of A or B
SIGN(X)	Returns -1 if X is negative, +1 otherwise
ROUND(X)	Round X to the nearest integer
TRUNC(X)	Remove fractional part of X
RAND(N)	Returns random number between 1 and N inclusive, different for each element in the variable

Conditional Expressions

Conditional expressions may be written using the `IF` function.

```
IF(P,T,F)
```

If the predicate expression P is true (has a non-zero value), returns the value of expression T, otherwise returns the value of expression F. Both T and F are fully evaluated regardless of the value of P; there is no "short-circuiting."

The predicate expression may include comparison and Boolean operators, as summarized in the following table.

Operator	Description	Example
==	Equal To	{A} = IF(X==2.0, 3, 4) assigns to variable A: 3 if X equals 2.0, otherwise 4
!=	Not Equal To	{A} = IF(X != Y,Z,2) assigns to variable A: Z if X does not equal Y, otherwise 2
>	Greater Than	{A} = IF(X > 1.0, 4, 5) assigns to variable A: 4 if X > 1.0, otherwise 5
>=	Greater Than or Equal To	{A} = IF(X >= Y, 4, Z) assigns to variable A: 4 if X >= Y, otherwise Z
<	Less Than	{A} = IF(X < Y, 4, Z) assigns to variable A: 4 if X < Y otherwise Z
<=	Less Than or Equal To	{A} = IF(X <= Y, 4, Z) assigns to variable A: 4 if X <= Y, otherwise Z
&&	Logical AND	{A} = IF(X < Y && X > 3, 8,9) assigns to variable A: 8 if X < Y AND X > 3, otherwise 9
||	Logical OR	{A} = IF(X == 2 || X > 3, 5,6) assigns to variable A: 5 if X == 2 OR X > 3, otherwise 6

These operators are valid only within the first argument of an `IF` function. They may not be used elsewhere in an expression. For example, `{A} = X > 1` is invalid. Instead, use `{A} = IF(X > 1, 1, 0)` or similar.

Python-style operator chaining (such as `A < B < C`) is not supported. Although this construction is syntactically valid, it will not produce the expected result. Use `A < B && B < C` instead.

`&&` has a higher precedence than (will be evaluated before) `||`. When in doubt, use parentheses.

Derivative and Difference Functions

The derivative functions can be called in the same manner as intrinsic functions. Derivative and difference functions can be calculated with respect to the following variables:

Variable	Definition	Restricted to:
x, y, z	variable assigned to the x-axis, y-axis or z-axis, respectively	XY Line[11], or 3D

Variable	Definition	Restricted to:
a	variable assigned to the theta-axis	Polar Line
r	variable assigned to the radial-axis	Polar Line
i, j, k	index range	Ordered Zones

The complete set of first and second-derivative and difference functions are listed below:

Type	Function Calls	Applicable Variables
First Order	ddx, ddy, ddz, dda, ddr	x, y, z, a, or r
Second Order	d2dx2, d2dy2, d2dz2, d2da2, d2dr2	x, y, z, a or r
Second-Order (cross derivatives)	d2dxy, d2dyz, d2dxz, d2dar	xy, yz, xz, or ar

The derivative function `ddx` is used to calculate $\frac{\partial}{\partial x}$; `d2dx2` calculates $\frac{\partial^2}{\partial x^2}$.

Type	Function Calls	Applicable Variables
First Order	ddi, ddj, ddk	i, j, or k
Second Order	d2di2 d2dj2, d2dk2	i, j, or k
Second Order (cross derivatives)	d2dij, d2djk, d2dik	ij, jk, or ik

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:

$$ddi(V) = \frac{V_{i+1} - V_{i-1}}{2}$$

For ordered zones, spatial derivative functions are calculated using the chain rule with difference functions. For example:

$$\frac{\partial \varphi}{\partial x} = \frac{\partial \varphi}{\partial i}\frac{\partial i}{\partial x} + \frac{\partial \varphi}{\partial j}\frac{\partial j}{\partial x} + \frac{\partial \varphi}{\partial k}\frac{\partial k}{\partial x}$$

or:

$$ddx(\varphi) = ddi(\varphi) * ddx(i) + ddj(\varphi) * ddx(j) + ddk(\varphi) * ddx(k)$$

The spatial derivatives of indices, such as ddx(i), are calculated from their inverses, such as ddi(x), using standard curvilinear coordinate transformations.

For finite-element zones, spatial derivatives can be calculated using either the Moving Least-Squares technique or the Green-Gauss method. For Moving Least-Squares, the variable at a particular node or cell center is assumed to vary quadratically in space about that point, and least-squares is used with all nodes surrounding the point to fit a polynomial function, whose coefficients then become the first and second derivatives of the variable. The Green-Gauss method uses surface intergals to compute derivatives and can give a better estimate of derivates for highly stretched and/or skewed grids. Note that there are limitations on when Green-Gauss will be used (see FE derivative method).

Boundary Values

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 (default) or complex [12]

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 (i.e., the second derivative is 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 (i.e. the first derivative varies linearly across the boundary).

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 that is 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.

The use of derivative and difference functions is restricted as follows:

- Derivatives and differences for IJK-ordered zones are calculated for the full 3D 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 active frame’s axis assignments. Be careful if you have multiple frames with different variable assignments for the same dataset.
- Derivatives at the boundary of two zones may differ since Tecplot 360 operates on only one zone at a time while generating derivatives.

Integration

Use the **Analyze** menu to calculate integrals with Tecplot 360. See Performing Integrations for information.

Auxiliary Data

You may use auxiliary data containing numerical constants in equations. The syntax for using auxiliary data in equations is:

```
AUXZONE[nnz]:Name
AUXDATASET:Name
AUXFRAME:Name
AUXVAR[nnv]:Name
AUXLINEMAP[nnm]:Name
```

Where:

nnz = the zone number(s)
nnv = the variable number(s)
nnm = the line map number(s)
Name = name of the auxiliary data

For example, a dataset auxiliary data constant called `Pref` would be referenced using `AUXDATASET:Pref`. Equations using this auxiliary data might appear as:

```
{P} = {P_NonDim} * AUXDataSet:Pref
```

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.

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 zone(s) the equation(s) will be applied to.

> If you do not specify a zone, the zone modified by the left-hand side of the equation is used.

Zone specification works only on the right-hand side of the equation.

Index Specification

By following a variable reference with parentheses, you can specify indices for ordered data only. Indices can be absolute or an offset from the current index.

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.

If the indices are not specified, the current index values are used.

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.

> Subsequent alteration of the variables may result in loss of sharing.

Equation Restrictions

The zone and index restrictions specified in the **Specify Equations** 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.
<V=valuelocation>	Set the value location (NODAL or CELLCENTERED) for the variable on the left hand side. Only applicable 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>
```

Macros and Equations

Tecplot 360 allows you to put your equations in macros. A macro with just equations in it may be referred to as an equation file, and in fact equation files saved using the Save Equations button are macro files with an `.eqn` extension. An equation in a macro file is specified using the `$!ALTERDATA` macro command. Equation files may also include comment lines and must start with the comment `#!MC 1410`, 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 by 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 3D 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 1410
$!ALTERDATA
EQUATION = "{Mag} = sqrt(U*U + V*V + W*W)"
```

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, select **Load Equations** on the **Specify Equations** dialog. In the **Load Equation File** dialog, 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 will be 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, the 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.

Refer to the Scripting Guide for more information on working with the `$!ALTERDATA` command.

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 and fourth variables 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)
```

You can also accomplish the above operation with the following equation (assuming you have already defined the vector components for the active 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}
```

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
```

Referencing variables by letter code:

```
V3 = I + J
V4 = cos(X) * cos(Y) * cos(Z)
{Dist} = sqrt(U*U + V*V + W*W)
{temp} = min(B,1)
```

Specifying the Zone number for a given variable:

```
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]
```

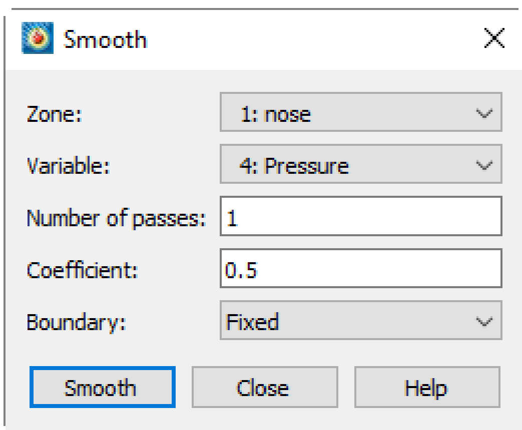
Data Smoothing

You can smooth the values of a variable of any zone (2D or 3D) or 1D line map (plotted in either XY or Polar) to reduce "noise" and lessen discontinuities in data.

Smoothing can be used after inverse-distance interpolation to reduce the artificial peaks and plateaus this type of interpolation can produce. 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.

> Zones must be mesh-based to be smoothed. If your data consists of disconnected points, see Inverse-Distance Interpoation and Irregular Data Point Triangulation for possible approaches to converting it to a form suitable for use with the Tecplot 360 smoothing feature.

To smooth data in Tecplot 360, select **Alter→Smooth** from the **Data** menu.



The **Smooth** dialog has the following options:

Zone

Specify the zone to smooth from the **Zone** drop-down. The zone should not intersect itself.

For a given plot type, the dimension of the zone must be less than or equal to the dimension of the plot type. That is, in 3D Cartesian plots, volume, surface and line zones may be smoothed, while in 2D Cartesian plots, volume zones cannot be smoothed.

Variable

Select the variable to smooth. For line plots, the variable must be a dependent variable for an active mapping for that zone.

Number of Passes

Specify the number of smoothing passes to perform. The default is 1. A greater number of passes results in greater smoothing, but takes more time.

Coefficient

Specify the relaxation factor for each pass of smoothing. 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.

Boundary

Select the boundary conditions by which to smooth from the Boundary drop-down.

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.

Smooth

Click to perform smoothing. While the smoothing is underway, the progress is displayed in the status bar. A Stop button displayed in the status bar allows you to interrupt the operation.

Smoothing Method

Tecplot 360 uses the Jacobi method with the Coefficient used as a relaxation factor (always < 1, so it's under-relaxed). For ordered zones, Tecplot 360 does this in generalized curvilinear coordinates, which has the effect of length- (or area- or volume-) weighting a node's neighboring values. For the fixed boundary condition, boundary values remain unchanged. For first-order boundaries, the boundary values are relaxed toward the neighboring interior values, and for second-order boundaries, they are relaxed toward a constant slope (zero curvature) relative to the two neighboring interior points. For finite-element zones, Tecplot 360 relaxes each value toward the inverse-distance weighted average of its neighbors. Only the fixed boundary condition is available for FE zones.

Limitations of Smoothing

- Finite element zones cannot be smoothed with anything other than Fixed boundary conditions.
- Tecplot 360 uses the active 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, Polar, and 2D or 3D Cartesian plot types. Be sure to select the correct frame if you have multiple frames with different variable assignments for the same dataset.

- Any axis scaling is ignored while smoothing.
- For I-ordered or finite element line segment zones, the active frame can be in the XY Line, 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 active frame can be a 2D or 3D Cartesian plot type, but you cannot smooth the variables assigned to the X and Y-axes in 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 3D 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.

Fourier Transform

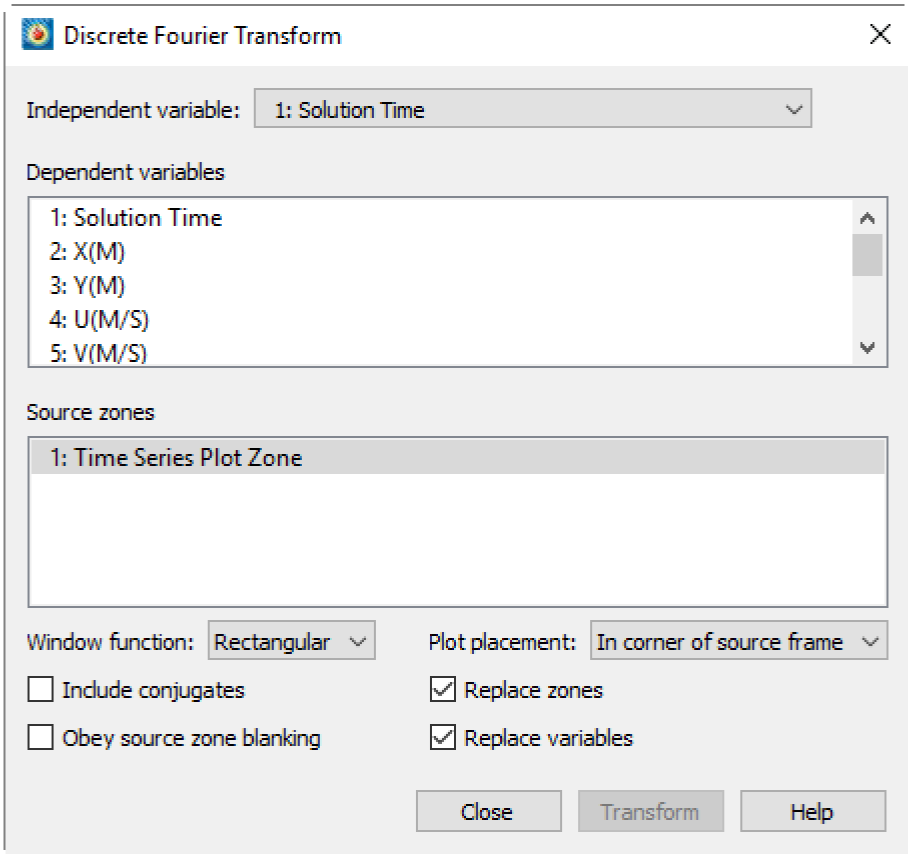
The Fourier transform feature (**Data→Fourier Transform**) allows you to transform one-dimensional ordered linear data into the frequency domain. One or more dependent variables in the data set are taken to be a representation of some function of a single independent variable, such as time, and this function is decomposed into its constituent sine waves. Variables are added to the data set to indicate these waves' frequency in Hertz, phase, and amplitude in each selected zone, and an XY of the resulting amplitudes is created in a new frame.

The dependent variables for a Fourier transform must be single-valued. The independent variable domain need not be constantly spaced; data will be resampled linearly if necessary.

The resulting frequency, amplitude, and phase variables (three for each dependent variable) are stored in Fourier transform result zones, one for each source zone. The result zones are given the name "Fourier Transform" followed by text indicating the source zone, independent variable, and window function used to calculate them. Result zones are assigned new time strands using the same groupings as the source zones if they belong to time stands; otherwise the resulting zones are static.

The result variables in the result zone are given the names Frequency, Amplitude and Phase followed by text indicating the dependent variable involved in the calculation. If the independent variable is non-uniform, Frequency is a uniform interpolation of the original data.

Newly-created zones are assigned passive variables for all variables that previously existed in the dataset, and all previously-existing zones are assigned passive variables for all new variables created by the Fourier transform.



Independent Variable

The variable used as the frequency domain. If this variable's spacing is non-uniform, this variable is used in conjunction with each dependent variable for interpolation to create a uniform frequency for the Fourier transform.

Dependent Variables

Choose the dependent variables to be transformed. Each will appear on the resulting XY plot as its own linemap.

Source Zones

Choose the zones for which the transform should be calculated.

Window Function

A function applied to the dependent variables before performing the transform (but after interpolation) to taper the data. Rectangular, triangular, Hann, and Hamming filters are provided.

Plot Placement

Where to initially place the result plot of the transform showing the distribution of frequencies in the data. The plot can be moved and resized as desired afterward.

In corner of source frame

A new frame is created for the frequency plot in the upper right corner of the frame containing the data being transformed.

Tile with existing frames

All frames, including the new one, are adjusted to the same size and arranged on the paper in a grid.

Include Conjugates

For purely real numbers, fully half of the results of the Fourier transform are conjugates (values with the same real part but opposite imaginary parts, which behave identically in situations where a real number is required). Toggling-off this option will allow the transform to be performed approximately twice as fast. as only one conjugate is calculated.

Obey Source Zone Blanking

If value blanking is active and this option is toggled-on, value blanking is applied to the values of both independent and dependent variables before interpolation is performed. If data values eliminated by blanking cause the data to be non-uniform, the values are interpolated appropriately. All blanked data values up to the first non-blanked value, and all blanked data values after the last non-blanked value, are ignored, providing a way to constrain the domain of the transform.

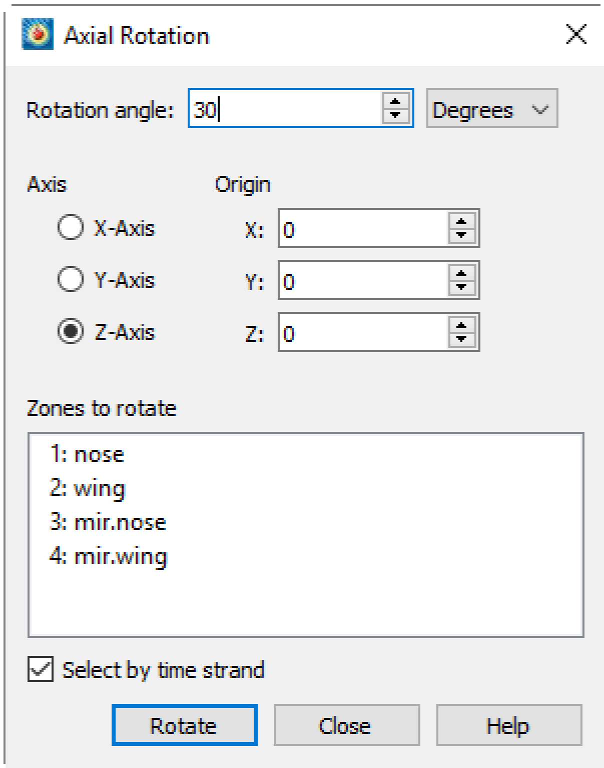
Replace Zones/Variables by Name

If either or both are toggled-on, the results of the transform are stored in existing result zones and/or variables, based on their names, if such zones and/or variables exist. Otherwise, new result zones and/or variables are created.

You may also access this feature by right-clicking a line map and choosing Fourier Transform from the context menu. In this case, the line map you click defines the independent and dependent variable used. All other settings are taken from last values entered in the dialog; the dialog does not appear.

Axial Rotation

The Axial Rotation dialog, available from the **Alter** submenu of the **Data** menu, allows you to rotate one or more 2D or 3D zones using the right-hand rule. The variables rotated are the spatial variables (X, Y, and, for 3D zones, Z) and the vector variables assigned using **Plot** → **Vector** → **Variables**. For 3D zones, you may also choose the axis of rotation.



The Axial Rotation dialog has the following options:

Rotation Angle

The angle by which the data should be rotated. May be specified in degrees or radians using the drop-down menu to the right of the entry field.

Axis

For 3D zones, choose the axis around which rotation should occur. Not available for 2D zones, which are always rotated about a notional axis perpendicular to the zone.

Origin

For rotation around a point other than (0, 0) or (0, 0, 0), enter the location of the origin using the X, Y, and (for 3D zones) Z fields.

Zones to Rotate

Click, Control-click, and/or Shift-click in the zone list to select one or more zones.

Select by **Time Strand** For transient data, if this checkbox is toggled on, the zones in the Zones to Rotate list are grouped by time strand, so that selecting a zone selects all the zones in that time strand.

Using a macro, it is possible to rotate around arbitrary axes rather than just X, Y, and Z, and to specify exactly which variables, if any, should be rotated as spatial variables and which as vector variables, including no spatial variables at all, or multiple vector variables. See `$!ROTATEDATA` in the Scripting Guide.

Zone Creation

The **Create Zone** submenu of the **Data** menu allows you to add data to your plot. The menu has the following options:

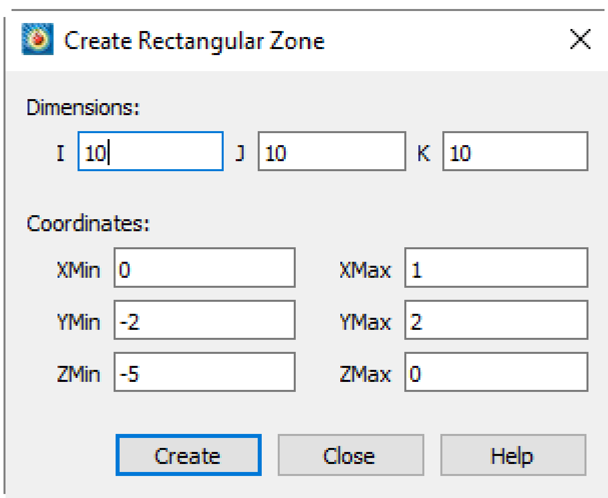
- Rectangular Zone Creation
- Circular or Cylindrical Zone Creation
- Zone Duplication
- Mirror Zone Creation

Rectangular Zone Creation

Creating a rectangular zone is often the first step in interpolating irregular data into an ordered grid

Tecplot 360 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 (only) or the **Create Rectangular Zone** dialog. The zone that you create has the same number of variables as other zones in the dataset.

To create a rectangular zone, select **Create Zone**→**Rectangular** from the **Data** menu.



The **Create Rectangular Zone** dialog has the following options:

Dimensions

Enter the number of data points in the I, J and K-directions.

- To create an I-ordered zone, enter 1 for both the J and K-dimensions.
- To create an IJ-ordered zone, enter 1 for the K-dimension. The z-axis variable will equal `ZMin` throughout the created zone.
- To create an IJK-ordered zone, enter a K-dimension greater than one.

Coordinates

Enter the start and end points of the physical coordinates (X,Y and Z).

Create

Select the [Create] button to create the zone.

Tecplot 360 uniformly distributes the data points in the I, J and K directions. Any variable not assigned to an axis is set to zero.

By using the **Specify Equations** dialog under the **Data→Alter** 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 Data Alteration through Equations.

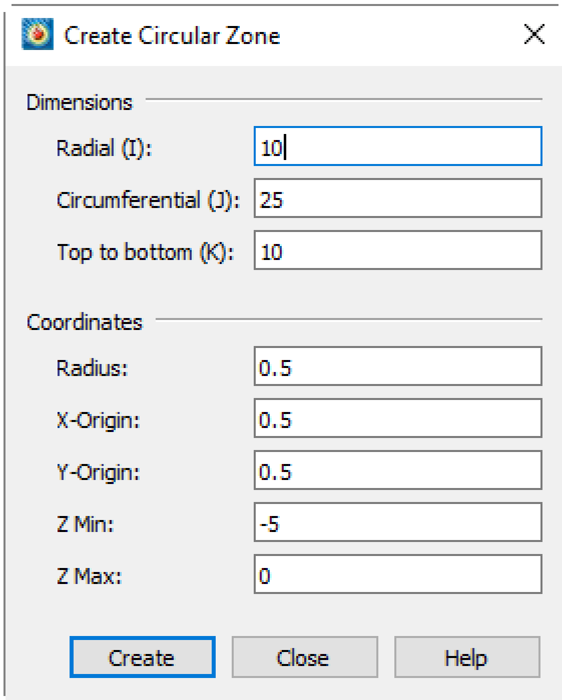
Circular or Cylindrical Zone Creation

Tecplot 360 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 (accessed via **Data→Create Zone**, or with the **Create Circular Zone** tool (only). The zone that you create has the same number of variables as other zones in the dataset.

If you have no current dataset, Tecplot 360 creates one with two or three variables, depending on the K-dimension. If you specify **K=1**, the dataset is created as IJ-ordered, and has two variables. If you specify **K** > **1**, the dataset is created as IJK-ordered, and has three variables.

To create a circular zone select **Create Zone→Circular** from the **Data** menu.



The **Create Circular Zone** dialog has the following options:

Dimensions

Enter the dimensions of your circular zone:

Radial (I)

Specify the number of points the radial direction (I)

Circumferential (J)

Specify the number of points in the circumferential direction (J)

Top to Bottom (K)

Specify the number of points for the height of the cylinder (K). Set K equal to one to create a circular zone.

Coordinates

Radius

Enter the length of the radius.

X-Origin and Y-Origin

Enter the coordinates for the zone center

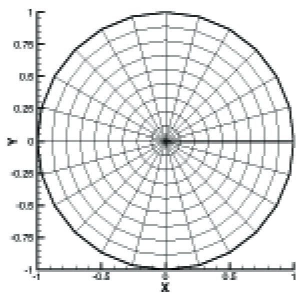
ZMin and ZMax

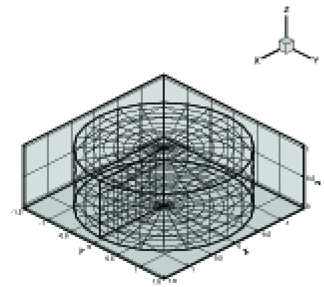
Enter the minimum and maximum Z-coordinates. For a circular zone (where K=1), the Z variable is set to ZMin for all points.

Create

Select the **Create** button to create the zone.

For IJ-ordered, Tecplot 360 creates a zone in which I-circles are connected by J-radial lines. For 3D **(K→1)**, Tecplot 360 creates a K-layered cylindrical zone having I-circles connected by J-radial planes. All other variables are set to zero These two scenarios are shown below

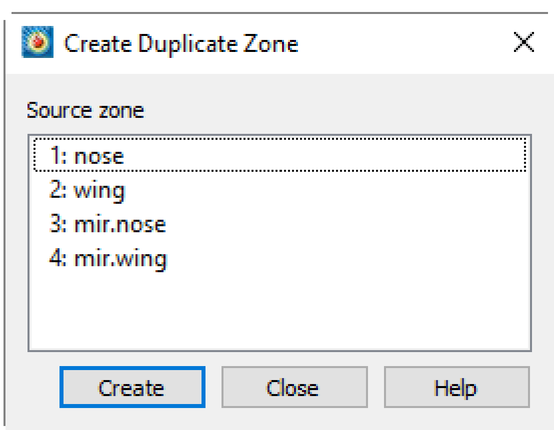




Using the **Alter** option from the **Data** menu, you can modify the X-, Y-, and Z-coordinates, and the values of the other variables as well, by using equations or equation files. See Data Alteration through Equations.

Zone Duplication

To create a full duplicate of one or more existing zones, select **Create Zone→Duplicate** from the **Data** menu. In the **Create Duplicate Zone** dialog, select the source zone(s). Each duplicate zone has the same name as its source zone.



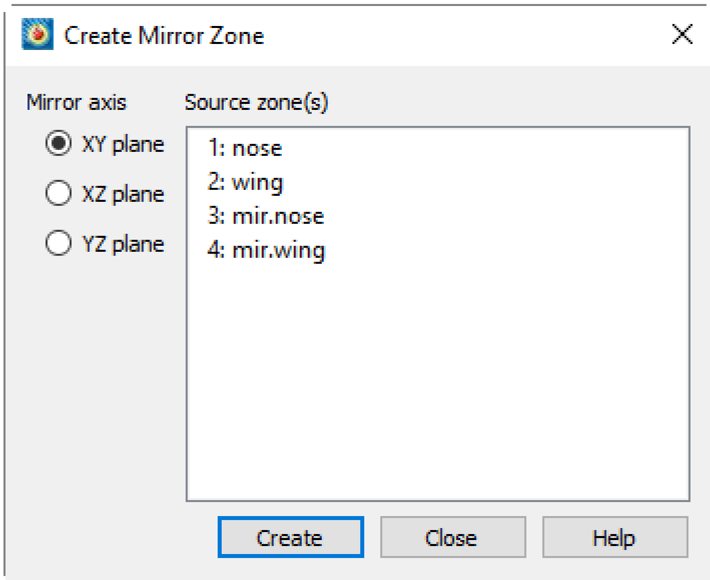
After a zone is duplicated, all variables in the newly created zone(s) will be shared with their corresponding source zone(s).

Mirror Zone Creation

To create a duplicate zone that is the mirror image of an existing zone, select **Create Zone→Mirror** from the **Data** menu.

You can only create mirrored zones along one of the standard axes (2D) or the plane determined by any two axes (3D).

The **Create Mirror Zone** dialog has the following options:



Source Zone(s)

Select the Sources Zone(s).

Mirror Axis

Specify the axis () or axis plane (3D) to mirror about.

Create

Select the [Create] button to create the zone.

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.

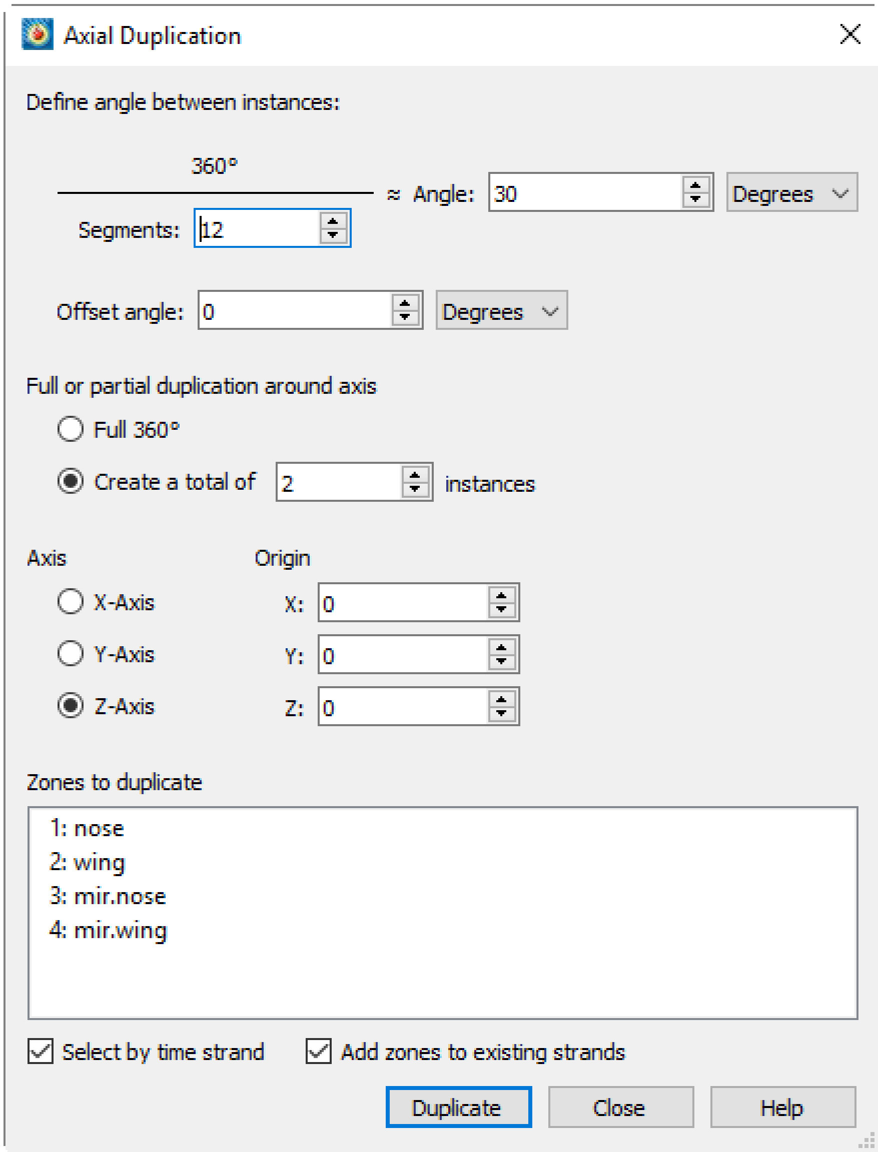
The variables in the newly created zone(s) are shared with their corresponding source zone(s), except for the coordinate and velocity normal to the symmetry plane.

Axial Duplication

The Axial Duplication dialog, available from the **Create Zone** submenu of the **Data** menu, allows you to create sucessively-rotated duplicates of one or more 2D or 3D zones using the right-hand rule. For example, if your data set contains a simulation of one blade of a turbine, and there are thirty blades in the full turbine, you can create 29 duplicates, each rotated twelve degrees from the previous, to be able to visualize the entire turbine.

The variables rotated are the spatial variables (X, Y, and, for 3D zones, Z) and the vector variables assigned using **Plot** → **Vector** → **Variables**. For 3D zones, you may also choose the axis of rotation.

Streamtraces are not duplicated; place streamtraces after zone duplication.



The Axial Duplication dialog has the following options:

Define Angle Between Instances

Contains two interdependent fields, Angle and Segments, which are used to specify how many duplicates are in a full 360° rotation of the zone.

The number of segments is determined by dividing the angle into 360,° and the angle is calculated by dividing the number of segments into 360°. Editing one automatically updates the other. You may choose to specify (or calculate) the angle in ethier degrees or radians.

Offset Angle

The additional amount by which the first duplicate should be rotated; may be specified in degrees or radians. For example, if you specify an angle between instances of 20° and an offset of 40°, the first duplicate will be made at 60°, the second at 80°, the third at 100°, and so on.

This field may be used, for instance, when you have already created some duplicates and want to create additional duplicates that do not overlap the ones you have already created. In the example just given of angle 20° and offset 40°, perhaps you have already created two duplicates (at 20'° and 40°) and wish to create additional duplicates. In this case, offset of 40° specifies that the last existing duplicate is at 40° and that new duplicates should be created in the next "slot" beyond that, at 60°.

Full or Partial Duplication

Choose to create as many new zones as are necessary for a full rotation by choosing Full 360° (this will result in the number of duplicates entered or calculated in the **Define Angle Between Instances**, less the original zone(s) being duplicated) or create a specified number of rotated duplicates.

Axis

For 3D zones, choose the axis around which rotation should occur. Not available for 2D zones, which are always rotated about a notional axis perpendicular to the zone.

Origin

For rotation around a point other than (0, 0) or (0, 0, 0), enter the location of the origin using the X, Y, and (for 3D zones) Z fields.

Zones to Duplicate

Click, Control-click, and/or Shift-click in the zone list to select one or more zones.

Select by Time Strand

For transient data, if this checkbox is toggled on, the zones in the Zones to Duplicate list are grouped by time strand, so that selecting a zone selects all the zones in that time strand

Add Zones to Existing Strands

For transient data, adds the duplicate zones to the same time strand as the original zones if toggled on. Otherwise, the duplicate zones are assigned new strand IDs if the soruce zones belonged to strands, otherwise they are made static.

> Using a macro, it is possible to rotate around arbitrary axes rather than just X, Y, and Z, and to specify exactly which variables, if any, should be rotated as spatial variables and which as vector variables, including no spatial variables at all, or multiple vector variables. See `$!AXIALDUPLICATE` in the Scripting Guide.

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, 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.

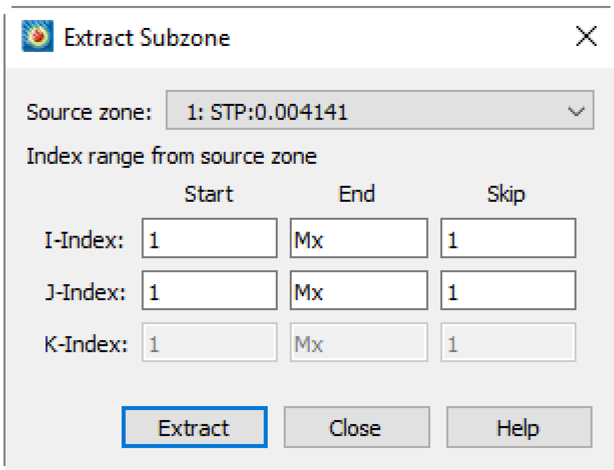
The procedures for extracting derived objects are discussed in the chapters related to those objects. For details see Contour Layer, Streamtrace Extraction as Zones, and Iso-Surface Extraction. Extracting slices, both derived and arbitrarily defined, is described in Extracting Slices to Zones.

Subzone Extraction

To create a subzone of an existing ordered zone, select **Extract→Subzone** from the **Data** menu.

> Subzone extraction is available for ordered zones only.

The **Extract Subzone** dialog has the following options:



Source Zone

Select the source zone (ordered zones only).

Index Range from Source Zone

Specify the desired subzone as a range of I, J, and K-indices. 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.

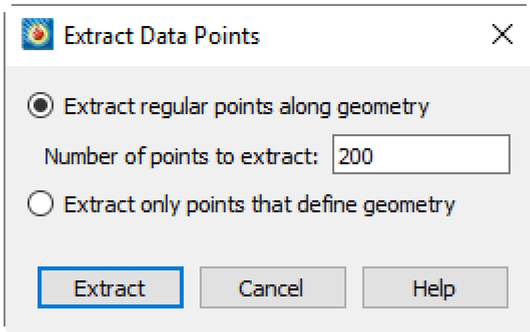
Extract

Select the **Extract** button to create the zone. Each extracted subzone is given the name "Subzone."

Data Point Extraction

You may create an I-ordered zone by extracting data points from the current dataset along a polyline that lies within the edges of a zone with connectivity. To extract points along a polyline:

1. If necessary, create the polyline geometry using the polyline tool. Click each of the points in the polyline, then double-click the last point (or press Escape) to finish the polyline. You may also extract points along an already-existing polyline.
2. Right-click the polyline geometry in the workspace and choose Extract Points from the context menu to open the **Extract Data Points** dialog.



3. Use the **Extract Data Points** dialog to control how data points are extracted. The dialog has the following options:

Extract regular points along geometry

Select this option to extract the specified number of points distributed uniformly along the geometry.

Number of points to extract

Enter the number of points to extract. This field is available only if you are extracting regular points along a geometry.

Extract only points which define geometry

Select this option to extract only the endpoints of the segments in the geometry.

Any necessary interpolation is performed using the same method used in probing. The spatial interpolation method is piecewise-linear by default. Cells are, in effect, divided by adding nodes at their center and face centers, until all cell pieces are tetrahedral. The extra nodes are calculated by arithmetically averaging the variable values at all nodes (for the cell-center node) or all face nodes (for the face-center nodes). Within each of these sub-cells, variables are interpolated linearly.

From triangular faces, linear interpolation is used to determine the value of the extracted points. On quadrilateral faces, a point of the center of mass of the quadrilateral is created and assigned a value equal to the average of the four corners. The surface is then broken down into four triangles using this center point and the four corners, and the extracted point value is interpolated linearly on the face of the triangle that surrounds it.

If a surface utilizes cell-centered values, the nodal values at the corners are interpolated, then the extracted points are interpolated from those nodal values.

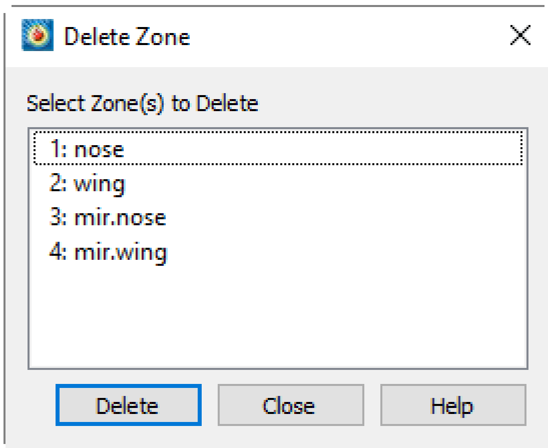
Volume interpolation, piece-wise by default, calculates to first-order accuracy. For second-order accuracy (tri-linear interpolation), add the following line to your `tecplot.cfg` file:

```
$!INTERFACE DATA {VOLUMECELLINTEROPLATIONMODE = TRILINEAR}
```

4. If you created a new polyline for the extraction operation, you may wish to delete it by again right-clicking it, then choosing **Delete** from the context menu.

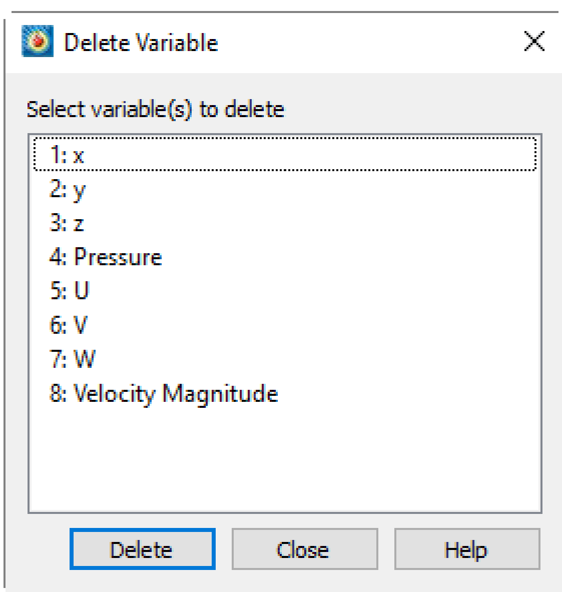
Zone Deletion

In any dataset with more than one zone, you can delete any unwanted zones. To delete a zone, select **Delete**→**Zone** from the **Data** menu. You cannot delete all zones; if you attempt to delete all zones, the lowest numbered zone is not deleted.



Variable Deletion

To delete a variable, select **Delete**→**Variable** from the **Data** menu. The **Delete Variable** dialog is shown below.



When deleting a variable, keep in mind that deleting a variable removes it from all zones. You cannot delete a variable used in a Calculate-on-demand function. See Calculate-on-demand Variables.

Data Interpolation

In Tecplot 360, interpolation refers to 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 randomly in the XY-plane. This type of data is sometimes referred to as unordered, ungridded, or random data. In Tecplot 360, it is referred to as 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 360, 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 3D, I-ordered irregular data into an IJK-ordered zone and create 3D volume plots from the IJK-ordered zone. You can even interpolate to a finite element zone.

The accuracy of the interpolation will depend on your data, the density of the destination grid, how well the grid fits the area of your unorganized zone and the settings used for interpolation.

There are three types of interpolation available:

Linear Interpolation
: Interpolate using linear interpolation from a set of finite element, IJ-ordered, or IJK-ordered zones to one zone.

Inverse-Distance Interpoation
: 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.

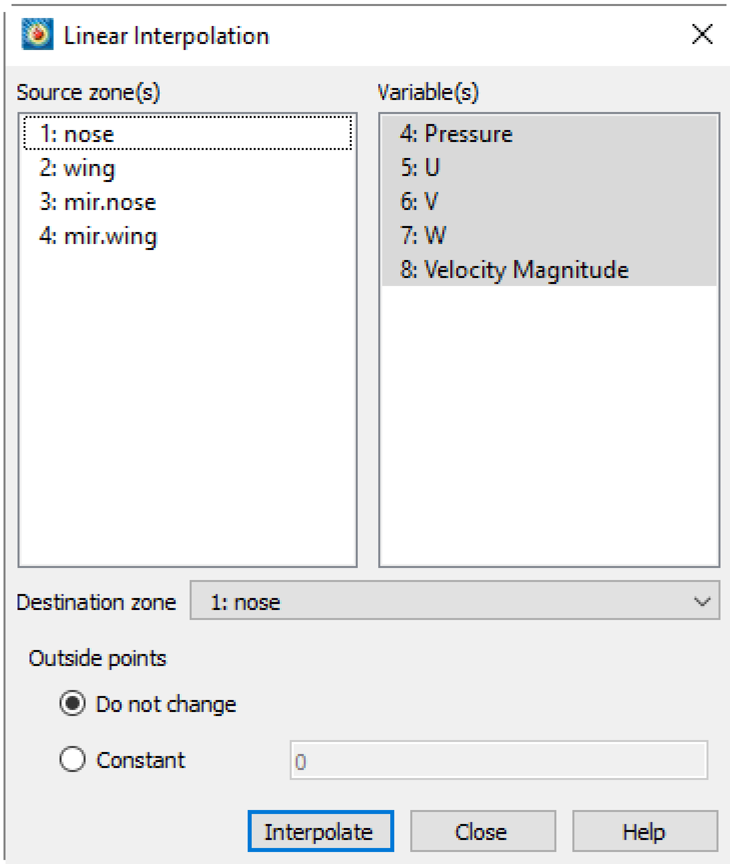
Linear Interpolation

Use the **Linear Interpolation** dialog to interpolate data from one or more ordered or finite element zones onto a destination zone. Irregular I-ordered data cannot be used for the source zones in linear interpolation; you may be able to first create a finite element zone from an irregular, I-ordered zone by using triangulation. (See 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:

1. Read the dataset to be interpolated into Tecplot 360 (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**→**Linear**.



4. From the **Linear Interpolation** dialog, 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.
6. Select the destination zone into which to interpolate. Existing values in the destination zone will be overwritten.
7. **Outside Points** - Optionally, choose how to treat points that lie outside the source-zone data field. You have two options:
    - **Do Not Change** - 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.
    - **Constant** - Sets all points outside the data field to a constant value that you specify.
8. Click the [Interpolate] button to perform the interpolation.
9. While the interpolation is proceeding, a working dialog appears showing the progress of the interpolation

> If you select [Cancel] during the interpolation process, the interpolation is terminated prematurely. The destination zone will be left in an indeterminate state, and you should redo the interpolation.

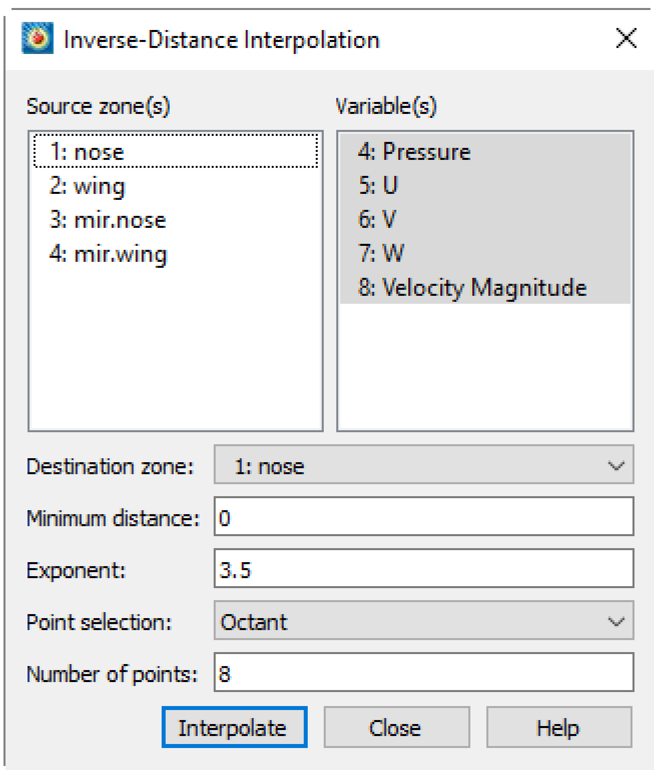
Inverse-Distance Interpoation

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 dataset—an I-ordered set of data points without any mesh structure (a list of points). Inverse-distance interpolation may be used to create a 3D surface or a 3D volume field plots of irregular data. The destination zone can, for example, be a circular or rectangular zone created within Tecplot 360 Zone Creation).

To perform inverse-distance interpolation in Tecplot 360, use the following steps:

1. Read the dataset to be interpolated into Tecplot 360 (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→Inverse Distance**.



4. From the **Inverse-Distance Interpolation** dialog, select the zones to be interpolated from those listed in the **Source Zone(s)** list.
5. Select which variables are to be interpolated from those listed in the **Variable(s)** list.
6. Select the **Destination Zone** into which to interpolate. Existing values 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. 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.

> 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.

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. 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. Click **Interpolate** to perform the interpolation.

> If you select the **Cancel** button during the interpolation process, the interpolation is terminated prematurely. The destination zone will be 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.

> Tecplot 360 uses the active frame's axis assignments to determine the variables to use for coordinates in interpolation. However, axis scaling is ignored.

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, accessed via **Data**→**Interpolate**→**Inverse-Distance**).

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:

$$\varphi_d = \frac{\sum w_s \varphi_s}{\sum w_s} \quad \text{(summed over the selected points in the source zone)}$$

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.

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 Data Smoothing for information on smoothing.

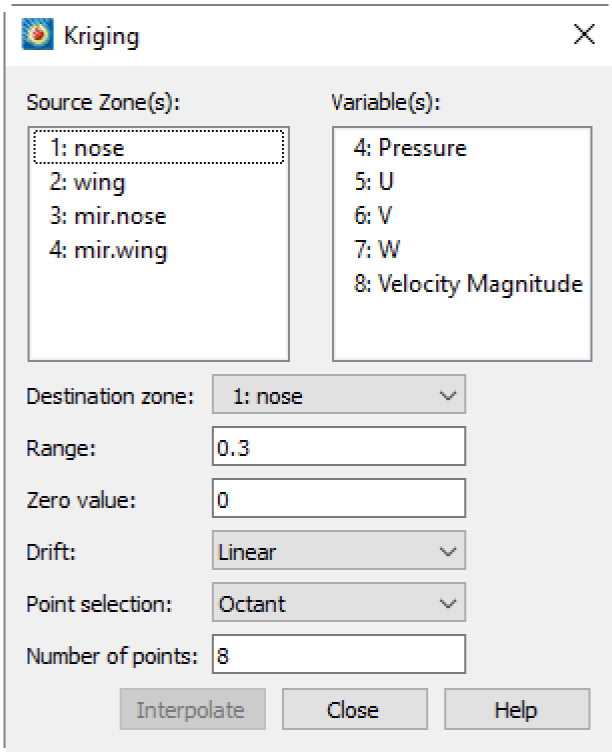
> *Kriging and Inverse Distance Interpolation Improvements*
>
> For better results with 3D data, try changing the range of your Z-variable to one similar to the X-range the Y-range. Also, set Zero Value to 0.05.

Kriging

Kriging is a more complex form of interpolation than inverse-distance. It generally produces superior results to the inverse-distance algorithm but requires more computer memory and time.

To perform kriging in Tecplot 360, perform the following steps:

1. Read the dataset to be interpolated into Tecplot 360 (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→Kriging**



4. From the **Kriging** dialog, select the zones to be interpolated from those listed in the **Source Zone(s)** scrolled list.

Tecplot 360 uses the active frame's axis assignments to determine the variables to use for coordinates in kriging. However, any axis scaling is ignored.

5. Select which variables are to be interpolated from those listed in the **Variable(s)** scrolled list.

6. Select the destination zone into which to interpolate. Existing values in the destination zone will be overwritten.

7. In the **Range** text field, optionally 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 dataset is statistically significant for each point. **In general, values between 0.2 and 0.5 should be used**.

8. In the **Zero Value** text field, optionally 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. Select the overall trend for the data in the **Drift** drop-down. This can be **Linear** (default), **None,** or **Quadratic**.

If the **Drift** is set to Linear or Quadratic, Tecplot 360 requires that the points selected be non-collinear (non-coplanar in 3D). To avoid this limitation, set **Drift** to None. Alternatively, you can eliminate coincident points by Irregular Data Point Triangulation before you interpolate.

10. 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 (default)

    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. In general, you should not use the All option unless you have very few source points

    The **Point Selection** option is very important for kriging, because 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.

11. Click **Interpolate** to begin kriging. While kriging is proceeding, a progress bar appears in the status bar, along with a Stop button that can be clicked to interrupt the operation.

    If you interrupt the kriging process, the destination zone is left in an indeterminate state, and you should redo the kriging or delete the destination zone.

The Kriging Algorithm

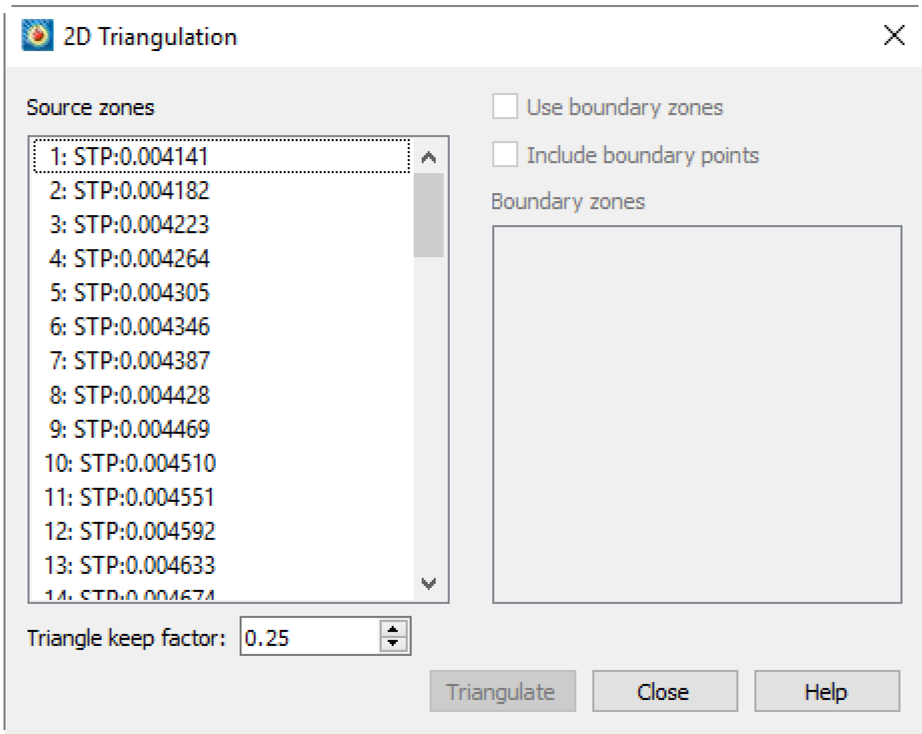
For a detailed discussion of the kriging algorithm refer to: Davis, J. C., *Statistics and Data Analysis in Geology*, Second Edition, John Wiley & Sons, New York, 1973, 1986.

Kriging and Inverse Distance Interpolation Improvements#: For better results with 3D 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.

Irregular Data Point Triangulation

Triangulation is a process that connects data points to form triangles. You can use triangulation to convert irregular I-ordered datasets into a finite element surface zone. Triangulation is one of the two options for creating field plots from irregular data. The other is interpolation, discussed in Data Interpolation. Triangulation preserves the accuracy of the data by creating a finite element surface zone with the source data points as nodes and a set of triangle elements. Triangulation works best for irregular data.

To triangulate 2D data, make sure your plot is in 2D Cartesian mode and select **2D Triangulation** from the **Data** menu. The **2D Triangulation** dialog has the following options:



Source Zone(s)

Select the zone or zones to triangulate from the list.

Use Boundary Zone(s)

Toggle-on to specify a boundary zone for the triangulation. Select the boundary zone or zones from the list. The boundary zones define the boundaries in the triangulation region. If you do not include boundary zones, Tecplot 360 assumes the data points lie within a convex polygon and that all points in the interior can be connected.

Include Boundary Points

Toggle-on to include the points in the boundary zones in the triangulated zone.

Triangle Keep Factor

This factor is used to define which triangles on the outside of the triangulated zone are "bad" and not included in the triangulation.

At the completion of triangulation, Tecplot 360 attempts to remove bad triangles from the outside of the triangulation. The keep factor is a description of the shape of the triangle 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 permitted. Bad triangles will not be removed if removing the triangle strands a data point.

Triangulate

Click the **Triangulate** button to perform the triangulation.

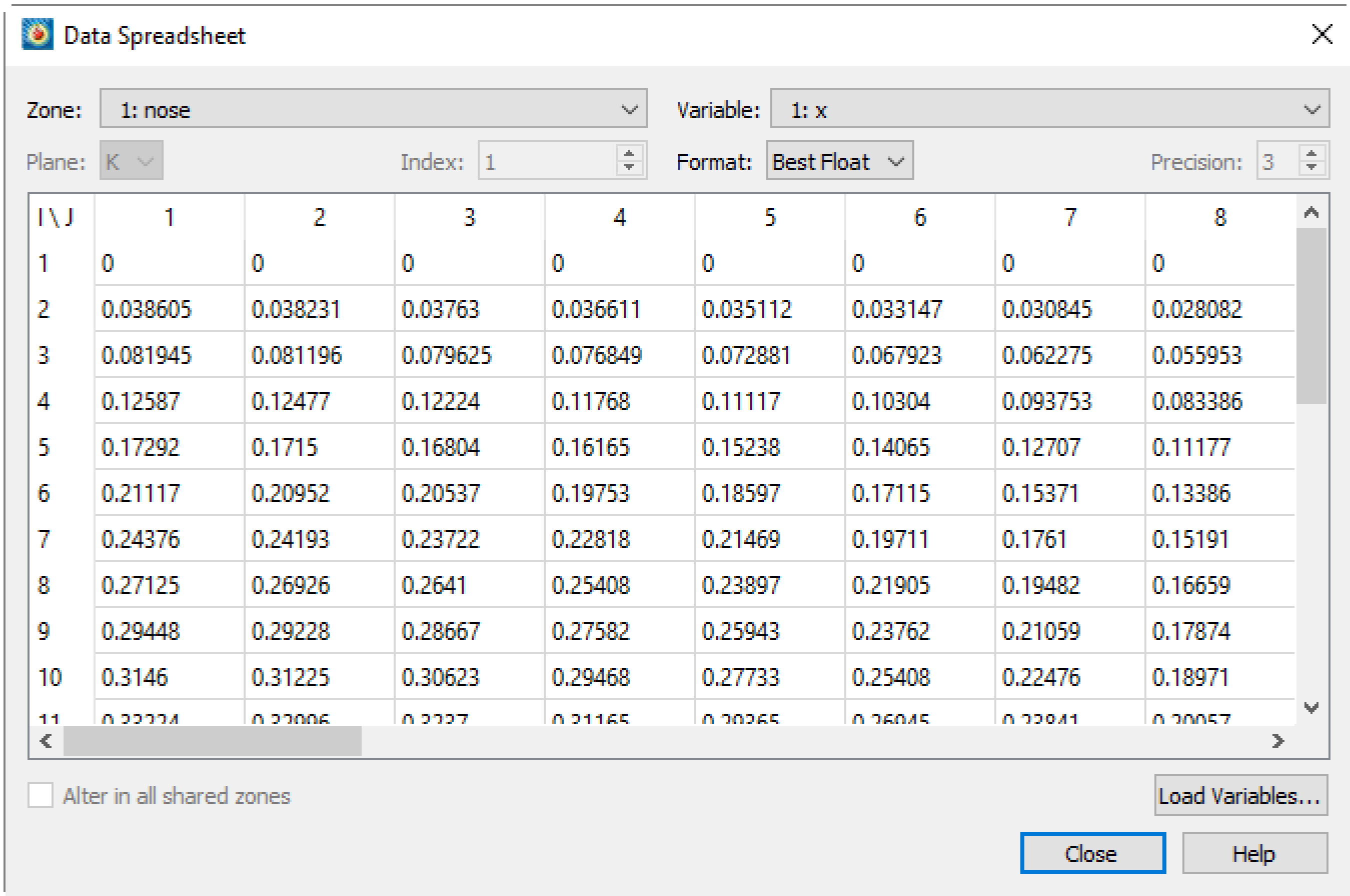
After triangulating your data, you can use the resulting finite element surface zone to create plots. Generally, you turn off the original zone(s) and plot the new zone only, but you can, for example, plot a scatter plot of the original zone(s) along with the contours of the new zone.

Data Spreadsheet

All ordered and finite element data can be viewed using Tecplot 360’s data spreadsheet (accessed via **Data**→**Spreadsheet**). The data may be modified within the spreadsheet in order to change the plots Tecplot 360 produces.

Changes to the spreadsheet do not immediately alter the original data file stored on disk. However, saving the plot of altered data as a layout file will save the changes in the data journal. You have the option of overwriting your original data file or creating a new file with the altered data.

The spreadsheet displays Tecplot 360’s data differently depending on the type of zone being examined. An example of the **Data Spreadsheet** dialog for a finite element zone is shown here; note the X, Y, and Z variables.



I \ J	1	2	3	4	5	6	7	8
1	0	0	0	0	0	0	0	0
2	0.038605	0.038231	0.03763	0.036611	0.035112	0.033147	0.030845	0.028082
3	0.081945	0.081196	0.079625	0.076849	0.072881	0.067923	0.062275	0.055953
4	0.12587	0.12477	0.12224	0.11768	0.11117	0.10304	0.093753	0.083386
5	0.17292	0.1715	0.16804	0.16165	0.15238	0.14065	0.12707	0.11177
6	0.21117	0.20952	0.20537	0.19753	0.18597	0.17115	0.15371	0.13386
7	0.24376	0.24193	0.23722	0.22818	0.21469	0.19711	0.1761	0.15191
8	0.27125	0.26926	0.2641	0.25408	0.23897	0.21905	0.19482	0.16659
9	0.29448	0.29228	0.28667	0.27582	0.25943	0.23762	0.21059	0.17874
10	0.3146	0.31225	0.30623	0.29468	0.27733	0.25408	0.22476	0.18971

I-ordered and finite element datasets are displayed with each zone's variable displayed in a column. IJ-ordered datasets are displayed in the spreadsheet with I along the rows and J along the columns. IJK-ordered datasets 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.

If a variable is not needed in your plot, it may not currently be loaded into memory. In this case, "Not Loaded" is displayed in every cell of that variable's column. Using the variable in your plot will cause it to be loaded. Click the **Load Variables** button to open the Load Variable dialog and load any variables you wish to view and edit in the spreadsheet.

To change the data spreadsheet's display format, select the desired format (Integer, Float, Exponent, or Best Float) from the Format drop-down menu. When Float or Exponent is selected, you may also specify the precision in the Precision field. These options do not affect the actual data, only how it is displayed.

To modify data:

1. In the **Data Spreadsheet** dialog, select a zone and variable to modify. Use the menu at the top of the dialog to choose the zone, then find the column representing the desired variable.
2. If the variable is shared with another zone or zones, the **Alter in all Shared Zones** toggle is enabled. Select this toggle to keep the variable shared as you modify data, propagating changes to

the other zones that share the variable. If this toggle is not selected, the variable will be changed in the selected zone and no longer shared with any other zones. See also Data Sharing.

3. Select the value of interest from the spreadsheet by finding its row. This will highlight and expand the value to its full precision.
4. To replace the highlighted value, simply enter the new value. The value is instantly replaced with the new digits entered.
5. To modify only a few digits from a highlighted value, double-click the cell to switch to edit mode. Click again or use the arrow keys to position the edit cursor at the desired position. Make any desired modifications to the existing value such as inserting or deleting digits.
6. To undo a modification of a given cell before you have committed it, press `Escape`. To commit a modification, press `Enter`, `Tab`, or `Shift-Tab` to move to the next cell, or simply select on another cell.

CFD Analysis

Tecplot 360 helps you analyze computational fluid dynamics and similar solutions. Data analysis capabilities are available via the **analyze** menu, and include:

- Function calculations, including grid quality functions (such as skewness) and flow variable functions (such as vorticity). Many of these functions duplicate functions that are available in NASA's PLOT3D and FAST plotting programs.
- Integration of input or calculated data, including scalar, vector-dot-normal and vector-dot-tangential integrands, as well as a special forces and moments option for calculating lift, drag and moments.
- Turbulence variable calculations.
- Particle path and streakline calculations, including particles with mass.
- Error analysis using Richardson extrapolation.
- Flow feature detection, including vortex cores, separation and attachment lines, and shock surfaces.

Units (Dimensions)

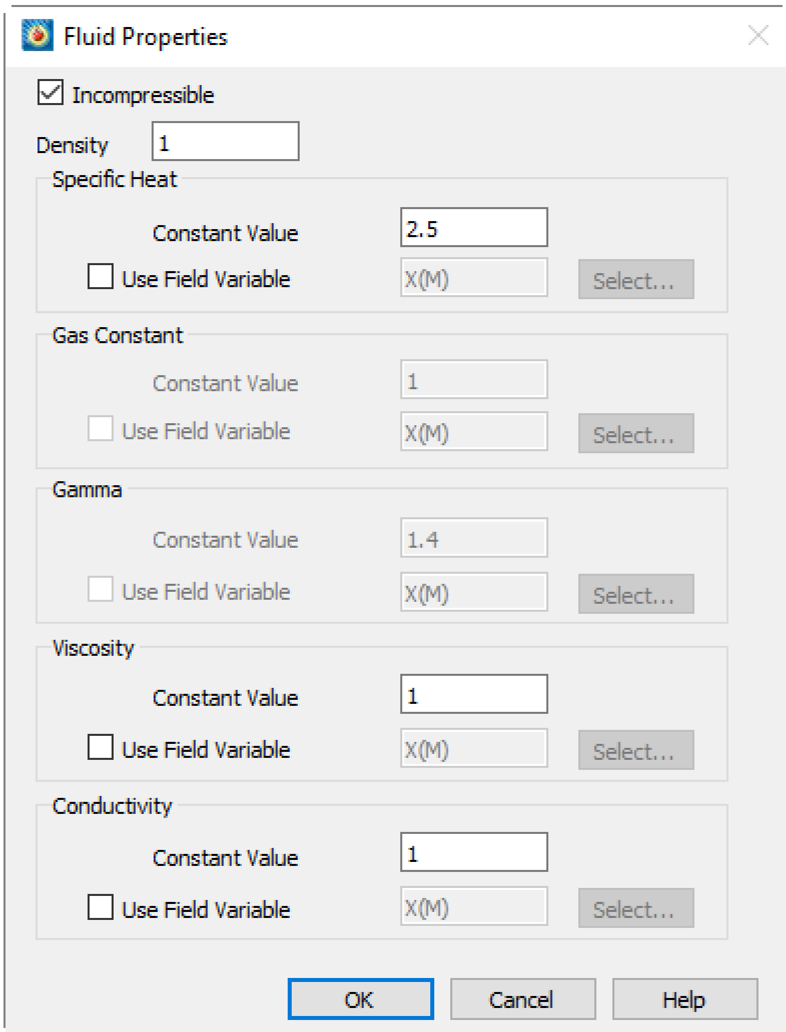
Analysis may be performed with data representing any system of units or dimensions, including non-dimensional data. All dataset variables and other parameters must, however, be in the same set of units. Unit conversions are not available. Linux and Mac users may wish to use the `units` utility for unit conversions. Analysis results will be in the same units as the data.

Specifying Fluid Properties

Fluid properties, such as viscosity, describe the fluid model used to create the dataset. These properties are required for many calculations performed by other dialogs. They are set via the **Fluid Properties** dialog. Values entered must be dimensionally consistent with each other and with your dataset. If you imported your data using the PLOT3D data loader, the default fluid properties will most likely suit your needs.

For a layout with multiple datasets, a separate set of fluid properties is maintained for each dataset. You can copy the settings from one dataset to another using the **Save Settings** and **Load Settings** options in the **Analyze** menu. These actions also transfer the settings made in the **Reference Values**, **Field Variables**, **Geometry and Boundaries**, and **Unsteady Flow Options** dialogs.

The **Fluid Properties** dialog is accessed by selecting **Fluid Properties** from the **Analyze** menu.



The **Fluid Properties** dialog allows you to specify properties for a compressible or incompressible fluid. For incompressible (uniform density) fluids, you specify density, specific heat, viscosity and conductivity. For compressible (variable density) fluids, you specify the gas constant, gamma (the ratio of specific heats), viscosity and conductivity.

By default, each fluid property is a constant. However, each property can be overridden by a field (dataset) variable (with the exception of density). When a field variable is assigned, the local value of that variable is used for field calculations using that property, and the constant value is used only for global calculations, such as the calculation of reference (free-stream) quantities. To assign a field variable for a particular property, set the **Use Field Variable** toggle and click **Select** to choose a variable from the current dataset from the **Select Variable** dialog.

Incompressible

Toggle-on to indicate the fluid is incompressible. For incompressible fluids, you must specify density, specific heat, viscosity and conductivity. For compressible fluids, you must specify gas constant, gamma, viscosity and conductivity.

Density (*for incompressible fluids only)*

Density represents the mass of fluid occupied by a unit volume. Its dimensions are $\frac{Mass}{Length^3}$

Specific Heat (*for incompressible fluids only*)

Specific heat is the amount of energy required to raise a unit mass of the fluid one degree in temperature. Dimensions are $\frac{Length^2}{Time^2 \times Temperature}$.

Gas Constant (*for compressible fluids only)*)

The specific gas constant has dimensions of $\frac{Length^2}{Time^2 \times Temperature}$.

Gamma (*for compressible fluids only*)

Gamma represents the ratio of the specific heat at constant pressure to the specific heat at constant volume, a non-dimensional quantity.

Viscosity

The dynamic viscosity's dimensions are $\frac{Mass}{Length \times Time}$.

Conductivity

The thermal conductivity's dimensions are $\frac{Mass \times Length}{Time^3 \times Temperature}$.

Specifying Incompressible Fluid Properties

When the Incompressible check box is selected, the density of the fluid and its specific heat (C_v), viscosity (μ), and conductivity (κ) must be entered. Gamma (γ), the ratio of specific heats at constant volume and pressure, is unity for incompressible fluids, so the Gamma section is inactive. Gas Constant (R) is also inactive. The thermal and caloric equations of state for incompressible fluids are shown below. ρ is density, and e represents the internal energy per unit mass.

$$\rho = const$$

$$e = C_v T$$

Since the density entered in the **Fluid Properties** dialog represents the density of the fluid throughout the physical domain, you are not allowed to enter a reference value for density in the **Reference Values** dialog, or choose a density field variable on the **Field Variables** dialog (see Identifying State Variables).

Specific heat (C_v) is the amount of energy required to raise a unit mass of the fluid one degree. It has dimensions of:

$$\frac{Length^2}{Time^2 \times Temperature} = \frac{Energy}{Mass \times Temperature}$$

Viscosity (μ) represents the dynamic viscosity coefficient, in units of

$$\frac{Mass}{Length \times Time}$$

Conductivity (κ) is the thermal conductivity of the fluid, in units of:

$$\frac{Mass \times Length}{Time^3 \times Temperature}$$

Specifying Compressible Fluid Properties

When the **Incompressible** check box is not selected, the specific gas constant, gamma, viscosity and conductivity must be entered. Since density is not a constant property of compressible fluids, the **Density** text field is inactive, as is the **Specific Heat** section of the dialog. The thermal and caloric equations of state for compressible fluids are shown below. p is pressure, and e is internal energy per unit mass:

$$p = \rho RT$$

$$e = C_v T$$

> The caloric equation of state assumes constant specific heats for the fluid. In situations where this assumption is not valid (such as high-temperature flows) Tecplot 360 will calculate inaccurate values of temperature. For these cases, it is best to have your solver output temperature, and then input it into Tecplot 360 for other calculations (see Identifying State Variables). If your solution represents a chemically reacting flow, your solver should also output R and γ as field variables, which you can identify as discussed earlier in this chapter in Specifying Incompressible Fluid Properties.

The gas constant is the universal gas constant divided by the molecular weight of the fluid:

$$R = \frac{\hat{R}}{M}$$

giving units of:

$$\frac{Length^2}{Time^2 \times Temperature}$$

Gamma is the ratio of the gas specific heats and is non-dimensional:

$$\gamma = \frac{C_p}{C_v}$$

Working with Non-dimensional Data

Consider a case where temperature is non-dimensionalized by dividing it by free-stream temperature:

$$T \Rightarrow \frac{T}{T_\infty}$$

and pressure is non-dimensionalized with gamma (the ratio of specific heats) and free-stream pressure:

$$p \Rightarrow \frac{p}{\gamma p_\infty}$$

We wish to know what to enter for the gas constant in the **Fluid Properties** dialog. We plug what we know into the thermal equation of state (where ρ is density and R is the gas constant):

$$p = \rho RT \Rightarrow \frac{p}{\gamma p_\infty} = \frac{\rho}{(1)} \times \frac{R}{(2)} \times \frac{T}{T_\infty}$$

Since the equation of state must hold for the free-stream conditions, we know:

$$p_\infty = \rho_\infty R_\infty T_\infty$$

From this, we see that the product of denominators (1) and (2) in the second-previous equation must equal $\gamma \rho_\infty R_\infty$ thus:

$$\rho R \Rightarrow \frac{\rho R}{\gamma \rho_\infty R_\infty}$$

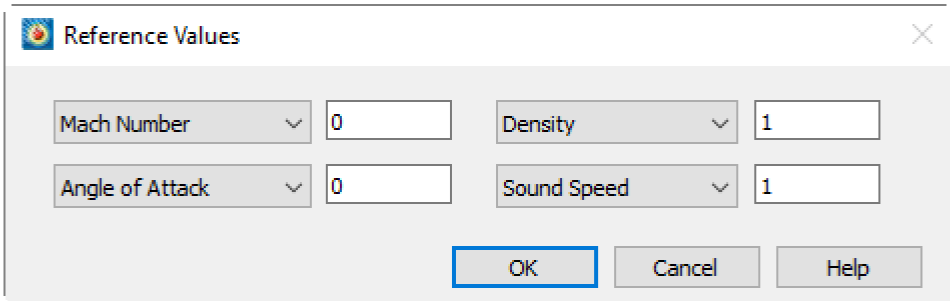
This doesn't entirely answer our question, however, and in the absence of additional information, we simply need to decide how $\rho$and R are each individually non-dimensionalized. The requirement we just determined is that the product of the two must be non-dimensionalized by $\gamma \rho_\infty R_\infty$ So we may decide to non-dimensionalize density by free-stream density, ρ_∞, which leaves the gas constant non-dimensionalized (that is, divided) by γR_∞. In the **Fluid Properties** dialog, we enter $\frac{1}{\gamma}$ for Gas Constant. If we chose to leave Gas Constant at unity, density would be non-dimensionalized by gamma and free-stream density, $\gamma \rho_\infty$.

Specifying Reference Values

Certain calculations, such as Pressure Coefficient (see Calculating Variables) require reference, or free-stream values. If you loaded your data with the PLOT3D loader, this information has probably been loaded along with the data. Otherwise, you may supply this information using the **Reference Values** dialog.

For a layout with multiple datasets, separate settings are maintained for each dataset. You can copy the settings from one dataset to another using the **Save Settings** and **Load Settings** options in the **Analyze menu**. These actions also transfer the settings made in the **Fluid Properties**, **Geometry and Boundaries**, **Field Variables**, and **Unsteady Flow Options** dialogs.

There must be data in the active frame for the **Reference Values** dialog to be displayed. The **Reference Values** dialog is shown below.



The dialog options are as follows:

U Velocity/Mach Number

the first two text fields, you may specify free-stream velocity as either U Velocity and V Velocity, or as Mach Number and Angle of Attack. (Z-velocity is assumed to be zero.) Angle of attack must be specified in degrees; flow proceeding in the +X- and +Y-direction has a positive angle of attack. For incompressible flow (see Specifying Incompressible Fluid Properties) only U and V-velocities may be specified.

Pressure/Density

The third text field allows you to specify either **Density** or **Pressure**. Select the corresponding option in the drop-down. For incompressible flow, you must specify **Pressure**, because density is specified in the **Fluid Properties** dialog.

V Velocity/Angle of Attack

If the first field is set to U Velocity, set this field to V Velocity. If it is set to Mach Number, set this field to Angle of Attack.

Temperature/Sound Speed

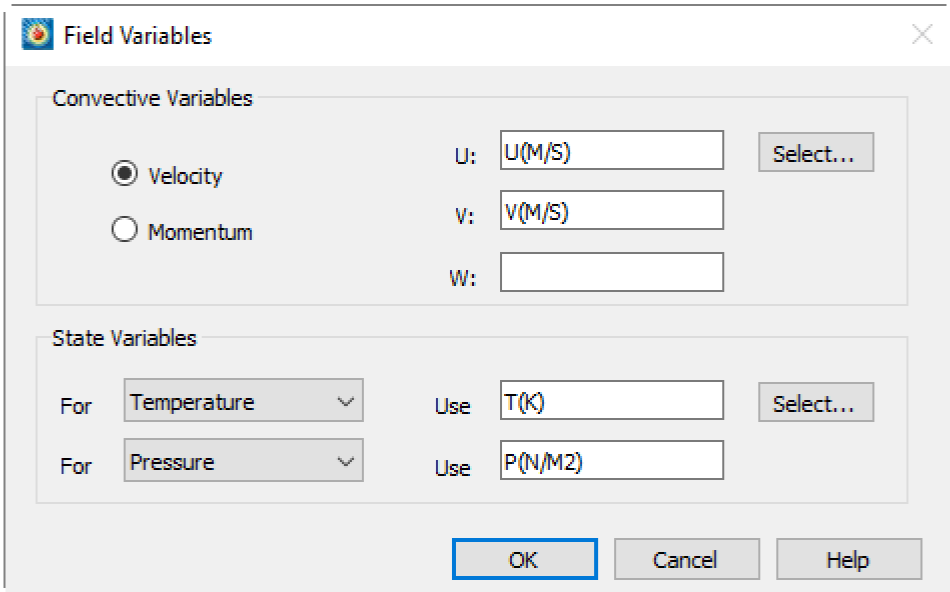
The final text field allows you to specify Temperature or Sound Speed. Temperature must be in absolute units, such as Kelvin or Rankine. For incompressible flow you must specify temperature. For incompressible fluids, the speed of sound is undefined and the density of the fluid is constant.

Identifying Field Variables

Data analysis is performed on data in the active frame. Many of these calculations require information about what the data represents. For example, if you wish to calculate pressure from your data you must identify two other thermodynamic state variables with which Tecplot 360 can perform the calculation using the thermal equation of the state. X, Y, and Z are taken from the axis assignments for the 2D or 3D plot in the active frame. The FLUENT and PLOT3D data loaders supply most or all of the remaining information to Tecplot 360. You may also supply this information using the **Field Variables** dialog.

For a layout with multiple datasets, separate settings are maintained for each dataset. You can copy the settings from one dataset to another using the **Save Settings** and **Load Settings** options in the **Analyze** menu. These actions also transfer the settings made in the **Fluid Properties**, **Geometry and Boundaries**, **Reference Values**, and **Unsteady Flow Options** dialogs.

You must have data in the active frame to open the **Field Variables** dialog. The **Field Variables** dialog is shown below



The top section of the dialog allows you to specify a vector of convective variables, either velocity or momentum (velocity multiplied by density). The bottom section of the dialog contains two drop-down menus and associated text fields for identifying two thermodynamic state variables in your dataset.

The variables selected in the Field Variables dialog are per unit volume.

Choosing the Convective Variables

Select the convective variables in your dataset by clicking the **Select** button in the top section of the **Field Variables** dialog. Choose one of the two options on the **Field Variables** dialog to indicate whether these variables represent pure velocity or momentum.

The convective variables used in data analysis are *not* the same variables that are used to create vector plots for your solution data, though their initial values may be set the same.

Identifying State Variables

The **State Variables** region of the dialog allows you to identify up to two variables, such as pressure and temperature, in your data. From the two drop-downs, select any two choices from the types: **Pressure**, **Temperature**, **Density**, **Stagnation Energy**, **Mach Number**, or **Not Used**. Then click **Select**, and choose the corresponding variable(s) from your data. If you have only one thermodynamic variable, select "Not Used" in one of the drop-downs. For incompressible flow, (see Specifying Incompressible Fluid Properties) you may specify Pressure for one variable, and you may specify Temperature or Stagnation Energy (per unit volume) for the other.

Temperature must be in absolute units, such as Kelvin or Rankin.

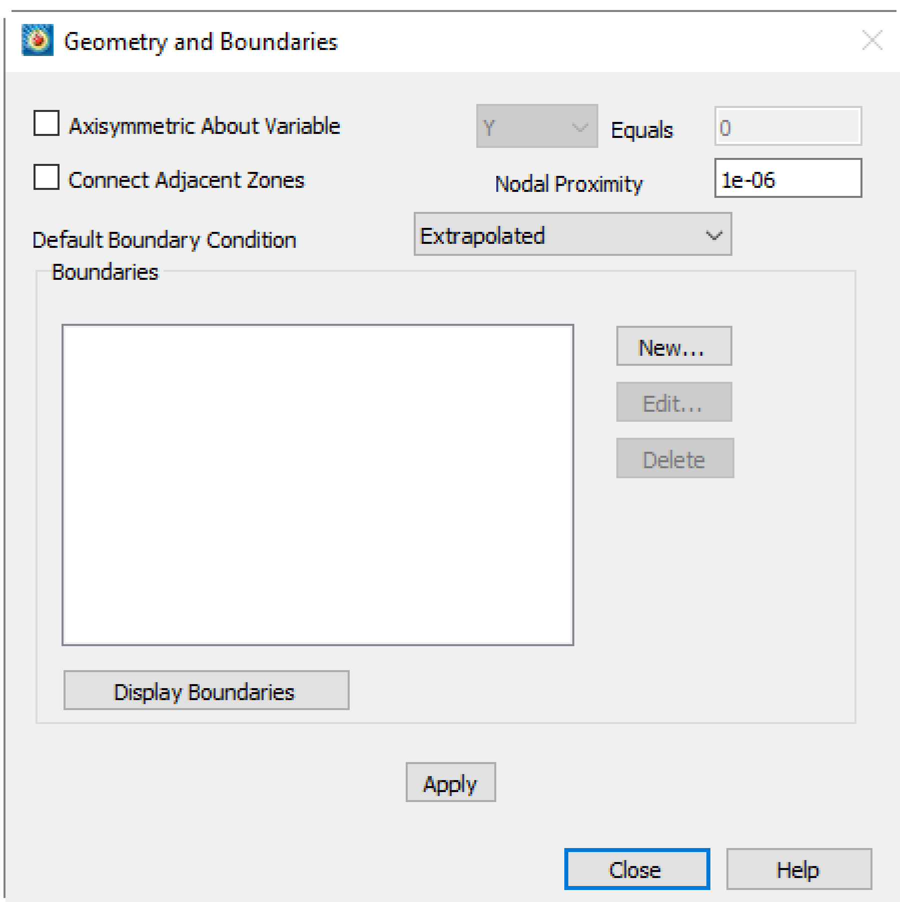
The **Select** button launches the **Select Variables** dialog which allows you to select variables in your dataset. The selections in the drop-down menus mentioned above determine whether these variables represent pressure, temperature, density, stagnation energy or Mach number.

Setting Geometry and Boundary Options

For certain calculations, you will need to specify information about your data that Tecplot 360 may not automatically detect. For example, a 2D solution may actually represent a 3D axisymmetric solution, affecting any integrations you perform. Adjacent zones may be connected, affecting other calculations such as grid stretch factors, gradients, and flow features such as vortex cores. Certain zones or zone surface regions may represent wall boundaries in your solution, on which separation and attachment lines may be calculated. The FLUENT data loader identifies most of these characteristics for you when you import FLUENT case and data files. You may also specify them with the **Geometry and Boundaries** dialog (accessed via the **Analyze** menu).

For a layout with multiple datasets, separate settings are maintained for each dataset. You can copy the settings from one dataset to another using the **Save Settings** and **Load Settings** options in the **Analyze menu**. These actions also transfer the settings made in the **Fluid Properties**, **Reference Values, Field Variables**, and **Unsteady Flow Options** dialogs.

You must have data in the active frame to launch the **Geometry and Boundaries** dialog. Select "Geometry and Boundaries" from the **Analyze** menu to display the dialog.



Specifying an Axisymmetric Solution

Selecting **Axisymmetric About Variable** enables the **Variable** drop-down menu and allows you to enter a value in the Equals field. Select X or Y from the **Variable** drop-down, and enter the constant value of this variable that defines the axis of symmetry. If you choose the axisymmetric option, all integrations will be performed as 3D axisymmetric integrations by multiplying the integrand by $2\pi r$ where r is the distance from the specified axis of symmetry. Integrations are described in Performing Integrations.

Connecting Adjacent Zones

Tecplot 360 can calculate whether nodes on the boundaries of adjacent zones (or the same zone) overlap. It uses this information in calculating the **Stretch Ratio** grid quality function (see I, J, K-stretch Ratio), calculating gradients, and extracting fluid flow features (see Extracting Fluid Flow Features). Connections between zones are calculated cell face by cell face. The two cells are considered connected wherever all nodes of a particular boundary cell face overlap all nodes of an adjacent boundary cell face.

For unsteady flows (see Unsteady Flow), only zones within the same time level are examined for connections. To enable this option, select the **Connect Adjacent Zones** option and enter the maximum distance at which two nodes will be considered to overlap in the **Nodal Proximity** text field. Note that this text field value is also used for zone-type boundaries, discussed below.

The zone connection feature is overridden, cell-by-cell, by any face neighbors contained in a dataset. Both connection mechanisms are overridden by any boundary conditions set on a particular face. That is, if you specify a boundary condition in the **Geometry and Boundaries** dialog that covers a specific cell face, that face will not be connected to an adjacent cell, irrespective of any face neighbors or overlapping nodes present.

Performance Considerations

Establishing connections across zone boundaries allows Tecplot 360 to calculate better gradient quantities at these locations. There may be a substantial performance penalty for ordered-zone calculations, because at these boundary locations, Tecplot 360 uses the finite element least-squares formulation for calculating the gradients. Refer to Gradient Calculations for a discussion of gradient calculations.

Specifying Boundaries and Boundary Conditions

You may associate cell boundary faces (cell faces on the exterior of a zone) with a boundary condition. There are two reasons why you might want to do this:

- To ensure that boundary faces are not connected to adjacent cells (see the above discussion on connections).
- To identify wall boundaries in 3D solutions for feature extraction (see Extracting Fluid Flow Features).

If you set a boundary condition on a particular cell boundary face, that face will not be considered connected to any other cells by the gradient calculation routines. This may be advantageous, for example, in solutions containing a thin flat plate, where nodes on either side of the flat plate overlap and would otherwise be connected by the connection mechanism.

For three-dimensional flow solutions, you can use the **Extract Flow Features** dialog to extract separation and attachment lines. These lines are only calculated on boundaries you have identified as wall boundaries. While other boundary conditions may be specified, this information is not currently used, aside from inhibiting connections.

Specifying the Default Boundary Condition

Tecplot 360 keeps track of all unconnected boundary cell faces (see Setting Geometry and Boundary Options) It applies the default boundary condition to any unconnected faces to which you do not specifically apply a boundary as described below. Choose the desired boundary condition from the **Default Boundary Condition** drop-down. The default boundary condition is at the bottom of the boundary 'pecking order.' If a cell boundary face is not covered by any other boundary condition, and is not connected to any other cells by either **Geometry and Boundaries** connection settings or Tecplot 360 face neighbors, then the default boundary condition is applied to it.

Identifying Zone Boundaries

Regions on the boundaries of zones may be explicitly identified and associated with particular boundary conditions. For ordered zones only, you may identify a boundary region by zone boundary (that is, the I=1 boundary) and index range on that boundary. For all zone types, you may identify a boundary region by selecting one or more boundary zones.

Boundary zones are zones of dimension one less than the current plot type. They are surfaces in 3D Cartesian plots, or lines in 2D Cartesian plots. Boundaries are considered to exist wherever the nodes of these boundary zones coincide with nodes on the boundaries of volume zones in 3D Cartesian plots, or surfaces in 2D Cartesian plots. For example, you can identify boundary regions on a tetrahedral (3D) zone using triangular zones that lie on the surface of the tetrahedral zone. The boundary is applied wherever the nodes of the triangular zone overlap boundary nodes of the tetrahedral zone. As with connecting adjacent zones, the matching is done cell face by cell face using the **Nodal Proximity** setting of the **Geometry and Boundaries** dialog to determine how close to each other nodes must be to be considered overlapping.

It is easy to create boundary zones by extracting subzones from ordered zones in your dataset. For finite element zones, it may be possible to extract the desired boundary region using blanking and FE-boundary extraction. In general, however, finite element boundary zones must come from your grid generator or flow solver.

New boundaries are created by clicking New on the **Geometry and Boundaries** dialog. This displays The Edit Boundary dialog, shown below.

Displaying Boundaries

The current settings of the **Geometry and Boundaries** dialog may be displayed by clicking the [Display

Boundaries] button. This creates a new frame and plots all zone boundaries. For each zone in your solution data, one zone will be created in the new frame for each boundary condition applied to the boundary faces of that zone. The names of these zones indicate their zone of origin in your solution data and the applied boundary condition.

For each boundary face in your solution, Tecplot 360 applies some simple rules to determine that face's boundary condition. First, all faces covered by the boundary definitions in the Boundaries list have the boundary conditions prescribed in the list applied to them. If a particular face is covered by more than one of these boundaries, the boundary lowest in the list takes precedence. If you have selected the **Connect Adjacent Zones** option, any faces not covered by the listed boundaries are then checked to see if they overlap faces of neighboring zones. Overlapping faces are assigned the boundary condition 'Interzone Boundary.' Finally, any boundary faces not assigned any other boundary condition will be assigned the default boundary condition you have chosen.

Since the **Geometry and Boundaries** dialog is modeless, you can explore the boundary definitions in this new frame prior to applying your settings. This is a convenient way to make sure you are applying the desired boundary settings.

Selecting the [Display Boundaries] button records a `DISPLAYBOUNDARIES` macro command if you are recording a macro file.

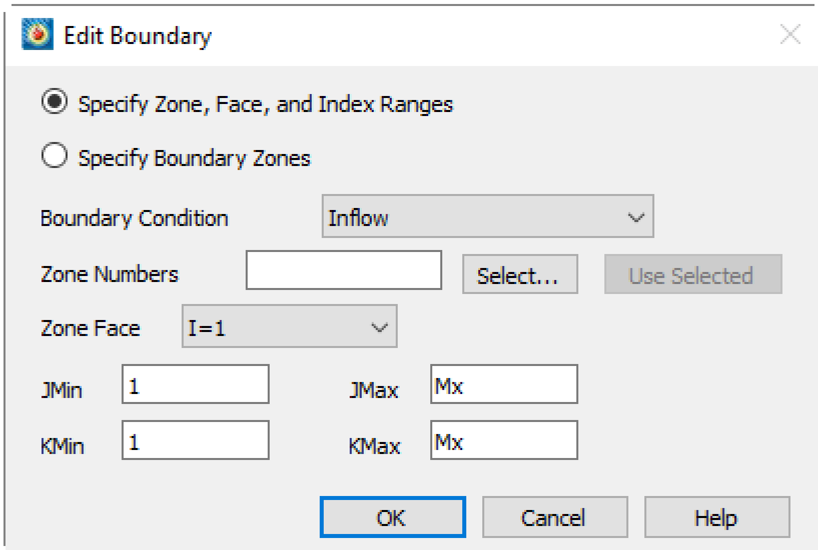
Since this feature creates a new frame, it cannot be saved in the data journal, and the current data journal is invalidated. If you subsequently save a layout file, you will be prompted to save a new data file.

Saving Geometry and Boundary Settings

Once you are satisfied with your geometry and boundary settings, you can save them by selecting the [Apply] button. When you apply your settings, a `SETGEOMETRYANDBOUNDARIES` macro is recorded (if you are recording a macro file).

The Edit Boundary dialog

The **Edit Boundary** dialog is displayed by clicking **New** on the **Geometry and Boundaries** dialog, or by selecting an existing boundary and then selecting **Edit**.



It allows you to identify a boundary of one or more zones, either by entering the zone number(s), face and index range on that face, or by entering the zone numbers of boundary zones, as discussed in Setting Geometry and Boundary Options. Enter the desired options and select **OK** to add the boundary to the **Geometry and Boundaries** dialog.

Using Index Range-type Boundaries

For ordered zones, you may identify boundary regions by choosing a zone boundary, or face, and index ranges to specify a region on the face. To create an index range-type boundary, select Zone, Face and Index Range, and choose the desired boundary condition from the Boundary Condition drop-down menu. Select the zones to which this boundary will apply by entering their zone numbers in the Zone Numbers text field, or clicking [Select] and choosing the zones from the resulting dialog. (See Performing Integrations for a description of the **Select Zones** dialog.) If you have selected zones by clicking in the work space, you may enter these zone numbers by clicking **Use Selected.** Choose a face from the Zone Face drop-down and enter the index ranges in the remaining text fields. When you select **OK**, the new boundary will appear in the Boundaries list in the following format:

```
<bc>,<set>,<face>,INDEX1MIN,INDEX1MAX,INDEX2MIN,INDEX2MAX
```

<bc> is the boundary condition, one of Inflow, Outflow, Wall, Slipwall, Symmetry, and Extrapolated. <set> is the set of zone numbers to which the boundary applies, enclosed in square brackets. <face> is one of I=1, I=IMAX, J=1, J=JMAX, K=1, and K=KMAX and the remaining parameters are the minimum and maximum indices on the face, with zero indicating the maximum index value, and negative numbers indicating offsets from the maximum index value. For example, the following line would indicate a wall boundary condition set on the J = 1 face of zones 2, 4, 5, and 6 from I = 1 to IMax and K = 3 to KMax - 2:

```
Wall,[2,4-6],J=1,1,0,3,-2
```

Using Boundary Zone-Type Boundaries

For all zone types, you may identify boundary zones, as discussed in Setting Geometry and Boundary Options. Toggle-on Specify Boundary Zones and choose the desired boundary condition from the Boundary Condition drop-down menu. Enter the zone numbers of the boundary zones, or click Select and choose them from the resulting dialog. The boundary will be applied to any volume (3D) or surface (2D) zones in the dataset. The boundary appears in the Boundaries list in the following format:

```
<bc>,<set>
```

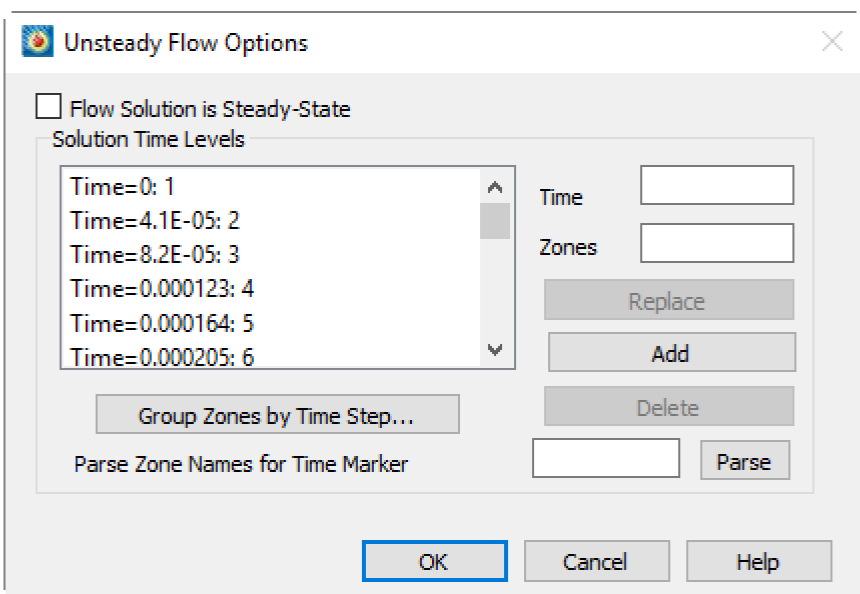
where `<bc>` is as described above, and `<set>` is the set of boundary zones that define the boundary.

Unsteady Flow

Tecplot 360 can perform particle path and streakline calculations for unsteady flow solutions. To enable this feature, it must know which zones correspond to which solution time levels in your unsteady solution. Each solution time level may comprise one or more zones, which may be ordered, finite element, or both. Many data loaders supply this information. You may also enter it in the **Unsteady Flow Options** dialog.

For a layout with multiple datasets, separate settings are maintained for each dataset. You can copy the settings from one dataset to another using the **Save Settings** and **Load Settings** options in the **Analyze** menu. These actions also transfer the settings made in the **Fluid Properties**, **Reference Values**, **Field Variables**, and **Geometry and Boundaries** dialogs.

The **Unsteady Flow Options** dialog, shown below, is displayed by selecting **Unsteady Flow Options** in the **Analyze** menu.



It contains an option allowing you to specify that your solution is steady-state, a list to display unsteady time levels you enter, as well as controls for entering new time levels.

Specifying a Steady-state Solution

To direct Tecplot 360 to treat your dataset as representing a steady-state solution, select the **Flow Solution is Steady-State** option. This setting disables the remainder of the dialog.

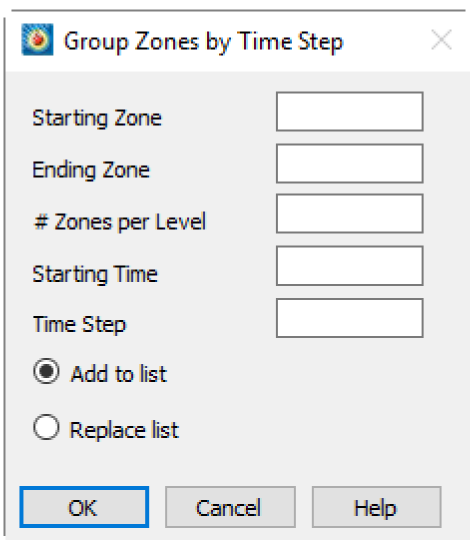
To direct Tecplot 360 to treat your dataset as an unsteady solution, toggle-off **Flow Solution is Steady-State.** This enables the remainder of the dialog, where you can identify your solution time levels.

An unsteady flow solution consists of a sequence of zones that represent successive solution times. Each time level may be represented by one or more zones. Identify solution time levels by entering the zone number(s) for a particular solution time level in the **Zones** text field and the time they represent in the **Time** text field, then selecting [Add]. The zones and associated time appear in the **Solution Time Levels** list. You may edit an existing time level by selecting it in the list. Its time and zones appear in the text fields, where you may edit them. Clicking Replace updates the currently selected list time level with the modified one.

By manually entering each time and associated zones in the text fields, you may identify all solution time levels in the current dataset. For large numbers of zones, two additional methods of entering time levels are provided. If your solution, or some portion of it was calculated with a constant time step, you may use the Group Zones by Time Step Dialog to enter all of these time levels at once. Alternatively, if your zone names contain the solution time each zone represents, you may enter all of your time levels by parsing the zone names for their corresponding solution time. These options are discussed below.

Group Zones by Time Step Dialog

The **Group Zones by Time Step** dialog allows you to enter a sequence of solution time levels into the Unsteady Flow dialog more easily than manually entering each time level.



Starting Zone

Enter the first zone of your solution data that you wish to be included in the grouping operation.

Ending Zone

Enter the final zone of your solution data that you wish to be included in the grouping operation.

Zones per Level

Enter how many zones represent each solution time level.

Starting Time

Enter the solution time which will be assigned to the first zone or group of zones identified in this operation.

Time Step

Enter the time step of your solution. The solution time of each time level will be calculated by adding this time step to the previous time level's solution time.

Add to List

Toggle-on to add all time levels identified by this operation to any time levels which already exist. If the time calculated for any of the new levels already exists in the list, this will generate an error.

Replace List

Toggle-onto replace any time levels in the list with the time levels identified in this operation.

Parsing Zone Names for Solution Time

If the names of your solution zones contain the solution time they represent, you may automatically enter all time levels by parsing the zone names for these times. Zones of the same solution time will be grouped together. The times must be preceded in the zone name by some identifiable text, such as "Time=." Enter this text (without quotes) in the text field, then select Parse.

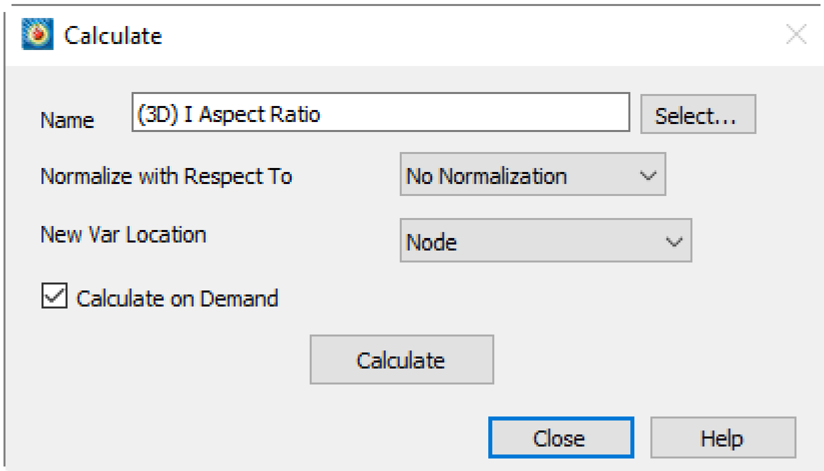
> This action will first delete all existing time levels, and then attempt to parse the zone names for new time levels. You may wish to view your zone names before attempting this action. You may view and edit zone names with the **Data Set Information** dialog (accessed via the **Data** menu).

Calculating Variables

The PLOT3D functions create dataset variables which are derived from CFD grids and solution data. This group of functions initially appeared in NASA's PLOT3D program and were expanded in PLOT3D's successor, FAST. The functions include grid quality measures, as well as scalar and vector flow variables. For a complete list of functions, refer to Calculate Variables Reference. The functions are calculated with the **Calculate** dialog.

Many of these calculations are affected by settings in the **Fluid Properties** dialog (see Specifying Fluid Properties), the **Reference Values** dialog (see Specifying Reference Values) and the **Field Variables** dialog (see Identifying Field Variables)

For the **Calculate** dialog to be displayed, the active frame must contain a dataset. The **Calculate** dialog, shown below, may then be displayed by selecting **Calculate Variables** in the **Analyze** menu.



Name

This text field indicates which function will be used for the calculation. Type in the name of the desired function, or click **Select** to choose from a list of all available functions (see the Selecting a Function dialog). Alternatively, you may enter the equivalent PLOT3D function number, as shown in the Calculate Variables Reference.

Normalizing a Function

A function may be normalized in one of two ways:

Maximum Magnitude

Divides the function value at each grid point by the maximum value in magnitude, such that the absolute value of the function is never greater than one. For vector functions, each vector component is divided by the maximum vector length.

Reference Values

Divides the function value at each grid point by the same function calculated with the reference values (the values entered in the **Reference Values** section of the dialog). This is the type of normalization performed by PLOT3D in its normalized functions. This option is not available for grid quality functions, since no meaningful reference values exist for these functions. It is also not available for functions whose reference value is zero, such as pressure coefficient.

No Normalization

Select to disable normalization.

New Var Location

You may select the location (nodal or cell-centered) of new variables created during a calculation with the **New Var Location** dropdown. Variables that already exist in the dataset keep their existing locations.

Calculate on Demand

This option adds the selected variable to the dataset, but delays the actual calculation until it is needed. This is discussed in more detail below.

Calculating the Function

Selecting **Calculate** performs the calculation for each zone in the active frame. If this is the first time the selected function has been calculated, a new variable is added to the dataset with the name of the function. Otherwise, you will be prompted to overwrite the previously calculated variable with new values. For vector functions, each component of the function is added to the dataset, with `X`, `Y`, and `Z` prefixed to the variable name, and `(vector)` removed from the name. If the function is normalized, `(Max-Normalized)` or `(RV-Normalized)` is appended to the variable name, depending on the option selected. Upon completion of the calculation, you will be informed of the new variable's minimum and maximum values and their locations.

Shared Variables

If variable sharing is enabled, all variables from which the function is calculated are shared between multiple zones, and they and the calculated variable are all at the same location (cell-centered or nodal), the new variable will be shared as well. You can see which variables in a dataset are shared in the **Data Set Info** dialog (accessed via the **Data** menu).

Calculate-on-demand Variables

Variables calculated with the Calculate-on-demand option are added to the dataset, but are not calculated until they are needed. This can save a lot of time when working with unsteady solutions where only a small number of zones are displayed at any given time. Displaying a contour plot of the calculated variable will only result in calculation of the variable for the currently active zones. Activating new zones (by, for example, advancing the solution time displayed in Tecplot 360) will result in the calculation being performed only for the newly displayed zones.

> If you wish to force the variable to be calculated for all zones at once, you may re-do the calculation with the calculate-on-demand toggle-off.

A calculate-on-demand variable is a function of other variables in the dataset and is calculated using the **Calculate** dialog. Calculate-on-demand variables are recalculated whenever a variable that they are a function-of is recalculated. For example, given **Pressure** = f(**Gas Constant**), if the value of **Gas Constant** changes, **Pressure** is recalculated.

> You cannot modify a variable that is calculated on demand.

To avoid circular data dependencies, you are prevented from selecting calculate-on-demand variables in the **Fluid Properties** or **Field Variables** dialogs. In addition, you cannot delete any variables on which a calculate-on-demand variable is dependent.

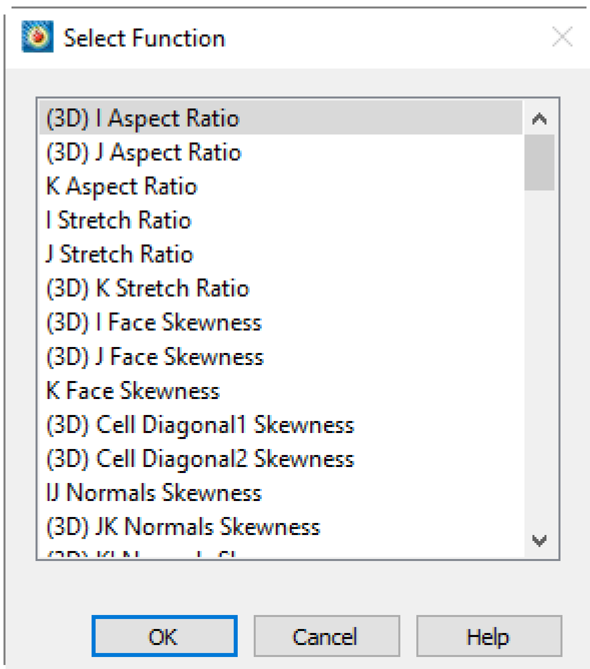
If you plan to make a sequence of changes to your data and analysis settings, you can inhibit these automatic recalculations by turning off Tecplot 360's Auto-Redraw feature. Recalculation will then take place only when you redraw the frame.

Undoing a Calculation

If the data journal is valid, alterations made to the dataset with the **Calculate** dialog may be undone by selecting **Undo** from the **Edit** menu. This will result in Tecplot 360 re-executing the data journal, which may be a lengthy process.

Selecting a Function

The function name may be typed into the **Name** text field, or selected from a list which contains all available functions. Click [Select] to display the **Select Function** dialog.



Selecting a function from this dialog and selecting **OK** enters that function in the appropriate area. Functions in this list which only apply to 3D solution data begin with `(3D)`. Vector functions, whose names are appended with `(vector)`, calculate three vector components. Each of the available functions is described in Calculate Variables Reference.

An alternative method of selecting a function is to enter its equivalent PLOT3D function number. These numbers may also be found in Calculate Variables Reference. If a valid function number is entered into the **Name** text field in the **Calculate** dialog, Tecplot 360 replaces the number with the name of the corresponding function and sets the Normalize drop-down to **None** or **Reference Values** as appropriate.

Gradient Calculations

Most of the PLOT3D functions are scalar functions. Gradient calculations are a notable exception to this rule, however, and depend on values at neighboring points. Understanding how these calculations are performed may help you interpret the results.

Gradients in Ordered Zones

With an exception for boundary nodes discussed below, gradients in ordered zones are calculated using standard finite-difference formulae. To calculate pressure gradient at a particular node in an ordered zone, for example, the following formula is used:

$$\nabla p = \begin{bmatrix} \frac{\partial p}{\partial x} \\ \frac{\partial p}{\partial y} \\ \frac{\partial p}{\partial z} \end{bmatrix} = \begin{bmatrix} \xi_x p_\xi + \eta_x p_\eta + \zeta_x p_\zeta \\ \xi_y p_\xi + \eta_y p_\eta + \zeta_y p_\zeta \\ \xi_z p_\xi + \eta_z p_\eta + \zeta_z p_\zeta \end{bmatrix}$$

Where ξ indicates the I-direction, η indicates the J-direction, ζ indicates the K-direction and subscripts indicate partial derivatives. In the zone interior, derivatives are estimated with second-order central differences, such as:

$$p_\xi \approx \frac{p_{i+1} - p_{i-1}}{2}$$

or

$$p_\xi \approx p_{i+\frac{1}{2}} - p_{i-\frac{1}{2}}$$

The left-hand form is used for calculating gradients at nodes, and the right-hand form is used at cell centers.

Boundary nodes of ordered zones that are not part of a boundary specified in the Geometry and Boundaries dialog are first examined to see whether they lie on a boundary face connected to other cells via face neighbors. If not, and if the "Connect Adjacent Zones" option is set in the Geometry and Boundaries dialog, the node is examined to determine if its location coincides with any boundary nodes of adjacent zones. If either is the case, the gradients for that node is calculated using the method described below for finite element zones. Otherwise, its gradients are calculated using standard one-sided (first-order) finite differences.

Gradients in Finite Element Zones

The coordinate transformation approach used in unconnected ordered zones is generally not possible for finite element zones. Instead, the variable, say pressure, is assumed to vary linearly in all dimensions, giving:

$$p - p_0 \equiv \Delta p = \Delta x p_x + \Delta y p_y + \Delta z p_z \equiv \overrightarrow{\Delta X} \cdot \nabla p$$

where p_0 is the pressure at the node or cell center in question. Next, a matrix equation is formed with the pressure difference for all nodes neighboring the current node (see below for how these neighboring nodes are found).

$$\begin{bmatrix} \Delta x_1 & \Delta y_1 & \Delta z_1 \\ \Delta x_2 & \Delta y_2 & \Delta z_2 \\ \Delta x_3 & \Delta y_3 & \Delta z_3 \\ \Delta x_4 & \Delta y_4 & \Delta z_4 \end{bmatrix} \begin{bmatrix} p_x \\ p_y \\ p_z \end{bmatrix} = \begin{bmatrix} \Delta p_1 \\ \Delta p_2 \\ \Delta p_3 \\ \Delta p_4 \end{bmatrix}$$

To reduce the influence of nodes far away from the value being calculated, each row i of this matrix equation is scaled by:

$$\frac{\|r\|^2}{r_i^2 + 0.1 \times \|r\|^2}$$

Where r_i is the distance from node *i* to the target location (node or cell center) and:

$$\|r\|^2 = \frac{1}{i}\sum_i r_i^2$$

This equation is generally over-specified and is inverted by least-squares to find the gradient vector.

If the cell-centered gradient is being calculated, each row in the above matrix equation is calculated from the values at the nodes that comprise the cell. If a node-centered gradient is being calculated, all nodes connected to that node by a cell edge are used. If the node lies on a zone boundary and is not covered by a boundary specified in the Geometry and Boundaries dialog, two additional steps are taken to give more continuous gradients across the zone boundary:

1. If the node is part of a face connected to other cells by face neighbors, then the nodes of those neighboring cells are also used;
2. Otherwise, if the "Connect Adjacent Zones" option is enabled in the Geometry and Boundaries dialog, the node is examined to see if its location coincides with a boundary node in an adjacent zone. If so, all nodes connected to that node by a cell edge are also used.

Surface Normal Calculations

With Tecplot 360’s CFDA variable calculation feature, you can calculate and display surface normal vectors on your plot. This includes the following steps:

1. With the **Calculate** dialog, calculate the "Grid K Unit Normal (vector)", using Cell Center as the New Var Location.
2. Turn on the Vector layer, selecting the components of the vector you just calculated as the vector components.
3. In the **Zone Style** dialog, on the Points page, choose "Cell Centers Near Surfaces" as the Points to Plot.

In detail, the steps above include the following.

To calculate the normal, choose "Calculate Variables" from the **Analyze** menu. In the **Calculate** dialog, choose "Grid K Unit Normal (vector)" as the variable to calculate (to do this, click the Select button, and scroll down in the list that appears to find "Grid K Unit Normal (vector)", and click it). Choose "Cell

Center" as the New Var Location, and click Calculate.

Next, toggle-on the Vector layer in the Plot sidebar to turn on the normal vectors, and choose the components of the calculated vector to display in the **Select Variables** dialog for vectors. This dialog appears when you toggle-on the vector layer for the first time; you can also open the dialog by going to **Plot** → **Vector** → **Variables** in the menu bar. To display the "Grid K Unit Normal (vector)" normal, choose the three components "X Grid K Unit Normal", "Y Grid K Unit Normal", and "Z Grid K Unit Normal" for the X, Y, and Z components of the vector layer, respectively.

Lastly, click the Zone Style button in the Plot sidebar to open the **Zone Style** dialog. In that dialog, switch to the Points page, and choose "Cell Centers Near Surfaces" from the **Points to Plot** menu.

> If you wish to display normal vectors on a plot that does not have an identifiable plane to use, choose "Extract" from the **Data** menu, and choose "FE-Boundary". In the Extract FE-Boundary dialog, extract a boundary zone from a source zone. You can then use the extracted zone to display the normal vectors.

Performing Integrations

Tecplot 360 provides a flexible integration feature. You can integrate scalar dataset variables as well as vector variables dotted with grid unit normal or unit tangential vectors, and you can integrate by zone in a single time step, or by time strand. Tecplot 360 also has several pre-defined integrations, such as mass flux, which simplify the integration process. In ordered zones, you can integrate these quantities over cell volumes, face areas, or lines. In finite element zones, you can integrate over cell volumes. In addition, you can calculate lift, drag, side force and moments due to pressure and viscous forces acting on a surface or a set of surfaces.

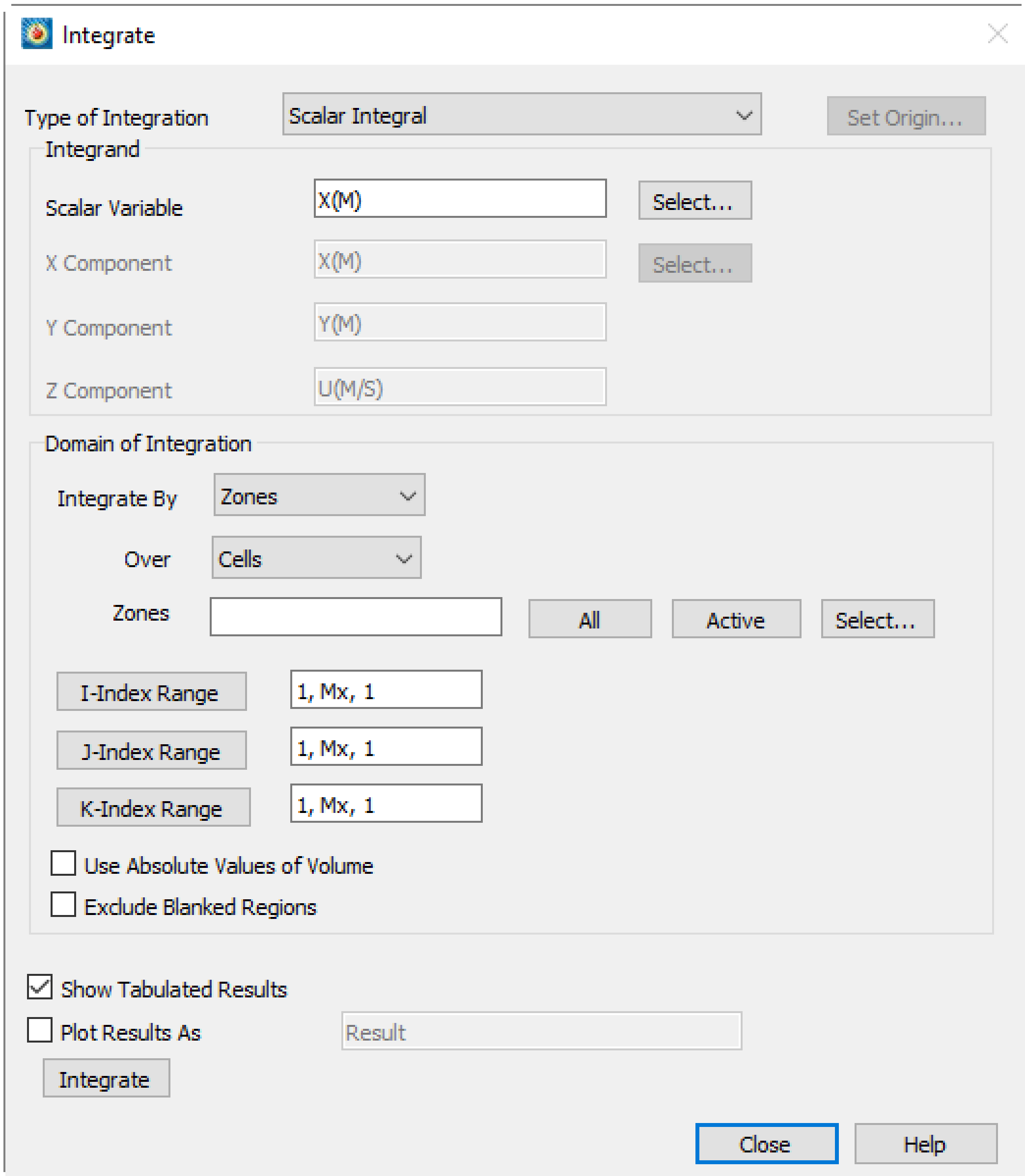
> The Integration feature refers to cell volumes in its user interface. In 2D or 1D zones, the cell area or length, respectively, is used in place of the volume.

The results of the integration may be displayed in a text window (and subsequently saved to a text file), or plotted in a frame. In the latter case, the solution time of the integration plot's frame is linked to the original frame's solution time and a marker gridline is displayed on the integration plot to indicate the time step. All of these features are accessed via the **Integrate** dialog (accessed via **Analyze**→**Perform Integration**).

> Many of these calculations are affected by settings in the **Fluid Properties** dialog (see Specifying Fluid Properties), the **Reference Values and Field Variables** dialog (see Identifying Field Variables) and the **Geometry and Boundaries** dialog (see Setting Geometry and Boundary Options).

Integrations of a variable or variable function use the trapezoidal method, and are second-order accurate. For each segment, face, or volume cell, the appropriate nodal or cell-centered values are averaged and multiplied by the cell length, area or volume. The calculation sums the resulting quantities over the zone or specified subset to produce the integrated result.

The **Integrate** dialog is displayed by selecting **Perform Integration** from the **Analyze** menu.



The resulting dialog provides options to specify the zone(s) of integration, the variable to be integrated, the domain of integration and display methods.

Type of Integration

Tecplot 360 can perform simple, path, surface, and volume integrals. Refer to Integrate Over to see how to select these using the current plot type. Tecplot 360 defines the following fourteen integration types:

Length/area/volume

The physical size of the integration domain.

Scalar

The integral of a single variable.

Average

The area or volume-weighted average of a single variable over the domain.

Mass weighted scalar

The integral of a single variable multiplied by density.

Mass weighted average

A weighted average of a single variable, with density as the weighting function.

Weighted average

A general weighted average—both the variable and the weighting function are specified.

Scalar flow rate

The convection of a scalar through a surface. It is calculated by integrating the dot product of the flow velocity and the surface unit normal multiplied by the scalar variable.

Mass flow rate

The convection of density through a surface. This is calculated by integrating the dot product of the flow velocity and the surface unit normal multiplied by the density.

Mass weighted flow rate

The convection of a scalar multiplied by density through a surface. This is calculated by integrating the dot product of the flow velocity and the surface unit normal multiplied by the scalar variable and density.

Mass flow weighted average

The weighted average of a scalar variable on a surface. Here the weighting function is the dot product of the flow momentum vector (velocity multiplied by density) and the surface unit normal.

Forces and moments

The integral of pressure and viscous stresses on a surface. The Forces and Moments option integrates pressure and shear stresses over lines (2D) and planes (3D). Pressure is assumed to act in the opposite direction of the unit normals. These are calculated by integrating the dot product of the stress tensor and the surface unit normal. This will correctly calculate lift and drag if, for example, you have a 2D airfoil defined by the `J=1` line and you integrate forces and moments over I-lines (or J-planes) for `J=1`.

For proper calculation of viscous forces, make sure you have set the value of viscosity in the Fluid Properties dialog. (See Specifying Fluid Properties.) If your flow is inviscid, you should exclude viscous forces from the integration by setting viscosity to zero.

Forces and Moments are calculated as six quantities: X, Y and Z-Force and X, Y and Z-moments about the origin. For backward compatibility, the forces are also displayed as Lift, Drag and Side force. Lift and Drag are the forces rotated in the XY-plane such that Lift is normal to the reference flow direction (specified on the **Reference Values** dialog) and Drag is parallel to it. Side force is equal to Z-Force.

If an I-ordered zone (in 2D) or a surface zone (in 3D) has been defined as a boundary to a surface (2D) or volume (3D) zone, then you can perform a Forces and Moments integration over this boundary zone. Tecplot 360 takes the shear stress and unit normal direction from the associated zone. This allows you, for example, to perform Forces and Moments integrations for finite element solutions, provided you have a line or surface zone that defines the surface, and you have identified this zone as a boundary zone in the **Geometry and Boundaries** dialog.

Vector-dot-normal

The integral over a surface of a vector dotted with the surface unit normals. Here the components of the vector are dataset variables.

Vector average

A weighted average of a scalar variable on a surface. The weighting function is the dot product of a vector with the surface unit normal. Both the scalar and the vector components are dataset variables.

Vector-dot-tangential

The integral on a line of a specified vector dotted with the line unit tangential vector.

> Options that involve a unit normal must be integrated over a domain where the unit normal direction can be determined. Acceptable domains include lines in 2D or planes in 3D, as well as triangular or quadrilateral zones in 3D. The vector-dot-tangential options can only be integrated over lines. Unit normals are discussed further in Surface Normal Calculations. If you have selected the 2D Cartesian plot type and have specified that the geometry is axisymmetric, an axisymmetric integration will be performed. Tecplot 360 multiplies each grid segment's or cell's contribution to the integration by $2\pi r$, where r is the distance from the centroid of the segment or cell to the axis of symmetry.
>
> Integrations involving surface unit normals, such as Mass Flow Rate and Forces and Moments integration, rely on surface unit normals pointing in a consistent direction (that is, toward the same side of the surface zone). This is guaranteed for ordered surface zones, but not for finite element surface zones (triangular, quadrilateral, or polygonal), including extracted slices. For these zones, the surface unit normal direction for each face is calculated using the right-hand rule with the node order for the face. If the nodes for some faces progress clockwise around the face while other faces' nodes progress counter-clockwise (as defined by the zone's connectivity), the faces' surface normals will point in inconsistent directions, and any integration that relies on these normals will not produce meaningful results. You can check for this condition using the technique for visualizing surface unit normals described in Surface Normal Calculations.

> Similarly, an integration that sums results from multiple surface zones may not be meaningful because the normals from one zone may be inconsistent with the normals of some other zone.

Integrand

Some of the available types of integrations require you to choose variables from your dataset to be integrated. Where required, fields in the **Integrand** section of the dialog will be enabled. You may type in the variable names, or click Select to choose variables.

For Forces and Moments integrations, pressure and the components of velocity are calculated from the field variables identified on the **Field Variables** dialog.

Specifying the Domain of Integration

The domain of integration is defined by zone or time strand numbers and index ranges. For ordered zones, you may choose whether to integrate over lines, planes, or volumes. You may also choose to use the absolute value of calculated volumes, which can be useful for finite element zones where the node ordering may result in erroneous calculations. Finally, you can choose to exclude regions not displayed due to index or value blanking. Please refer to Blanking for more information on blanking.

Integrate By

The **Integrate By** drop-down menu lets you specify whether to integrate over specific zones or specific time strands.

Integrate Over

The **Over** drop-down menu allows you to specify cells, planes of constant I, J, or K, or lines of varying I, J, or K. For tetrahedral and brick finite element zones, only volume integration is allowed. For quadrilateral and triangular finite element zones, only K-planes are allowed (selecting Cells for these zones is equivalent to selecting K-planes, since they are logically 2D). For 2D and 3D Cartesian plot types, integrations over lines are performed as path integrals and integrals over planes are performed as surface integrals. Integrals in XY line plots integrate the chosen variable along the X axis to calculate the area between the curve and the X axis. Volume integrations should be done in 3D Cartesian plots—volume integrations in 2D Cartesian plots will give zero results.

If a vector dot product is to be integrated, then the domain must have an identifiable normal or tangential direction. In 3D Cartesian plots, this usually means I, J, or K-planes will be selected. The normals in these cases will point in the +I, +J, and +K-directions, respectively, or the reverse for a left-handed grid. I,I, J, and K-planes do not have an identifiable tangential direction, so vector-dot-tangential integration over planes generates an error.

If I, J, or K-Lines are selected, the tangential vectors point in the positive-index direction. Vector-dot-normal integration is also available, but may not be meaningful—the normal is calculated by taking the cross-product of the tangential and the +Z-axis.

In 2D Cartesian plots, I-planes are equivalent to J-lines, J-planes is equivalent to I-lines, and K-planes is equivalent to cells. (It may be better to ignore planes in two dimensions.) Both normal and tangential directions are available in all cases. However, the normal to K-planes points in the third dimension; it may not be meaningful.

For quadrilateral and triangular finite element zones, the normal direction is found with the right-hand rule—if the fingers of the right hand are curled in the direction of a line drawn from cell node 1 to node 2, thence to node 3, then the thumb will point in the direction of the normal.

Zones/Time Strands

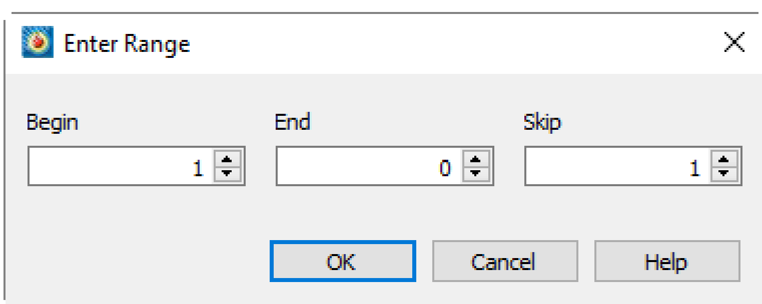
Depending on whether you have chosen to integrate by zones or by time strands, this text field allows you to specify which zones or time strands the variable will be integrated over. You may enter a single zone or strand, a range with a hyphen (for example, 3-5), or a combination of these, separated by commas (,). For convenience, the [All] button will set this text field to indicate all zones or time strands. The [Active] button will list all zones or time strands currently active. You may also select items from a list by clicking **Select**, which calls up a separate selection dialog.

Specifying Index Ranges

Below the **Zone** or **Time Step** field are I, J, and K-index ranges. These ranges will be applied to each zone over which the integration is performed. The three comma separated items in each index range indicate the starting index, the ending index and the skip factor, respectively.

For finite element zones, only the J-index settings have effect. These indicate the range of cells over which the integration will be performed. For reasons discussed below, a skip factor of 1 is probably desirable for these cases.

To enter or change an index range, select the button over the desired range's text field. The **Enter Range** dialog will be displayed.



Enter the starting index in the **Begin** field, the ending index in the **End** field, and the skip factor in the **Skip** field.

You have two options for entries into the **End** field. You can enter a number, in which case the maximum allowable value is displayed at the top of this dialog, and indicates the smallest size of the given index for all of the zones listed on the **Integrate** dialog. Alternatively, you can enter "Mx" to use the maximum index for each individual zone, "Mx - 1" to use one less than the maximum and so on. A skip factor of 1 means "use every point in the range," a skip of two means, "use every other point", and so forth.

For linear and planar integration, skip factors are ignored along the line, or within the plane of integration. For example, if you are integrating along I-lines, the I-skip factor will be ignored. If you are integrating along an IJ-plane (for example), both I- and J- skip factors are ignored. For volume cells, all skip factors are ignored. Minimum and maximum index values are always used.

Time Min/Max

When integrating by time strands, these fields appear to the right of the Index Range, allowing you to specify the starting and ending time steps for the integration. Click the Reset Min/Max button to set these fields to the first and last time steps in your data set, respectively.

Use Absolute Values of Volume

Takes the absolute value of the volumes of 3D grid cells used for integration. This is useful if you have a finite element grid with arbitrary node ordering such that the calculated volume of cells may be positive or negative. Negative grid cell volumes occur when left-handed grids are used in Tecplot 360. A right-handed ordered zone will have the +J-direction proceeding to the left of the +I-direction when viewed from the +K-direction. For finite element zones, the nodes of each cell will proceed counter-clockwise when viewed from the direction of the highest-numbered node.

Exclude Blanked Regions

Removes from the integration domain portions of any zones that are hidden due to value or index-blanking. (Note that 3D depth blanking has no effect.)

Excluding blanked regions can lead to unexpected results, depending on the blanking settings. In particular, note that blanking options allow for a cell to be blanked when any of its nodes is blanked, when its "primary" (or lowest-numbered) index is blanked, or only when all of its nodes are blanked. As a result, cells may still be displayed where some nodes have been blanked. Figure 59 illustrates this effect. Index-blanking has been used to blank all nodes along the J=1 line, but all cells are still displayed. An integration over volumes or K-planes would include the entire mesh, while integrations over I-lines or J-lines would exclude the J=1 line. In general, display the Mesh layer to see the domain of integration if you are integrating over volumes in 3D or planes in 2D, and display the Scatter layer to see the remaining types of integration domain. See Blanking for more information on blanking.

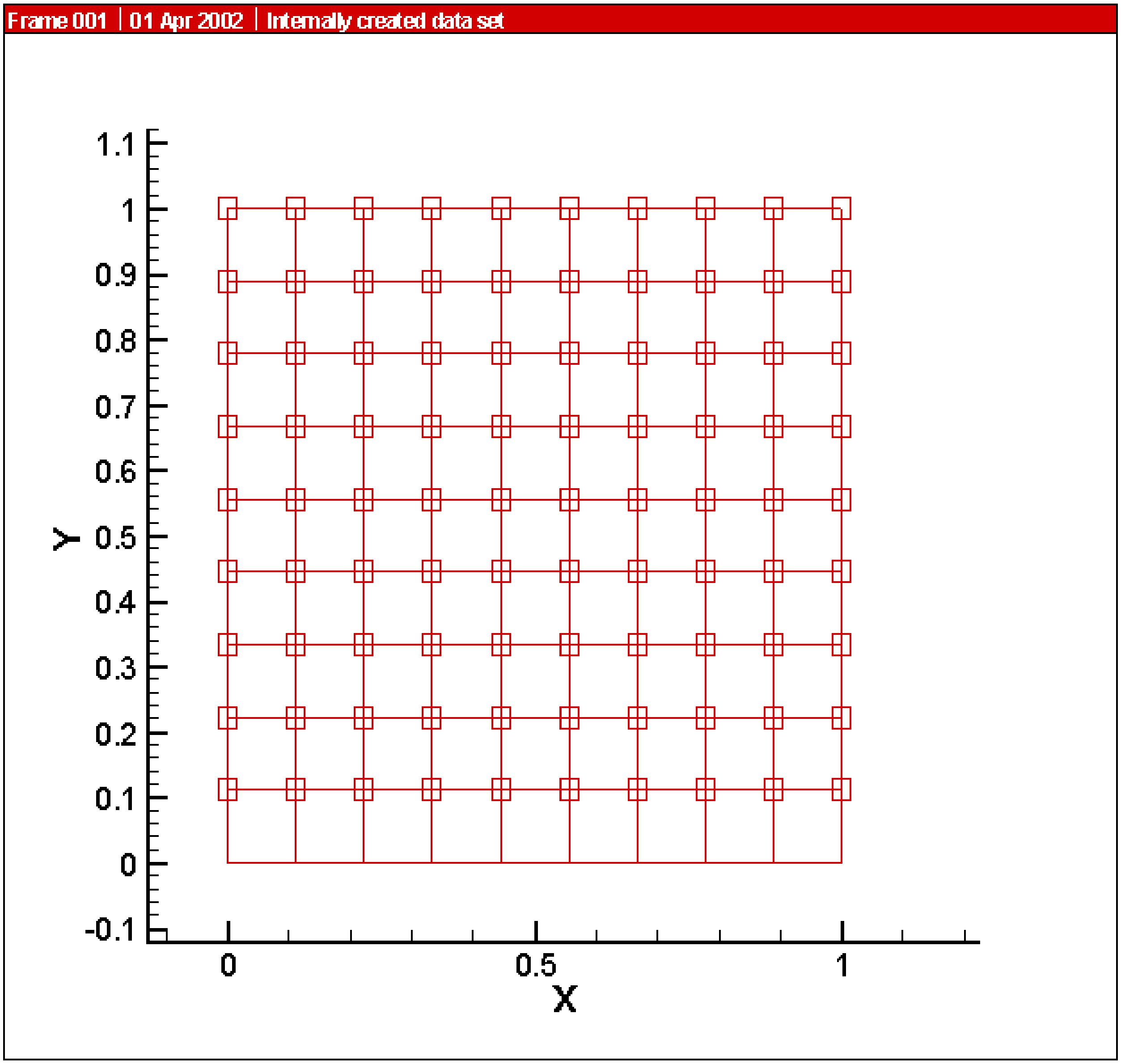


Figure 59. The effect of blanking on nodes and cells.

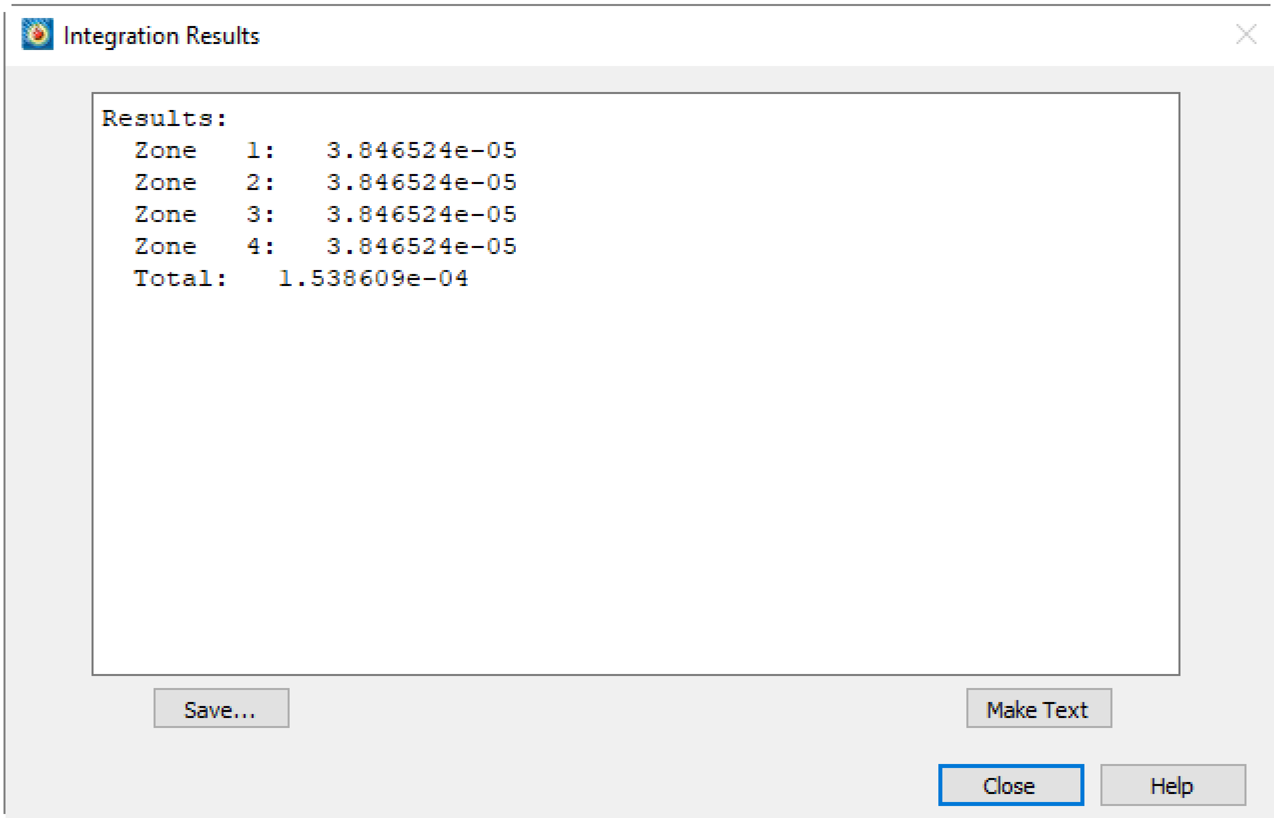
Performing the Integration

Selecting **Integrate** at the bottom of the **Integrate** dialog will perform the integration and display the results. Tecplot 360 uses the trapezoidal method, a second-order method which averages nodal values to cell, face, or edge centers, then sums the products of these values with the corresponding cell volumes, areas, or lengths.

Specifying Display Options

Displaying Tabulated Results

You can display the results of an integration in a text dialog, plotted, or both. The options at the bottom of the **Integrate** dialog (accessed via **Analyze**→**Perform Integration**) control these settings. When you have "Show Tabulated Results" toggled-on, integration results will appear in a text dialog, as shown below.



This dialog presents two additional options. Selecting the **Save** button displays a file selection dialog which allows you to save the integration results to a text file. The [Make Text] button places a text field containing the results into the active frame. Make sure you have the frame in which you wish to place the results selected as the active frame before you select this button.

Plotting Results

Setting the **Plot Results As** check box results in the integration results being plotted in a new frame. Each zone or time strand used in the integration results in a corresponding zone being created in this frame. For Cell integrations, the plot will not be useful, because it will contain only a single point in each zone. For plane (in 3D) or line integrations where multiple planes or lines are integrated in each zone or time strand, plotting can be very useful. In these cases, the results for each plane or line are plotted versus the corresponding index or indices.

When integrating by time strand, a new variable called "Solution Time" will be generated and plotted as the independent variable. Integration results for co-relevant zones are summed to a single point for each solution time. If no relevant zones exist at a given solution time, the integration is zero at that time step.

For all integrations except Forces and Moments, the text field to the right of the Plot Results As check box may be used to name the variable used to hold the integration results in the results plot. For Forces and Moments, the nine variable names will be Lift, Drag, Side, X-Moment, Y-Moment, Z-Moment, X-Force, Y-Force and Z-Force, with Lift initially being the only variable displayed.

Because the plotting feature creates a new frame, it cannot be saved to the data journal, and the current data journal is invalidated. If you subsequently save a layout file, you will be prompted to save a new data file.

Accessing Integration Results in Macros

Macro commands may access the results of the most recent integration through specific environment variables. Each of these variables represents the total over all zones (the final number shown in the **Integration Results** dialog). For all integration types except Forces and Moments, the single result is stored in the variable `INTEGRATION_TOTAL`. Table 7 shows the variable names for forces and moments.

Table 7. Environment variables for integration results.

Integration Types	Environment Variables
Forces and Moments	INTEGRATION_LIFT INTEGRATION_DRAG INTEGRATION_SIDE INTEGRATION_XMOMENT INTEGRATION_YMOMENT INTEGRATION_ZMOMENT INTEGRATION_XFORCE INTEGRATION_YFORCE INTEGRATION_ZFORCE
All other types	INTEGRATION_TOTAL

Environment variables are accessed in macros in the same way as regular macro variables, except that a `$` is prefixed to the variable name. For example, the following macro command would display the result of the most recent scalar integration:

```
$!PAUSE "Integration total = |$INTEGRATION_TOTAL|"
```

You can also access integration results as frame auxiliary data. For example, to access the `INTEGRATION_TOTAL` variable as aux data, use the following syntax:

```
INTEGRATION_TOTAL = &(AUXFRAME:CFDA.INTEGRATION_TOTAL)
```

Integration Examples

The following sections demonstrate potential uses of the **Integrate** dialog.

Calculating the Volume Under a Surface

Figure 60 shows a 3D surface. We want to calculate the volume between that surface and the Z=0 plane. To do this, integrate Z over the projection of the surface onto the Z=0 plane. To get this projection, switch to 2D Cartesian plot type. Ensure that the same variables used for X and Y in 3D are used for X and Y in 2D using the **Assign XYZ** dialog (available in the **Plot** menu).

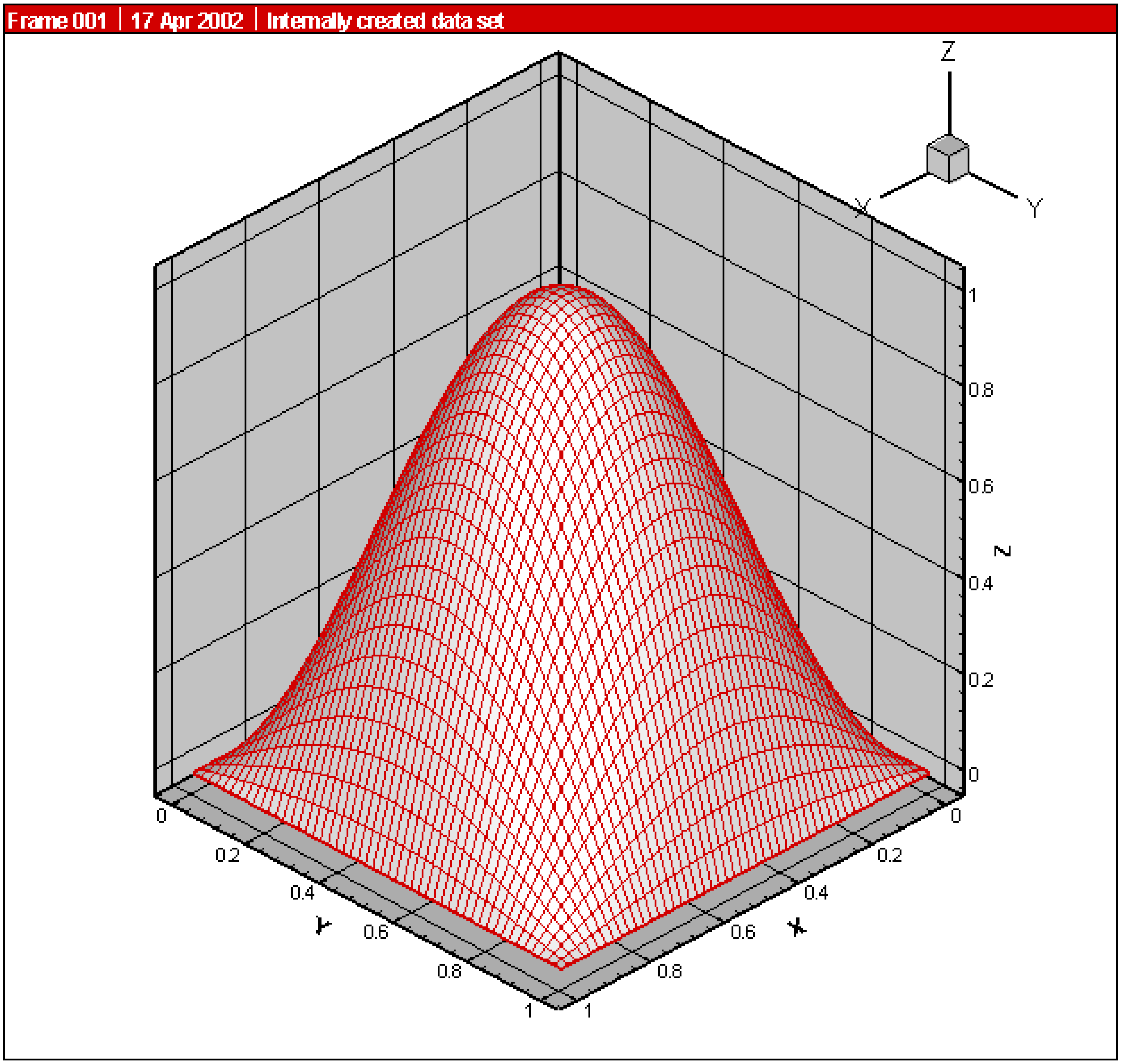
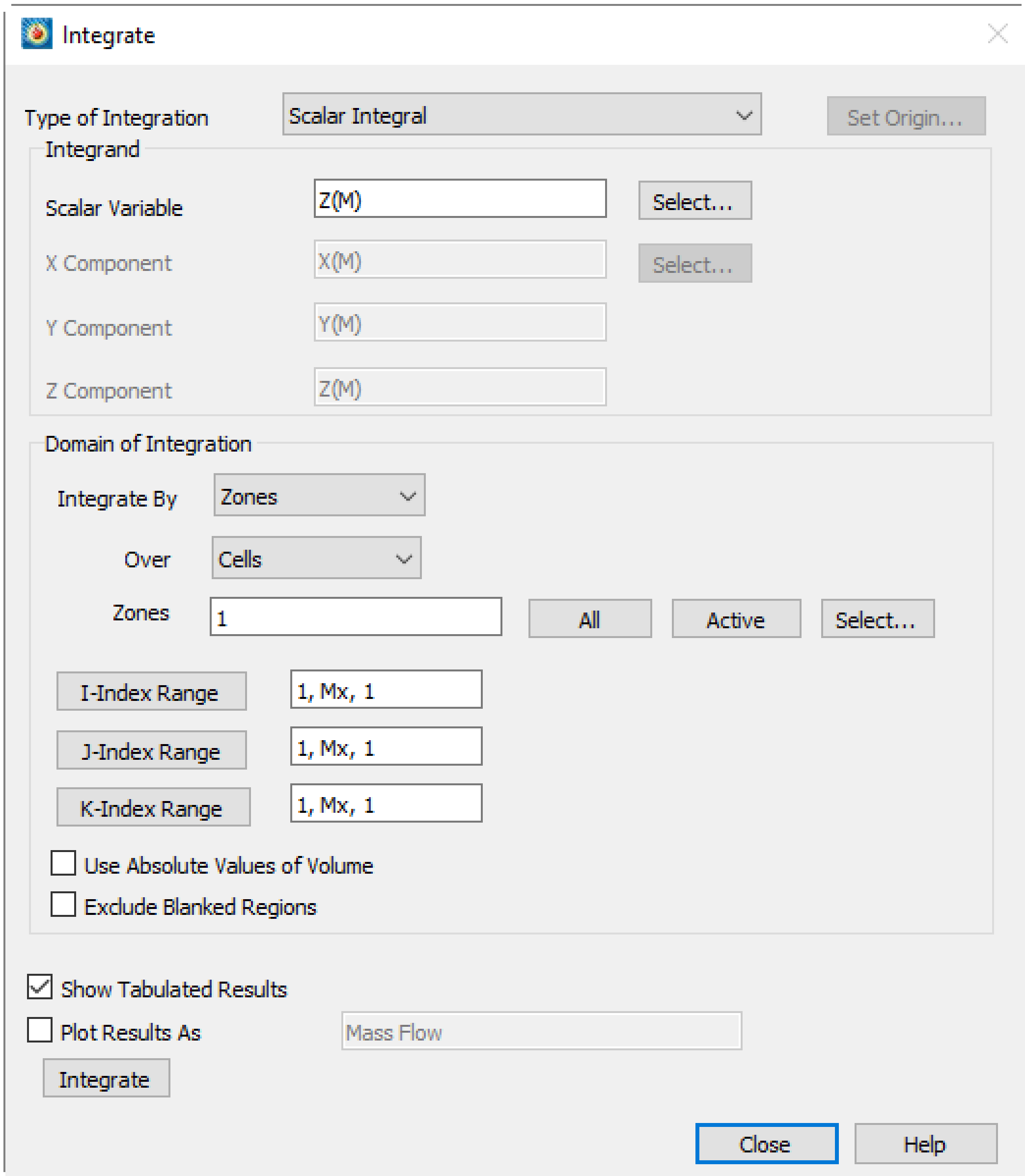


Figure 60. A 3D surface.

To set up the **Integrate** dialog to perform the integration, choose Scalar as the integration type and Z as the scalar variable. The remaining controls are left at their default settings. Selecting Integrate displays the volume under the surface. The **Integrate** dialog and the results are shown in Figure 61].


Integrate
Type of Integration
Scalar Integral
Set Origin...
Integrand
Scalar Variable
Z(M)
Select...
X Component
X(M)
Select...
Y Component
Y(M)
Z Component
Z(M)
Domain of Integration
Integrate By
Zones
Over
Cells
Zones
1
All
Active
Select...
I-Index Range
1, Mx, 1
J-Index Range
1, Mx, 1
K-Index Range
1, Mx, 1
Use Absolute Values of Volume
Exclude Blanked Regions
Show Tabulated Results
Plot Results As
Mass Flow
Integrate
Close
Help

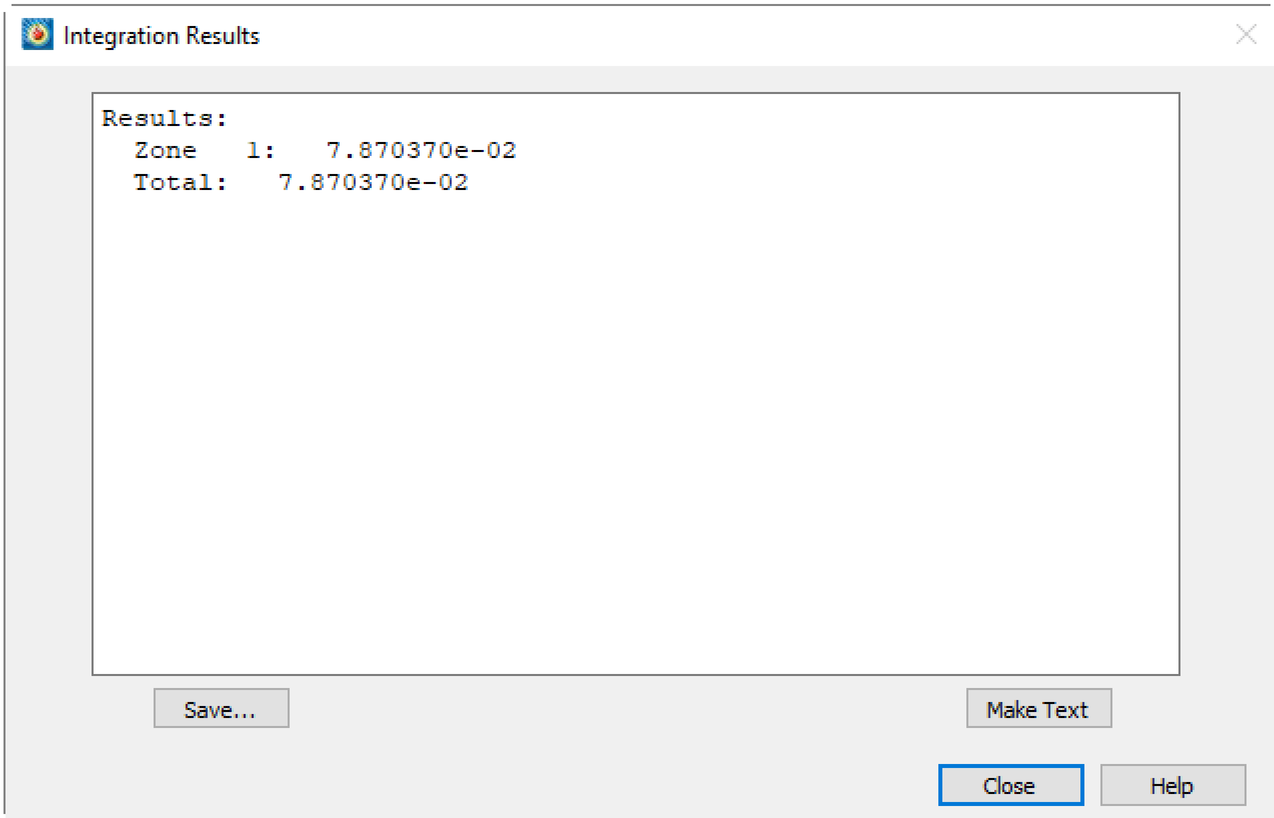


Figure 61. The Integration dialog and the integration results for calculating the volume under the surface shown in Figure 60.

Internal Flow Examples

The next few examples will demonstrate some uses of the **Integrate** dialog for internal flows, such as flow through a jet engine or a pipe. Our dataset consists of a single I-J ordered zone. It is shown with the mesh and contours of pressure in Figure 62.

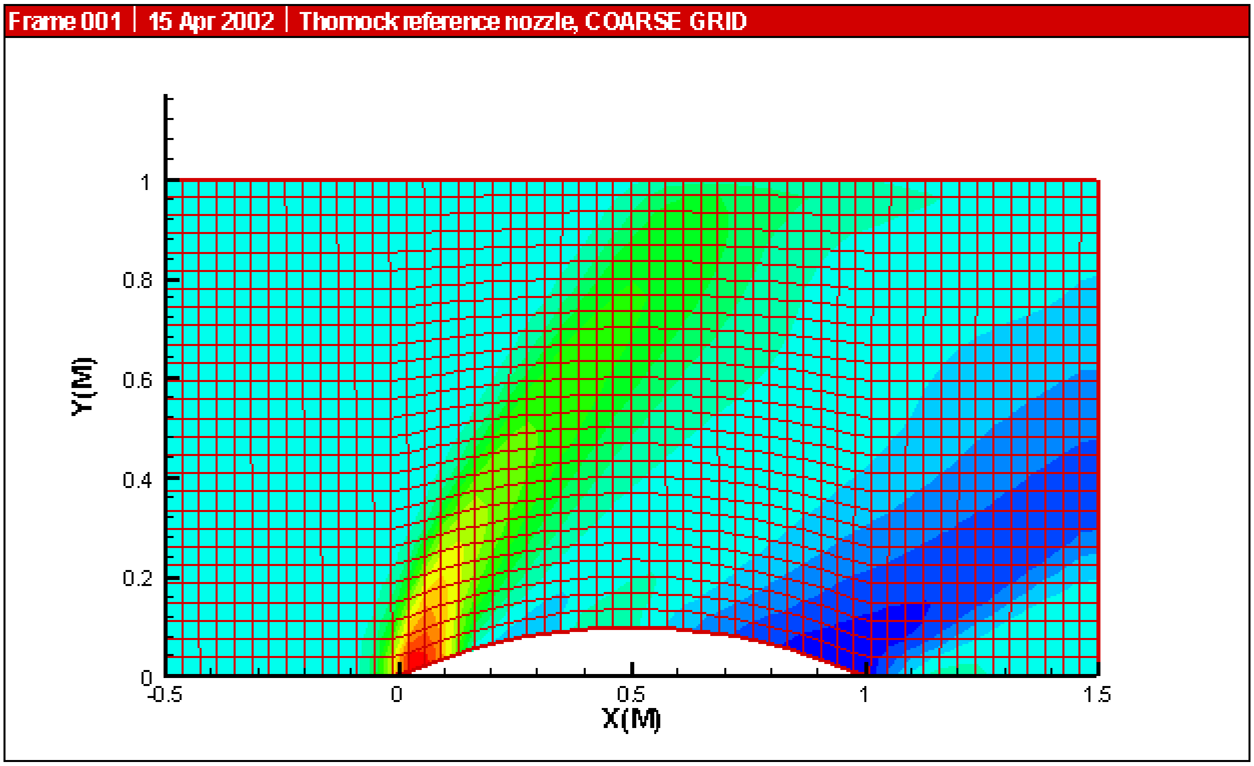
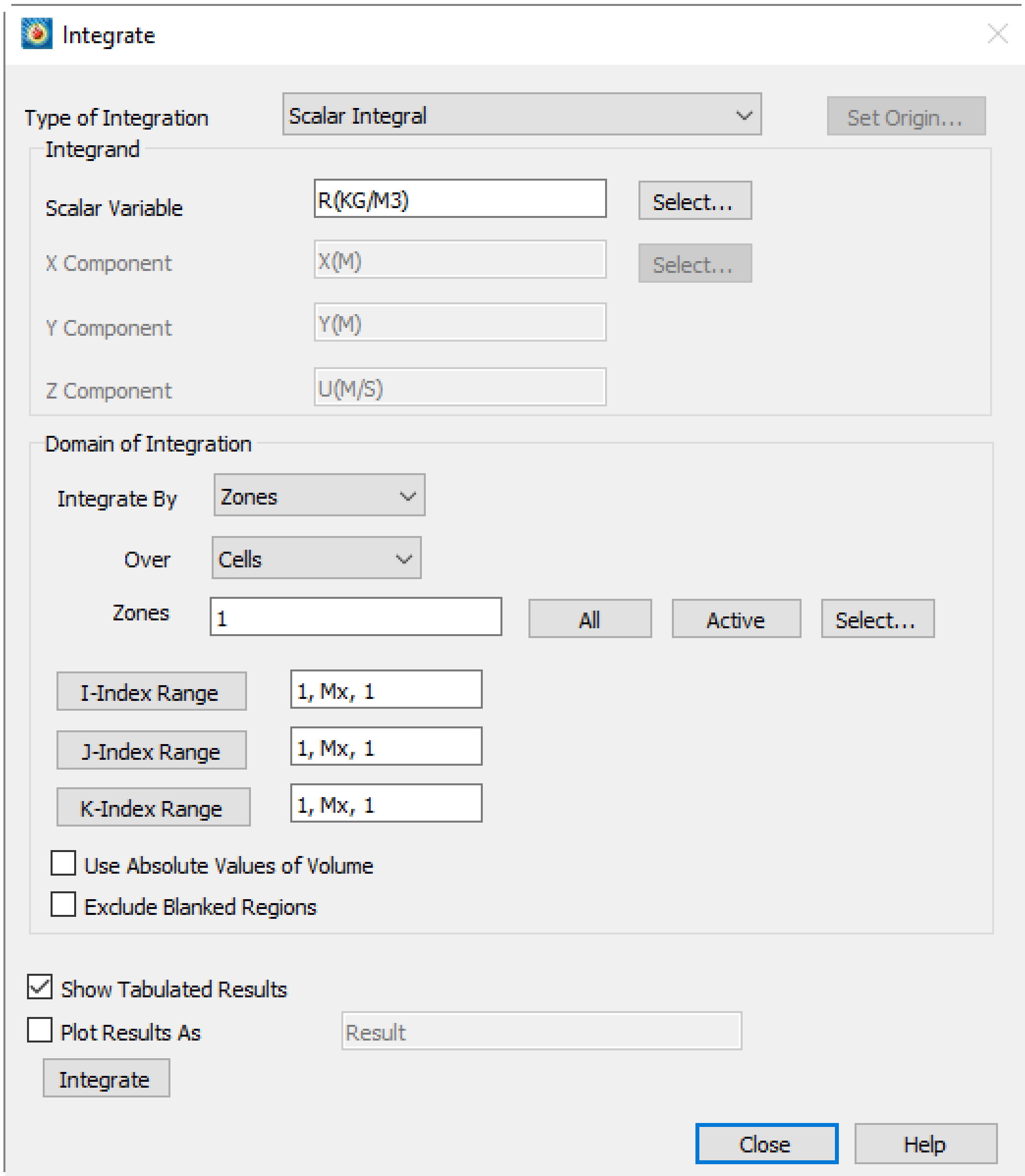


Figure 62. An internal flow solution.

Calculating Total Mass

To calculate the total mass we must integrate density over volume (or area in 2D). If your dataset does not contain density, it may be determined using the **Calculate** dialog. (See Calculating Variables) Select the Scalar Integral integration type, choose the density variable as the scalar, then integrate over Cells (which is demoted to K-planes for our IJ-ordered data). When we click Integrate, the total mass appears as the result of the integration. The **Integrate** dialog and the results are shown in Figure 63.


Integrate
Type of Integration
Scalar Integral
Set Origin...
Integrand
Scalar Variable
R(KG/M3)
Select...
X Component
X(M)
Select...
Y Component
Y(M)
Z Component
U(M/S)
Domain of Integration
Integrate By
Zones
Over
Cells
Zones
1
All
Active
Select...
I-Index Range
1, Mx, 1
J-Index Range
1, Mx, 1
K-Index Range
1, Mx, 1
Use Absolute Values of Volume
Exclude Blanked Regions
Show Tabulated Results
Plot Results As
Result
Integrate
Close
Help

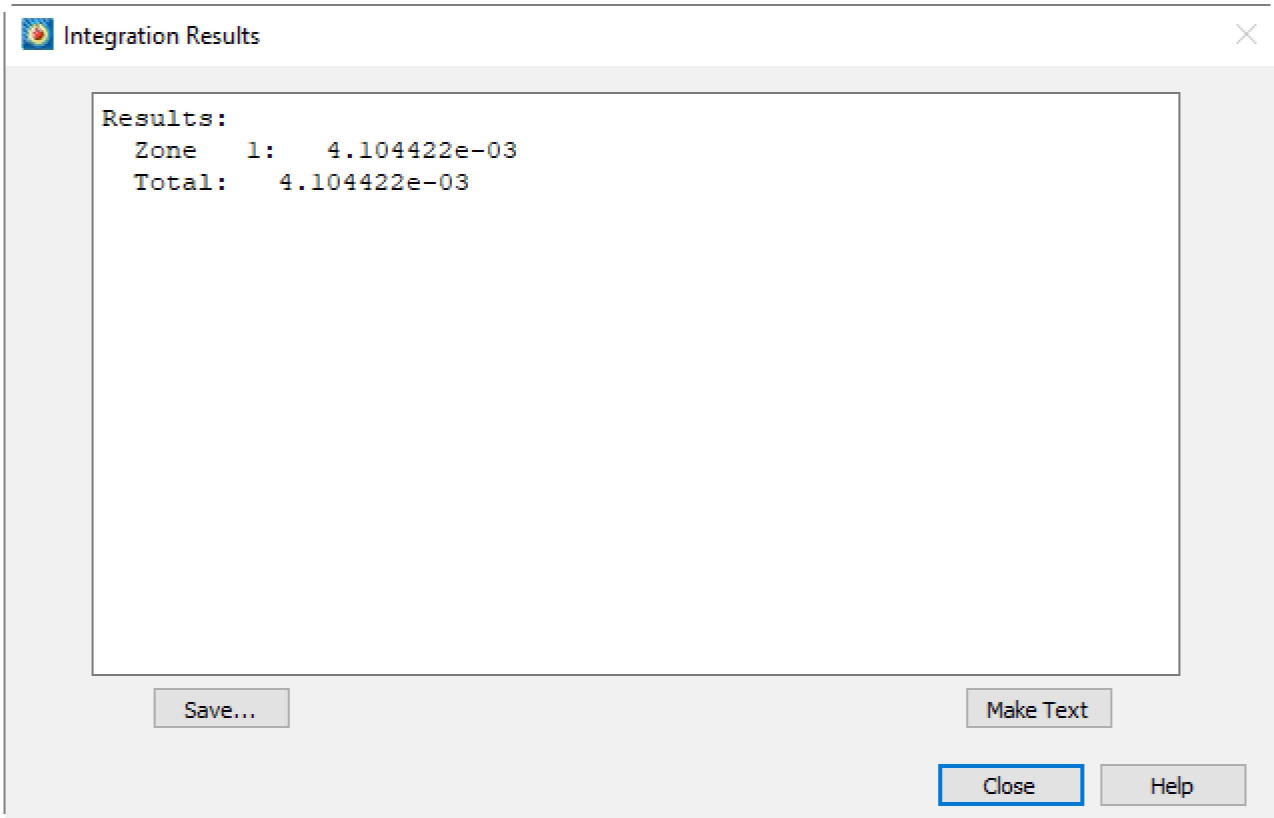


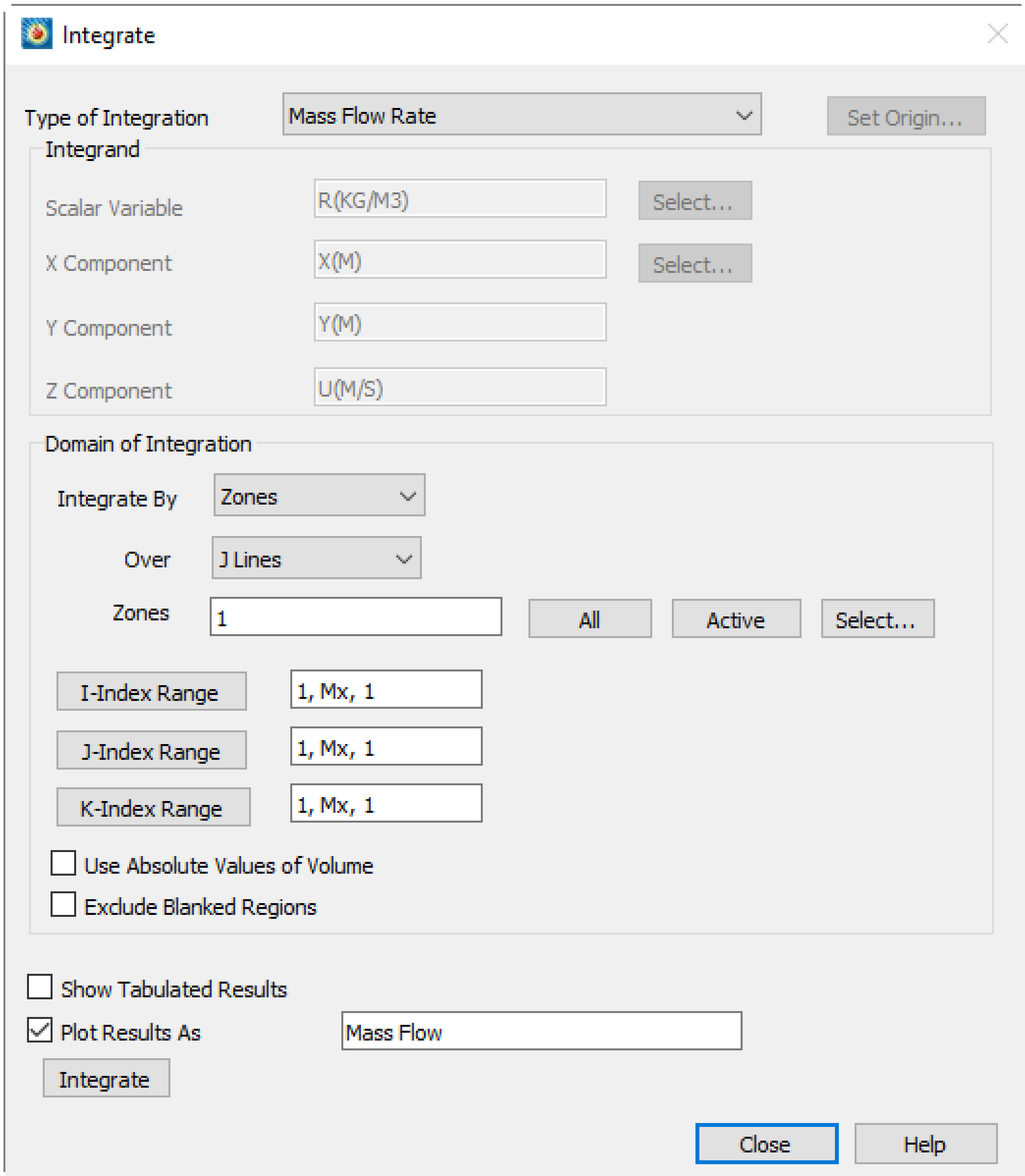
Figure 63. The ***Integration*** *dialog and the integration results for calculating the total mass under the surface shown in Figure 62.*

Calculating Mass Flow Rate

> To calculate mass flow rate, you must first set your convective variables in the **Field Variables** dialog. See Choosing the Convective Variables for information on setting these variables.

We will now calculate the mass flow rate at various stations in the streamwise direction. This will give us an indication of how well converged our solution is to steady-state. The **Integrate** dialog makes this easy with the Mass Flow Rate integration type. We select this option and specify integration over J-lines (which is equivalent to I-planes in 2D). Note that the entire Integrand section of the dialog is disabled. Tecplot 360 calculates the necessary variable (momentum) from information entered in the **Fluid Properties** and the **Field Variables** dialogs.

We only wish to plot the results, so we select this option at the bottom of the **Integrate** dialog, specifying that the result be named "Mass Flow." When we select Integrate, the mass flow rate is plotted versus I-index in a new frame. The **Integrate** dialog and the plotted results are shown in Figure 64. From the results, we see that our solution was not fully converged.


Integrate
Type of Integration
Mass Flow Rate
Set Origin...
Integrand
Scalar Variable
R(KG/M3)
Select...
X Component
X(M)
Select...
Y Component
Y(M)
Z Component
U(M/S)
Domain of Integration
Integrate By
Zones
Over
J Lines
Zones
1
All
Active
Select...
I-Index Range
1, Mx, 1
J-Index Range
1, Mx, 1
K-Index Range
1, Mx, 1
Use Absolute Values of Volume
Exclude Blanked Regions
Show Tabulated Results
Plot Results As
Mass Flow
Integrate
Close
Help

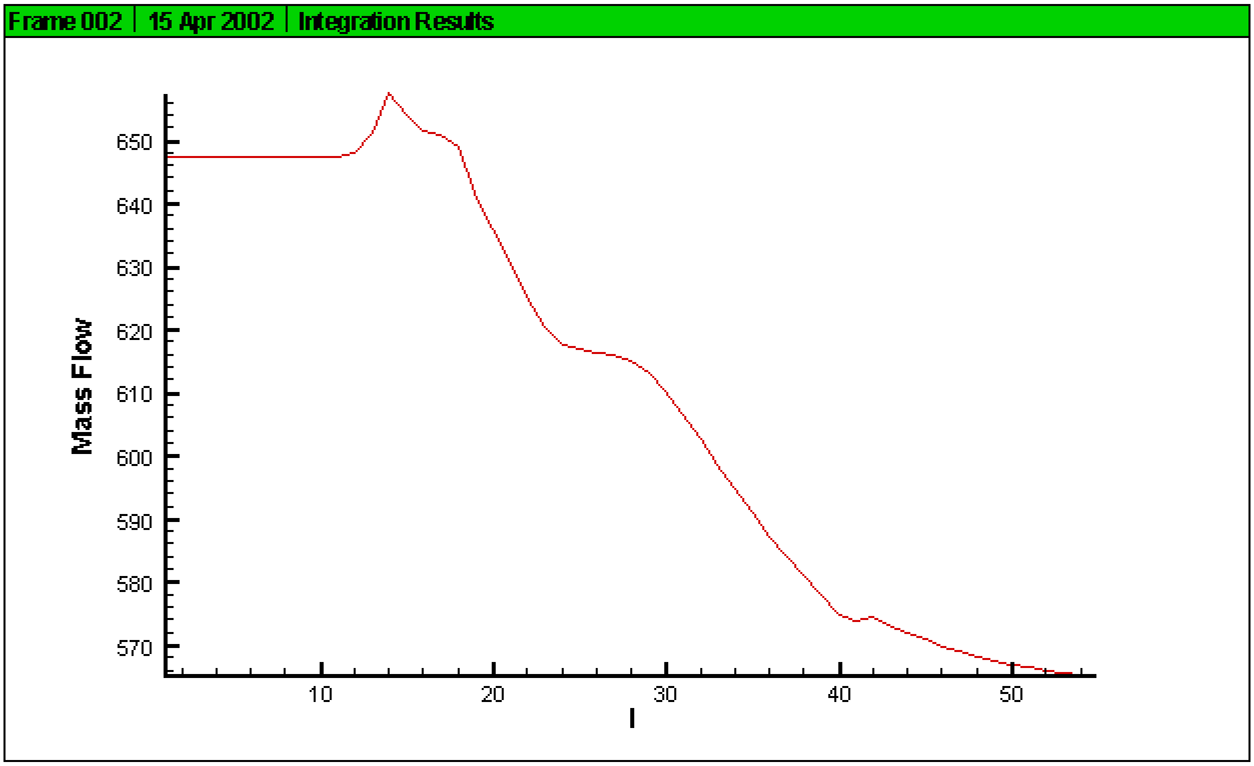
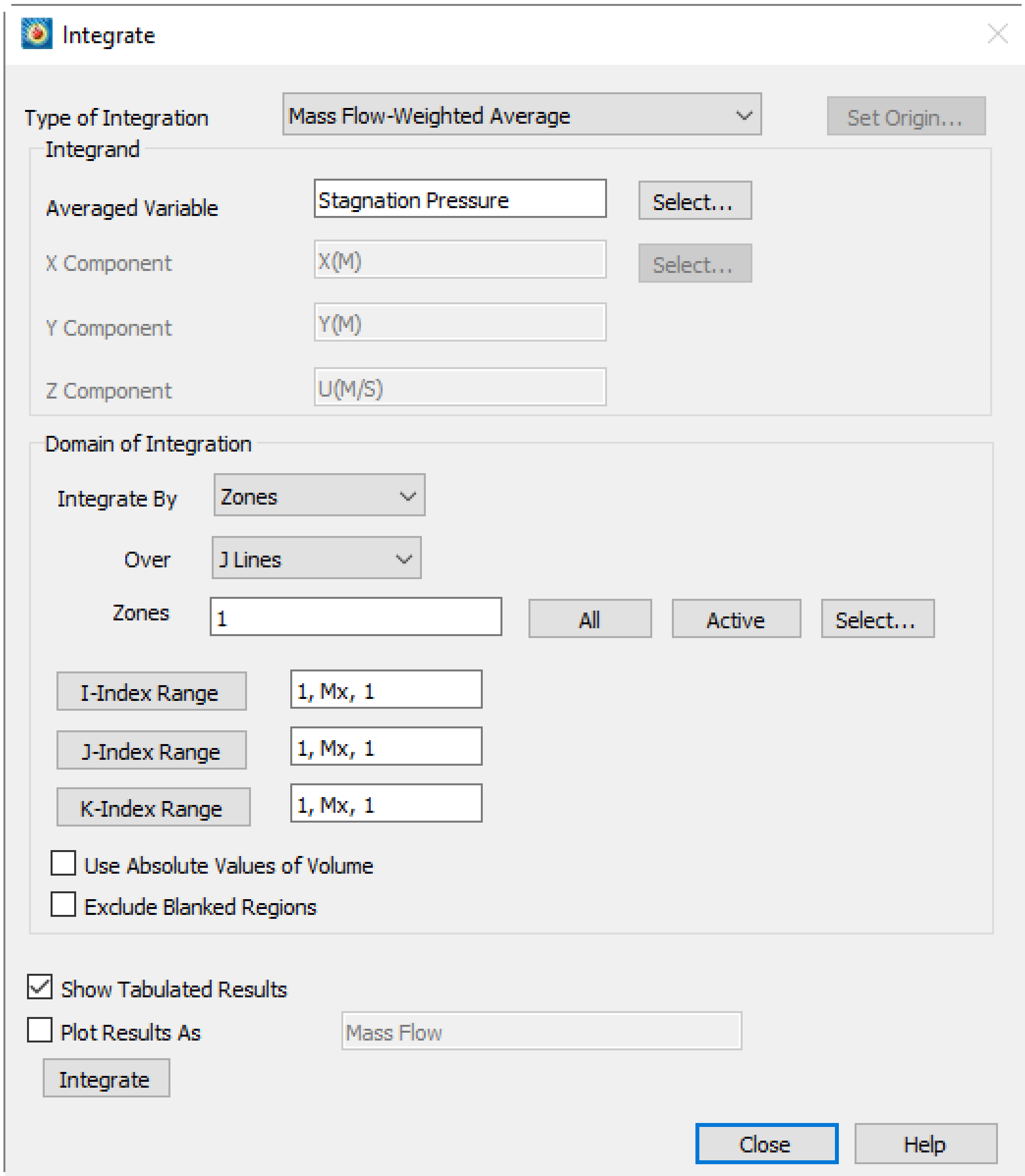


Figure 64. The **Integration** *dialog and the results of calculating the mass flow rate of the object in Figure 62.*

Calculating Mass-weighted Stagnation Pressure

We will now calculate a quantity commonly used in engine analysis, the mass-weighted stagnation (or total) pressure. Although it is referred to as "mass-weighted," the weighting function is actually the mass flow rate. Accordingly, select Mass Flow-weighted Average for the integration type, choosing the Stagnation Pressure variable from our dataset (previously calculated with the **Calculate** dialog). Since we are only interested in this value at the exit plane, we again select J-lines (from the "Integrate Over" drop-down menu), but now specify an I-range of (Mx, Mx, 1) to integrate only the I=IMax plane. We choose only to display the result in a text dialog. Select [Integrate] to perform the calculation. The **Integrate** dialog and the result are shown in Figure 65.


Integrate
Type of Integration
Mass Flow-Weighted Average
Set Origin...
Integrand
Averaged Variable
Stagnation Pressure
Select...
X Component
X(M)
Select...
Y Component
Y(M)
Z Component
U(M/S)
Domain of Integration
Integrate By
Zones
Over
J Lines
Zones
1
All
Active
Select...
I-Index Range
1, Mx, 1
J-Index Range
1, Mx, 1
K-Index Range
1, Mx, 1
Use Absolute Values of Volume
Exclude Blanked Regions
Show Tabulated Results
Plot Results As
Mass Flow
Integrate
Close
Help

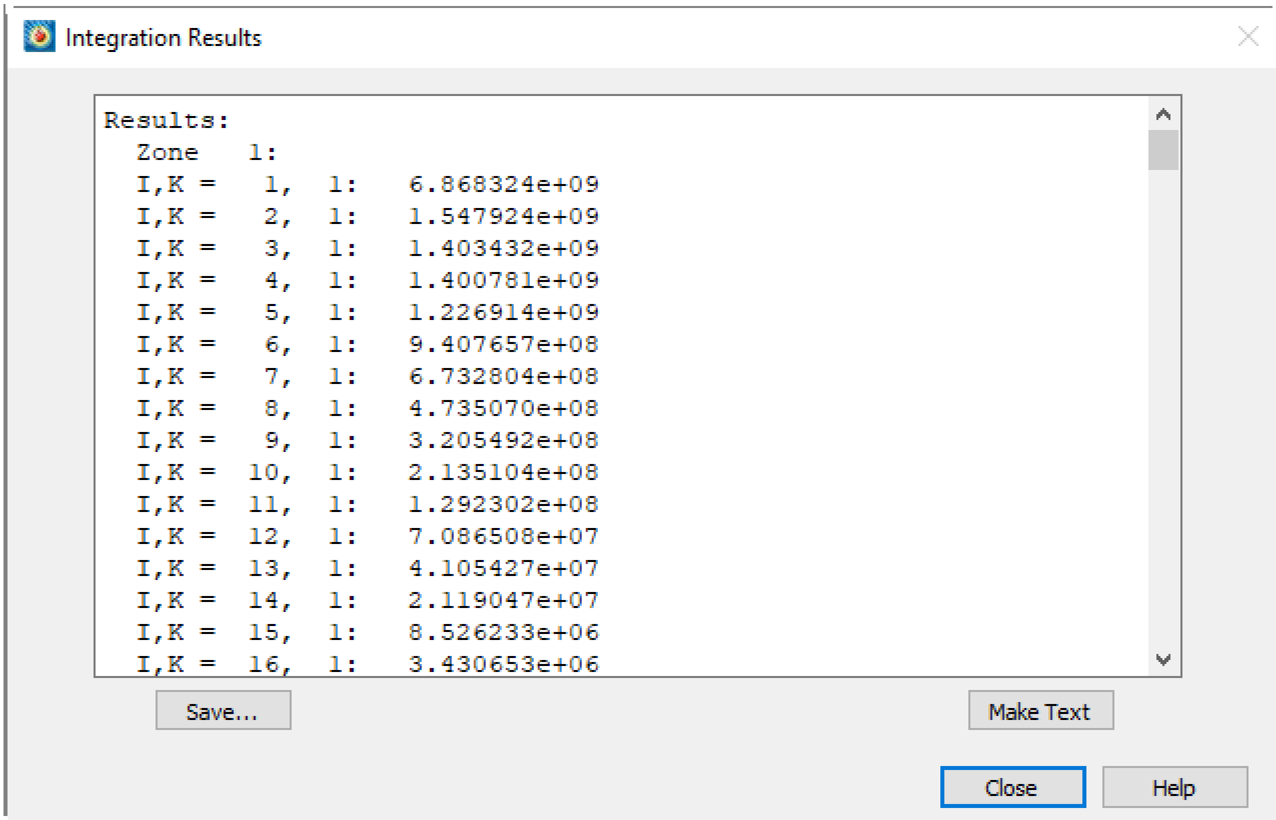


Figure 65. The **Integration** *dialog and results for calculating the mass-flux weighted average integral for the data in Figure 62.*

Circulation in a Vortex Core

If you can define a closed path in your data over which to integrate, you can calculate circulation using integration. In the Integrate dialog, for Type of Integration, select Vector Dotted with Unit Tangential, and select your velocity components for the Integrand. You then need to identify a line—a closed path—as your Domain of Integration. Click Integrate to display circulation.

Calculating Lift and Drag

Our final example makes use of a three-element airfoil solution, an example of an external flow solution. Our data consists of four zones. Three zones are IJ-ordered zones which capture the Edge layer about each of the elements. The fourth zone is a triangular finite element zone that fills the remaining airspace about the elements. Pressure contours and streamtraces of this solution are shown in Figure 66.

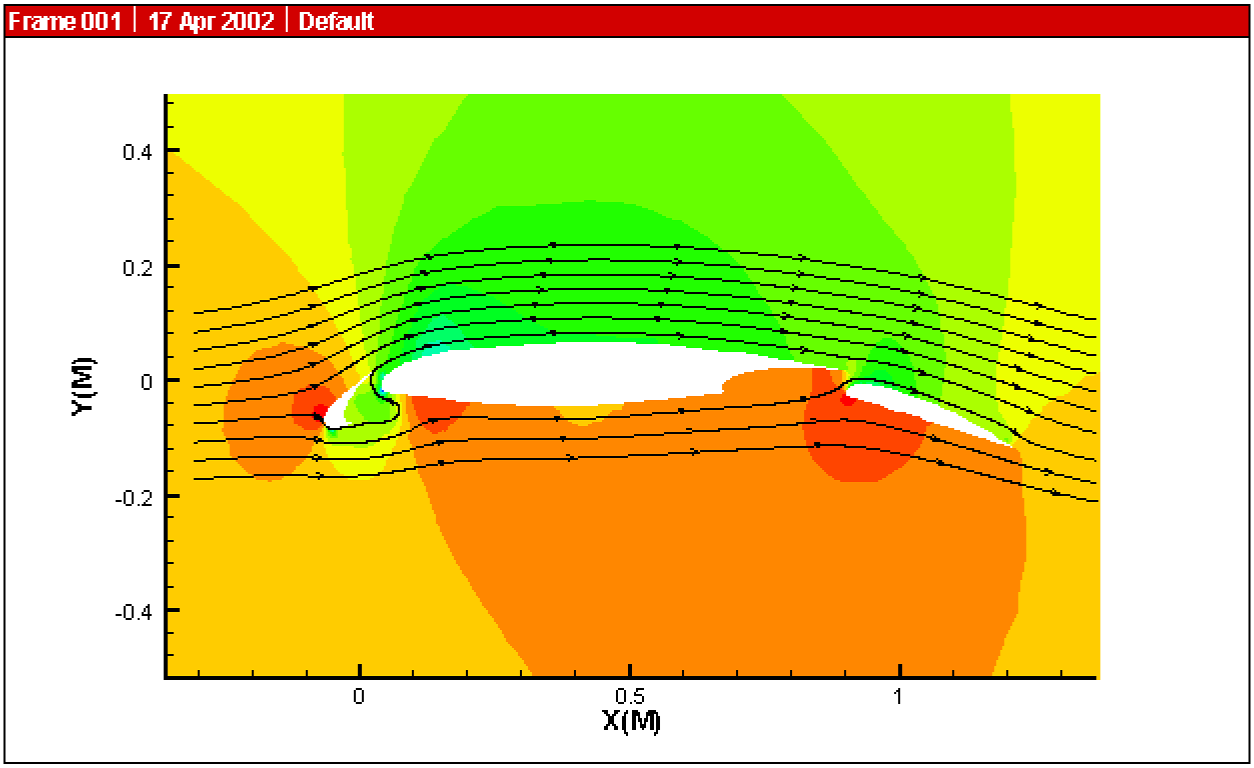
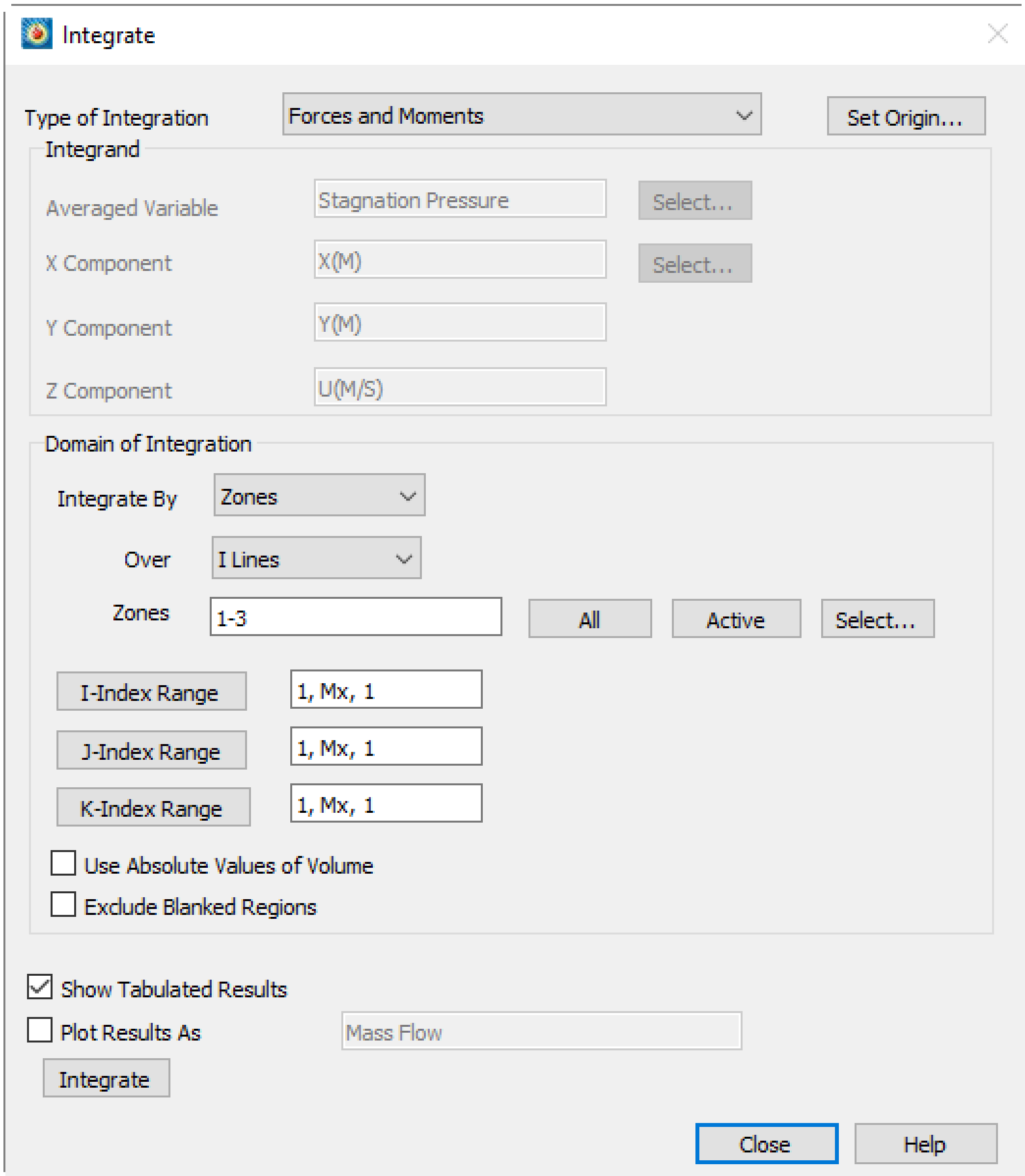


Figure 66. A three element airfoil solution.

To calculate lift and drag for this airfoil configuration we use the Forces and Moments integration type. As with Mass Flow Rate, the entire Integrand portion of the **Integrate** dialog is disabled, because Tecplot 360 will derive the required values (pressure, velocity gradient, and viscosity) from settings in other dialogs. We choose integrate over the surface (J=1) line for each of the three Edge layer zones, the click Integrate. The **Integrate** dialog and results appears as in Figure 67.


Integrate
Type of Integration
Forces and Moments
Set Origin...
Integrand
Averaged Variable
Stagnation Pressure
Select...
X Component
X(M)
Select...
Y Component
Y(M)
Z Component
U(M/S)
Domain of Integration
Integrate By
Zones
Over
I Lines
Zones
1-3
All
Active
Select...
I-Index Range
1, Mx, 1
J-Index Range
1, Mx, 1
K-Index Range
1, Mx, 1
Use Absolute Values of Volume
Exclude Blanked Regions
Show Tabulated Results
Plot Results As
Mass Flow
Integrate
Close
Help

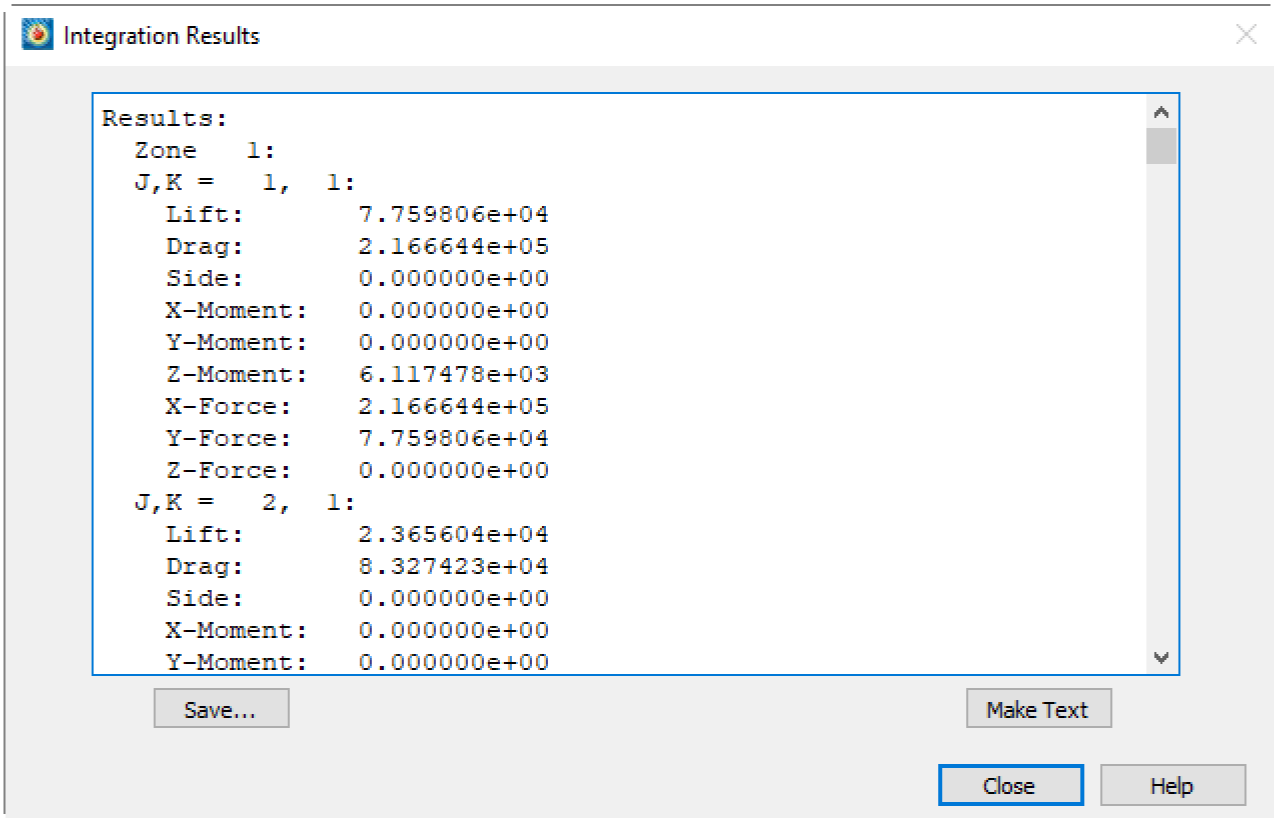
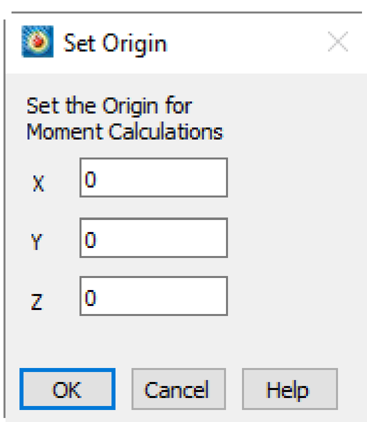


Figure 67. The **Integration** *dialog and the integration results for calculating the lift and drag for the data shown in Figure 66.*

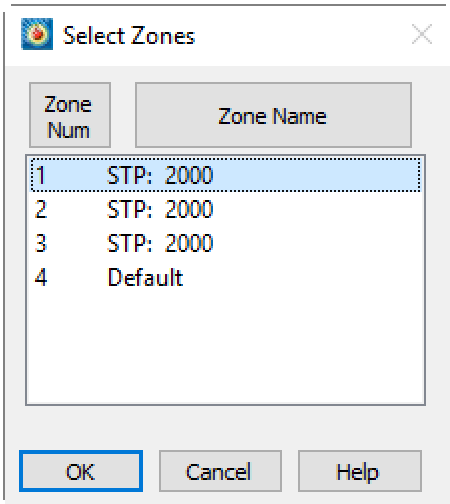
The results of each zone are listed separately. Scrolling to the bottom of the **Integration Results** dialog, we see the total lift and drag, along with other force and moment data.

Specifying the Origin for Moments



When the **Forces and Moments** integration type is selected, the [Set Origin] button is active. Selecting this button displays the **Set Origin** dialog. In this dialog, you may specify the X, Y, Z-location of the origin about which the moments will be calculated.

Select Zones

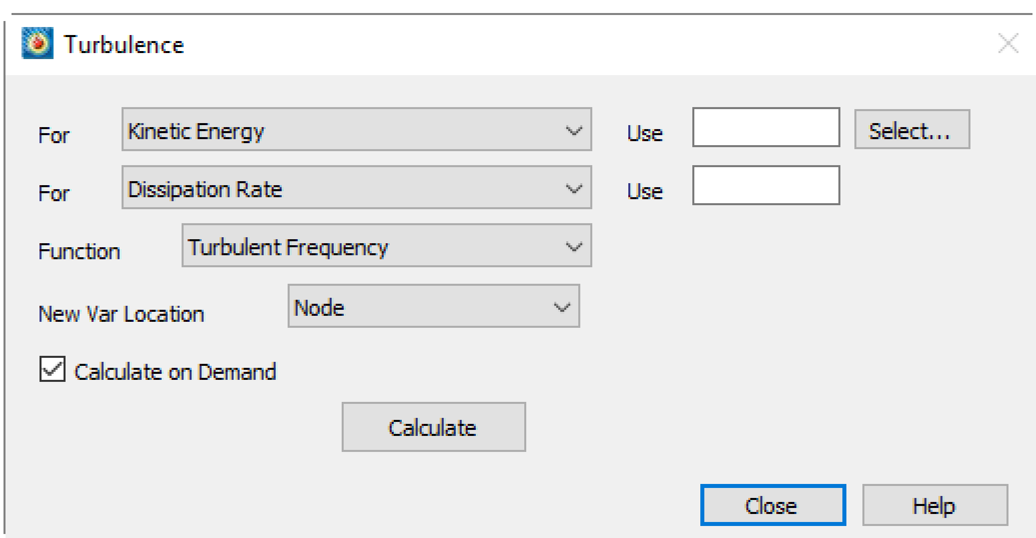


To select zones, choose [All], [Active], or [Select] from the Domain of Integration portion of the **Integrate** dialog. By choosing [Select], you may select the zones you want by clicking in the list, or by selecting [Zone Number] or [Zone Name] in the resultant **Select Zones** dialog. Selecting [Zone Number] calls up the **Enter Range** dialog, allowing you to indicate the desired zones by a numeric range. Selecting [Zone Name] prompts you for a pattern string, which is matched against the names of all zones. You may use the asterisk as a wildcard when entering the zone name pattern. All zones whose names match the pattern are then selected in the `list.ht`.

Calculating Turbulence Functions

Tecplot 360 allows you to calculate and add to your dataset any of four turbulence-related quantities, provided you already have any given two in your dataset. Turbulent kinetic energy, dissipation rate, frequency and kinematic viscosity, and dynamic viscosity are available via the **Turbulence** dialog.

The **Turbulence** dialog is displayed by selecting **Calculate Turbulence Functions** from the **Analyze** menu.



It contains two drop-down menus and associated text fields for you to identify the two turbulence-related variables in your dataset, drop-downs for you to select the function you wish to calculate and the location of the calculated variable, a toggle to select calculate-on-demand, and a **Calculate** button to perform the calculation.

Identifying Turbulence Variables

The first two drop-down menus on the **Turbulence** dialog allow you to specify which turbulence variables are contained in your dataset. The options are Kinetic Energy (κ), Dissipation Rate (ε), Turbulent Frequency (ω), Turbulent Kinematic Viscosity (ν_t), and Turbulent Dynamic Viscosity(μ_t). This last option is the kinematic viscosity, which is equal to the dynamic viscosity divided by the density.

Selecting the Variable Location

You may select the location (nodal or cell-centered) of new variables Tecplot 360 creates during a calculation with the New Var Location dropdown. This setting only affects new variables added to the dataset when you click Calculate. Variables that already exist in the dataset keep their existing locations. If you wish to change the location of an existing variable, you can delete or rename the variable and then perform the calculation with the desired setting for New Var Location.

Calculating on Demand

Selecting the **Calculate on Demand** option results in the calculated variable being added to the dataset when you click the **Calculate** button, but the actual calculation is delayed until it is actually needed. Please refer to the discussion of calculate-on-demand in Calculating Variables.

Performing the Calculation

Once you have identified two turbulence variables in your dataset, you may calculate either of the other two. Select the desired function from the Function drop-down menu and click **Calculate**. The function is calculated and added to your dataset as a variable with the same name as the function selected. If your dataset variables are κ and ε, the following formulae will be used for the calculations of ω and ν_t:

$$\omega = \frac{\varepsilon}{C_\mu \kappa}$$

$$\nu_t = \frac{C_\mu \kappa^2}{\varepsilon}$$

with $C_\mu = 0.09$. Equations for other input variables are derived from these.

Shared Variables

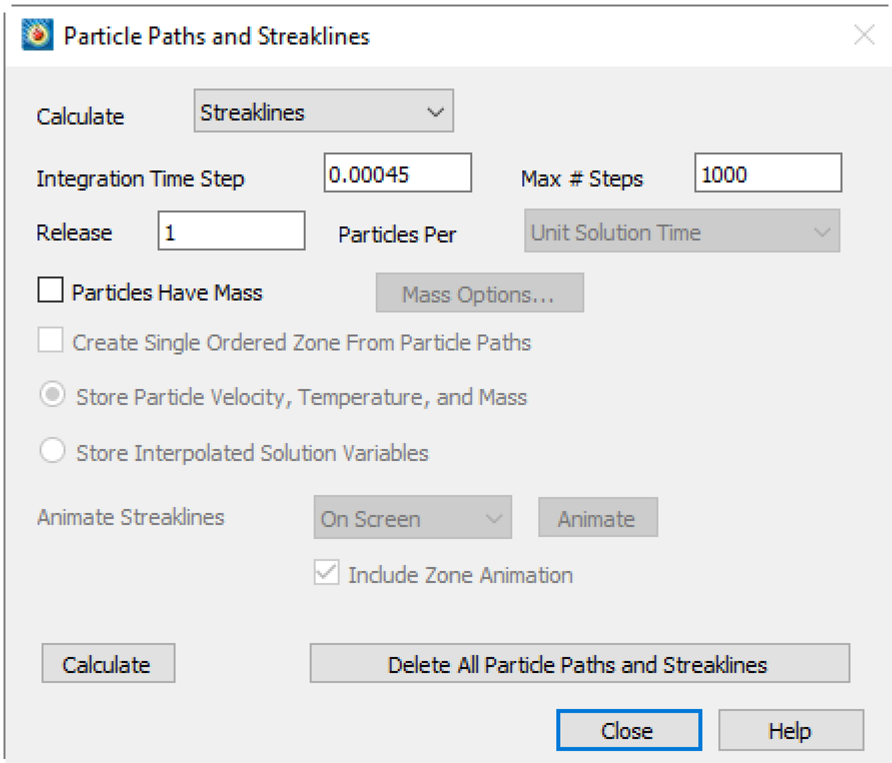
If both variables from which the turbulence function is calculated are shared between multiple zones, and they and the calculated variable are all at the same location (cell-centered or nodal), the new variable will be shared as well. This mimics the behavior of the **Data→Alter→Specify Equations** dialog.

Calculating Particle Paths and Streaklines

For steady-state solutions, Tecplot 360 allows you to track the paths of massless particles by placing streamtraces in the flow. The **Particle Paths and Streaklines** dialog augments this capability by providing two additional visualization methods, particle paths and streaklines, for particles with or without mass.

Please note that these calculations, particularly for streaklines, may be very lengthy to perform, especially for cases with large grids.

The **Particle Paths and Streaklines** dialog is displayed by selecting **Calculate Particle Paths and Streaklines** from the **Analyze** menu.



It contains a drop-down menu allowing you to choose particle paths or streaklines, as well as options pertaining to the path integrations, particles with mass, storage and display of the calculated particle paths. In addition, the results of streaklines may be animated.

Calculating Particle Paths

A particle path is the path that a single particle follows through a solution. In steady flow, particle paths are the same as streaklines and streamtraces for massless particles. To calculate particle paths, you must:

1. Place streamtraces at the locations where you wish particles to be released, then select Particle Paths from the drop-down menu at the top of the dialog. (Details on placing streamtraces may be found in Streamtraces).
2. Specify an integration time step. For steady-state calculations, specify the maximum number of

time steps to be performed (see Unsteady Flow for specifying steady or unsteady flow).

3. Set the Particles Have Mass option for particles with mass. Click Mass Options to set mass-related options.
4. Optionally, set the Create Single Ordered Zone From Particle Paths toggle to create a single IJ-ordered zone from all particle paths instead of a separate I-ordered zone from each path.
5. Select **Calculate**.

Specifying the Integration Time Step and Maximum Number of Steps

Particle Paths are calculated by integrating the velocity field of your solution using a constant time step, which you enter in the Integration Time Step text field. A smaller time step will result in more accurate particle paths but will take longer to calculate. For unsteady calculations, the time step is set equal to the time interval between your solution time levels by default. If you specify so large a time step that a particle passes out of your solution domain in the first integration time step, you will get a warning message.

If you have set the Flow Solution is Steady-state option, you must also enter the maximum number of integration time steps to be performed (see also Unsteady Flow).

Specifying Mass-Related Options

For particles with mass, set the Particles Have Mass option. This enables other mass-related controls in the dialog. Click Mass Options to display the **Particle Mass Options** dialog. (See Particles with Mass) In addition, you have the option of storing the particle's velocity and other particle properties or the local flow properties along the calculated particle path. Select Store Particle Velocity, Temperature, and Mass to store these values along the particle path. Select Store Interpolated Solution Variables to store these values instead. Following the calculation, you will be informed of which dataset variables contain these values.

Performing the Particle Path Calculation

When you select Calculate, a particle is placed at the starting point for each streamtrace you have placed. If you did not place any streamtraces, you will get an error message. From these starting locations, beginning with the time equal to the time of your first solution time level (or zero for steady-state calculations), the particle positions are advanced by performing a second-order Runge-Kutta integration of the velocity field. For unsteady calculations, linear interpolation is performed between solution time levels. Integration for each particle is continued until the final time level is reached (unsteady calculations), the specified number of time steps has been performed (steady-state calculations), or until the particle passes out of your solution domain. The particle paths are displayed as new I-ordered zones in your dataset, with each integration step represented by a node in the new zones, unless you selected the Create Single Zone From Particle Paths option, which results in a single IJ-ordered zone.

Examining the Particle Paths

Each I-ordered zone created by a Particle Path calculation represents a path through space and time. The paths' non-grid variables will hold interpolated values of your solution data that the particle "saw" as it passed through your solution, except as discussed in Specifying Mass-Related Options. You can visualize this by coloring the particle zones' mesh plots with one of your solution variables. The following steps will accomplish this:

1. Turn on the Mesh plot layer by toggling-on the Mesh in the Plot sidebar.
2. Call up the **Zone Style** dialog (accessed via the **Plot** menu or the Plot sidebar).
3. Turn off mesh plotting for your solution zones by selecting the solution zones, clicking Mesh Show and selecting No.
4. If necessary, turn on mesh plotting for the Particle Path zones by selecting them, clicking Mesh Show and selecting Yes.
5. Color the Particle Path zones with a variable by selecting these zones, clicking Mesh Color and selecting Multi-color. If you had not previously chosen a contour variable, the **Contour Variable** dialog will open to allow you to select it. Choose the variable you wish to use to color the particle paths.
6. If Auto Redraw has not been selected, click Redraw to redraw your plot. You will see the particle paths displayed and colored with the contour variable.

You may wish to turn on the Scatter plot layer to see the size of these steps. If you do this, you will first want to turn off scatter plotting for your solution zones. You can also do this with the **Zone Style dialog**.

Calculating Streaklines

Streaklines simulate experimental techniques which involve the periodic or continuous release of a tracer substance, such as oil drops or smoke. Tecplot 360 produces streaklines by releasing a sequence of particles from the release points and integrating the unsteady velocity field to find their positions in the flow at the end solution time. The final positions of all particles emitted from a particular release point form one streakline. Once streaklines have been calculated, they may be animated on screen or to a file.

To calculate Streaklines, perform the following actions:

1. Identify the solution time levels in your dataset. (See Unsteady Flow.)
2. Place streamtraces at the locations where you wish particles to be released.
3. Select Streaklines from the drop-down menu at the top of the **Particle Paths and Streaklines** dialog.
4. Enter the integration time step as with particle path calculations. (See Specifying the Integration Time Step and Maximum Number of Steps).
5. Specify the particle release frequency. (See Specifying the Particle Release Frequency.)

6. For particles with mass, set the Particles Have Mass option. Click Mass Options to set mass-related options.
7. Select **Calculate**.

It is not reasonable to calculate streaklines for steady-state flow, because in steady-state flow, even for particles with mass, streaklines are the same as particle paths (just more time consuming to compute).

Specifying the Particle Release Frequency

For Streakline calculations, a sequence of particles is released throughout the solution time. Each particle's position is integrated using the specified integration time step. The frequency with which particles are released is specified by the controls just above the **Calculate** button. In the Release text field, enter the number of particles to be released in the specified time interval. In the particles per drop-down menu, identify this time interval by selecting either Solution Time Level or Unit Solution Time.

If you select Solution Time Level, the indicated number of particles will be released, evenly spaced in time, between each pair of solution time levels you have identified. If you select Unit Solution Time, the particles will be released at regular intervals throughout the time covered by your solution. In either case, a particle will be released at the final time of your solution, so that the streaklines will include the release points themselves. Releasing particles more frequently will produce more detailed streaklines (the accuracy is determined by the Integration Time Step), but will take longer to calculate.

Performing the Streakline Calculation

When you click Calculate, the streaklines are calculated and added to your dataset as new I-ordered zones. To see them, turn on the Mesh plot layer and disable mesh plotting for your solution zones. See Examining the Particle Paths.

Animating Streaklines

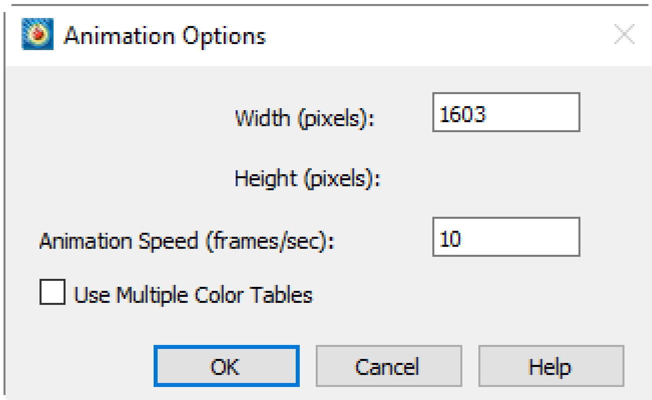
Once you have performed a streakline calculation, the animation controls of the **Particle Paths and Streaklines** dialog are enabled. A streakline animation displays each successive step in the integration, and can be an effective means of visualizing the unsteadiness of a flow. Toggle-on **Include Zone Animation** in the **Particle Paths and Streaklines** dialog to animate the zones along with the streaklines.

> Please note that subsequent particle path or streakline calculations will replace the current streakline calculation, making it unavailable for animation.

You may display the animation in the frame in which the streaklines were calculated or save it to a video file in a number of formats. To perform a streakline animation, perform the following steps:

1. Delete the I-ordered zones of any streaklines you do not wish to be part of the animation using **Data→Delete→Zone**.
2. Select the animation destination from the Animate Streaklines dropdown.

3. Select Animate.
4. If you chose to save the animation to a file, the Animate Options dialog will be displayed. Enter your choices for the animation and select **OK**. Then choose a file name in the resulting file selection dialog.



While animating on the screen, the **Animate** button's text will change to Cancel, allowing you to stop the animation. While animating to a file, a progress dialog will be displayed that allows you to cancel the animation.

Deleting Particle Paths and Streaklines

Particle paths and streaklines are saved either as I-ordered zones or as a single IJ-ordered zone. You may delete these individually using the **Delete Zone** dialog (accessed via **Data**→**Delete**→**Zone**). If you wish to delete all previously calculated particle paths and streaklines, you may do so using the [Delete All Particle Paths and Streaklines] button. This deletes all zones whose names begin with 'Particle Path' or 'Streakline.'

Animate Options

The **Animation Options** dialog allows you to specify options for saving the streakline animation to a file. The following options are available:

Width (pixels)

Enter a value in the text field for your exported image's width. The image region is rendered to the image file to exactly fit a size of Width by Height. This text field initially displays the frame's actual width.

Height (pixels)

Displays the height of the image based on the value entered for Width, preserving the shape of the region to be exported. (Calculated by Tecplot 360.)

Animation Speed (frames/sec)

Applicable only to AVI files. Enter a value in the text field to set your speed in frames per second.

Use Multiple Color Tables

Selecting this check box will create a color table for each frame of the animation. If this check box is not selected, Tecplot 360 will scan each frame in your Raster Metafile and create an optimal color table from 256 colors for the entire animation.

Particles with Mass

Whereas massless particles always travel with the local fluid velocity, particles with mass travel according to a more complicated equation of motion where the fluid creates drag on the particle. In addition, particles with mass may have a temperature that is different from the local fluid temperature, and they may lose mass due to ablative processes such as vaporization. The **Particle Mass Options** dialog allows you to enter coefficients and particle properties to indicate how these mass-related effects are calculated.

The **Particle Mass Options** dialog is displayed by selecting Mass Options on the **Particle Paths and Streaklines** dialog. It allows you to specify either general or detailed coefficients related to the particle trajectory and heat transfer calculations, plus options related to gravity and the initial particle velocity. If you choose to calculate the particle temperature, you may choose to terminate the particle at a specified temperature, or, with the detailed coefficient option, to ablate the particles until their mass reaches zero.

Selecting a Coefficient Set

You may enter either general coefficients or detailed coefficients. General coefficients are a convenient way of characterizing the particles, but result in less accurate calculations. They should only be used when the particle drag coefficient and heat transfer coefficient (if particle temperature is being calculated) are essentially constant. Detailed coefficients result in more accurate calculations, and should be used whenever the drag coefficient or heat transfer coefficient may not be constant, such as when the particle Reynolds Number is less than 1000. In addition, if you wish to calculate particle ablation, you must specify detailed coefficients. Indicate your choice of coefficients by making the appropriate selection in the option box at the top of the **Particle Mass Options** dialog.

Calculating Particle Temperature

If you wish to calculate each particle’s temperature along its path, set the Calculate Particle Temperature option. Particles begin with their temperature equal to the local fluid temperature at their insertion point (the beginning of each streamtrace you have placed). If you have chosen to enter general coefficients, enter the Temperature Time Constant in the General Coefficients section of the dialog. Otherwise, enter the specific heat (per unit mass) and the Nusselt number in the Detailed Coefficients section of the dialog. Also, select from the available options in the Termination Options section of the dialog. All of these options are discussed below.

Specifying the Effects of Gravity and Buoyancy

If you wish to include the effects of gravity in your calculation, enter the value in the gravity field and select the axis direction in which gravity acts.

If you choose the detailed coefficient set and non-zero gravity, the effects of buoyancy will also be included. Buoyancy acts in the opposite direction of gravity. It is included by subtracting from the particle mass the mass of the fluid it displaces, and multiplying the result by the gravitational constant to calculate the force due to gravity.

Buoyancy effects are not included if you choose the general coefficient set because the particle size is not specified. In this case, the value for gravity is simply added to the particle acceleration that is calculated from the general coefficients and local flow conditions.

Specifying the Initial Particle Velocity

Each particle injected into the flow begins either at the velocity of the flow at the point where the particle is injected, or at zero velocity or at a specified velocity. Select one of these options from the drop-down menu and, if you have chosen Specified Velocity, enter the U, V, and W velocities.

General Coefficients

Figure 68 shows the **Particle Mass Options** dialog with the general coefficients displayed. The General Coefficients consist of the Ballistic Coefficient and, if you are calculating particle temperature, the Temperature Time Constant.

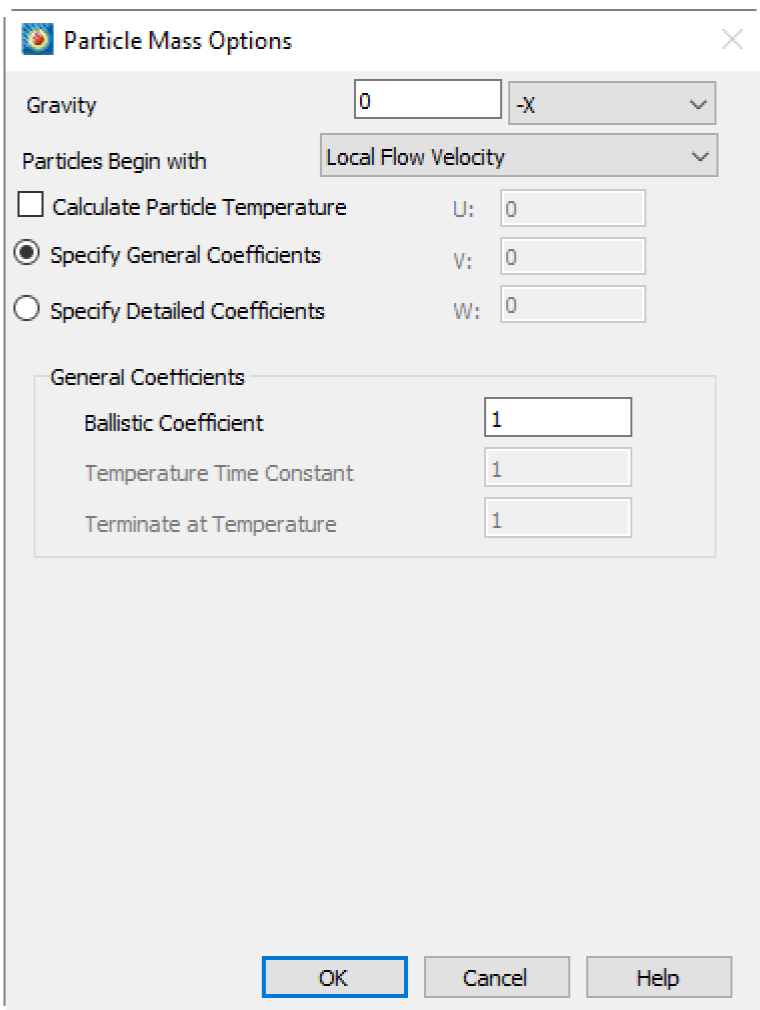


Figure 68. The **Particle Mass Options** *dialog with general coefficients.*

Ballistic Coefficient

The Ballistic Coefficient is defined by the following:

$$B = \frac{m_p}{SC_D}$$

where B is the Ballistic Coefficient, S is the frontal area of the particle, C_D is the particle's drag coefficient and m_P is the particle's mass. Given the Ballistic Coefficient, the acceleration of a particle due to fluid drag is calculated from

$$a_i = \frac{0.5\rho_f(u_{f_i} - u_{p_i})\|(u_f - u_\rho)\|}{B}$$

where a is particle acceleration, i stands for each spatial dimension, ρ_f is the local fluid density and u_{f_i} and u_{p_i} are the velocity components of the fluid and the particle. If non-zero gravity has been specified, the acceleration in the specified direction is augmented by the value for gravity. For example, if a gravitational constant, g_c, acts in the $-Z$ direction, the acceleration in the Z direction becomes:

$$a_z = \frac{0.5\rho_f(u_{f_z} - u_{p_z})\|(u_f - u_\rho)\|}{B} - g_c$$

Temperature Time Constant

For the general coefficient option, particle temperature is calculated with a simple relaxation:

$$\frac{dT_p}{dt} = \frac{1}{\tau_T}(T_f - T_p)$$

where T is temperature, and τ_T is the Temperature Time Constant you enter in this text field. τ_T has units of time, and indicates the "e-folding" time of this relaxation—the amount of time it takes to reduce the difference between the fluid temperature and the particle temperature by a factor of about 2.7.

Comparing with the convective heat transfer equation,

$$Q = hA(T_w - T_\infty) = -m_p c_p \frac{dT_p}{dt}$$

we see that τ_T may be thought of as a combination of the convective heat transfer coefficient, h and the surface area, mass, and specific heat of the particle:

$$\tau_T = -\frac{\pi r_p^2 h}{m_p c_p}$$

Note that the Temperature Time Constant is only constant if the heat transfer coefficient is also constant. In general, however, this coefficient will vary with the particle's velocity relative to the fluid, so this approximation should be viewed with skepticism.

Detailed Coefficients

Figure 69 shows the **Particle Mass Options** dialog with detailed coefficients displayed. The detailed coefficients consist of particle mass radius and drag coefficient. In addition, if particle temperature is being calculated, the detailed coefficients consist of particle specific heat and Nusselt number.

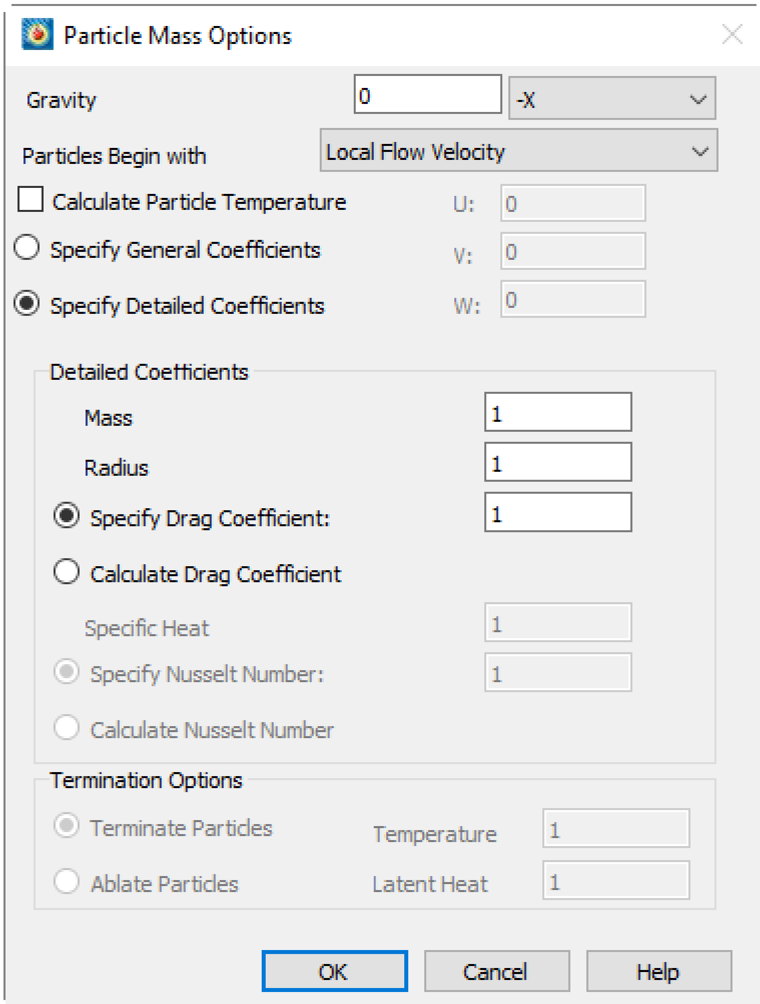


Figure 69. The **Particle Mass Options** *dialog with detailed coefficients.*

Mass

Each particle begins with the same mass, entered in this text field. If ablation is being calculated, the particle's mass may be reduced by the ablative process as it travels through the flow field.

Radius

As with Mass, each particle begins with the same radius, entered in this text field and may be reduced by ablation.

Specify/Calculate Drag Coefficient

You may elect to specify a constant drag coefficient or have Tecplot 360 calculate it. If you specify a constant drag coefficient, enter its value in the corresponding text field. For calculated drag coefficient, Tecplot 360 uses a formula from *Multiphase Flow and Fluidization: Continuum and Kinetic Theory Descriptions* (D. Gidaspow, 1994):

$$C_D = \frac{24}{Re}(1 + 0.15(Re)^{0.687}) \quad for\ Re<1000$$

$$C_D = 0.44 \ \ for\ Re \geq 1000$$

with the particle Reynolds number:

$$Re = \frac{\rho_f d_p |\vec{U}_p - \vec{U}_f|}{\mu_f}$$

where d_p is the particle diameter, $|\vec{U}_p - \vec{U}_f|$ is the speed of the particle relative to the fluid and μ_f is the dynamic viscosity of the gas. The acceleration then becomes:

$$a_i = \frac{F_i}{m_p} = \frac{\frac{\pi}{2} r_p^2 \rho_f (u_{f_i} - u_{p_i}) \|(u_f - u_p)\| C_D}{m_p}$$

If non-zero gravity has been specified, the acceleration in the specified direction is augmented by the gravitational constant adjusted for buoyancy. For example, if a gravitational constant, g_c, acts in the Z direction, the acceleration in the Z direction becomes:

$$a_z = \frac{\frac{\pi}{2} r_p^2 \rho_f (u_{f_z} - u_{p_z}) \|(u_f - u_p)\| C_D}{m_p} - g_c(1 - \frac{\rho_f}{\rho_p})$$

where ρ_p is the density of the particle.

Specific Heat

If particle temperature is being calculated, enter the specific heat per unit mass of the particles, in units of energy per mass per degree.

Specify/Calculate Nusselt Number

The Nusselt number is a non-dimensional measure of heat transfer. The temperature change of the particle is calculated from this number using the following formula:

$$\frac{dT_p}{dt} = \frac{-Q}{m_p c_p} = \frac{2\pi r_p \kappa_f Nu (T_f - T_p)}{m_p c_p}$$

where κ_f is the conductivity of the fluid.

If you specify a constant Nusselt number, enter its value in the text field. Otherwise, Tecplot 360 will calculate it using a formula from *An Eulerian-Lagrangian Analysis for Rocket Motor Internal Flows* (Jayant S. Sabnis, et al., 1989):

$$Nu = 2 + 0.53(Re)^{0.5} \quad \text{for} \quad Re \leq 278.2$$

$$Nu = 0.37(Re)^{0.6} \quad \text{for} \quad Re > 278.2$$

Termination Options

When solving for particle temperature, you may terminate particles when they reach a specified temperature, or calculate particle ablation (mass reduction due to off-gassing or some sort of sloughing of material from the particle).

Terminate/Ablate Particles

If you elect to terminate the particles at a particular temperature, you must enter the temperature. When the particle reaches this temperature, its path will be terminated at that location. If you elect ablation, you must enter the temperature at which ablation begins, and the latent heat of the ablative process. If you wish to model boiling of initially solid particles, enter the latent heat of fusion plus the latent heat of vaporization, as a positive number. Once the particle reaches the specified temperature, any additional heat transferred to the particle will result in ablation instead of an additional temperature rise. If the particle's mass reaches zero, it will be terminated at that location.

Temperature

For temperature-based termination, this is the temperature in absolute units at which the particle will be terminated. For ablation, this is the temperature at which the ablation begins.

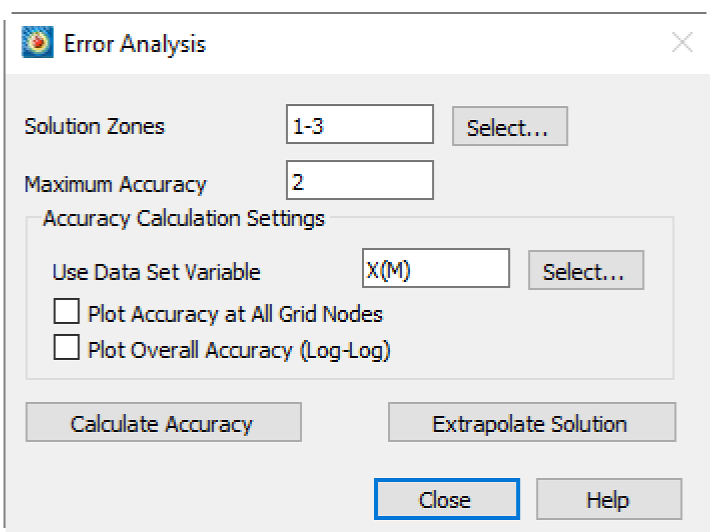
Latent Heat

This is the combined latent heat of fusion and vaporization for the particle, used only for particle ablation. Its units are energy per unit mass.

Analyzing Solution Error

Tecplot 360 allows you to examine a sequence of CFD solutions on successively finer meshes, estimate the order of accuracy of the solutions, as well as perform Richardson extrapolation to improve the accuracy of the solutions. These features are applicable only to smooth solutions (solutions with no discontinuities). They are available via the **Error Analysis** dialog.

The **Error Analysis** dialog is displayed by selecting **Analyze Error** from the **Analyze** menu.



It contains controls for specifying the solution zones to analyze, the maximum accuracy of your CFD solver, some options specific to accuracy calculation, and buttons to perform the analyses.

> You cannot perform error analysis on polygonal or polyhedral zones.

Calculating Solution Accuracy

The accuracy of a sequence of three solutions is estimated using Richardson extrapolation on a particular dataset variable you select. The resulting accuracy in both the 1-norm and the Max- (infinity-) norm is reported in a text dialog. You also have the options of plotting the overall error versus grid spacing, or plotting the calculated accuracy at each grid node.

Selecting Solution Zones

To calculate solution accuracy, you must identify three zones from your dataset. The zones must represent coarse, medium, and fine grid solutions of the same problem. The order in which you enter the zone numbers does not matter. The medium grid must have twice the number of cells in each index direction as the coarse grid, or twice, plus one, the number of nodes. The fine grid must have four times the number of cells, or four times, plus one, the number of nodes as the coarse grid.

Since finite element zones do not have identifiable index directions, the requirement for the coarse, medium and fine grid sizes is only in terms of the total number of cells. It is assumed that successively finer grids have been refined equally in all directions. The requirement is that the medium grid have eight times as many cells (four in 2D) as the coarse grid and the fine grid have 64 times as many cells (sixteen in 2D).

For all zone types, the medium and fine grids must have nodes that overlap the coarse grid nodes.

You may type the zone numbers in the text field, or select them by clicking Select and choosing three zones from the resulting list.

Error Analysis is not supported for polygonal or polyhedral zone types.

Specifying the Solver's Maximum Accuracy

Under some circumstances, Richardson extrapolation can report an accuracy in excess of the solver's theoretical maximum accuracy. For this reason, Tecplot 360 limits the accuracy used by this technique to the value you enter in the Maximum Accuracy text field. Although fractional values are allowed in this text field, you should enter the theoretical maximum order of accuracy of your solver as an integer. That is, two for a second-order accurate solver.

Selecting the Dataset Variable

For the accuracy calculation, Tecplot 360 performs Richardson extrapolation on one variable in your dataset. It must not be a grid variable. Enter the name of the variable in the Use Data Set Variable text field, or click Select to choose the variable.

Plotting the Solution Accuracy

You can plot the results of the accuracy calculation in either or both of two ways. First, you can plot the accuracy at each grid node as a contour plot (XY-plot for 1D data) by setting the Plot Accuracy at All Grid Nodes check box. Second, you can plot the overall error as a log-log XY-plot by setting the Plot

Overall Accuracy (log-log) check box. If you select either of these options, new frames will be created to display the plots when you perform the calculation.

The plot of overall accuracy plots the error in the 1-norm and max- (infinity-) norm versus grid spacing for each of the three zones. The grid spacing of the coarse grid zone is taken as unity for this plot. The 1-norm is the average absolute value of the difference between the extrapolated solution and the solutions of the input zones. The max-norm is the maximum absolute value of this difference. Figure 70 shows an example of this plot. The slopes of the two lines represent the accuracy of the solver. A significant difference in the slopes may indicate discontinuities in your solution, or other problems with the calculation.

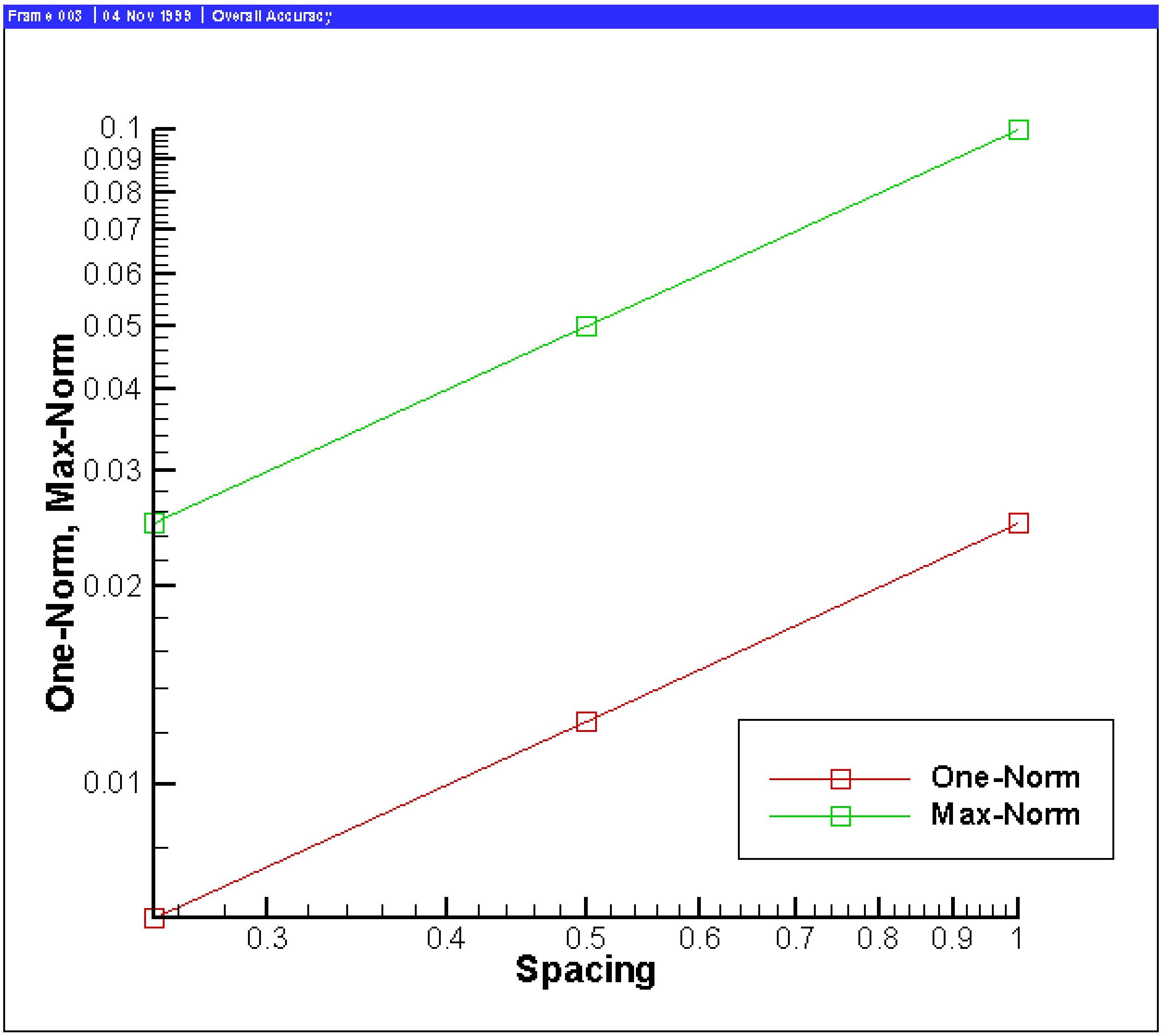


Figure 70. A plot of the overall accuracy.

The plot of accuracy at all grid nodes plots the calculated accuracy on the grid from your coarse solution. For 2D and 3D grids, it is plotted as a contour plot. For 1D solutions, it is plotted as an XY-plot.

Because this feature creates a new frame, it cannot be saved in the data journal, and the current data journal is invalidated. If you subsequently save a layout file, you will be prompted to save a new data file.

Performing the Calculation

When you select [Calculate Accuracy], the accuracy calculation is performed. The accuracy in the 1-norm and max-norm is reported in a text dialog. If you selected either of the plot options, the plots are created in new frames.

Extrapolating a Solution

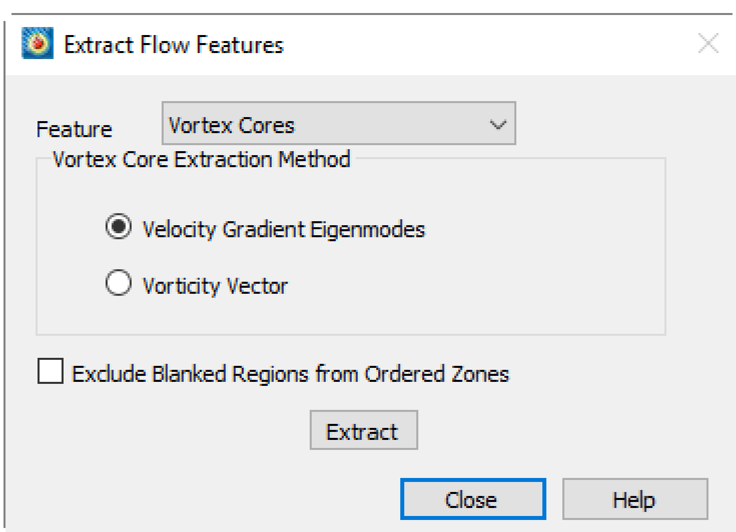
Given three solutions on successively finer grids, Tecplot 360 can perform Richardson extrapolation to improve the accuracy of the solution, and report the difference between the extrapolated solution and the original, fine grid solution.

To perform this extrapolation, three zones must be identified in the **Error Analysis** dialog as previously discussed (see Selecting Solution Zones) and the maximum accuracy of the solver entered (see Specifying the Solver's Maximum Accuracy). Once these are entered, clicking Extrapolate Solution creates two new zones in the solution dataset. The first new zone contains the extrapolated solution on the coarse grid. The second new zone contains the difference between the extrapolated solution and the original fine grid solution.

Extracting Fluid Flow Features

Tecplot 360 can display important features in 3D fluid flow solutions that make analyzing the solutions much easier. For trans-sonic flow, it can display shock surfaces. For all flows, including incompressible flows, it can display lines indicating the location of vortex cores, as well as separation and attachment lines. These calculations make use of MIT's FX library. These features are accessed through the **Extract Flow Features** dialog.

The **Extract Flow Features** dialog is displayed by selecting **Extract Flow Features** from the **Analyze** menu.



It contains a drop-down for selecting the desired feature, options for specifying the algorithm to use when extracting vortex cores, as well as an **Extract** button, which performs the desired task.

Flow features are identified using field variables you have identified on the **Field Variables** dialog. (See Identifying Field Variables) and may be affected by settings on the **Fluid Properties** dialog. (See Specifying Fluid Properties) The feature extraction may also be affected by your boundary settings. In particular, separation and attachment lines are only calculated on boundaries you have identified as wall boundaries. Refer to Setting Geometry and Boundary Options for more information on specifying boundary conditions for your data.

Extracting Shock Surfaces

To extract shock surfaces, select Shock Surfaces from the Feature drop-down, then click Extract. The remaining controls on the dialog are disabled. After calculation, shock surfaces are then displayed as iso-surfaces of a new dataset variable named ShockFeature. This variable is similar to the **Shock** variable available on the **Calculate** dialog.

> Shock Surface values are calculated for the current time step only. The ShockFeature variable will equal zero for all other time steps.

You may note that the displayed shock surface is obscured by clutter due to the sensitivity of the shock function capturing minor oscillations in the solution. A useful technique for displaying only the true shock is to use the value blanking feature to eliminate regions where this clutter appears. Use Tecplot 360's **Calculate** dialog to calculate the Pressure Gradient Magnitude variable, then use the value blanking to blank the plot where this variable is less than some constant. A good value to use is $0.1\rho_\infty c_\infty^2$, or for PLOT3D non-dimensional data, just 0.1.

Extracting Vortex Cores

To extract vortex cores, select Vortex Cores from the Feature drop-down, choose from the two available extraction methods, then click Extract. The cores consist of a group of line segments that may not all be connected. As a result, they are displayed using a line segment finite element zone. Display the Mesh or Edge plot layer to see the new zone.

> the new vortex core zone is a static zone.

If you are using value blanking, you may need to interpolate the blanking variable to the new zone. Refer to Data Interpolation for information on interpolation and Value Blanking for information on value blanking.

Due to the properties of the algorithm used, vortices that happen to exactly align with grid lines may not be properly extracted. This is unlikely to occur in real-world solutions, but is common in test data generated by extruding 2D solutions to produce artificial 3D solutions.

Choosing a Vortex Core Extraction Method

Two algorithms for determining the location of the vortex cores are available. These methods are represented by the Vorticity Vector and Velocity Gradient Eigenmodes options. The Vorticity Vector method determines the location of vortex cores by examining the vorticity vector. The Velocity

Gradient Eigenmodes method is more sophisticated and a little more expensive, using the eigenvalues and eigenvectors of the velocity gradient tensor. The eigenmode method tends to give fewer spurious vortex cores.

Visualizing the Vortex Core Strength

If you have chosen a contour variable for your dataset, the vortex strength returned by the FX library will be stored in this variable in the new zone. You may visualize this vortex strength by turning on the Mesh plot layer and choosing to color the mesh of this zone with the contour variable. You may need to modify the contour levels to get an acceptable display of the vortex strength. You may also wish to use the value blanking feature using this variable to blank out the vortex cores where they are very weak or unrealistically menu (as can happen at a no-slip wall boundary).

Extracting Separation and Attachment Lines

Separation and attachment lines show where a fluid flow separates from or reattaches to a no-slip wall boundary. These lines can give you an indication of where separation bubbles or recirculation regions appear in your data. To calculate them you must first identify one or more Wall boundaries using the **Geometry and Boundaries** dialog. (See Setting Geometry and Boundary Options.) The separation and attachment lines will be calculated on these boundaries.

Due to the algorithm used by the FX library to detect separation and attachment lines, these lines may not be detected for flows that are essentially two-dimensional. (That is, flows which contain no variation along one of the three spatial dimensions.)

To calculate separation and attachment lines, select this option in the Feature drop-down and click Extract. The lines, if any, will be displayed in new static zones, one zone for separations lines and a separate zone for attachment lines. As with vortex cores, the lines consist of sets of possibly unconnected line segments, which are displayed using line segment finite element zones. Display the Mesh or Edge layer to see the lines.

Excluding Blanked Regions

For vortex core and separation/attachment line calculations in ordered zones, you may choose to exclude blanked regions from the calculation. Select this option by selecting the Exclude Blanked Regions from Ordered Zones toggle. This will prevent lines from being calculated in regions of ordered zones that are not plotted due to blanking. Note, however, that this will invalidate the data journal. If you subsequently save a layout file, you will be prompted to save a new data file as well.

Probing Plots

The **Probe** tool allows you to select a location in your plot and view the values of all variables at that location in the Probe sidebar. You can also view information about the dataset itself while probing. With the **Probe At** dialog (accessed via **Data**→**Probe At**), you can specify the location of the probe as a 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** sidebar.

When you probe with the mouse, you can probe in either of two modes: **Interpolate** or **Nearest Point**. In **Interpolate** mode (accessed by a single mouse click) the value returned is the linearly interpolated value for the specified locations. In Nearest Point mode, accessed by Control-click, the value returned is the exact value at the closest data point in the field.

> If Tecplot 360 appears to be unloading variables that you are trying to Probe, you may need to adjust your memory threshold in **Options**→**Performance**→**Misc**. Refer to Load On Demand for additional information.

Field Plot Probing with the Mouse

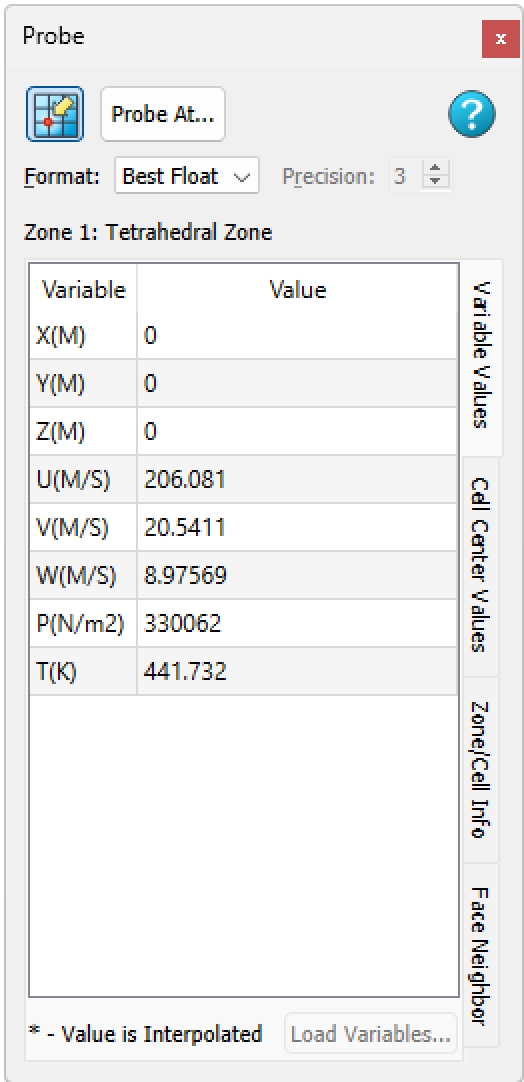
The most direct method of probing is to use the **Probe** tool. Click at any point in your plot to probe in **Interpolate** mode, which displays the Probe sidebar with information interpolated for that point. Control-click at any location to probe in **Nearest Point** mode, which will obtain probe information for the data point closest to the crosshair cursor and display it in the Probe sidebar.

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, the value returned is the interpolated field values from all nodes in the cell. 2D Cartesian plots - If multiple cells are candidates, the cell from the highest-numbered zone is used. 3D Cartesian plots - The closest cell in a zone, slice, iso-surface or streamtrace is selected. If multiple cells are candidates, the cell closest to the viewer is used, with priority given to surfaces drawn with mesh, flooded contours, or shading. Translucent zone surfaces are excluded from probing priority.
Control-Click	If the pointer is over a valid cell, the field value from the nearest node in the cell is returned. If multiple cells are candidates: • 2D Cartesian plots - The cell from the highest number zone is used • 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.

Probe Action	Information Returned
Shift-Control-Click	Return the field values from the nearest point on the screen (ignoring surfaces, zone number or depth of the point). This is useful in 3D for probing on data points that are on the back side of a closed surface without having to rotate the object. In 2D 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 only)	Same as Click except ignore zones while probing. (Probe only on streamtraces, iso-surfaces, or slices.)
Alt-Control-Click	Same as Control-Click except ignore zones while probing. (Probe only on streamtraces, iso-surfaces, or slices.)
Alt-Control-Shift-Click	Same as Shift-Control-Click except ignore zones while probing. (Probe only on streamtraces, iso-surfaces, or slices.)

The probe results are displayed in the **Probe** sidebar. To copy data from the Probe sidebar, select the range of cells to be copied by dragging (or by clicking a start position, then Shift-clicking an end position), then right-click and choose Copy from the context menu.



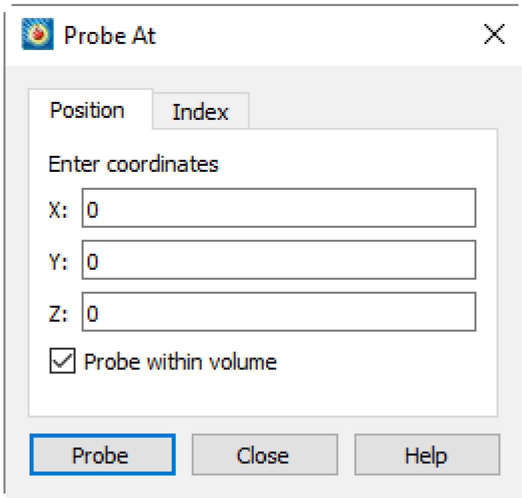
Interpolate mode does not work for I-ordered data displayed in a 2D or 3D Cartesian plot; if you probe such data you will always get the error message "Point is outside of data field," because Tecplot 360 cannot interpolate without a field mesh structure. You should instead use the Nearest Point mode in such situations.

Field Plot Probing by Specifying Coordinates and Indices

Use the **Probe At** dialog for precise control over your probe location, probing using I, J, and K-indices, or probing inside a 3D volume. You can launch the **Probe At** dialog from the **Data** menu.

Probe at Position

To probe at a specified location using spatial coordinates (in Interpolate mode), launch the **Probe At** dialog via the **Data** menu or the **Probe** sidebar.



The **Position** page of the **Probe At** dialog has the following options:

Enter Coordinates

Enter the X, Y, and Z-coordinates of the desired probe location.

Probe Within Volume [DEFAULT]

If the zone you are probing is a 3D volume zone, toggle-on **Probe Within Volume** to ensure that the probe is performed at the indicated point. If you specify a position within a 3D volume zone and the **Probe Within Volume** is not selected, Tecplot 360 probes at the surface of the zone nearest to the user.

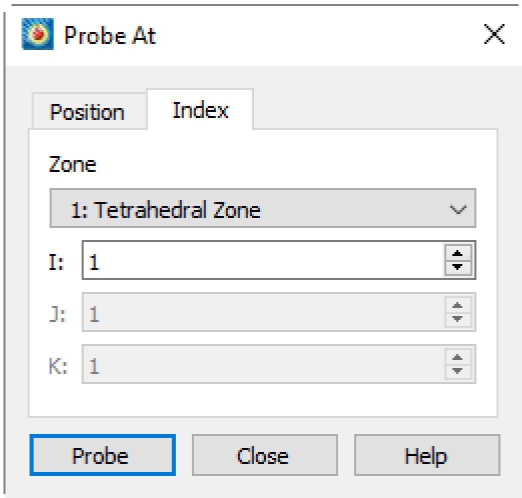
Probe

Select the [Probe] button to perform the probe. The **Probe** sidebar will display the interpolated values for the specified location.

> If the entered location is not within a cell, Tecplot 360 will return the closest surface to the entered location along the line-of-sight ray.

Probe at Index

To probe at a specified location using dataset indices (in **Nearest Point** mode), launch the **Probe At** dialog (accessed via the **Data** menu or the Probe sidebar) and select the **Index** page (shown below).



The **Index** page of the **Probe At** dialog has the following options:

Mapping/Zone

Select the desired zone or mapping from the drop-down.

I, J, K

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.)

Probe

Select the [Probe] button to perform the probe. The **Probe** sidebar will display interpolated values for the specified location.

Probe Sidebar

You can view probed data in the **Probe** sidebar, which appears automatically when probe results need to be displayed. It can also be opened from the Data menu.

The Probe sidebar initially appears docked on the right side of the workspace, but it can be docked to the left side as well, or combined with other sidebars. It may also be "torn off" from the workspace and dragged anywhere on any of your displays.

You can choose what the **Probe** sidebar displays via the tabs along its side. (The tabs may appear on either the right or left side of the sidebar depending on where it is docked.) Available tabs are:

Variable Values

Examine values of all variables at any selected location.

Cell Center Values

Examine values of all variables at the center of the clicked-on cell.

Zone and Cell Information

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).

Face Neighbor

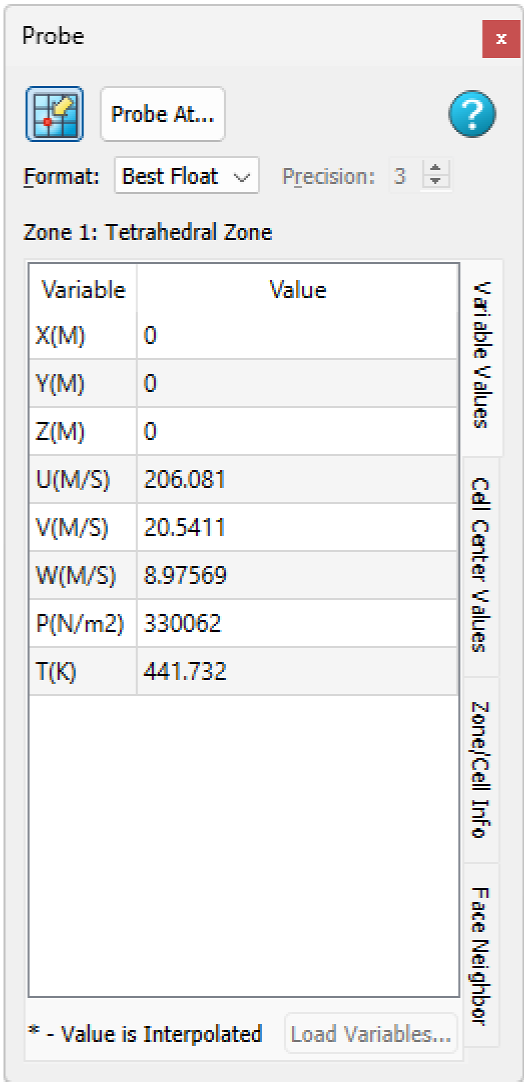
Examine neighboring cells of the clicked-on cell.

The display formatting of variable values can be selected in the **Variable Values** or **Cell Center Values** tabs. Select Integer, Float, Exponent, or Best Float from the Format drop-down menu. Floating-point precision can be specified in the Precision field when either the Float or Exponent format is selected. These options do not affect the actual data, only how it is displayed.

To copy data from the Probe sidebar, select the range of cells to be copied by dragging (or by clicking a start position, then Shift-clicking an end position), then right-click and choose Copy from the context menu.

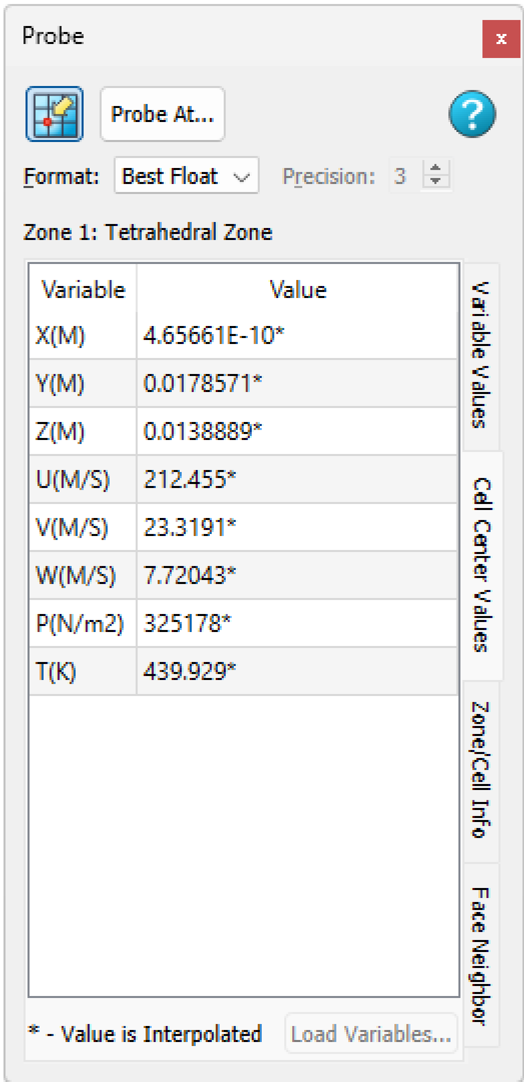
Variable Values

Choosing the **Variable Values** tab in the **Probe** sidebar lists every variable in the current dataset, together with its value at the specified probe point. The zone name and number and the current solution time (if the data set contains transient data) also appear.



The **Probe At** button opens the Probe At dialog (see Field Plot Probing by Specifying Coordinates and Indices). Click the **Load Variables** button to open the **Load Variable** dialog and load any variables you wish to view when probing.

Cell Center Values

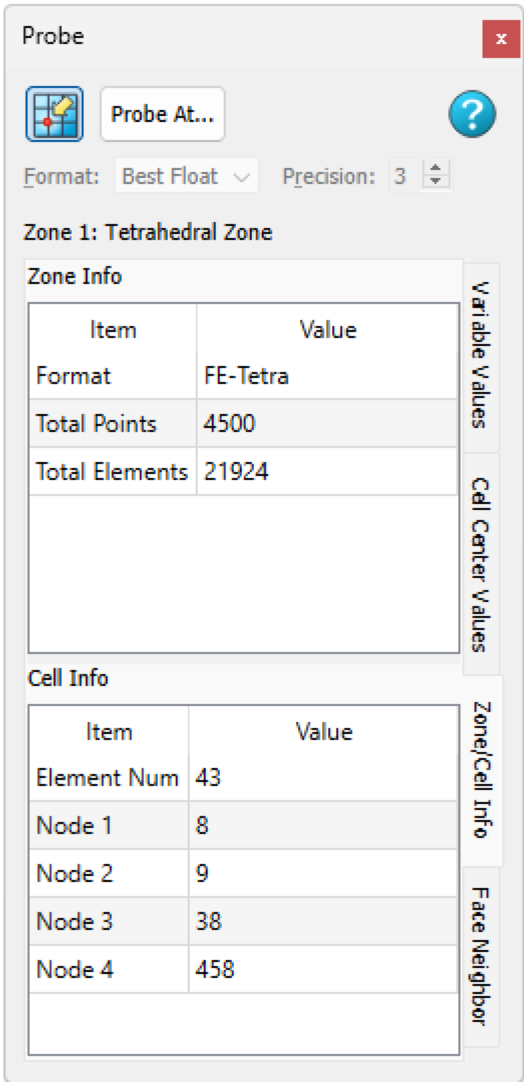


Choosing the **Cell Center** tab in the **Probe** sidebar lists the value of every variable in the current dataset at the center of the cell that was selected. The Zone name and number and the current solution time (if the data set contains transient data) are also displayed.

The **Probe At** button opens the Probe At dialog (see Field Plot Probing by Specifying Coordinates and Indices). Click the **Load Variables** button to open the **Load Variable** dialog and load any variables you wish to view when probing.

Zone and Cell Information

Choosing the **Zone/Cell Info** tab in the **Probe** sidebar lists the following information about any probed data point, regardless of the format of the data.



The following information appears:

- 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 or FE-Brick).
- The Current Solution time of the probed point.

For ordered zones, the following additional information is displayed:

- **I-Max** - Maximum I-index of the zone.
- **J-Max** - Maximum J-index of the zone.(J-Max is one for I-ordered data.)
- **K-Max** - Maximum K-index of the zone. (K-Max is one for IJ-ordered data).
- **Plane** - Shows the type of plane. I, J, or K displays the index of the point at the principal data point of the cell containing the probed point. (If the point is probed using Control-click for **Nearest Point**, the label reads "I,J or K-Index.")
- **Face Plane** - The I, J, or K-plane that is probed.

- **Face Indices** - The planes that are not mentioned in Face Plane, these are the other faces that are showing in 3D, or are the axes in 2D.

> Index values are not displayed for subzone data sets `(.szplt)` since these values cannot currently be reliably determined with this file format.

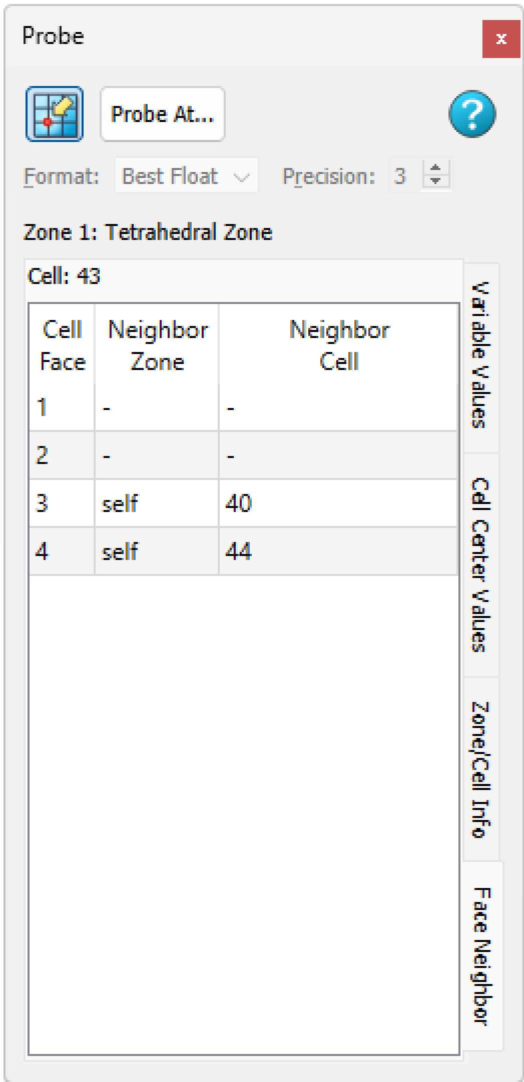
For cell-based finite element zones (FE-Triangle, FE-Quad, FE-Tetra or FE-Brick), the following additional information is displayed:

- **Total Pts** - Total number of points in the zone.
- **Total Elems** - Total number of elements (cells) in the zone.
- **Node Num** - Number of the probed node. This field is filled in only if the point is probed using Control-click for Nearest Point.
- **Elem Num** - Number of the probed element.
- **Node 1 - 8** - Number of the node defining Node 1-8 of the cell.
- **Node 4** - FE-Quad, FE-Tetra, and FE-Brick only.
- **Node 5-8** - FE-Brick only.

The **Probe At** button opens the Probe At dialog (see Field Plot Probing by Specifying Coordinates and Indices).

Face Neighbor

Choosing **the Face Neighbor** tab in the **Probe** sidebar displays cells that neighbor the selected cell.



The **Probe At** button opens the Probe At dialog (see Field Plot Probing by Specifying Coordinates and Indices).

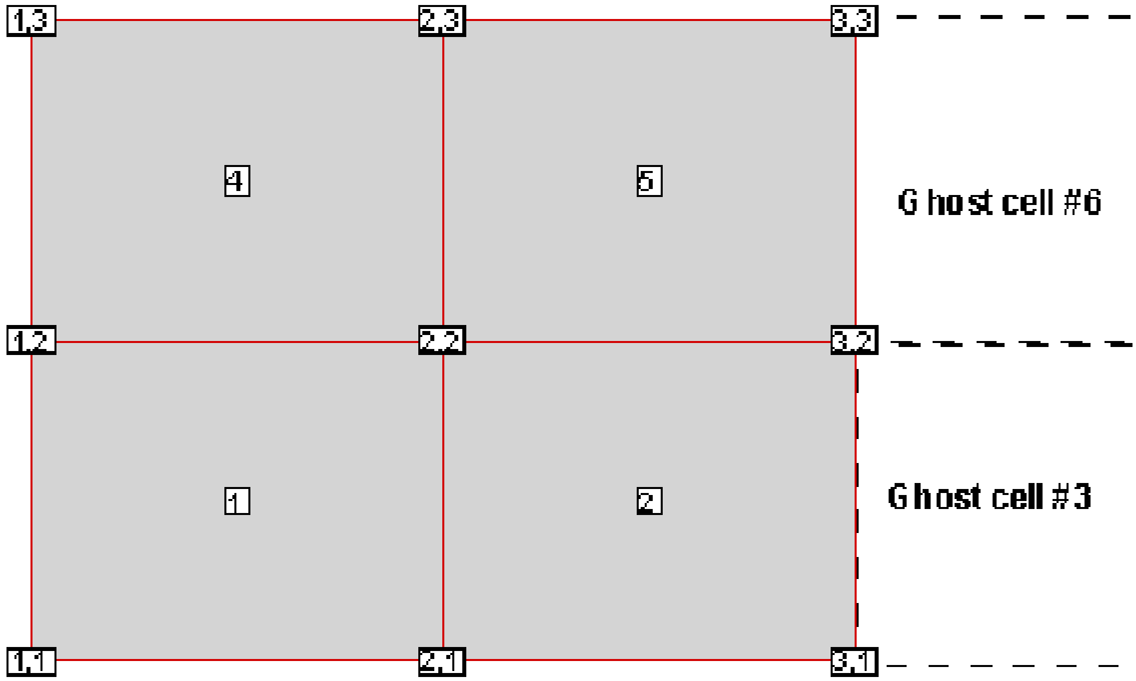
A cell is considered a neighbor if one of its faces shares all nodes in common with the selected cell, or if it is identified as a neighbor by face neighbor data in the dataset. The current solution time is also displayed.

For ordered (IJ or IJK) zones, cell numbers are defined by the index value of the first node

$$Index = i + (j-1)I_{max} + (k-1)I_{max}J_{max}$$

where, i, j, and k are the i, j, and k values for the location of the first node.

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).



For FE zones, the cells are numbered in the order that they appear in the connectivity list. In the following example "7 8 19 11" is the cell number 2:

```
Example connectivity list of 3 cells:
2 4 8 19
7 8 19 11
1 2 4 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 360 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).

To probe in interpolate mode: activate the probe tool and click anywhere on your plot. Axis variable values of all active mappings that lie along the probe line are interpolated and displayed in the Probe sidebar.

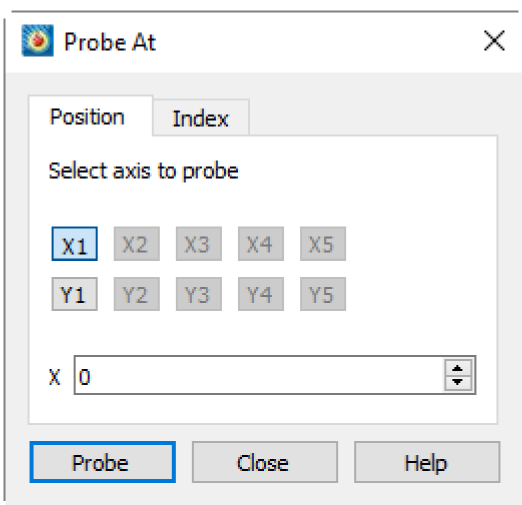
To probe in **Nearest Point** mode, activate the probe tool and Control-click anywhere on your plot. When you Control-click, Tecplot 360 displays the exact X and Y, or Theta and R-values of the data point closest to the location clicked.

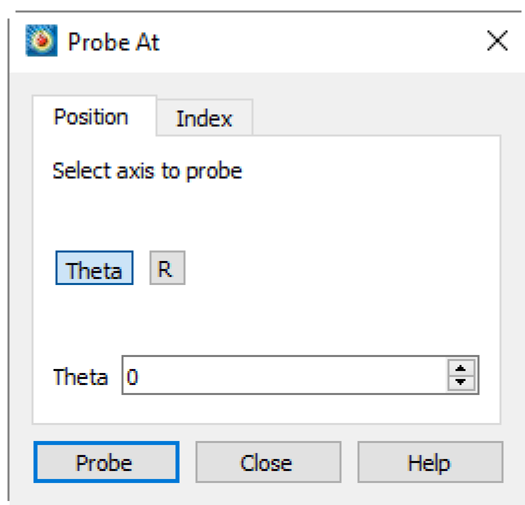
Line Plot Probing in Interpolate Mode

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 360's five X-axes, or along any of Tecplot 360'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.

In Polar Line plots, 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 360 uses the Theta-value within the current Theta-axis range to determine the corresponding R-values reported in the Probe sidebar. This behavior may 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 360 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, choose Probe At from the Data menu. the **Probe At** dialogs for XY and Polar Line plots are shown below:





Select the button corresponding to the axis you want to probe along, then enter the position. Click **Probe** to perform the probe.

The **Probe** sidebar appears if it is not already visible, For interpolated values, the **Probe** sidebar 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 Probe sidebar.

In the **Probe** sidebar, 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.

X-Value, Y-value

X, Y-value of the nearest data point to the probe position.

I, J, of K-Index

I, J or K-index of the nearest data point to the probe position.

Map

Number and name of the nearest map to the probe position.

Zone

Number and name of the nearest zone to the probe position.

I, J, or K-Max

Maximum I, J or K-index of the current zone.

X or Y-Axis

X or Y-axis associated with the current map.

> Index values are not displayed for subzone data sets `(.szplt)` since these values cannot currently be reliably determined with this file format.

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 select the **Probe** tool, from the Toolbar and Control-click at the desired probe location. The nearest point is calculated from the actual location of the cross-hair and is independent of the axis you were probing along.

The following information about the nearest data point is displayed in the Probe sidebar:

- X or Theta-value.
- Y or R-value.
- I, J, or 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, J, and K-indices of the zone.
- For XY Line plots, the X-axis and Y-axis associated with the mapping.

Data Editing

Using the **Adjustor** tool, you can actually modify the coordinates of your data with the mouse. See Adjustor Tool for more information.

> If you attempt to double-click, but move the mouse between clicks, you may find that you have moved your data point.

Part 5: Final Output

Output

Tecplot 360 provides a variety of formats for you to output and export your complete plots. This chapter discusses saving your settings using layout files or stylesheets, preparing plots for web publishing, and writing data files to a file.

For information on exporting or printing your completed plot(s), please refer to: Exporting Plots or Printing, respectively.

Layout Files, Layout Package Files, Stylesheets

Tecplot 360 has three different types of files for storing plot information:

Stylesheets (`.sty`)
: Stylesheets store information about a single frame and do not include any information about the data used by the frame.

Layout Files (`.lay`)
: Layout files store information about all the frames in the workspace, including identification of, and links to, the data used by each frame.

Layout Package Files (`.lpk`)
: Layout package files are an extension to layout files where data and an optional preview image are included.

Layout and layout package files are the preferred method for saving the style of your plot. They save a complete picture of the workspace and are quick-and-easy to load and save. Stylesheets contain the style of a single frame in Tecplot 360.

Creating Layouts and Stylesheets for Tecplot Focus

If you are working with stylesheets or layouts in Tecplot 360 that you wish to share with Tecplot Focus users, please be aware that the following features **are not available in Tecplot Focus**:

- Any aspect of the CFD Analyzer (i.e. options available from the **Analyze** menu in Tecplot 360) including integration
- Data files loaded using a loader not available in Tecplot Focus
- Data files that contain polygonal or polyhedral zones or more than 5 million data points.
- Versions of Tecplot Focus prior to Tecplot Focus 2016 R1 do not support more than one slice group or more than one contour group. This limitation does not apply to newer versions.

If any of the above items are included in your stylesheet or layout file, it will not load in Tecplot Focus.

However, stylesheets and layouts created by Tecplot Focus will load in either product.

Working with Layout Files from Previous Releases

If you are working with layout files from previous releases of Tecplot 360, Tecplot Focus, or their precursor called simply Tecplot, please note the following:

Zone Numbers

Due to changes in a third-party library used by the FEA Loader, which is used for several data formats, zones may load in a different order in than they were saved. If this happens, a warning will alert the change in zone order. If you have any layouts that refer to zones in an FEA data set, you should double-check to make sure that these still refer to the desired zone.

Exporting Backward Compatible Data Files

While Tecplot 360 layouts and layout package files are not backward compatible, you do have the option to export data files written in one version of Tecplot 360 to be read by an earlier version. To do this, use the **Write Data Options** dialog as described in Data File Writing.

Stylesheets

Stylesheets are useful when:

- Pre-processing must be done to a dataset prior to attaching a style. You may need to load a dataset 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.

  > Tecplot 360's data journaling capabilities together with layout files eliminate this situation in many cases.

- Switching styles on large datasets. You may want to load a large dataset 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 dataset.
- Copying the style of one frame to another frame in the same layout.
- Saving just part of a frame's style, such as just the contour levels.

A stylesheet includes the following attributes (see Figure 71):

- Type of plot (a 2D contour plot in 2D or an XY Line plot)
- Colors used
- Current view of the data
- Axes display
- Text and geometry

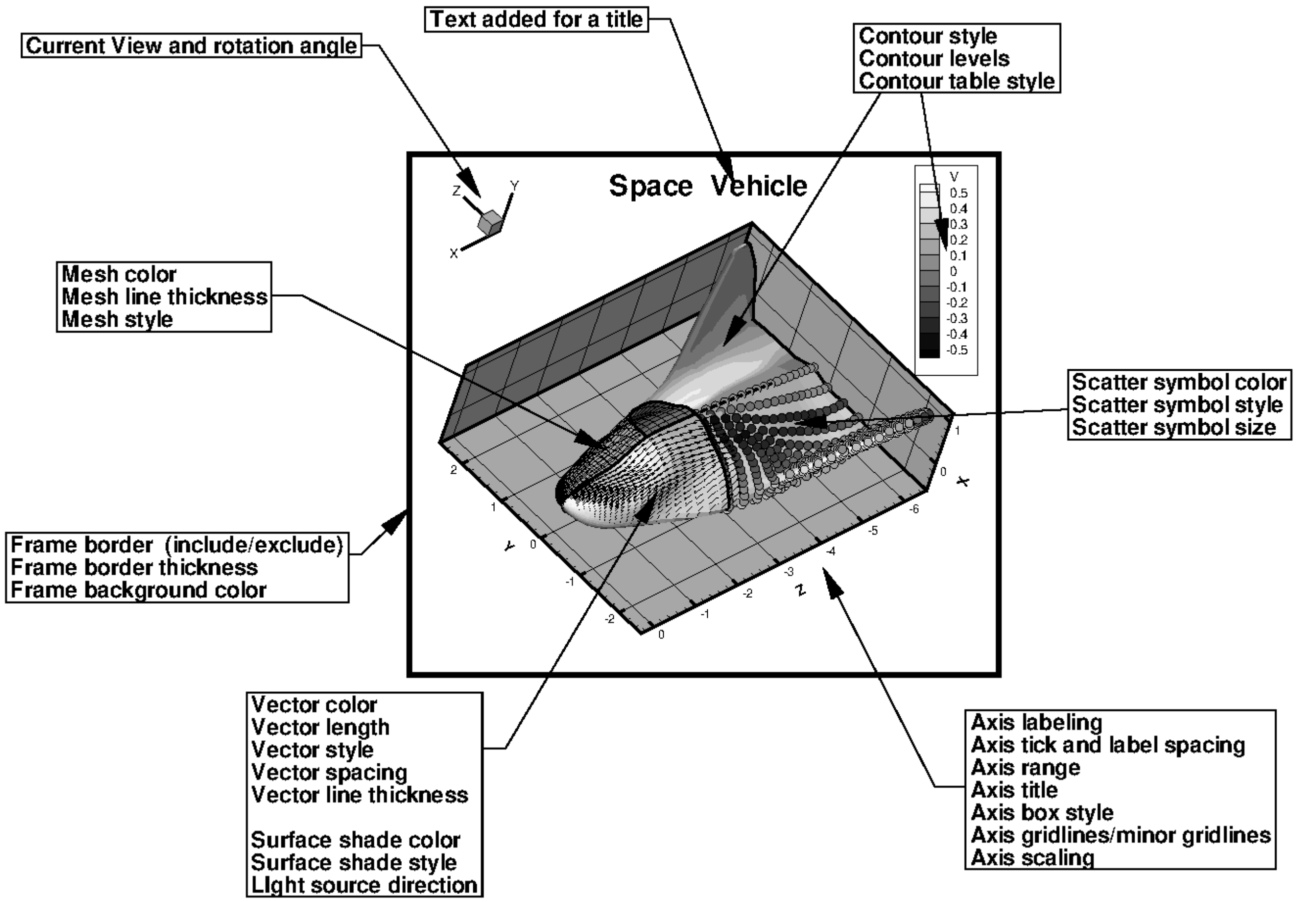


Figure 71. Some of the items considered part of the frame style.

To learn how to use a stylesheet, see Save Frame Style and Load Frame Style.

Layout Files

A plot often consists of multiple frames or even multiple datasets. Layout files allow you to capture all the information on the plot. Layout files include instructions on how to create the data used in the plot, the frame layout and dataset attachments, axis and plot attributes, the current color map, and so forth.

Figure 72 shows a layout with four frames. The frame in the upper left-hand corner is attached to dataset 1. The two frames on the right are both attached to dataset 2. The frame in the lower left is not attached to a dataset.

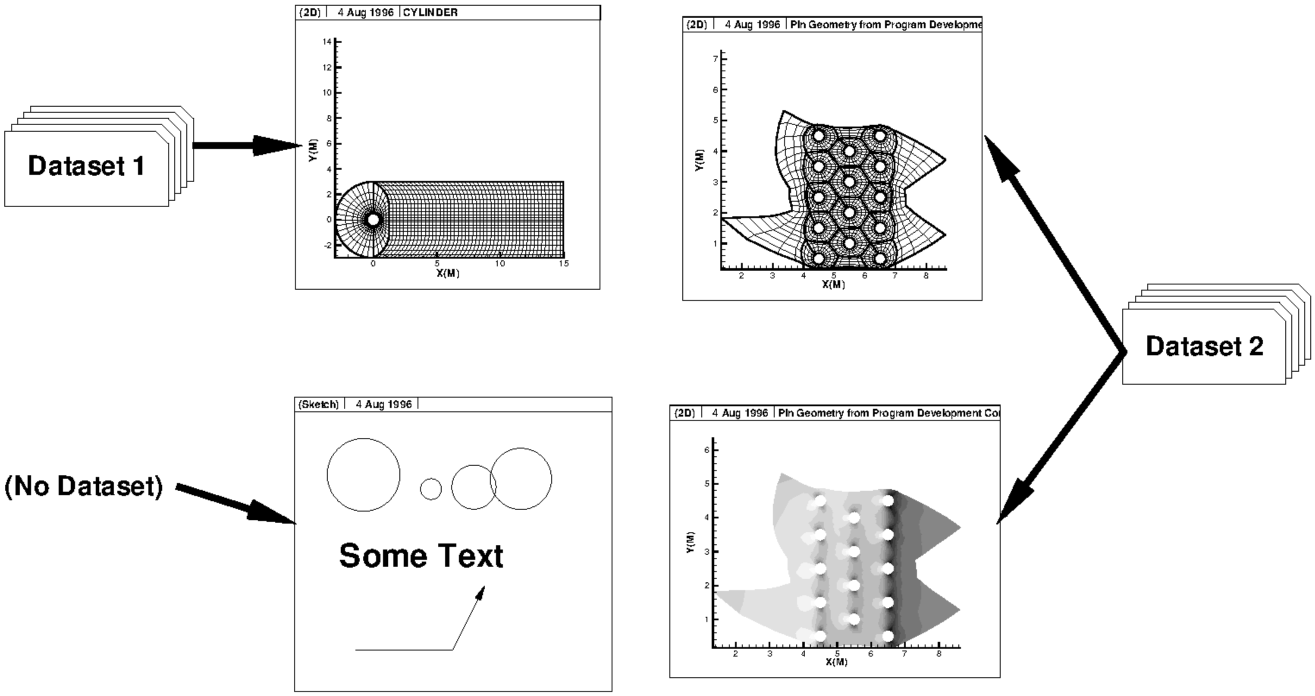


Figure 72. Layout of four frames using two datasets.

If a frame defined in a layout file requires an attached dataset, the data files necessary to build the dataset are referenced in the layout file. These data files can be referenced using absolute paths or relative paths. When using relative paths on Windows operating systems, 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 360 sessions. Not all data operations are supported by the data journal. For more information, see Data Journaling.

In addition to storing the individual style of each frame, layout files record:

- Page layout information (including the size and orientation of the paper).
- Color spectrum information, including the color maps in use.

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

Layout Package Files

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

Layout package files have the same properties as standard layout files. (See Layout Files). Layout package files also contain all data associated with frames in the layout, and an optional preview image of the Tecplot 360 workspace. An extension of `.lpk` is used.

Working with Layout and Layout Package Files

New Layout

File→New Layout creates a new layout in your workspace after removing any existing frames and resetting the paper setup to the default configuration. Anything not saved before this action will be lost.

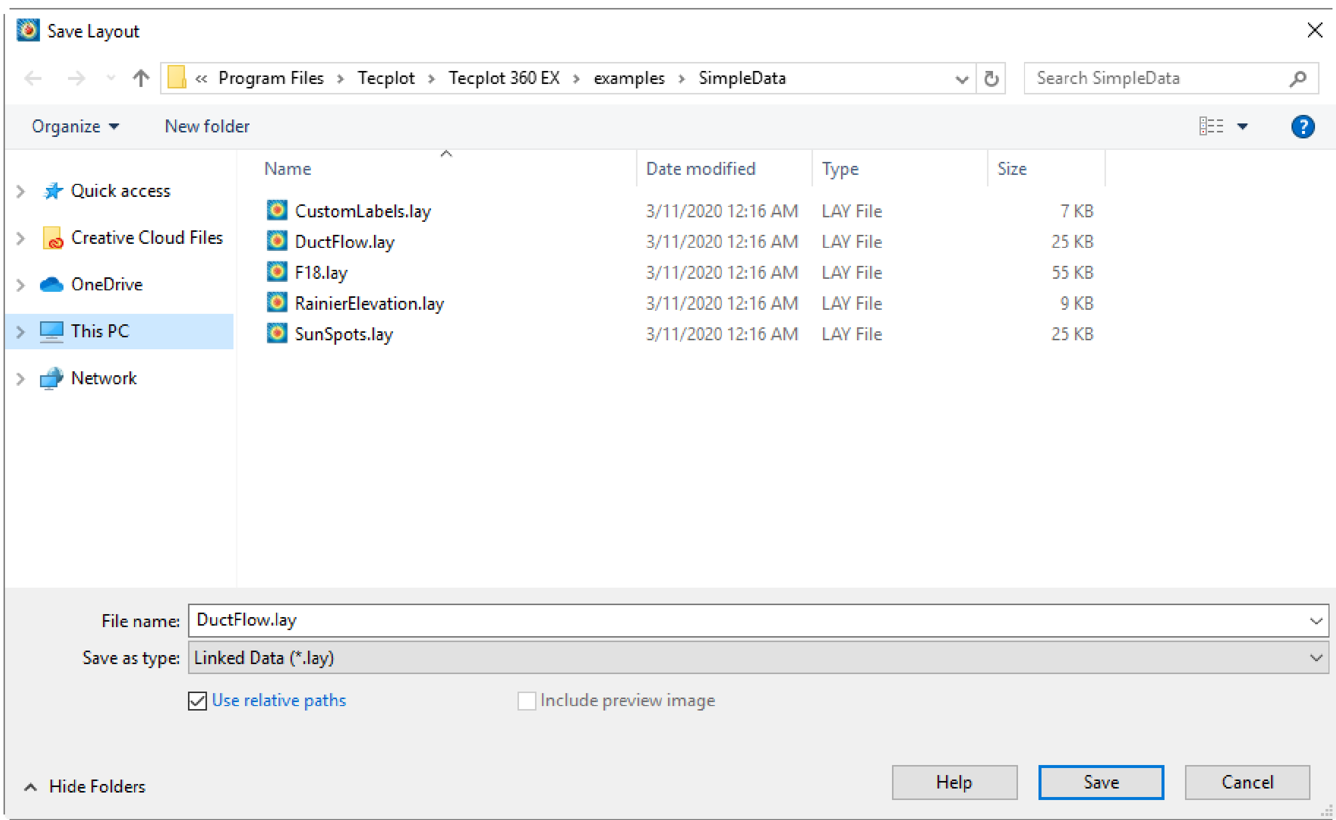
Layout Saving

Save layout files using the **Save Layout (Control-S)** or **Save Layout As (Control-W)** options under the **File** menu.

The **Save Layout** dialog has the following options:

Save As Type

Choose "Linked Data" (`*.lay`) or "Packaged Data" (`*.lpk`)



Use Relative Path (Linked Layout Only)

By default, Tecplot 360 saves the name of the data files used in the layout with their relative file paths. To save your layout using absolute file paths, toggle-off **Use Relative Path**.

Include Preview Image (Layout Package Files Only)

Toggle-on **Include Preview Image** to include a preview image with the file.

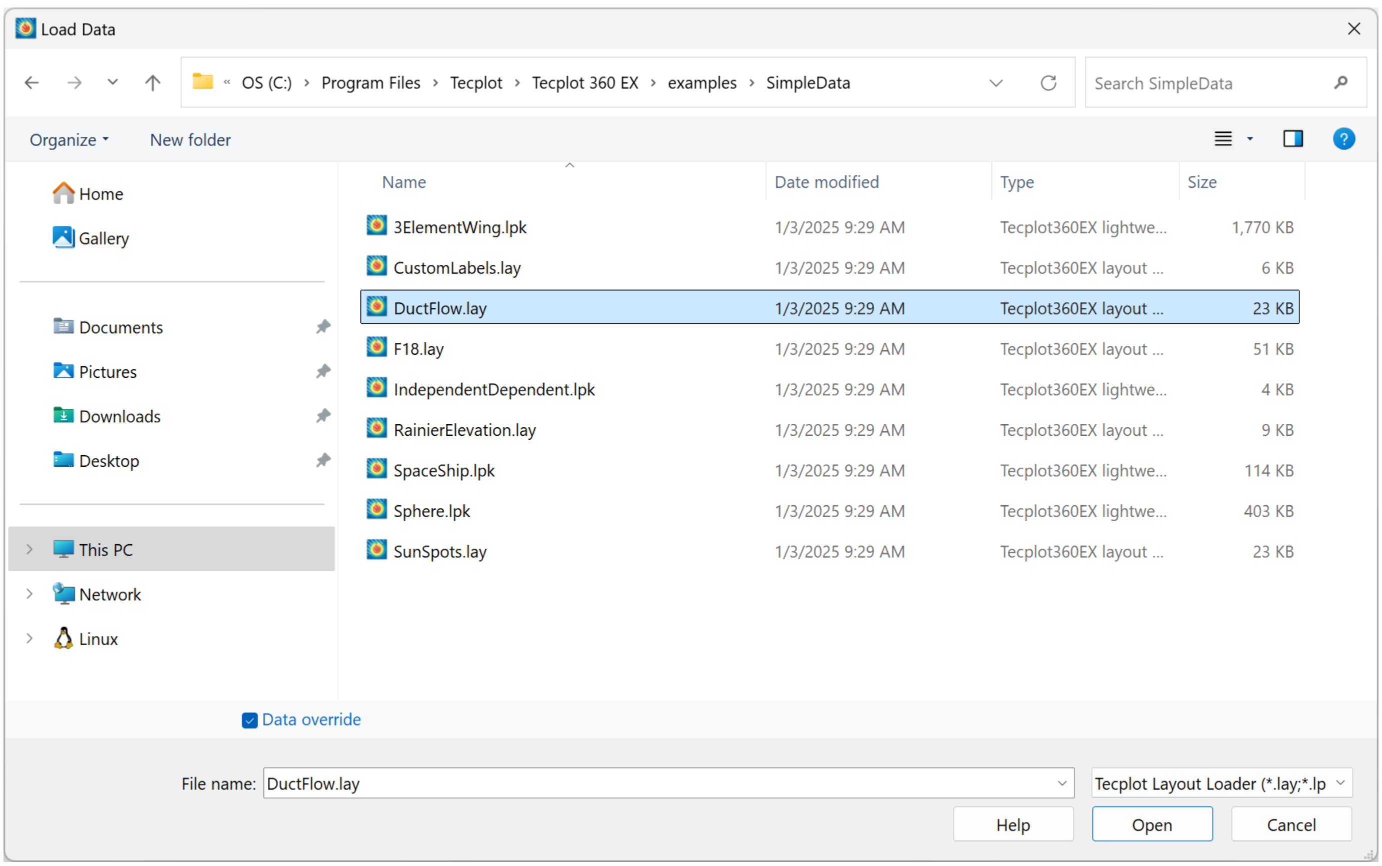
After saving the layout file, you will be asked how to handle any dataset changes. See Dataset Changed for more information.

Dataset Changed

In Tecplot 360, layout files contain references to the data files in use. The datasets are not copied directly into the layout file. Therefore, if you make changes to the dataset using Tecplot 360 and wish to save the layout, Tecplot 360 will ask you whether you want to create new data files reflecting the changes made. If you answer in the affirmative, Tecplot 360 prompts you for a file name under which to save the changed data. (If your layout has multiple datasets, Tecplot 360 prompts you for a file name for each modified dataset.) The new data is then referenced in the layout instead of the original.

Layout File Opening

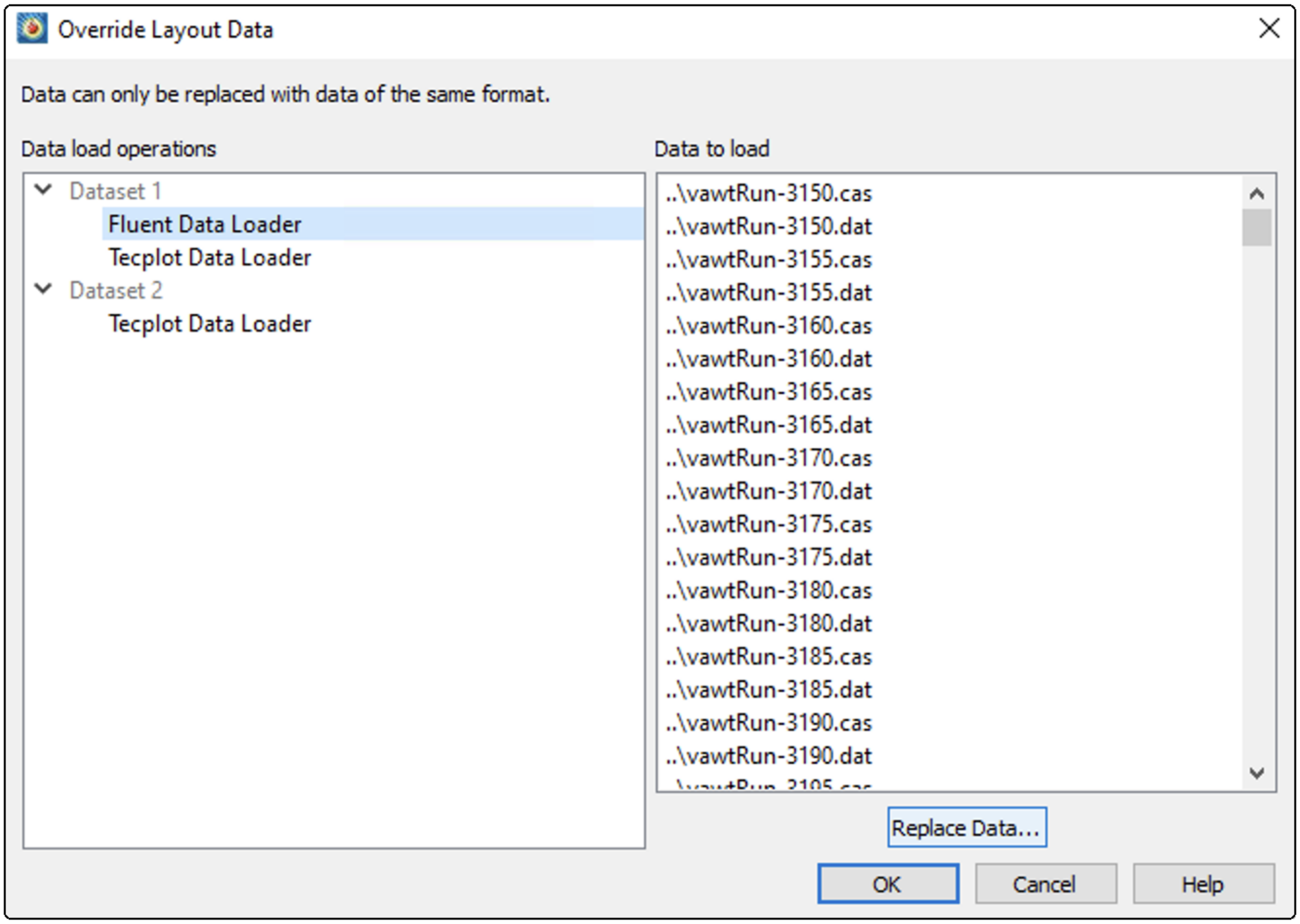
Open layout files using the **Open Layout** (Control-O) option from the **File** menu. You can also access this function by choosing **File**→**Load Data** and selecting Tecplot Layout Loader from the file type menu.



Data Override

When opening a layout you are given the opportunity to override the data source instructions for the layout file. This allows you to apply a given layout to different data. The data overriding capability is supported for Tecplot's native loaders as well as any loaders which contain both the macro command keyword `STANDARDSYNTAX` and keywords beginning with `FILELIST_` or `FILENAME_`. See Using a Loader for a complete list of loaders and their macro command keywords.

To use layout override, toggle-on the **Data override** checkbox within the **Load Data** dialog, then open your desired linked layout (.lay) file. This will open the **Override Layout Data** dialog.



On the left panel, you will see the datasets contained in the selected layout file. When selecting a dataset, you will see the linked files pertaining to that dataset on the right panel. To change the data files associated with a given dataset, highlight the dataset on the left panel and select **Replace Data...** where you can choose new data to supply. New data must be of the same type as the original dataset.

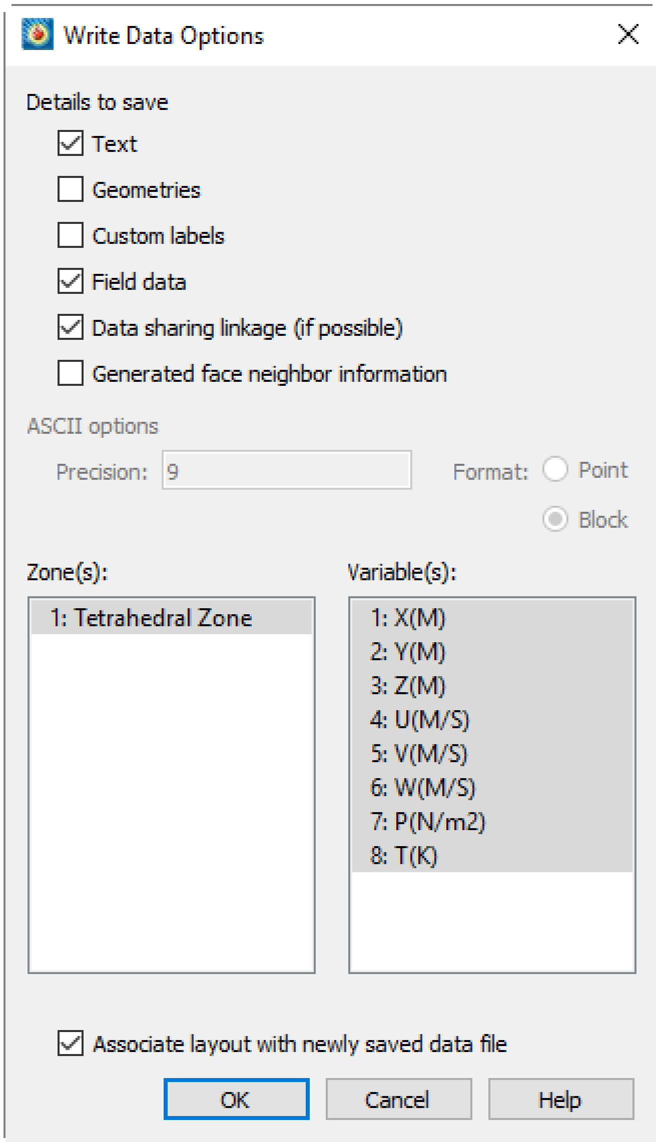
Data File Writing

You can write out the dataset in the active frame in various Tecplot file formats, including backward-compatible binary formats. You must choose which zones and variables of the data to write, as well as the format for the saved file (if applicable).

To write the dataset in the active frame to a file, choose **Write Data** from the **File** menu. The Write Data dialog appears to let you name the file and to choose its format from the following:

- *Tecplot Binary Data Writer (current)* - Use the current Tecplot binary file format, the best choice for small to moderately-sized data sets. Files are saved with the filename extension `.plt.`
- *Tecplot Binary Data Writer (older)* - Several previous versions of the Tecplot binary format are supported if you need to exchange data with users of older version of Tecplot products.

- *Tecplot ASCII Data Writer* - Uses the Tecplot ASCII data format. You will generally not want to use this loader except in very limited circumstances, as it is much slower to load than binary format and does not support load-on-demand. Files are saved with the filename extension `.dat`.
- *Tecplot Subzone Data Writer* - Write data in the Tecplot Subzone (`.szplt`) format. This format is intended for large data files; it loads the least amount of data needed for each operation, and loads data only when needed, improving interactive performance. Files are saved with the filename extension `.szplt`.



When writing data in Tecplot binary or ASCII format, the **Write Data Options** dialog appears with the following options:

Details to Save

Specify which record types you want to include in the data file by selecting the appropriate checkboxes. By default, all record types present in the active frame's dataset are selected. These details include:

Text, Geometries, Custom Labels

Choose extra items to include in the datafile.

Field Data

Select this detail to save zone data.

Data Linkage (If Possible)

Save the variable and connectivity sharing between zones existing in the dataset, reducing its size and loading time.

Generated Face Neighbor Information

Automatically save the face neighbor information generated by Tecplot 360 for finite element zones. This increases the dataset size and loading time, but improves performance after loading.

Zone/Geometry Format

For ASCII, choose to write the file in `POINT` format or `BLOCK` format (`BLOCK` is required if any variables are cell-centered).

Zone(s)/Variable(s)

Select the zones and variables to include.

Associate Layout with Newly Saved Data File

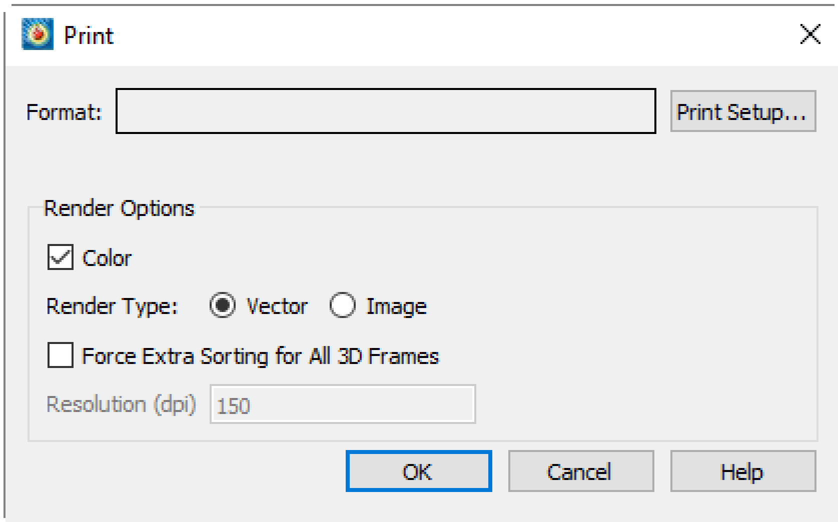
If activated, Tecplot 360 replaces internal references to loaded data with a reference to the new file. When you later save a layout, this layout file will refer to the new file, not to the original data file.

Printing

Printing your plot is the process of sending the plot image to an output device, print spooler, or a file. Typically the output device is a printer, but it may be a plotter, film recorder, file or typesetter. If you are creating files for use in another program, you should use Tecplot 360's **Export** option (accessed via the **File** menu) to create your files—**Export** includes all the supported print file types, as well as several standard graphics formats. See Exporting Plots for complete details.

Plot Printing

To print a plot, select "Print" from the **File** menu.



Tecplot 360 supports the standard printer drivers for the supported operating systems.

The **Print** dialog has the following options:

Format

Indicates the printer chosen to print the plot. You can choose a different printer in the **Print Setup** dialog.

Print Setup

Calls up the Print Setup Dialog in which you can choose a printer and other printing settings.

Color

Select this check box for color output; deselect the check box for monochrome output.

Render Type

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 are not available.

> Vector graphics formats do not support: **Translucency, Contour flooding with Gouraud shading, Contour flooding using the continuous color distribution method**

Image

Select this option to create print output using an image. Rendering is done by Tecplot 360 at the specified resolution, usually less than the printer’s resolution. However, all plot options are available.

> To preserve quality of color in your plot (translucency, contour flooding with Gouraud shading, or continuous contour flooding), select the **Image** render type.
> To preserve the quality of text in your plot, select the **Vector** render type.

Force Extra Sorting for all 3D Frames

This option is available when the Vector option has been selected and overrides the setting in the Advanced 3D Control dialog. If this check box is not selected, Tecplot 360 will choose sorting algorithms based on the advanced 3D options that were chosen for each frame. When printing 3D plots in a vector graphics format, Tecplot 360 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 360 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 360 uses a slower, more accurate approach that detects and resolves such 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. Lower resolutions may yield less-attractive output images.

Control-P is the shortcut command for the **Print** dialog.

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.

Printing Setup for Windows

Print Setup Dialog

On Windows platforms, use the **Print Setup** dialog to set up your paper. The **Print Setup** dialog is accessed via the **Print** dialog (accessed via the **File** menu) and has the following options:

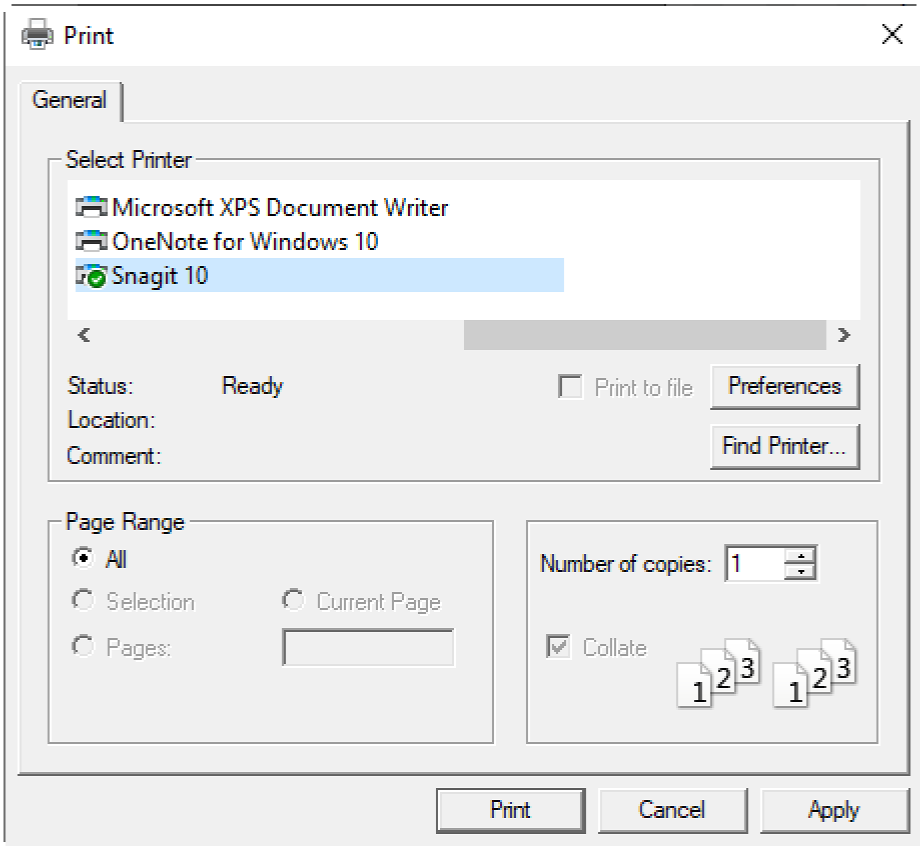


Figure 73. Print Setup Dialog

Printer

Choose the printer and set its properties.

Paper

Specify the paper size and source tray using the following drop-downs:

Size

Select the paper size. The choices are printer-dependent.

Source

Choose a paper tray from the drop-down.

Orientation

Specify one of the following options:

Portrait

The horizontal axis of the plot is aligned with the short side of the paper.

Landscape

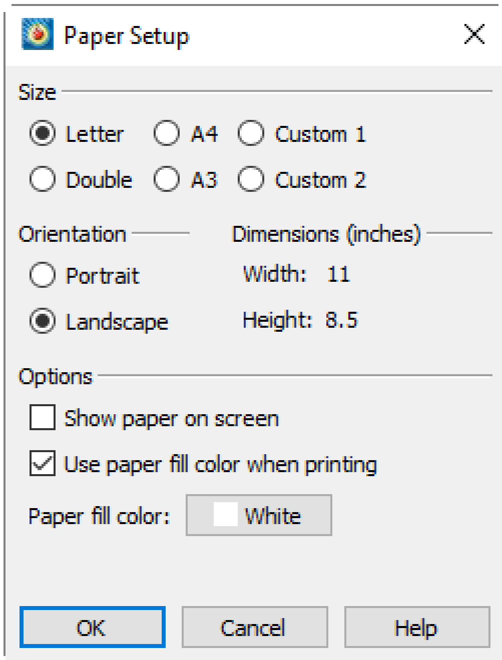
The vertical axis of the plot is aligned with the short side of the paper.

By default, Tecplot 360 uses the standard Windows print drivers. You can choose to use Tecplot's print drivers by adding the following line to your tecplot.cfg file:

```
$!INTERFACE USETECPLOTPRINTDRIVERS=YES
```

If you use Tecplot’s print drivers, you will use the **Print Setup** dialog for subsequent printing.

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 (accessed via the **Options** menu). This check box is selected by default.)

The **Paper Setup** dialog, in contrast with the **Print Setup** dialog on Windows platforms, 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 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).

On Windows systems, paper size **Custom 2** is overwritten with the size selected in **Print Setup** if that size does not exist in Tecplot 360.

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

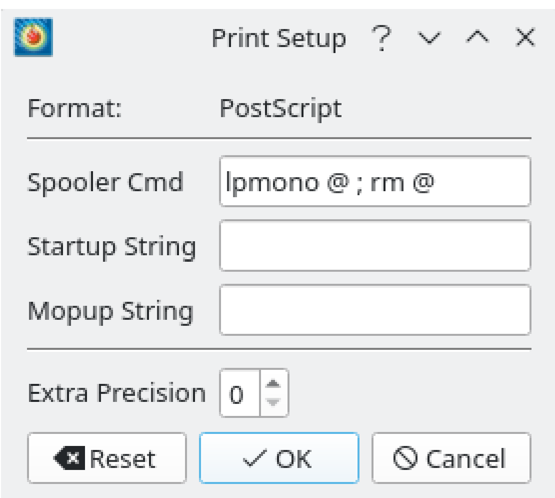
Choose the paper orientation. You have two options: Portrait and Landscape. In Portrait orientation, the long axis of the paper is aligned with the vertical axis of the plot. In Landscape orientation, the long axis of the paper is aligned with the horizontal axis of the plot.

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 360 print this background color on the hard-copy as well.

Printing Setup for Linux

The Print Setup dialog on Linux operating systems includes a field for entering a spool command, along with startup and mopup strings. Contact your system administrator if you have any questions about how to set up these commands.



Spool Command

Printers on Linux systems are accessed via print spoolers that manage the print queue. Typically you 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.

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 360 creates temporary files with names of the form `tp??????`, where the `?'s` are randomly generated characters. Tecplot 360 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.

Startup and Mopup Strings

A startup string is an initialization string that sets up your output device to accept the plot created by Tecplot 360. 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.

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.

Extra Precision

For PostScript 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.The maximum setting for the precision is eight.

Exporting Plots

The **Export** dialog, accessible from the **File** menu, can be used to create files for export to other applications. Tecplot 360 can export files in variations of three types: vector graphics, image, and movie file formats.

Tecplot 360 exports the following **vector** formats:

EPS Export

Vector or image graphics in a special type of PostScript file designed for inclusion in other applications.

PostScript (PS) Export

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.

WMF Export

Vector graphics to import into various Windows applications.

Tecplot 360 exports the following **image** formats:

BMP Export

Image in Windows Bitmap format.

JPEG Export

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 Export

Also common on the internet, PNG images have a high image quality but larger file size than JPEG.

TIFF Export

Image in Tagged Image File Format.

Tecplot 360 can also export animations to various movie file formats. For more information on exporting animations, see Animation. Tecplot 360 can also export animations to sequenced image files. See Sequenced Image Files for more information.

Certain image formats support anti-aliasing, a feature that smooths jagged edges on text, lines and edges. This feature is discussed at the end of this chapter. See Antialiasing Images.

Tecplot 360 can also export images directly to the clipboard instead of to a file. See Clipboard Exporting to Other Applications for more information.

Performance Tips

If exporting is taking an unusually long time, or you get an error message saying that the image cannot be exported, the most likely cause is that the image width you are trying to export is too large. Selecting a smaller image width should speed up the export process.

For an image export size of Length x Width, the file size for an uncompressed true color image is approximately Length x Width x 3. Memory requirements to export such an image can be up to twice this size.

For 256 color images, the maximum file size is approximately Length x Width, but is usually less since all 256 color image files are compressed. However, the memory requirements for exporting are the same as they are for a true color uncompressed image.

Anti-aliasing can dramatically increase the memory and time requirements of image export, since a large image is rendered first and then downsized to the final image dimensions. Furthermore, some graphics card drivers may not report an error if they cannot allocate enough memory for this operation, instead producing a black image or other rendering issues. See Antialiasing Images.

Layout Packages

As an alternative to exporting image and vector representations of plots, Tecplot 360 supports the Packaged Data layout format, or layout packages. A layout package (exported from Tecplot 360 as an `.lpk` file rather than as a `.lay` file) contains the data needed for the layout, and only that data. You can reload the resulting file into Tecplot 360 for viewing and rotation. The plot will appear visually identical to the original plot, although you cannot manipulate the variables and other plot elements as in the source plot.

By streamlining the elements included in the exported file, the Packaged Data export reduces the size of the resulting LPK to an average of 25% of the original file size (although the reduction varies greatly depending on what the original layout includes).

To export your current (3D Cartesian) plot with the Tecplot Viewer export format, choose **Save Layout As** from the **File** menu. In the dialog that appears, select **Packaged Data** as the format, and click OK.

To reduce the file size, the lightweight file includes only the following.

- Data for only the current surfaces and lines drawn for the current plot
- Derived elements (such as slices and iso-surfaces) as zones
- The variables necessary to draw each zone (For example, if an iso-surface displays a contour flood, only the spatial variables and the contour variable data will be output to the file for that iso-surface's zone)
- Text, geometries, custom labels, and all visible data elements displayed on your plot

The following restrictions apply.

- Only the active frame of a 3D Cartesian plot may be exported
- Only the current time step of a transient data set may be exported
- The plot must include at least one zone, slice, streamtrace, or iso-surface
- Several plot features will not be reflected in the exported plot:
- Finite element volume zones with the Contour or Shade layer enabled and a "Surfaces to Plot" setting other than "None" or "Boundary Faces" will not display.
- Volume zones with the Scatter layer enabled and a "Points to Plot" setting other than "Nodes on Surface" or "Cell Centers Near Surfaces" will not display.
- Finite element volume zones with the Scatter layer enabled and a point index I-Skip other than one

(1) may render scatter symbols in different locations.

- IJK blanking of ordered zones will not display.
- Streamtrace arrowheads may render at different locations and with different sizes.

Saving the Exported File

When saving an exported file, the Select Exported File dialog appears. Navigate to the directory where you want to save the file and name it as desired, then click **Save**. The standard filename extension for the type of file being exported is automatically added to the name entered when appropriate.

Vector Graphics Format

Vector export files have device-independent resolution and thus can be easily resized, but they have the same limitations as vector print output. Table 8 provides a summary of the advantages and disadvantages of using vector graphics file formats.

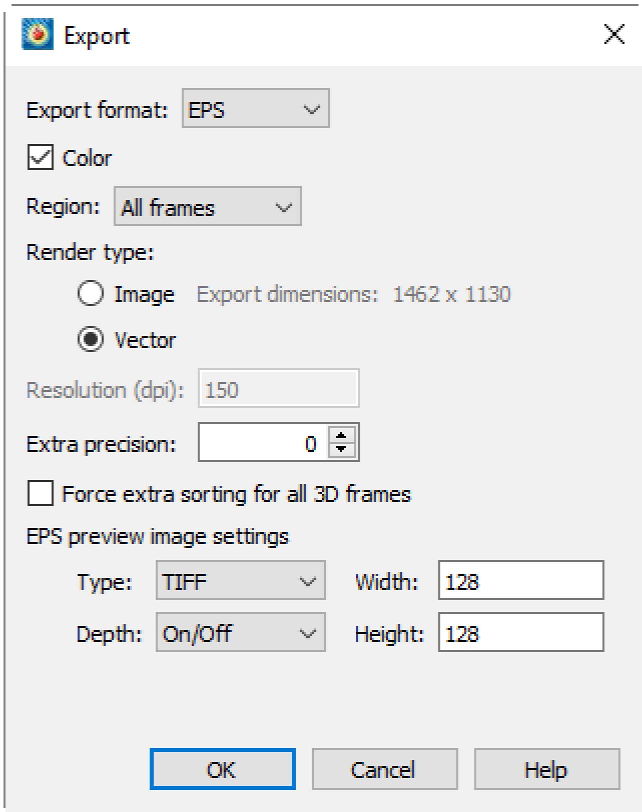
Table 8. Advantages and Disadvantages of Vector Graphics format

Advantages	Disadvantages
Resolution Independent (can be re-sized over and over to any size)	Does not support translucency
Always journal quality (suitable for publication)	Lines can appear different than on screen
Small file sizes for XY and 2-D output	Very large 3D output files
	Not Web-friendly
	Can be manipulated in third-party programs, but those programs are usually more expensive
	Needs a PostScript printer for proper output

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. Import the EPS to a word processor or page layout software to position it on the page. Use the PostScript export format to create files to send directly to a printer.



The **Export** dialog for EPS format has the following options:

Color

Mark the checkbox to output in color. Otherwise output will be in grayscale.

Region

Choose to export only the current frame, the smallest rectangle containing all frames, or the full work area.

Render Type

If you choose Vector, the individual PostScript commands required to draw the image are saved. If you choose Image, a bitmap image is embedded in the file.

Vector mode generally results in a smaller file, and such graphics remain smooth when enlarged, but Image mode can accurately represent translucency or smooth color gradations, like other bitmap formats. In most cases, however, a TIFF, BMP, or PNG is a better choice than an Image EPS, as such files are more widely compatible.

When printing a vector EPS file to a non-PostScript printer, only a low-resolution preview image may be printed, depending on the operating system and print drivers. On Windows platforms, you must specify that the printer is a PostScript printer to print vector EPS files at their full quality. In most cases, Image mode EPSs will be printed correctly on any printer.

If you choose the **Image** render mode, the following option is available:

Resolution

Enter the resolution of the image in dots per inch. Larger values create more accurate plots, but result in larger file sizes.

If you choose the **Vector** render mode, the following options are available:

Extra Precision

Specify the number of decimal places to specify the size and position parameters in the vector-based EPS output. Larger values create more accurate plots, but result in larger file sizes.

Force Extra Sorting for All 3D Frames

Toggle on to use extra sorting in all 3D frames. Overrides the setting in the Advanced 3D Control dialog. If selected, Tecplot 360 uses a slower, more accurate approach that detects and resolves problems with how, for example, intersecting objects are rendered.

EPS Preview Image Settings

Tecplot 360 provides the following options for the preview image:

Type

The type of the preview image, or None.

None

No preview image information is included. If your application will not use such information, this minimizes the file size. However, the graphic will appear blank in most applications; it will appear correctly only when printed.

TIFF

Include a monochrome or grayscale TIFF preview image. This is the most common preview image format. You may specify an image depth for the preview image in the **Depth** drop-down.

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 compatible with older versions of Adobe® FrameMaker®. This preview image type is rarely necessary, as newer versions of FrameMaker support TIFF previews.

Depth

Specifies the number of shades of gray for the preview by how many bits of information is used per pixel. The larger the number of bits per pixel, the larger the resulting file, but the larger the number of shades available. 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 smaller 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.

Width,Height

The size of the preview image.

When using a Render Type of Image, these preview image width, height, and depth values are separate from the characteristics of the EPS image itself. The EPS image size is determined by the Resolution setting and the depth is determined automatically.

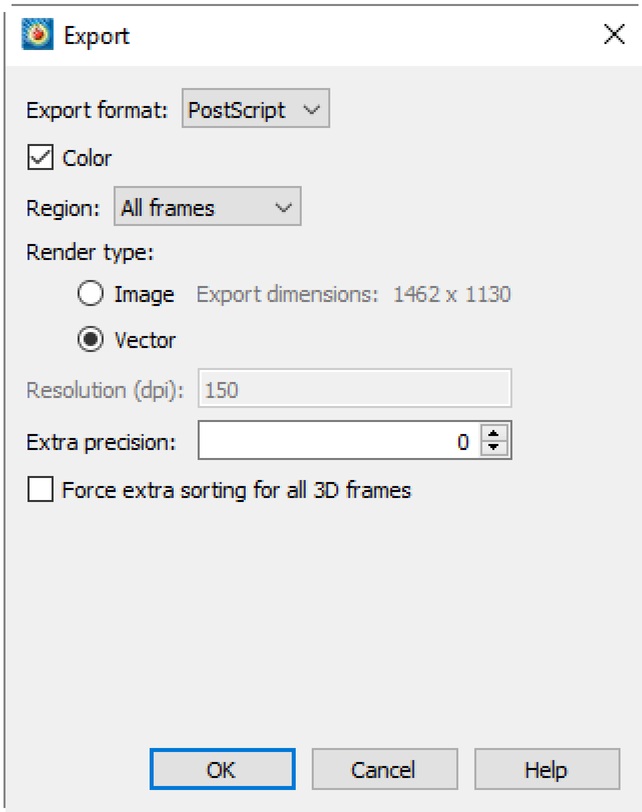
Performance Tips If exporting is taking an unusually long time, or you get an error message saying that the image cannot be exported, the most likely cause is that the image width you are trying to export is too large. Selecting a smaller image width will greatly speed up the export process.

For an image export size of Length x Width, the file size for an uncompressed true color image is approximately Length x Width x 3. Memory requirements to export such an image can be up to twice this size.

For 256 color images, the maximum file size is approximately Length x Width, but is usually less since all 256 color image files are compressed. However, the memory requirements for exporting are the same as they are for a true color uncompressed image.

PostScript (PS) Export

The **Export** dialog allows you to export plots in PostScript (PS). 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, for example, page layout or illustration applications. See EPS Export for details.



The Export dialog for the PostScript format has the following options:

Color

Mark the checkbox to output in color. Otherwise output will be in grayscale.

Region

Choose to export only the current frame, the smallest rectangle containing all frames, or the full work area.

Render Type

If you choose Vector, the indivdual PostScript commands required to draw the image are saved. If you choose Image, a bitmap image is embedded in the file.

Vector mode generally results in a smaller file, and such graphics remain smooth when enlarged, but Image mode can accurately represent translucency or smooth color gradations, like other bitmap formats. If you choose the **Image** render mode, the following option is available:

Resolution

Enter the resolution of the image in dots per inch. Larger values create more accurate plots, but result in larger file sizes.

If you choose the **Vector** render mode, the following options are available:

Extra Precision

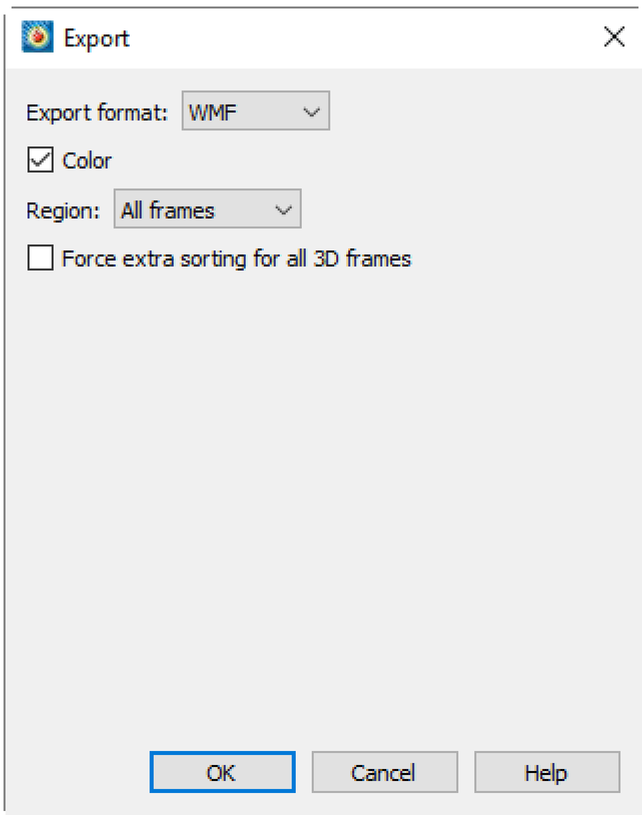
Specify the number of decimal places to specify the size and position parameters in the vector-based EPS output. Larger values create more accurate plots, but result in larger file sizes.

Force Extra Sorting for All 3D Frames

Toggle on to use extra sorting in all 3D frames. Overrides the setting in the Advanced 3D Control dialog. If selected, Tecplot 360 uses a slower, more accurate approach that detects and resolves problems with how, for example, intersecting objects are rendered.

WMF Export

WMF (Windows Metafile) is a Microsoft vector graphics format widely accepted by Windows applications. Since WMFs are vector graphics, they can be easily resized by the importing application without the introduction of visual artifacts, but they cannot accurately represent plots with translucency or smooth color gradations.



Selecting WMF from the **Export** Format drop-down displays WMF options:

Color

Mark the checkbox to output in color. Otherwise output will be in grayscale.

Region

Choose to export only the current frame, the smallest rectangle containing all frames, or the full work area.

Force Extra Sorting for All 3D Frames

Toggle-on to use extra sorting in all 3D frames. Overrides the setting in the Advanced 3D Control dialog. If selected, Tecplot 360 uses a slower, more accurate approach that detects and resolves problems with how, for example, intersecting objects are rendered.

Image Format

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, as may be necessary for printing. Image files are sometimes called raster or bit-mapped. Table 9 provides a summary of the advantages and disadvantages of image file formats.

Table 9. Advantages and Disadvantages of Image file formats

Advantages	Disadvantages
Looks like the screen image, or better	Resolution-dependent (starts to lose quality if enlarged too much)
Adjustable export size	For professional-quality printing, images will be very large
Supports translucency	
Relatively small file size	
Easily managed by presentation packages	
Web friendly	
Easily manipulated in inexpensive 3rd party programs	
Prints on any printer	

Exporting Images with a Specific DPI

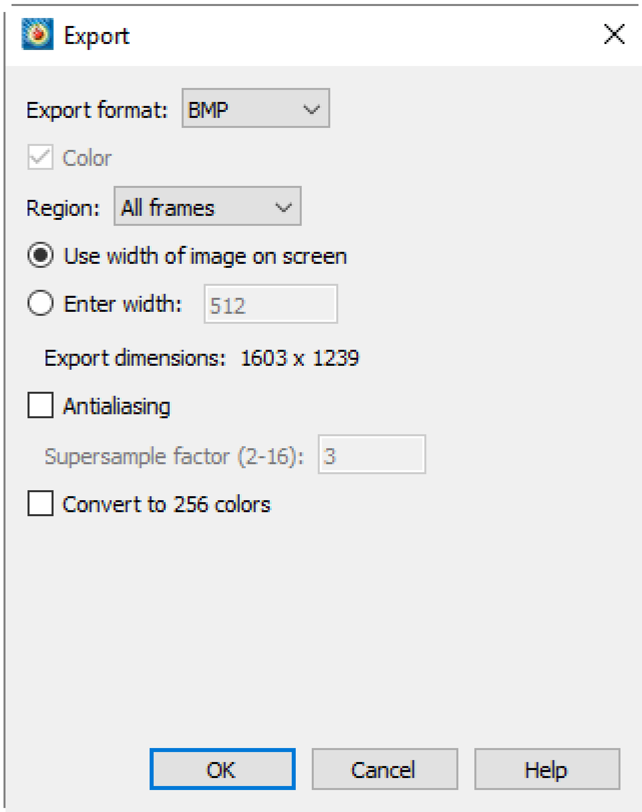
A characteristic of image files is a dots-per-inch setting (DPI), which interacts with the pixel dimensions of the image to determine how large it will be printed. To customize the DPI of your exported image, first set the width of the frame to the size of the image that you wish to export, using the **Edit Active Frame** dialog (accessible by selecting "Edit Active Frame" from the **Frame** menu). Next, when actually exporting the image, multiply the frame width (in inches) by the desired DPI, and enter that number in the Enter Width field of the **Export** dialog.

For example, to export a 4 inch by 6 inch image at 300 DPI, set the frame dimensions to 4 inches by 6 inches, and then when exporting, enter a width of 1200 (4 times 300).

BMP Export

BMP (Bitmap) is an image format, and thus accurately represents plots with translucency and smooth color gradations. Although it originated on Windows, it is widely accepted on other platforms by many applications. However, it does not perform any compression, so file sizes are larger than more modern formats such as JPG and PNG.

BMP is always exported in color. The BMP export options are shown below.



Region

Choose to export only the current frame, the smallest rectangle containing all frames, or the full work area.

Use Width of Image on Screen

Select this option to generate an image file the same size as the current plot on the screen.

Enter Width

Select this option to specify a width (in pixels) for the generated image. A larger width increases the quality of your image. However, the greater the width you specify, the longer it will take to export the image and the larger the exported file.

Antialiasing

Select this option to smooth jagged edges in the image. See Antialiasing Images for details.

Supersample Factor

Control the amount of antialiasing used in the image. See Antialiasing Images for details.

Convert to 256 Colors

Select this option to generate an image with only 256 colors (down from a possible 16 million colors). Tecplot 360 selects the colors. This option can help reduce file size, but the results may be suboptimal; using it with transparency, smooth color gradations, or antialiasing may result in poor image quality.

Performance Tips

If exporting is taking an unusually long time, or you get an error message saying that the image cannot be exported, the most likely cause is that the image width you are trying to export is too large. Selecting a smaller image width will greatly speed up the export process.

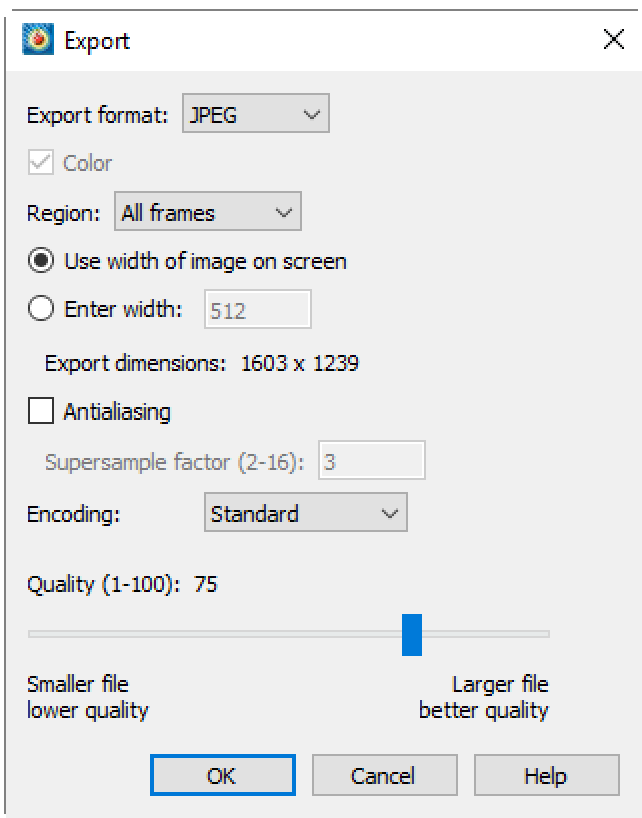
For an image export size of Length x Width, the file size for an uncompressed true color image is approximately Length x Width x 3. Memory requirements to export such an image can be up to twice this size.

For 256 color images, the maximum file size is approximately Length x Width, but is usually less since all 256 color image files are compressed. However, the memory requirements for exporting are the same as they are for a true color uncompressed image.

JPEG Export

JPEG (Joint Photographic Experts Group) is an image format, and thus accurately represent plots with translucency and smooth color gradations. However, JPEG is a highly-compressed, "lossy" format, meaning the exported file may not be identical to the plot. This 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 levels of compression, and Tecplot 360 allows you to control the image quality (and thus, inversely, the file size).



JPEG images are always exported in color. When you select JPEG in the **Export** dialog, you have the following options:

Region

Choose to export only the current frame, the smallest rectangle containing all frames, or the full work area.

Use Width of Image on Screen

Select this option to generate an image file the same size as the current plot on the screen.

Enter Width

Select this option to specify a width (in pixels) for the generated image. A larger width increases the quality of your image. However, the greater the width you specify, the longer it will take to export the image and the larger the exported file.

Antialiasing

Select this option to smooth jagged edges in the image. See Antialiasing Images for details.

Supersample Factor

Control the amount of antialiasing used in the image. See Antialiasing Images for details.

Encoding

Choose an encoding method for the JPEG file.

Standard

Encodes a JPEG one line at a time, starting at the top line. In most Web browsers, this means nothing appears until the image has been completely loaded.

Progressive

Encodes the JPG in two passes, first at lower quality and then at higher quality. In a Web browser, this means that a lower-quality version of the image appears quickly, then is replaced with a larger version of the image when the file has been completely loaded.

Given the same Quality level,**Standard** encoded JPEG files generally look better than equivalent **Progressive** encoded JPEG files. However, they have a larger file size.

Quality

Select the quality of JPEG image. Higher quality settings produce larger files and better looking exported images. Lower quality settings produce smaller files. A quality setting of around 75 is usually optimal; settings much below this may visibly degrade image appearance, while higher settings usually do not provide much visible improvement but create larger files.

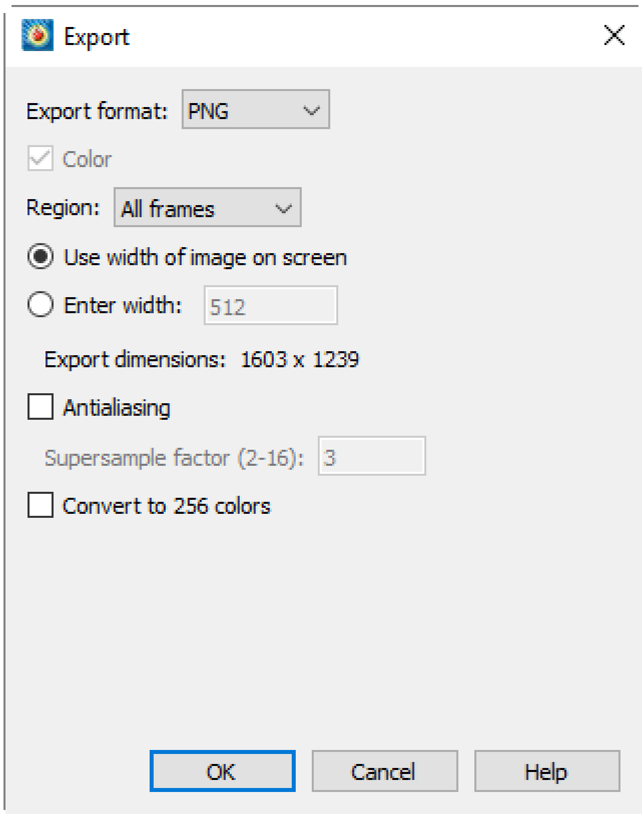
Performance Tips

If exporting is taking an unusually long time, or you get an error message saying that the image cannot be exported, the most likely cause is that the image width you are trying to export is too large. Selecting a smaller image width will greatly speed up the export process.

For an image export size of Length x Width, the file size for an uncompressed true color image is approximately Length x Width x 3. Memory requirements to export such an image can be up to twice this size.

PNG Export

PNG (Portable Network Graphics) is an image format, and thus accurately represent plots with translucency and smooth color gradations. Unlike JPEG images, PNGs can exactly represent a plot without losing any nuance.



PNG images are always exported in color. When you select PNG in the **Export** dialog, you have the following options:

Region
: Choose to export only the current frame, the smallest rectangle containing all frames, or the full work area.

Use Width of Image on Screen
: Select this option to generate an image file the same size as the current plot on the screen.

Enter Width
: Select this option to specify a width (in pixels) for the generated image. A larger width increases the quality of your image. However, the greater the width you specify, the longer it will take to export the image and the larger the exported file.

Antialiasing

Select this option to smooth jagged edges in the image. See Antialiasing Images for details.

Supersample Factor

Control the amount of antialiasing used in the image. See Antialiasing Images for details.

Convert to 256 Colors

Select this option to generate an image with only 256 colors (down from a possible 16 million colors). Tecplot 360 selects the colors. This option can help reduce file size, but the results may be suboptimal; using it with transparency, smooth color gradations, or antialiasing may result in poor image quality.

Performance Tips

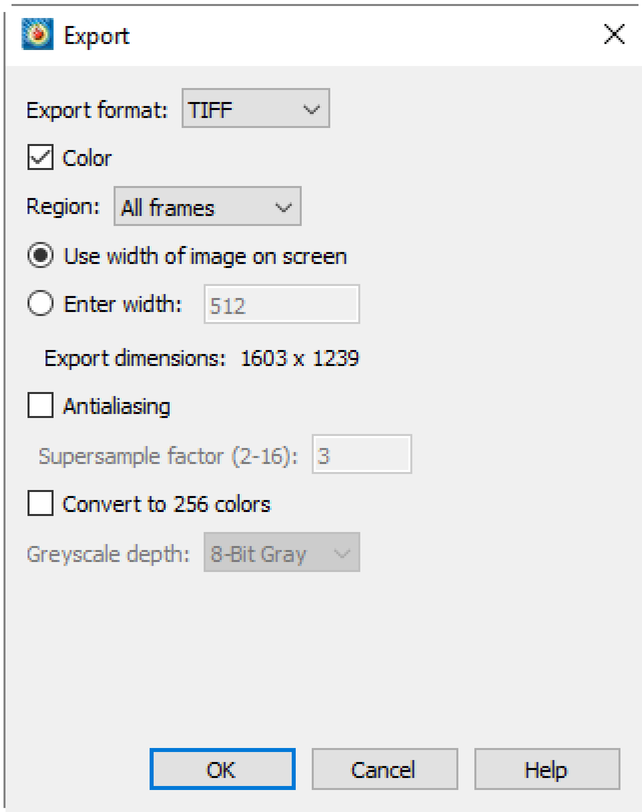
If exporting is taking an unusually long time, or you get an error message saying that the image cannot be exported, the most likely cause is that the image width you are trying to export is too large. Selecting a smaller image width will greatly speed up the export process.

For an image export size of Length x Width, the file size for an uncompressed true color image is approximately Length x Width x 3. Memory requirements to export such an image can be up to twice this size.

For 256 color images, the maximum file size is approximately Length x Width, but is usually less since all 256 color image files are compressed. However, the memory requirements for exporting are the same as they are for a true color uncompressed image.

TIFF Export

TIFF (Tagged Image File Format) is an image format, and thus accurately represent plots with translucency and smooth color gradations. Tecplot 360 generates both color and gray-scale TIFF images.



When you select TIFF in the **Export** dialog, you have the following options:

Color

Mark the checkbox to output in color. Otherwise output will be in grayscale.

Region

Choose to export only the current frame, the smallest rectangle containing all frames, or the full work area.

Use Width of Image on Screen

Select this option to generate an image file the same size as the current plot on the screen.

Enter Width

Select this option to specify a width (in pixels) for the generated image. A larger width increases the quality of your image. However, the greater the width you specify, the longer it will take to export the image and the larger the exported file.

Antialiasing

Select this option to smooth jagged edges in the image. See Antialiasing Images for details.

Supersample Factor

Control the amount of antialiasing used in the image. See Antialiasing Images for details.

Convert to 256 Colors

Select this option to generate an image with only 256 colors (down from a possible 16 million colors). Tecplot 360 selects the colors. This option can help reduce file size, but the results may be suboptimal; using it with transparency, smooth color gradations, or antialiasing may result in poor image quality.

Grayscale Depth

For grayscale 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.

Antialiasing Images

Antialiasing smooths jagged edges on 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 called 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 360 cannot antialias if the intermediate image would be larger than this limit. Some graphics drivers do not report an error in such situations, instead producing blank or garbled images.

Antialiasing uses many colors. If you are exporting with the **Convert to 256 Colors** option activated, antialiasing works for plots with a very limited selection of colors (like a red mesh on a black field). Otherwise, antialiasing to 256 colors wastes time and may decrease plot quality.

Using animation formats can amplify the antialiasing and 256-color problem, as the same 256 colors must generally be used for colors in all frames of the animation. For these formats, try a test animation of a few steps with antialiasing on before creating the entire animation.

Clipboard Exporting to Other Applications

In general, the **Cut**, **Copy**, and **Paste** commands on the **Edit** menu work only within Tecplot 360. However, it is possible to copy the plot in a frame to the clipboard as an image using the following procedure:

1. With the  selector tool active, click the frame border to select the frame. Black dots appear at the frame's corners and at the midpoints of its edges.
2. From the **Edit** menu, choose **Copy**.

An image of the frame's plot is now on your system clipboard and can be pasted into documents in other applications. Note that the image captures the plot exactly as it appears on the screen, including its size.

Text may be copied and pasted into other applications using a similar procedure. Select the text object, choose **Edit**→**Copy**, then paste the text into the other application.

Part 6: Scripting

Macros

Tecplot 360 provides two methods of scripting or automating your workflow: macros and batch processing. This chapter focuses on the Tecplot 360 menu options for recording and playing back macros. The Scripting Guide describes the Tecplot 360 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. Macros can also be used to drive Tecplot 360 batch jobs. For information on batch processing, see Batch Processing.

Macro Creation

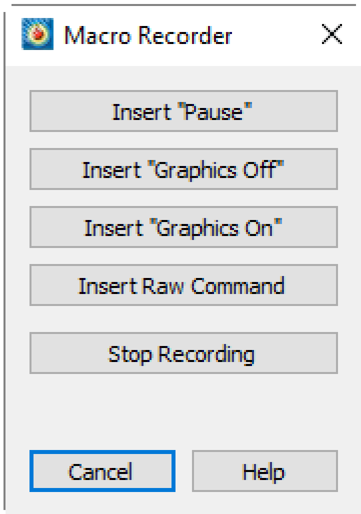
Tecplot 360's **Macro Recorder** records a macro as you perform a sequence of actions interactively. Macros can be recorded in either Tecplot `(.mcr)` or PyTecplot Python `(.py)` format. PyTecplot Python scripts run outside of Tecplot 360, and may be run from either the command line or any Python IDE. See the PyTecplot Guide for more information.

After recording your macro, you can edit your macro file with a plain text editor to remove redundant operations, compress repetitive actions into loops, replace references to files with variable names, and otherwise modify the macro to make it more generic and flexible.

To record a macro with the **Macro Recorder** dialog, select "Record Macro" or "Record PyTecplot" from the **Scripting** menu. Specify a macro file name in the **Write Macro File (.mcr)** or **Write Script (.py)** dialog and click Save to initiate the recording. The **Macro Recorder** dialog remains open during the recording session.

> Auto Redraw is disabled during macro recording. If necessary, you can manually redraw your plot by clicking the Redraw button in the Plot sidebar.

While recording macros, you can use any of the following 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 360 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 360 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 Scripting Guide for information on the Tecplot 360 macro language.

Stop Recording

Select when you have completed the sequence of actions you want recorded.

The commands in a macro file typically rely on Tecplot 360 being in a particular state. It is a good practice to use commands at the start of a macro that force Tecplot 360 into a known state. For example, the `$!NEWLAYOUT` command deletes all data sets and frames and creates a single empty frame with a default size and position.

Macros are more likely to be forward compatible (i.e. work with future releases of Tecplot 360) if the macro begins with a new layout or stylesheet. Refer to the Scripting Guide for more information.

Macro Functions

When editing a macro file in a text editor, 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`.

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
  $!FIELDMAP [1-3] CONTOUR{CONTOURTYPE = BOTHLINESANDFLOOD}
  $!GLOBALCONTOUR 1 COLORMAPNAME = "Grayscale"
  $!REDRAW
$!ENDMACROFUNCTION
```

The `RETAIN` parameter tells Tecplot 360 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 360, and continue to use it throughout your Tecplot 360 session.

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"
```

You can use the `$!RUNMACROFUNCTION` command within other macro functions; calls may be nested up to ten deep.

To access parameters from within a macro function use "|*n*|", where *n* is the parameter number (do not include the double quotes). For example, the following function uses two parameters for the assignments to SHOWCONTOUR and SHOWMESH:

```
$!MACROFUNCTION
  NAME = "AssignContourAndMesh"
  $!FIELDLAYERS SHOWCONTOUR = |1|
  $!FIELDLAYERS SHOWMESH = |2|
$!ENDMACROFUNCTION
.
.
.
$!RUNMACROFUNCTION "AssignContourAndMesh" (YES,NO)
```

Linking to Geometries

Each 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 geometry. Macro functions are specified with the **Link to Macro function** field in the **Geometry** dialog. Only one macro function may be specified for each geometry.

The macro function must be defined in a macro file using the `$!MACROFUNCTION` instruction as described in Macro Functions. The macro file containing the variable must then be executed so that the function is loaded into memory and available for use in the current Tecplot 360 session.

A convenient way to handle this is to record a macro that opens the layout that contains the linked objects and then edit this macro using a text editor to also include the necessary macro functions. You can also simply include the macro functions in your `tecplot.mcr` file.

Macro Playback

Once you have created a macro file, you have four methods in Tecplot 360 for playing it back:

From the command line

You can play a macro when Tecplot 360 is launched by including the name of the macro file on the command line, i.e.:

```
tecplot mymacro.mcr
```

If your macro file does not have the `.mcr` extension, run Tecplot 360 with the macro file by including the `-p` flag on the command line, such as:

```
tecplot -p mymacro.mmm
```

From the Tecplot 360 interface

You can play a macro from within Tecplot 360 by using the "Play Macro/Script" option on the **Scripting** menu.

Using the Macro Debugger

Use the **Macro Debugger** (accessed via **Scripting→View/Debug Macro**) to open, inspect, step through, and debug your macro file. See Macro Debugger for more information.

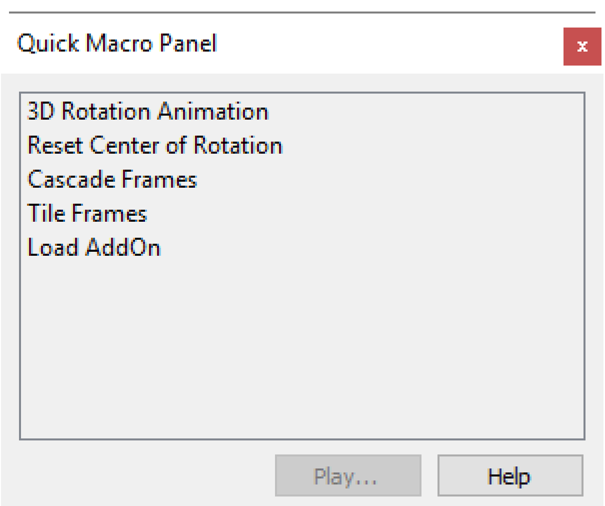
Using the Quick Macro Panel

The **Quick Macro Panel** (accessed via the **Scripting** menu) allows you to quickly play a macro function by clicking on the button in the panel that is linked to that macro function. See Quick Macro Panel for more information.

> On Windows operating systems, you can also launch Tecplot 360 and run a macro by dragging and dropping a macro file onto the Tecplot 360 icon. However, in this case, the macro file must have the .mcr extension. Otherwise, the file will be treated as an ASCII data file.

Quick Macro Panel

The **Quick Macro Panel**, accessible by selecting "Quick Macros" from the **Scripting** menu, is a sidebar for playing your favorite or frequently-used macros. It initially appears docked to the right side of the main Tecplot 360 window, but like the other sidebars, it can be dragged out of the workspace to become a floating window, or docked with one or more other sidebars to suit your working habits. (See Sidebars for more information on working with sidebars.)



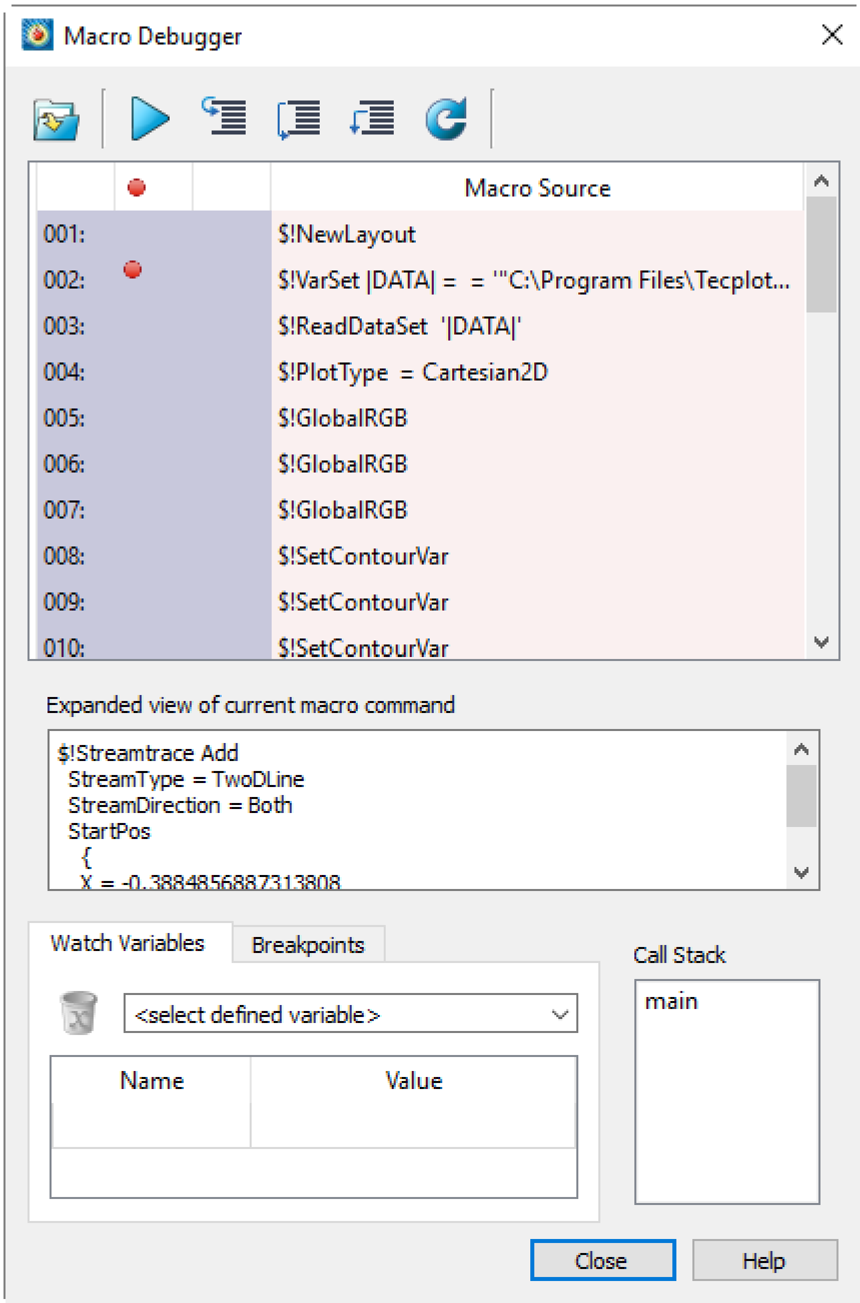
This panel comes with a number of useful macro functions already defined. To run any of these macros, select it in the list, then click the Play button, or simply double-click the macro name.

The Quick Macro Panel is linked to a special macro file that should contain only macro function definitions (`$!MACROFUNCTION` instructions). The macro definitions having `SHOWINMACROPANEL = TRUE` appear in the Quick Macro Panel. Other functions may defined in this file without this flag; these can be called by the macro(s) displayed in the panel. You may also use the `$!INCLUDEMACRO` instruction to include macro instructions from other files into the quick macro file.

For information on how to load a quick macro file, see Custom Files loaded on Startup.

Macro Debugger

Use the **Macro Debugger**, accessed via **Scripting→View/Debug Macro,** to step through and debug a macro file. This dialog allows you to add and delete breakpoints, view and set watch variables, and view the call stack.



The **Macro Debugger** displays the text of the currently loaded macro file at the top of the dialog in the Macro Source area. The black arrow  marks the line about to be executed; it moves to the next command after the current command is executed. Below this is a second area (ExpandedView) showing the complete command about to be executed, as many macro commands are lengthy and may not be readable in their entirety in the Macro Source area.

The Macro Debugger has the following buttons in its toolbar:

Load a Macro file

Loads a macro file using the **Open Macro** dialog, in which you can specify which macro file to load. Tecplot macro files typically have the extension `.mcr`. Once you have loaded a macro file, it stays loaded in the Macro Debugger until you load a different one.

Continue

Plays the macro without stopping after each step, until the macro finishes or the Macro Debugger encounters a breakpoint.

Step buttons

These three related buttons help you step through your macro line by line, pausing after each instruction has been processed. Checking the values of variables after each step is very helpful in finding and fixing problems. These buttons have slightly different functions, described below.

Step Into

Executes the current macro command. When a $!RUNMACROFUNCTION command is encountered, the **Macro Viewer** steps into the called function, so it too can be executed line by line.

Step Over

Executes the current macro command. When a `$!RUNMACROFUNCTION` command is encountered, the entire function is processed as a single operation, instead of allowing you to step through it line by line.

This is useful for functions that you have already verified to be working correctly and/or do not wish to step through, especially long ones.

Step Out

Executes the rest of the current function at full speed, rather than line by line, until the end of the function (`$!ENDMACROFUNCTION`) is encountered, then resumes single-stepping at the line after the `$!RUNMACROFUNCTION` instruction that called the function.

This is useful when you have verified that a function is working correctly, and don't need to continue stepping through it (especially in functions with loops) or have accidentally stepped into a function you don't want to step through.

Reset

Stops the running macro, if any, and resets the macro to start running at the beginning again.

If your macro relies on Tecplot 360 being in a particular state when it starts processing (that is, it does not begin with loading or creating a layout), make sure Tecplot 360 is in this state before you click **Reset**. Otherwise, the results of the macro may not be what you expect.

> A convenient way to handle state with macro resets is to save a layout before beginning debugging, then reload this layout if you need to reset.

The Macro Debugger has the following three areas at the bottom of the dialog. Watch Variables and Breakpoints share the same space; use the provided tabs to switch between them.

Watch Variables

Displays the values of selected macro variables as you step through the code, and allows you to add and remove variables to be watched. See Watching Variables.

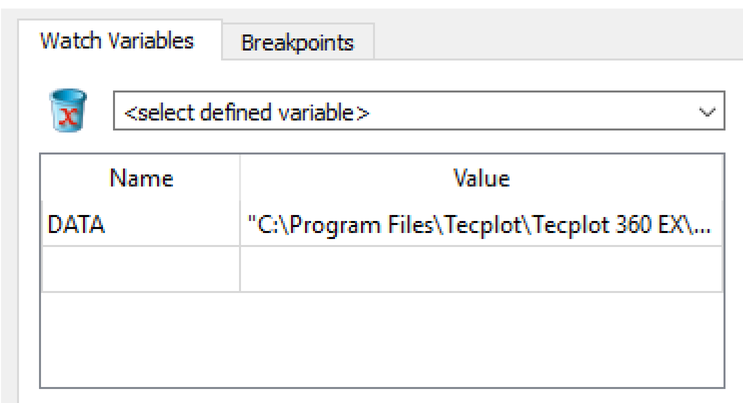
Breakpoints

Displays the breakpoints set in your macro and allows you to delete any or all of them easily. See Using Breakpoints.

Call Stack

Displays all function calls currently executing. When one function calls another, which calls another, and so on, this area displays the name of all these functions in the order they were called. The name *main* represents the top-level script (instructions outside of any functions).

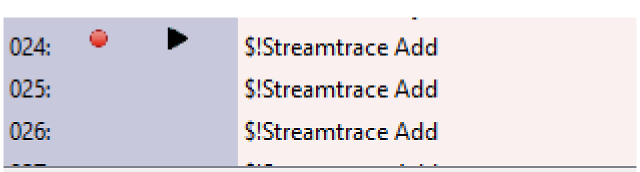
Watching Variables



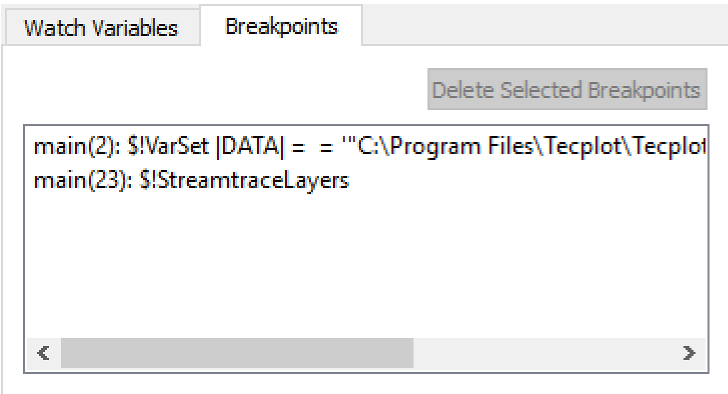
The Watch Variable panel at the bottom of the Macro Debugger allow you to set variables to be displayed while your macro runs. You can add a variable to be watched by choosing it from the pop-up menu here. To stop a variable from being watched, click it in the list of watched variables, then click the delete (trash) icon.

Using Breakpoints

A *breakpoint* is a point in your code where you wish to pause execution, usually just before some macro instructions you are having trouble with. Using the Continue button, your macro runs at full speed until it reaches a breakpoint. You can then inspect variables or the call stack and see the next several lines to be executed, then step through the macro a line at a time using the appropriate Step buttons in the toolbar to see how they behave.



Breakpoints are shown as red discs to the left of each line of code in the Macro Source field. To set a breakpoint at a particular line of your macro, click to the left of that line. A red disc appears next to that line to indicate that execution will pause just before that line is executed. You can remove the breakpoint by clicking again.



The Breakpoints area at the bottom of the Macro Debugger dialog display information about where each breakpoint is set: the function it's in ("main" if it's outside a function), its line number, and the macro command. You can delete breakpoints from this area by selecting them and clicking **Delete Selected Breakpoints**. (This is the fastest way to delete all breakpoints.)

Batch Processing

You can run Tecplot 360 in batch mode to create plots without displaying any graphics to the screen. This saves time when processing multiple files for printing or export. In batch mode, Tecplot 360 can be executed locally on your workstation computer or remotely using a terminal (Linux only).

Batch Processing Setup

1. Create a macro file to control the batch processing. You may do this either by using **Scripting**→**Record Macro** and recording a Tecplot 360 session, or by using any text editor. See Macros.
2. Create layout and stylesheet files, as necessary.
3. Prepare data files.
4. Debug the macro file by running Tecplot 360 while not in batch mode.

> Macros are more likely to be forward compatible (i.e. work with future releases) only if the file is started with a layout or stylesheet. Refer to the Scripting Guide for more information.

When you launch Tecplot 360 in batch mode, you provide the name of a macro file to execute. The minimal command to launch Tecplot 360 in batch mode is as follows:

```
tec360 -b -p mymacrofile.mcr
```

The `-b` flag instructs Tecplot 360 to run in batch mode and the `-p` tells Tecplot 360 to run the filename that follows (mymacrofile.mcr in this case).

> The `-p` flag can be omitted if the macro file uses the `.mcr` extension (thus in the example above it could be omitted).

Batch Mode and Linux

See Tecplot 360 Command Line for information on batch mode command line arguments for Linux. If you have any issues with running in batch mode, see the Rendering and Export Troubleshooting guide for more information.

Batch Mode and Windows

Under Windows, invoking Tecplot 360 from the command line or in a `.bat` file returns immediately rather than waiting for the application to finish working, as is normal on Linux and Mac.

When using batch mode, however, you generally want to wait until Tecplot 360 has finished one operation before proceeding with the next. To do this, precede the command with `start /wait`. For example:

```
start /wait tec360 -b -p mymacrofile.mcr
```

Batch Mode and Mac

When running batch mode on a Mac, the full path to the executable must be called:

```
"/Applications/Tecplot 360 EX 2025 R1/Tecplot 360 EX 2025 R1.app/Contents/MacOS/Tecplot
360 EX 2025 R1" -b mymacro.mcr
```

If you are running many files in batch mode, it may be helpful to set up an alias for the full executable path. To set up an alias, type the following in `~/.bashrc`:

```
alias tec360='"/Applications/Tecplot 360 EX 2025 R1/Tecplot 360 EX 2025
R1.app/Contents/MacOS/Tecplot 360 EX 2025 R1"'
```

With the alias defined, running macros in batch mode becomes easier. Simply add the -b flag followed by the macro you wish to use:

```
tec360 -b mymacro.mcr
```

For more information on setting an alias and running Tecplot from the command line in Mac see the Installation Guide.

Batch Processing Using a Layout File

Combining layout files with batch processing is both powerful and flexible, and is also the recommended procedure. With layout files, you can organize a plot using one or more frames in a single file. The layout file manages datasets and can be altered on the fly, either on the command line or within a macro that loads the layout file.

> The layout file must contain a `$!READDATASET` macro instruction compatible with the data being loaded. The easiest way to create such a layout is to load a file similar to the file(s) you wish to process into Tecplot 360, then set up the plot as desired and save the layout. The filename specified in the layout's `$!READDATASET` instruction will be overridden by the filename specified on the command line (see below).

For example, 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 script as follows:

1. Obtain a representative data file to be plotted.
2. Create a layout of the style of plot you want. (For this example, name the file `batch.lay`).
3. Use a text editor to create the following macro (For this example call this macro `batch.mcr`):

```
#!MC 1410
$!EXPORTSETUP
  EXPORTFORMAT = PS
  PRINTRENDERTYPE = VECTOR
$!PRINTSETUP
  PALETTE = MONOCHROME
$!EXPORT
  EXPORTREGION = CURRENTFRAME
$!Quit
```

4. Use the following command to run the job in batch mode:

```
tec360 -b -p batch.mcr -y psoutput.ps batch.lay mydatafile
```

tec360 -b	Launches Tecplot 360 in batch mode
-p	Tells Tecplot 360 to use the following macro file. (Note: This is not needed if the macro file uses the `.mcr` extension).
batch.mcr	Macro file
-y	Tells Tecplot 360 to use the following export file
psoutput.ps	Export file
batch.lay	Layout file to use
mydatafile	Data file to use; overrides the file specified in the layout

Multiple Data File Processing

In Batch Processing Using a Layout File, we set up Tecplot 360 to process a user-supplied data file (or 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 360 in the operating system using your shell's looping constructs, or within Tecplot 360 using the flow-of-control commands in the Tecplot macro language.

Looping Outside Tecplot 360

The following examples show the command files for launching Tecplot 360 in an operating system loop on two different operating systems. Tecplot 360 processes five data files named `d`*nn*`.plt` and creates ten output files named `d`*nn*`.out` where *nn* goes from 1 to 10.

Looping Outside Tecplot 360 (Linux)

Create a shell script with the following commands:

```
#!/bin/sh
n=1
while test $n -le 10
do
  tec360 -b -p batch.mcr -y d$n.out batch.lay d$n.plt
  n=`expr $n+1`
done
```

Looping Outside Tecplot 360 (Windows)

Create a batch file with the following commands:

```
for %%f in (d1 d2 d3 d4 d5 d6 d7 d8 d9 d10)
do tec360 -b -p batch.mcr -y %%f.out batch.lay %%f.plt
```

Looping Inside Tecplot 360

In Looping Outside Tecplot 360 we set up Tecplot 360 to process multiple data files using the shell to do the looping. There are two drawbacks to this procedure:

- The shell languages are not portable between different operating systems.
- Tecplot 360 must be repeatedly started and stopped to process each dataset.

A more efficient approach is to loop through the data files inside Tecplot 360. Here, the layout file and the data files are all named within the macro. The command line in this example is simple, as follows:

```
tec360 -b -p batch.mcr
```

Where the macro `batch.mcr` contains:

```
#!MC 1410
$!EXPORTSETUP EXPORTFORMAT = PS
$!PRINTSETUP PALETTE = MONOCHROME
$!LOOP 10
  $!OPENLAYOUT "batch.lay"
    ALTDATALOADINSTRUCTIONS = "d|LOOP|.plt"
  $!EXPORTSETUP PRINTRENDERTYPE = VECTOR
  $!EXPORTSETUP EXPORTFNAME = "d|LOOP|.out"
  $!EXPORT
    EXPORTREGION = CURRENTFRAME
$!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.

If you want to make many different plots using the same dataset, stylesheets will be more efficient than layout files.

Batch Processing Diagnostics

Each time Tecplot 360 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 360 was started. If the name given in the `BATCHLOGFILE` environment variable is a relative path, the directory name where Tecplot 360 was started is prefixed. A running commentary on actions performed in Tecplot 360, as well as warning and error messages, are sent to the `batch.log` file.

Batch Converting to SZL Format

Two of Tecplot 360's command line options, `-o` and `-convert`, allow you to more easily convert your existing datasets to SZL, Tecplot's high-performance subzone data format (filename extension `.szplt`). Since it is command-line driven, this capability is suitable for converting large numbers of files without user intervention, or as a step in solver postprocessing.

Tecplot 360 does not require a license when launched with either of these options, so you can perform data file conversion on any available workstation without concern. Only a Tecplot 360 installation is required.

In a restricted batch session started with `-o` or `-convert`, Tecplot 360 can read data files (using any of its data loader add-ons) and write data in SZL format, but cannot perform most other operations. As with all batch operations, there is no graphical user interface; batch conversion is strictly a command-line function.

Most of the time, you'll want to use the `-o` option for SZL conversion. See Writing SZL Files With Macros for more information on the use cases for the `-convert` option.

Loading Data Files by Name

The `-o` option outputs any loaded data to the specified `.szplt` output file. Just add `-o` followed by the output filename or path to whatever command line you would normally use to load your data. Tecplot 360 will then write that data in SZL format (and exit) instead of displaying a plot in its workspace. For example, if you usually start exploring data in two Tecplot `.plt` files using the following command line:

```
tec360 file1.plt file2.plt
```

You can instead have Tecplot 360 convert those data files to a single `.szplt` file using the following command line:

```
tec360 file1.plt file2.plt -o output.szplt
```

If your data is in a non-Tecplot format, you may or may not be able to load it all into Tecplot 360 using a single, simple command line of the type shown. Some formats (e.g. CGNS) support loading multpile files, but require that the files all be part of a single case, and require additional cases to be appended via further load operations. Other loaders do not support multiple files at all and require all loads after the first to be appended. In such cases, you can record a macro to load your data.

Loading Data Files with Macros

Create a macro to load some or all of your data when one or more of the following apply:

- The data cannot be loaded using a single command line because it is in multiple files and the required loader does not support loading all of them using a single command line.
- The data files do not have filenames that allow Tecplot 360 to unambiguously determine the correct loader for the data. (In some cases, though, you can simply rename the files.)
- You need to set the loader's advanced options to successfully load the data. In other words, in the Load Data dialog, you use the Open with Advanced Options command (accessed by clicking the menu triangle next to the Open button) or mark the Advanced Options checkbox.

Simply record a macro that loads the data, using the procedure you normally use with the Tecplot 360 graphical user interface. See Macro Creation for more information on recording macros.

The loading procedure you record can include multiple load operations using the append option, any number of different file formats, and selection of advanced loader options. After you have recorded the macro, specify it on the command line when invoking `tec360`. As with the previous case, simply add `-o` followed by the name of SZL file to be created:

```
tec360 loaddata.mcr -o output.szplt
```

Your recorded macro contains fixed file names and directory paths for the files you loaded, which makes it difficult to use the macro with other data files. We will see how to address this with a few simple changes to the macro shortly.

Using Multiple Macros to Load Files

Tecplot 360 allows you to specify multiple macros on the command line, which will be executed in the order they appear in the command. Instead of creating one macro that loads multiple files, then, you could record multiple macros, each loading one data file, then specify just the ones you actually need for a given project. For example:

```
tec360 loaddata1.mcr loaddata2.mcr loaddata3.mcr -o output.szplt
```

Combined with the techniques in the following sections to allow the macros to load data files other than the ones used when recording the macro, this lets you create a macro "tool set" for converting the various kinds of files you typically work with.

There is one hitch to using multiple macros on the command line. When you record a load operation, the resulting macro may include an instruction telling the loader to erase any data that has already been loaded. When you attempt to load your data, the second macro file specified will erase the data loaded by the first, and so on for the third, fourth, etc., wasting time and making it seem like only the last macro loaded anything.

Fortunately, this is easy to fix. Simply load the macro into any text editor (`Notepad.exe` will do) and find any lines that says `READDATAOPTION = NEW`. Change it to say `READDATAOPTION = APPEND` instead. Such a macro will still work fine even when specified first on the command line, because each conversion is a new Tecplot 360 session that begins with no data loaded.

Mixing Macros and Data Files

Macros are just another kind of data file: one that Tecplot 360 happens to execute as instructions. For this reason, it is possible to mix data files and macros on the command line to load your data, using macros only for files that can't be loaded by specifying them directly. Files are processed in the order they appear on the command line. For example:

```
tec360 cube.plt loadcircle.mcr -o output.szplt
```

This loads `cube.plt` (a regular Tecplot data file) and then executes `loadcircle.mcr`, which, presumably, loads another data file (or even multiple files). The loaded data are then written to `output.szplt`.

Using Relative Paths

As we mentioned, the paths of the files you loaded and saved become a part of recorded macros. The macro will then always load input files with the same names, and in the same directory, as the file(s) used when the macro was recorded. The output file, similarly, is always given exactly the same name and stored in the same directory as the one you wrote while recording the macro.

If you have set up your simulation process so that the solver always produces an output file of the same name, but in a different directory for each case, you can simply edit the recorded macro to use relative paths. Then the macro will work with files in the current directory, whatever it may be, and you can simply change to the directory that contains the file you wish to convert before performing the conversion.

For example, here is a simple macro, `loadcube.mcr`, that we recorded to load a single `.plt` file. (A real macro would probably be longer, since you don't even need a macro to load a `.plt` file, but let's keep it simple.)

```
#!MC 1410
$!VarSet |MFBD| = 'C:\Users\kelvin\Desktop'
$!READDATASET '"|MFBD|\cube.plt" '
  READDATAOPTION = NEW
  RESETSTYLE = YES
  VARLOADMODE = BYNAME
  ASSIGNSTRANDIDS = YES
  VARNAMELIST = '"X" "Y" "Z"'
$!RemoveVar |MFBD|
```

All recorded macros contain a definition of the variable `MFBD`, which refers to the current (or working) directory that was in effect when the macro was recorded. Since we loaded `cube.plt` from our Desktop folder, which was also the working directory, the macro uses the `MFBD` variable in the `READDATASET` command. If we had loaded our data file from a directory other than the working directory, the `READDATASET` command would have the full path (for example, on Windows, it would probably start with `C:\`). Here's what the macro looks like when `cube.plt` is in `` `C:\ ``:

```
#!MC 1410
$!VarSet |MFBD| = 'C:\Users\kelvin\Desktop'
$!READDATASET '"C:\cube.plt" '
  READDATAOPTION = NEW
  RESETSTYLE = YES
  VARLOADMODE = BYNAME
  ASSIGNSTRANDIDS = YES
  VARNAMELIST = '"X" "Y" "Z"'
$!RemoveVar |MFBD|
```

We don't actually need the `MFBD` variable for our purposes, so we can remove the `VarSet` command and the corresponding `RemoveVar` command. (It doesn't hurt to leave them, however.) We will also remove everything but the filename from the `READDATASET` and `WRITEDATASET` commands. We will also change the `READDATAOPTION` to `APPEND`, as suggested in the preceding section. Here is the macro after these changes.

```
#!MC 1410
$!READDATASET '"cube.plt" '
  READDATAOPTION = NEW
  RESETSTYLE = YES
  VARLOADMODE = BYNAME
  ASSIGNSTRANDIDS = YES
  VARNAMELIST = '"X" "Y" "Z"'
```

This macro now reads the `cube.plt` file from the current (or working) directory. After saving it as `loadcube.mcr`, you can invoke the conversion as follows:

```
cd C:/path/to/files
tec360 C:/path/to/loadcube.mcr -o output.szplt
```

Note that, when invoking the conversion, we must specify the full path to `convert.mcr` since it is not in the working directory. If the Tecplot 360 `bin` directory is not in your system's `PATH` variable, you must also supply the full path to the `tec360` executable. (For simplicity, our examples assume `tec360` is in your `PATH`.)

Using Environment Variables in Loader Macros

When the data files you want to convert may have different names, but are in the same format and need the same loader options, it is possible to write a macro that accepts paths specified outside of the macro. The most straightforward way to do this is to set environment variables before launching Tecplot 360. The macro file then uses these environment variables as the input file path(s). Here, we will use the variable `INFILE` with the macro from the preceding section. Since it will no longer load only `cube.plt`, we will name the macro `loaddata.mcr` rather than `loadcube.mcr`.

```
#!MC 1410
$!READDATASET '"|$INFILE|" '
  READDATAOPTION = APPEND
  RESETSTYLE = YES
  VARLOADMODE = BYNAME
  ASSIGNSTRANDIDS = YES
  VARNAMELIST = '"X" "Y" "Z"'
$!EXTENDEDCOMMAND
  COMMANDPROCESSORID = 'Tecplot Subzone Data Tools'
  COMMAND = 'WRITEDATASET FILENAME="|$OUTFILE|"'
```

To invoke it, set the environment variables before starting the conversion:

Windows:

```
SET INFILE=C:\path\to\cube.plt
tec360 C:\path\to\loaddata.mcr -o output.szplt
```

Linux or Mac (bash shell):

```
INFILE=/path/to/cube.plt
tec360 /path/to/loaddata.mcr -o output.szplt
```

Either absolute or relative paths may be used in environment variables; relative paths are, as always, relative to the current or working directory.

If your file loading process requires you to specify more than one input file to a macro, simply add an `INFILE` variable for each, appending numbers such as `INFILE1`, `INFILE2`, etc. and change your macro accordingly. If you will be using multiple macros, use different variables in each so that they don't conflict.

A Batch File for Conversion

The command line instructions in the preceding section can be put into a batch file (Windows) or a shell script (Linux or Mac) so that you can specify all the necessary paths in a single command line. This file (`convert.bat` or `convert.sh`) can be placed in the same directory as our `loaddata.mcr` from the preceding section, as long as that directory is in your system's `PATH`.

Windows — convert.bat:

```
SET INFILE=%1
tec360 loaddata.mcr -o %2
```

Now you may invoke it as follows:

```
convert.bat C:\path\to\input\file.plt C:\path\to\output\file.szplt
```

Linux or Mac OS — convert.sh:

```
#!/usr/bin/env bash
INFILE=$1
tec360 loaddata.mcr -o $2
```

After creating `convert.sh`, execute the command below to make it executable:

```
chmod +x convert.sh
```

The script may then be invoked as follows:

```
convert.sh /path/to/input/file.plt /path/to/output/file.szplt
```

The same technique can be used with additional macro files and additional filenames; you will simply need to set the environment variable(s) needed by each macro based on the arguments to the script (`%1`, `%2`, etc. in Windows batch files or `$1`, `$2`, etc. in Linux or Mac `bash` scripts).

Writing SZL Files With Macros

The principal difference between `-o` and `-convert` is that when you use `-o`, Tecplot 360 writes SZL data for you, whereas with `-convert`, you write the data file using a macro. All that we have already learned about using macros still applies; the following are valid scenarios:

- A single macro loads all the data and also writes a `.szplt` file.
- You load some data by specifying the filenames on the command line and the rest by specifying one or more macros, the last of which also writes the .`szplt` file.
- Same as above, except the macro that writes the .`szplt` file is a separate macro.
- A group of macros reads multiple groups of data files and writes mulipte .szplt files in a single Tecplot 360 operation (`READDATAOPTION = NEW` is used to start fresh for each group of loaded files).
- Any other combination of data files and macros you can imagine.

You can create a macro file that writes a `.szplt` file by recording a save operation, but here, for the record, is the necessary command:

```
#!MC 1410
$!EXTENDEDCOMMAND
  COMMANDPROCESSORID = 'Tecplot Subzone Data Tools'
  COMMAND = 'WRITEDATASET FILENAME="C:\Users\kelvin\Desktop\out.szplt"'
```

You can adapt this macro to use relative paths or an environment variable, as we have already done with the macros we created to load data. Here is the macro rewritten to use the environment variable `OUTFILE`:

```
#!MC 1410
$!EXTENDEDCOMMAND
  COMMANDPROCESSORID = 'Tecplot Subzone Data Tools'
  COMMAND = 'WRITEDATASET FILENAME="|$OUTFILE|"'
```

The circumstances in which you might use -convert instead of -o are fairly limited. One use might be to write multiple copies of the `.szplt` file, for example to make a backup (though using the `COPY` or `cp` shell command is probably better).

Another use might be to automatically name the `.szplt` file based on some attribute of the loaded data, rather than requiring a name to be specified. Keep in mind that in the restricted macro environment provided when using `-convert`, the ability to inspect and analyze data is extremely limited. However, you could name a file based on the date, or on an auxiliary data element (using the `AUXDATASET`, `AUXFRAME`, or `AUXZONE` intristic variables) — or a combination of these:, such as

```
#!MC 1410
$!EXTENDEDCOMMAND
  COMMANDPROCESSORID = 'Tecplot Subzone Data Tools'
  COMMAND = 'WRITEDATASET FILENAME="|AUXDATASET:caseid| |DATE|.szplt"'
```

The variable `DATASETFNAME` contains the full paths of loaded data file(s) and can be used to give the output file the same base name as an input file. Since this variable can contain multiple paths if you have loaded multiple files, we recommend always using an index with it to specify which path you want; otherwise, the paths of *all* loaded files are substituted into the command, which will cause the save to fail.

```
#!MC 1410
$!EXTENDEDCOMMAND
  COMMANDPROCESSORID = 'Tecplot Subzone Data Tools'
  COMMAND = 'WRITEDATASET FILENAME="|DATASETFNAME[1]|.szplt"'
```

You may use `|DATASETFNAME[2]|`, `|DATASETFNAME[3]|`, and so on to refer to the path of a specific file, indexed by loading sequence.

For a full list of available macro variables, see the Scripting Guide for more information.

PyTecplot

TecPLUS™ subscribers are given access to PyTecplot, a high level API that connects your Python script to the power of the Tecplot 360 visualization engine. Familiarity with Tecplot 360 and the Tecplot 360 macro language is helpful, but not required.

> PyTecplot requires 64-bit Python versions 2.7 or 3.4+ and Tecplot 360 version 2017 R1 or later with TecPLUS™ maintenance service. PyTecplot does not support 32 bit Python.

Please refer to the PyTecplot Guide for installation instructions, environment setup, and the full reference manual. The sections below refer only to the User Interface portions of PyTecplot located in Tecplot 360.

PyTecplot Recording

Tecplot 360's **PyTecplot Recorder** records a Python script as you perform a sequence of actions interactively. Scripts are PyTecplot Python (`.py`) format. PyTecplot Python scripts run outside of Tecplot 360, and may be run from either the command line or any Python IDE.

After recording your script, you can edit your script file with a plain text editor to remove redundant operations, compress repetitive actions into loops, replace references to files with variable names, and otherwise modify the script to make it more generic and flexible.

To record a macro with the **PyTecplot Recorder** dialog, select "Record PyTecplot" from the **Scripting** menu. Specify a file name in the **Write Script (.py)** dialog and click Save to initiate the recording. The **Macro Recorder** dialog remains open during the recording session.

> Auto Redraw is disabled during recording. If necessary, you can manually redraw your plot by clicking the Redraw button in the Plot sidebar.

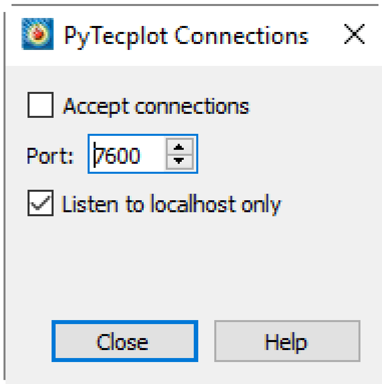
For information on the Macro Recorder dialog, see Macro Creation.

PyTecplot Connections

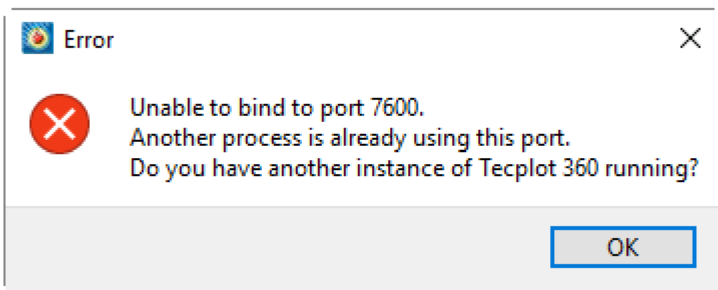
The PyTecplot Connections dialog allows a remote Python instance to control the GUI and interact with the loaded data. Currently, the only available client for this server is PyTecplot, the "tecplot" Python module available on Python's official package repository, PyPI.

Server Configuration

To allow the server to process incoming requests, go to Tecplot 360’s main menu and choose **Scripting→PyTecplot Connections…** and the PyTecplot Connections dialog will appear:



Clicking on the checkbox next to "Accept connections" will enable the server immediately. If the chosen port is already in use by another process, even by another running instance of Tecplot 360, the following error message will be shown:



In this case, you may choose another unused port number and attempt to re-enable the "Accept connections" option.

By default, the TecUtil Server will only listen for requests from localhost. Unchecking the "Listen to localhost only" option will cause the server to process incoming requests from any remote machine, subject to the firewall and security settings of the computer running Tecplot 360.

Connecting from the PyTecplot Client Script

To connect to the server from a Python script, you must have the PyTecplot python package installed. Please see PyTecplot’s documentation for detailed instructions on how to get started with PyTecplot. In the Python script, use the tecplot.session.connect() method to connect to the running instance of Tecplot 360:

```
import tecplot
tecplot.session.connect(port=7600)
```

Once connected, the Python script will continue to run and the Tecplot 360 will be updated in real-time.

Activating Server with a Macro File

To activate the TecUtil Server addon in Tecplot 360 with a macro, add this macro command to a separate `.mcr` file:

```
$!EXTENDEDCOMMAND
 COMMANDPROCESSORID = "TecUtilServer"
 COMMAND = R"(
 AcceptRequests = Yes
 ListenOnAddress = localhost
 ListenOnPort = 7600
 )"
```

To use this function on start-up, simply play the macro file as you launch Tecplot 360:

```
tec360 -p startConnector.mcr
```

Part 7: Advanced Topics

Animation

Tecplot 360 provides a variety of methods for creating animated plots and exporting them to movie or sequences of static image files files for playback at a later time. The basic animation methods available include:

Animation Tools

Create animations using the **Animate** menu, the Slice Details dialog, or the Streamtraces dialog. The animation can be viewed within Tecplot 360 or exported to a movie file.

Movie File Creation with Macros

Use a macro to perform multiple, repetitive changes, and write each image to a movie file.

Animation Tools

Use a built-in animation tool to have Tecplot 360 cycle through your data, automatically displaying zones, slices, or streamtraces one after the other. The following plot elements may be animated using the dialogs in the **Animate** menu:

- Time Animation
- IJK-plane Animation
- IJK Blanking Animation
- Iso-surfaces Animation
- Mapping Animation
- Slice Animation
- Streamtrace Animation
- Zone Animation

> When you need a particular size for your animation image, such as 300 by 250 pixels, first resize your frame to the desired width and height. Then export only the active frame.

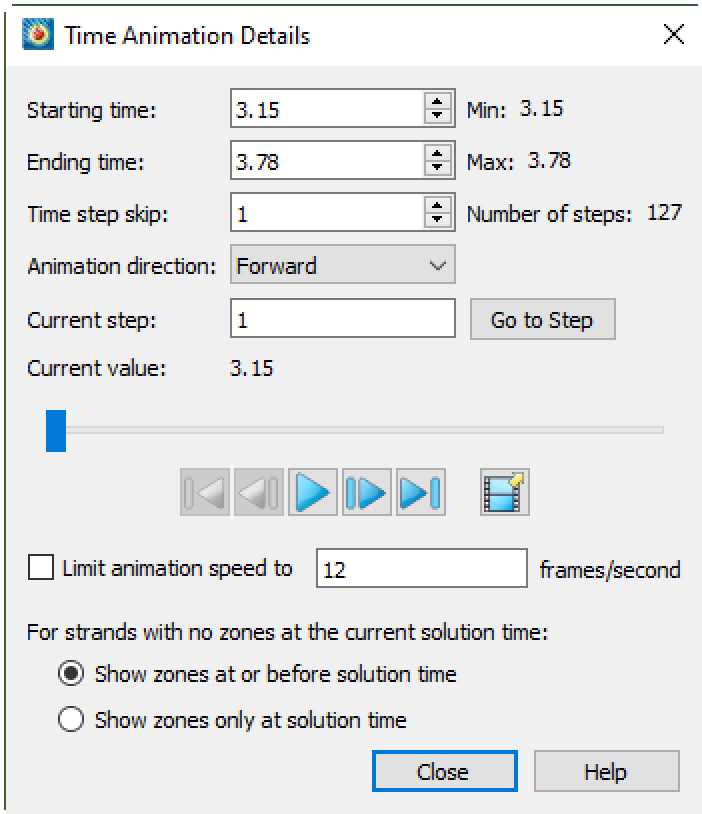
Time Animation

To animate over time, simply press the play button in the Plot sidebar or in the **Time Animation Details** dialog. (This dialog can be launched from the Animate menu or by clicking the button above the Solution Time slider in the Plot sidebar.) The active frame will be animated from the current time step to the last time step. The current Solution Time determines which transient zones are displayed in the active frame. The slider control can be used to interactively change the Current

Solution Time.

This option is available for transient field plot data only.

The **Time Animation Details** dialog has the following options:



Starting Time

Enter the value of the first solution time to include in the animation. If the Solution Time entered does not exist, the nearest Solution Time less than the entered time is used, or the first solution time if no such time exists. The default value is the first Solution Time. The value of the first Solution Time in the dataset is displayed in **Min**.

Ending Time

Enter the value of the last Solution Time to include in the animation. If the Solution Time entered does not exist, the nearest Solution Time less than the entered time is used. The default value is the last Solution Time. The value of the last Solution Time in the dataset is displayed in **Max**.

Time Step Skip

Enter the skip number between time steps. A value of 2 results in every other time steps being animated, a value of 3 animates every 3 time steps, and so on.

Number of Steps

Displays the number of time steps in the data between the Start Time and the End Time.

Animation direction

Select the direction for the animation: forward, backward, loop (forward repeatedly), or bounce (forward, then backward, then forward, then backward…).

Current Step

This field displays the time step for the active frame of the animation. The field is updated while an animation is in progress.

Go To

Use the **Go To** button to jump to the animation step entered in the Current Step field.

Slider

The slider can be dragged to change the current solution time. The following buttons may also be used.

- \- Jumps to the value in Start Time.
- – Moves one step toward the value in Start Time.
- – Plays the animation as specified by the Animation direction field. The Play button becomes a Pause button while the animation is playing.
- – Moves toward the value in End Time.
- – Jumps to the value in End Time.

Click the filmstrip icon to export the animation to a file. See Animation Export. Note that some of the settings in this dialog (such as direction and speed) do not apply when exporting to a file. Instead, you may add these effects later using a video editor, if desired.

Limit Animation Speed to

Toggle-on to limit the animation speed to the value specified in the text box.

For strands with no zones at the current solution time

Determines what is shown when no zones with a given Strand ID exist at the current solution time, within a small time tolerance:

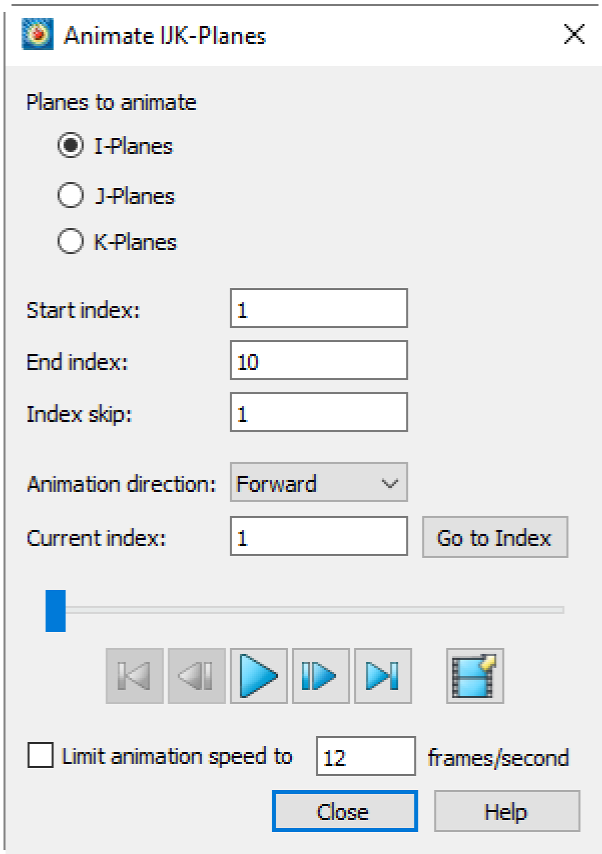
- Show zones at or before solution time (default)- Show zones for the closest prior solution time where the zone with that Strand ID does exist.
- Show zones only at solution time: Only show zones for the current solution time.

Note that, in addition to the above setting, the fieldmap must be active for it to be shown.

IJK-plane Animation

Use the **Animate IJK-Planes** dialog to display all or a specified subset of the IJK-planes in the current dataset, one at a time. You can choose to animate either the I, J, or K-planes.

To animate IJK-planes, select "IJK-planes" from the **Animate** menu. The **Animate IJK-Planes** dialog has the following options:



Planes to Animate

Specify the set of planes to animate: I, J, or K-planes.

Initial Index, End Index, Index Skip

Specify an Initial Index (the first plane you want to display), an End Index (the last plane you want to display), and an Index Skip in the fields provided. A skip of 1 displays every index, 2 displays every other index, etc.

Animation direction

Select the direction for the animation: forward, backward, loop (forward repeatedly), or bounce (forward, then backward, then forward, then backward...).

Current Index

Displays the number of the current plane index, which can be edited. Click **Go to Index** after editing to move to the chosen step.

Slider

The slider can be dragged to change the current plane index. The following buttons may also be used.

-  - Jumps to the first index.

- – Moves one step toward the first index.
- – Plays the animation as specified by the Animation direction field. The Play button becomes a Pause button while the animation is playing.
- – Moves toward the last index.
- – Jumps to the last index.

Click the filmstrip icon to export the animation to a file. See Animation Export. Note that some of the settings in this dialog (such as direction and speed) do not apply when exporting to a file. Instead, you may add these effects later using a video editor, if desired.

Limit Animation Speed to

Toggle-on to limit the animation speed to the value specified.

Figure 74 shows an example of animating I-planes in an IJK-ordered zone.

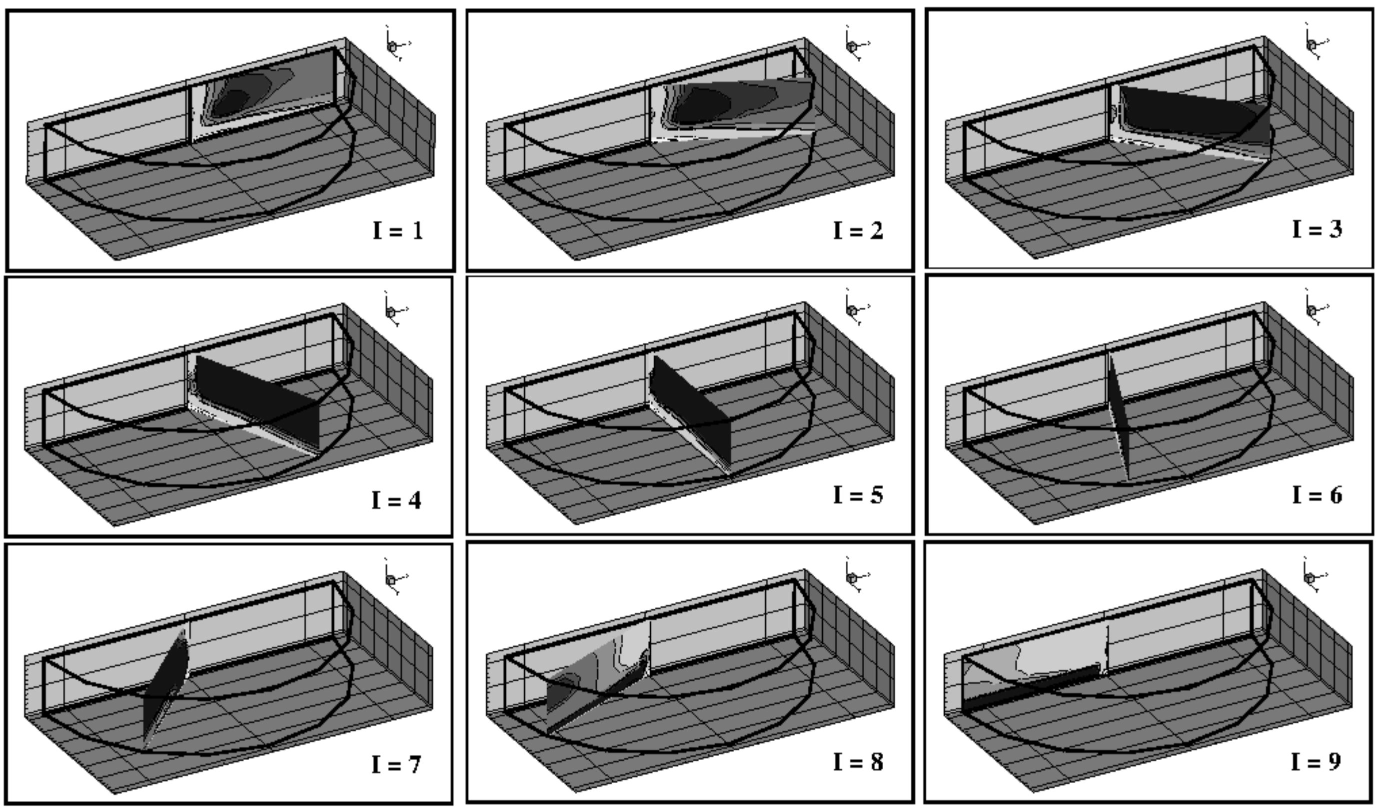


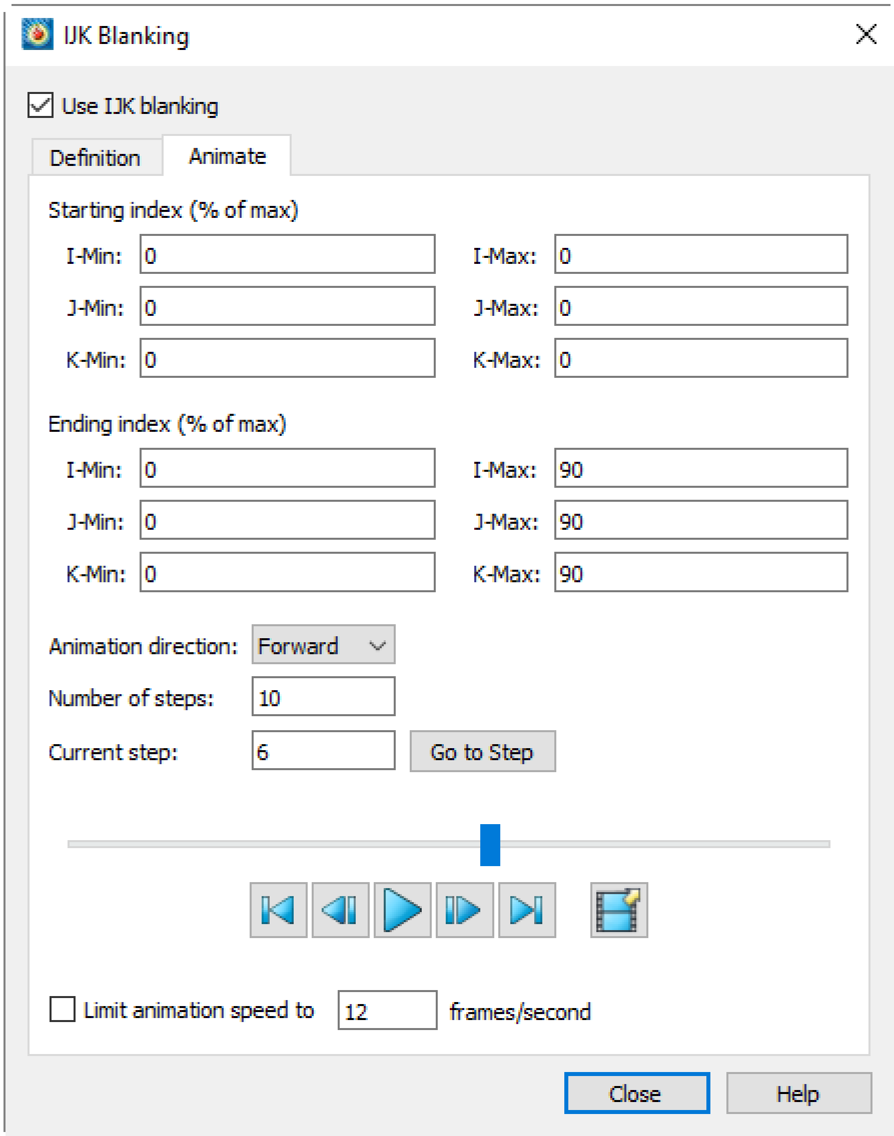
Figure 74. An animated sequence of I-planes.

> To restore the workspace back to its normal view after the IJK animation, go to **Edit**→**Undo Style Change**.

IJK Blanking Animation

Use the Animation page of the **IJK Blanking** dialog (**Plot**→**Blanking**→**IJK Blanking**) to animate a sequence of Tecplot 360 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.

To animate a sequence of IJK blankings, you must first turn on IJK blanking using the checkbox at the top of the IJK Blanking dialog. For more information on initially setting up IJK blanking, see IJK Blanking.



The Animation page of the IJK Blanking dialog has the following settings:

Starting Index (% of Max)

Specify an initial set of blanked IJK-indices in the text fields. Enter a range of indices for: I, J, and K (index values are entered as percentages of the maximum index).

Ending Index (% of Max)

Specify a final set of blanked IJK-indices. Enter a range of indices for each: I, J, and K.

Animation direction

Select the direction for the animation: forward, backward, loop (forward repeatedly), or bounce (forward, then backward, then forward, then backward…).

Number of Steps

Specify the number of steps. The minimum is two.

Current Step

Displays the number of the current step, which can be edited. Click **Go to Step** after editing to move to the chosen step.

Slider

The slider can be dragged to change the current step. The following buttons may also be used.

- - Jumps to the first step.
- – Moves one step toward the first step.
- – Plays the animation as specified by the Animation direction field. The Play button becomes a Pause button while the animation is playing.
- – Moves toward the last step.
- – Jumps to the last step.

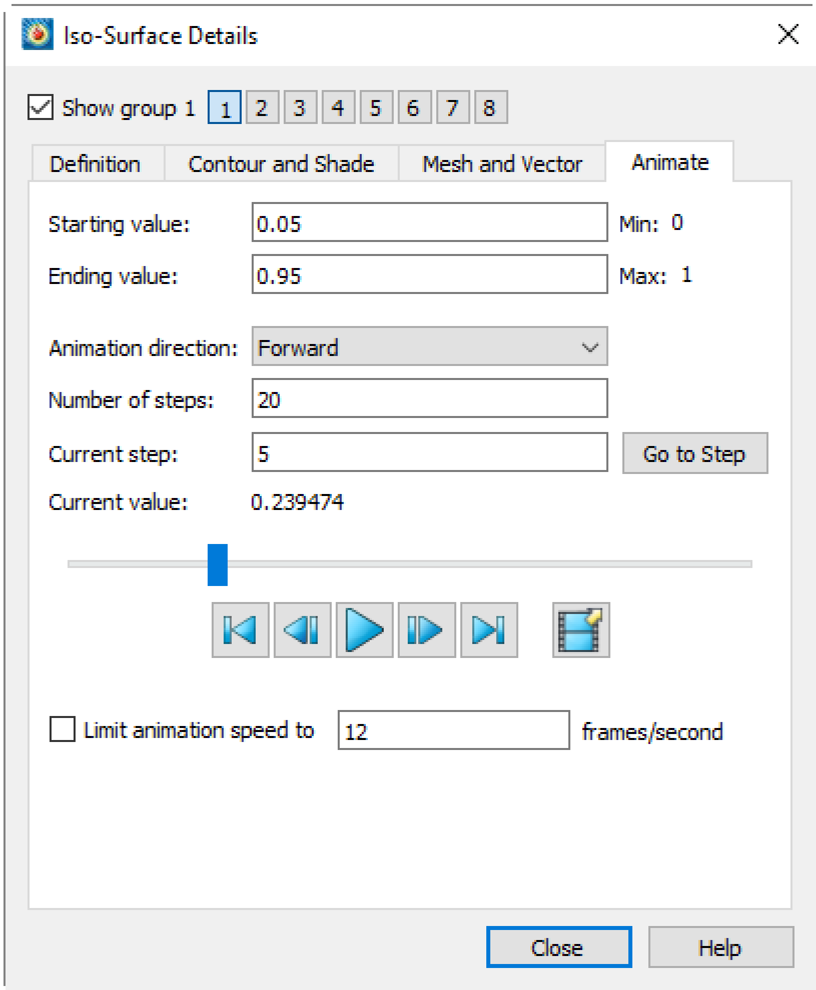
Click the filmstrip icon to export the animation to a file. See Animation Export. Note that some of the settings in this dialog (such as direction and speed) do not apply when exporting to a file. Instead, you may add these effects later using a video editor, if desired.

Limit Animation Speed to

Toggle-on to limit the animation speed to the value specified in the field.

Iso-surfaces Animation

Use the Animation page of the **Iso-surface Details** dialog to define iso-surfaces to animate either on screen or to a file. You can reach this page directly by choosing "Iso-Surfaces" from the **Animate** menu, or you can open the **Iso-Surfaces Details** dialog from the **Plot** menu or Plot sidebar and click the Animation tab.



Starting Value

The starting iso-surface value for the animation.

Ending Value

The ending iso-surface value for the animation.

Animation direction

Select the direction for the animation: forward, backward, loop (forward repeatedly), or bounce (forward, then backward, then forward, then backward…).

Number of steps

The range defined by the Starting and Ending value fields above is divided into the specified number of steps for the animation.

Current Step

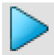
Indicates the active step of the animation. You can jump to a specific step by entering the desired value and clicking **Go to Step**.

Current Value

Displays the iso-surface value at the current step.

Slider

The slider can be dragged to change the current step. The following buttons may also be used.

-  - Jumps to the first step.
-  – Moves one step toward the first step.
-  – Plays the animation as specified by the Animation direction field. The Play button becomes a Pause button while the animation is playing.
-  – Moves toward the last step.
-  – Jumps to the last step.

Click the filmstrip icon to export the animation to a file. See Animation Export. Note that some of the settings in this dialog (such as direction and speed) do not apply when exporting to a file. Instead, you may add these effects later using a video editor, if desired.

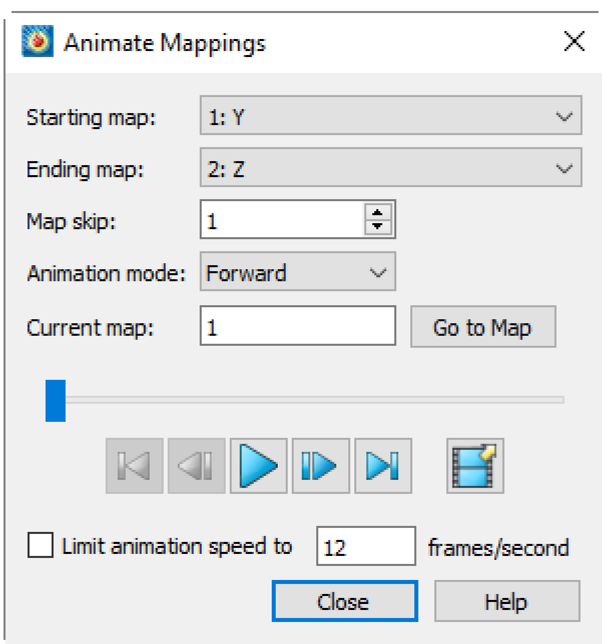
Limit Animation Speed to

Toggle-on to limit the animation speed to the value specified in the field.

Mapping Animation

Use the **Animate Mappings** dialog to display all or a specified subset of the XY or Polar Line mappings defined in the active frame, one at a time.

To animate mapping, select "Mappings" from the **Animate** menu. The **Animate Mappings** dialog has the following options:



Start Map

Specify the first line mapping you want displayed.

End Map

Specify the last line mapping you want displayed.

Map Skip

Specify the number of maps to skip per step.

Animation Mode

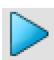
Select whether to animate forward, backward, loop (forward, then back to the beginning for another pass forward), or bounce (forward, then backward, then forward, then backward…).

Current Map

Indicates the map currently displayed. You may jump to a different map by editing the value in the text box and clicking **Go to Map**.

Slider

The slider can be dragged to change the current step. The following buttons may also be used.

- - Jumps to the first map.
-  – Moves one step toward the first map.
-  – Plays the animation as specified by the Animation mode field. The Play button becomes a Pause button while the animation is playing.
-  – Moves toward the last map.
-  – Jumps to the last map.

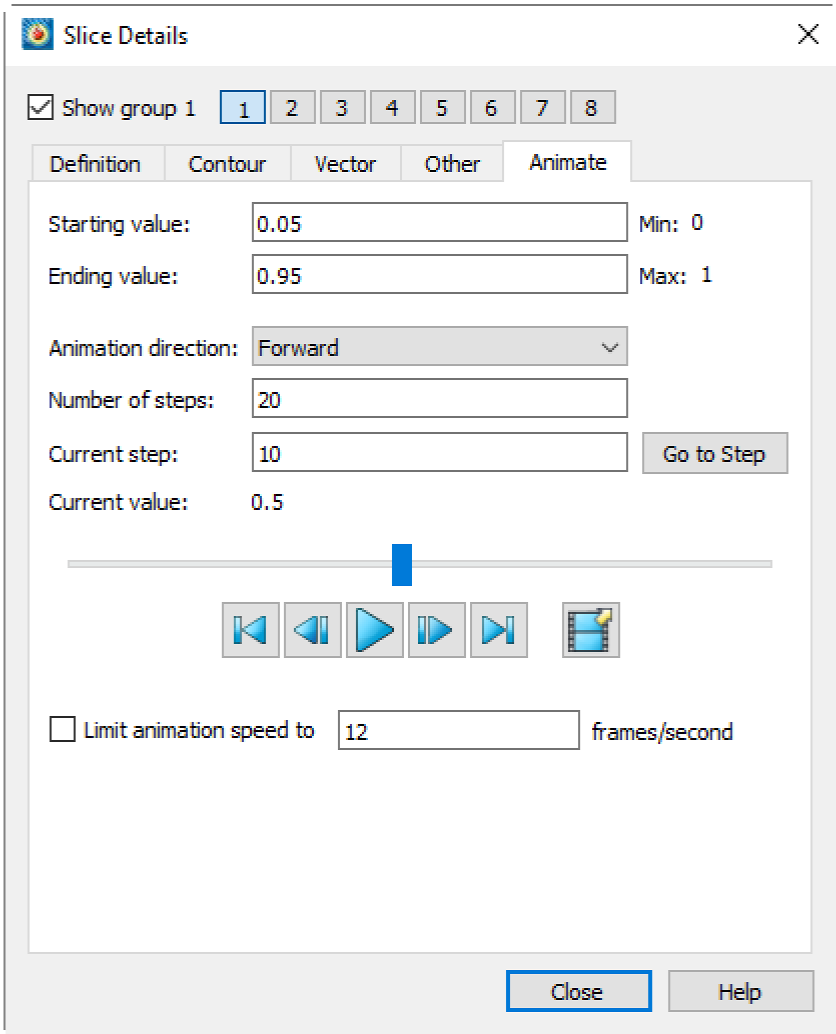
Click the filmstrip icon to export the animation to a file. See Animation Export. Note that some of the settings in this dialog (such as direction and speed) do not apply when exporting to a file. Instead, you may add these effects later using a video editor, if desired.

Limit Animation Speed to

To limit the animation speed, toggle on the checkbox and enter a frame speed value in the text box.

Slice Animation

To animate slices, choose **Animate**→**Slices**. You may also get to this feature by opening the **Slice Details** dialog from the **Plot** menu or the Plot sidebar and choosing the Animate page.



Specify a Starting Value, an Ending Value, and the Number of Steps in the fields provided.

> Only the primary slice of the current slice group (specified on the Position page) is changed during animations. The start and end slice and any intermediate slices of the current slice group are unchanged. It is possible that the animated primary slice will overlap the start slice, end slice, or an intermediate slice. The animation will proceed, without changing those values.

For arbitrary slices, the start and end origin positions are specified in XYZ coordinates. Arbitrary slices are oriented using the normal defined on the Definition page of the Slice Details dialog (see Arbitrary Slice Orientation). The slice's origin point is moved between the start and end origins specified here; at each step of the animation, the slice is drawn through the origin point and oriented against the normal.

After choosing the starting and ending positions and the number of steps, you can animate on the screen or export the animation to a file using the following options:

Animation Direction

Select whether to animate forward, backward, loop (forward, then back to the beginning for another pass forward), or bounce (forward, then backward, then forward, then backward...).

Current Step

Indicates the active step of the animation. This may be edited to jump to a specific step. Click the **Go to Step** button to display the entered time step in the workspace.

Current Value or Origin

Displays the value being varied (e.g. X for an X-plane slice) at the current step. For arbitrary slices, shows the XYZ coordinates of the slice origin (through which the slice will be drawn) at the current step.

Slider

The slider can be dragged to change the current step. The following buttons may also be used.

- - Jumps to the first step.
- – Moves one step toward the first step.
- – Plays the animation as specified by the Animation direction field. The Play button becomes a Pause button while the animation is playing.
- – Moves toward the last step.
- – Jumps to the last step.

Click the filmstrip icon to export the animation to a file. See Animation Export. Note that some of the settings in this dialog (such as direction and speed) do not apply when exporting to a file. Instead, you may add these effects later using a video editor, if desired.

Limit Animation Speed

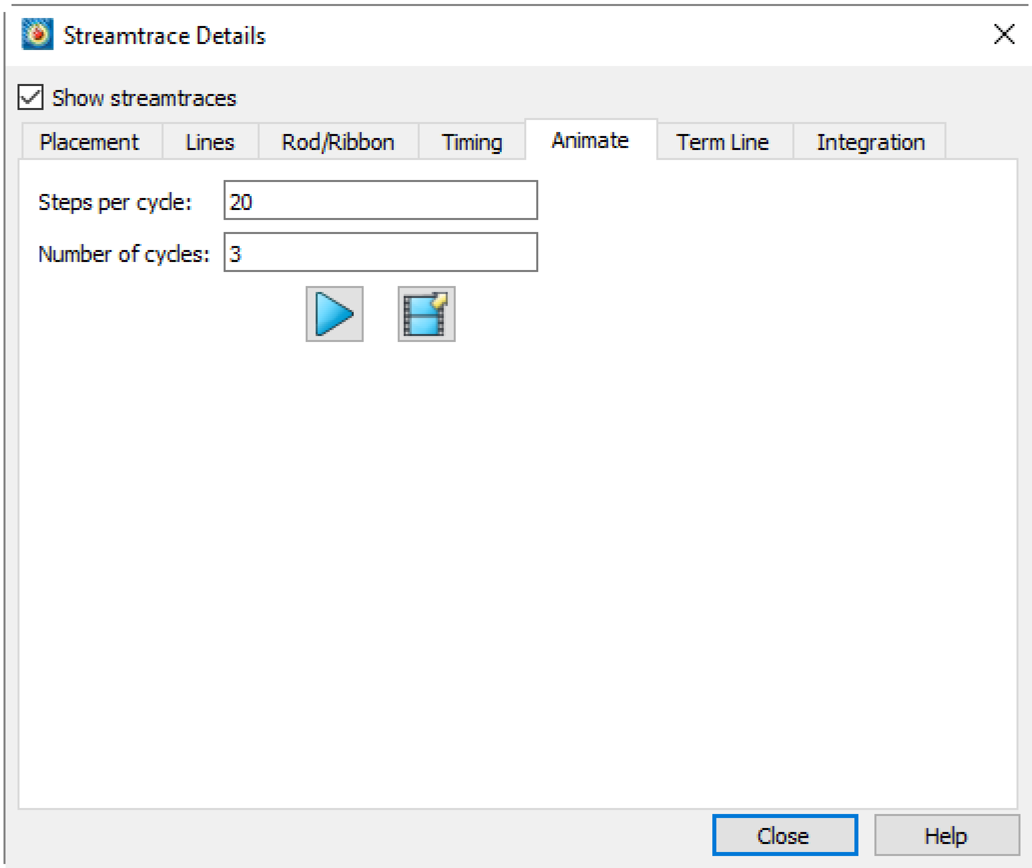
Toggle-on to limit the animation speed to the value specified in the text field.

Streamtrace Animation

To animate your streamtraces, choose **Animate→Streamtraces.** You may also access this feature by opening the Streamtrace Details dialog from the Plot menu or the Plot sidebar and switching to the Animate page.

> When you animate streamtraces, timing markers are automatically turned on. You can change the appearance of these markers in the Streamtrace Details dialog. See Timing Page for details.

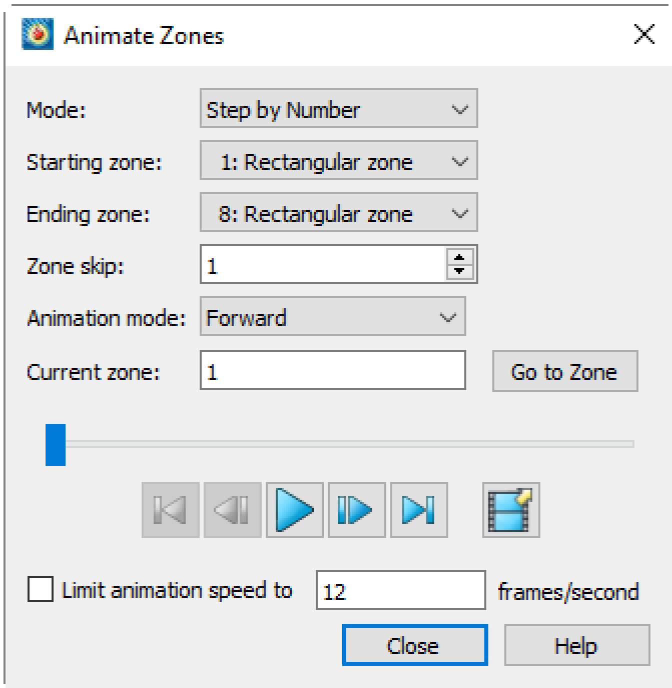
Specify the number of steps per cycle and the number of cycles in the fields provided on the **Animate** page of the **Streamtrace Details** dialog.



Click the Play button to view the streamtrace animation on the screen, or click the filmstrip icon to export the animation to a file. See Animation Export.

Zone Animation

To animate zones, select "Zones" from the **Animate** menu. From the **Zone Animation Details** dialog, specify the animation mode, Start Zone, End Zone, and Zone Skip in the fields provided.



You can animate to a file or "On Screen". (See also Animation Export.)

Use the **Mode** drop-down menu to select one of the following two options:

Step by Number

Animate all zones from the first zone to the last zone with a skip specified in the Zone Skip field. A Zone Skip of 1 animates all zones.

Group Step by Number

Animate zones in groups (as specified by the Group Size field). For example, a Group Size of 2 will animate all zones in groups of 2 (i.e. zones 1 and 2, followed by zones 3 and 4, and so on).

The following fields are also available:

Starting zone

The first zone in the animation.

Ending zone

The last zone in the animation.

Zone Skip or Group Size

The label on this field changes depending on the setting of the Mode drop-down menu, described previously. Decides the increment for switching zones (for Step by Number) or the size of the zone group (for Group Step by Number).

Animation Mode

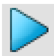
Select whether to animate forward, backward, loop (forward, then back to the beginning for another pass forward), or bounce (forward, then backward, then forward, then backward…).

Current Zone

Indicates the zone displayed. This may be edited to jump to a specific zone; click Go to Zone after editing.

Slider

The slider can be dragged to change the current step. The following buttons may also be used.

-  - Jumps to the first zone.
-  – Moves one step toward the first zone.
-  – Plays the animation as specified by the Animation Mode field. The Play button becomes a Pause button while the animation is playing.
-  – Moves toward the last zone.
-  – Jumps to the last zone.

Click the filmstrip icon to export the animation to a file. See Animation Export. Note that some of the settings in this dialog (such as direction and speed) do not apply when exporting to a file. Instead, you may add these effects later using a video editor, if desired.

Limit Animation Speed

Toggle-on to limit the animation speed to the value specified in the text field.

The Animate Zones dialog is not available for transient datasets.

Movie File Creation with Macros

The Tecplot macro language expands the capabilities of the standard animation features. The macro commands allow you to do anything you can do interactively—and more. You can also use loops to repeatedly rotate 3D objects, cycle from one active zone to another, and so on, to create your movie. See Macros for further information about the Tecplot macro language.

A typical macro file for making movies has the following form:

```
#!MC 1410
... optional commands to set up the first image
$!EXPORTSETUP
  EXPORTFORMAT = AVI
  EXPORTFNAME = "mymovie.avi"
$!EXPORTSTART
  EXPORTREGION = CURRENTFRAME
$!LOOP 50
  ... commands to set up next image
  $!REDRAWALL
  $!EXPORTNEXTFRAME
$!ENDLOOP
```

```
$!EXPORTFINISH
```

For example, the following macro file can be used to animate zones:

```
#!MC 1410
#
# Set up Export file type and file name.
#
$!EXPORTSETUP
  EXPORTFORMAT = AVI
  EXPORTFNAME = "C:\temp\timeseries.avi"
#
# Begin Animating
#
$!LOOP |NUMZONES|
  #
  # 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
      EXPORTREGION = CURRENTFRAME
  $!ENDIF
  $!IF |Loop| != 1
    $!EXPORTNEXTFRAME
  $!ENDIF
$!ENDLOOP
$!EXPORTFINISH
```

Advanced Animation Techniques

Text Changes

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 two ways this can be done.

Using Dynamic Text

The best way to do this is to add dynamic text to your text box. See Dynamic Text.

Attaching Text to Zones

This method works best if you are animating zones. First, create several text strings in your data file, and use the `ZN=` parameter to attach each text string to a zone or mapping. You should have a separate text string for each zone that will be used in your animation. For example

```
ZONE T= "Temp. distribution, Distance = 0.5 m" I=51, J=51 F=POINT
.
.
.
.....list of variable values........
.
.
.
TEXT X=70, Y=90, T= "Distance = 0.5 m", F=COURIER, CS=FRAME, H=2, ZN=1
ZONE T= "Temp. distribution, Distance = 1.0 m" I=51, J=51 F=POINT
.
.
.
.....list of variable values........
.
.
.
TEXT X=70, Y=90, T= "Distance = 1.0 m", F=COURIER, CS=FRAME, H=2, ZN=2
```

You can also use Tecplot 360’s dynamic text feature (see Dynamic Text) to insert a zone name into your text strings. For example:

```
ZONE T= "Distance= 1.0 m" 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
```

Multiple Frame Animation

Synchronized animation of plots in multiple frames requires the use of a macro. Macros can use the

`$!FRAMECONTROL` commands to switch between each frame. The following template demonstrates how this is done with a layout where each frame contains a similar plot:

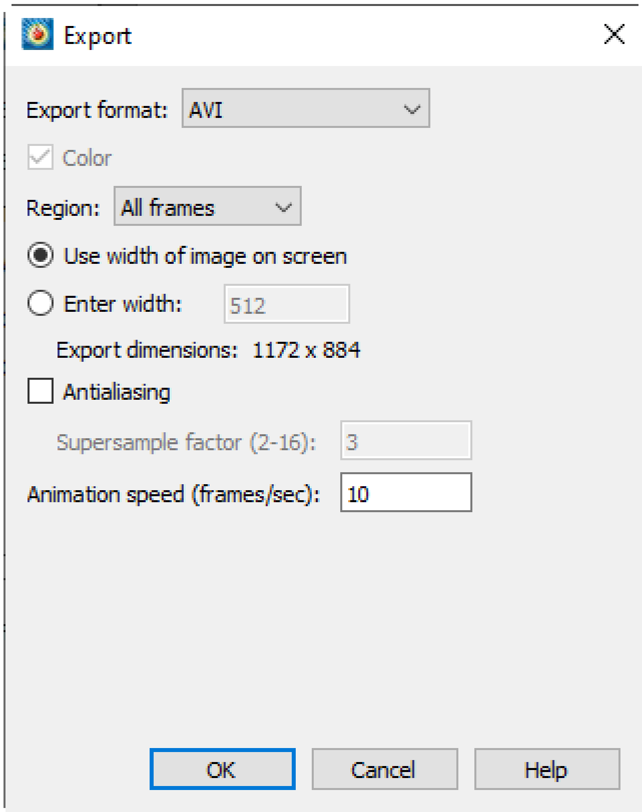
```
#!MC 1410
##Set the number of images (movie frames) in the animation.
$!VARSET |NumCycles| = 10
$!EXPORTSETUP
  EXPORTFORMAT = RASTERMETAFILE
  EXPORTSETUP EXPORTFNAME = "2frames.rm"
  BITDUMPREGION = ALLFRAMES
.
.....Insert commands to set up first frame, if necessary.
.
## Outer loop
$!LOOP |NumCycles|
  ## Inner loop cycles through each frame in the current layout.
  $!LOOP |NumFrames|
    .
    .....Insert commands here to change the plot in the active frame.
    .
    ## Push the top (active) frame to the back.
    $!FrameControl MoveToBottomByNumber
      Frame = |NumFrames|
    #
    ## Activate new top frame
    $!FrameControl ActivateTop
  $!EndLoop
  $!IF |Loop| == 1
    $!EXPORTSTART
      EXPORTREGION = CURRENTFRAME
  $!ELSE
    $!EXPORTNEXTFRAME
  $!ENDIF
$!ENDLOOP
$!EXPORTFINISH
```

Animation Export

When you choose to animate to a file in a dialog with animation options, the Export dialog appears. Choose the desired export format from the menu at the top of the dialog, then set the options as desired. Two different types of export formats are available:

- Movie formats (e.g. mpeg-4), which produce a single file for a given animation.
- Sequenced image formats (e.g. png etc.), which produce a sequence of static image files for a given animation. See Sequenced Image Files.

Most of the available formats share a set of similar options, which are described here.



Region

Select the region of the workspace to animate.

Current Frame

Captures only the active frame.

All Frames

Captures the smallest rectangular area containing all frames.

Work Area

Captures 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.

Enter Width

Select this option to specify a width (in pixels) for the generated image. A larger width increases the quality of your image. However, the greater the width you specify, the longer it will take to export the image and the larger the exported file.

Antialiasing

Select this option to smooth jagged edges in the image. The image is rendered at a higher resolution than that at which it will eventually be displayed, then reduced in size.

Supersample Factor

Control the amount of antialiasing used in the image. The higher this number, the larger the temporary image used for antialiasing will be, and the more time may be required.

Animation Speed (frames/sec)

Enter a value in the text field to set your animation's speed in frames per second.

The following sections describe any settings unique to specific formats, as well as providing additional useful information about each format (references, viewers, etc.).

- AVI Files
- Flash Files
- MPEG-4 Files
- Raster Metafiles

Performance Tips

If exporting is taking an unusually long time, or you get an error message saying that the animation cannot be exported, the most likely cause is that the width of the image you are trying to export is too large. Choosing a smaller width may greatly speed up the export process. Note that antialiasing multiplies the image's width and height by the entered supersampling factor, so choosing a lower factor also speeds up export.

For an export size of Length x Width, with a supersampling factor of SSF, the size of an uncompressed true color frame is approximately Length x SSF x Width x SSF x 3. Memory requirements to export such a frame can be up to twice this.

AVI Files

AVI format is an older video format for Windows platforms, and is widely compatible with Windows applications. Below are some applications that can be used to view and/or edit AVI files:

Windows Media Player

A standard movie viewer included with Windows distribution. More information is available at support.microsoft.com/en-us/windows/windows-media-technologies-ebf04fdc-dc47-9944-c160-ae22d8413116

QuickTime

Included with Mac, Apple's QuickTime can play AVI files and export them to other formats as necessary.

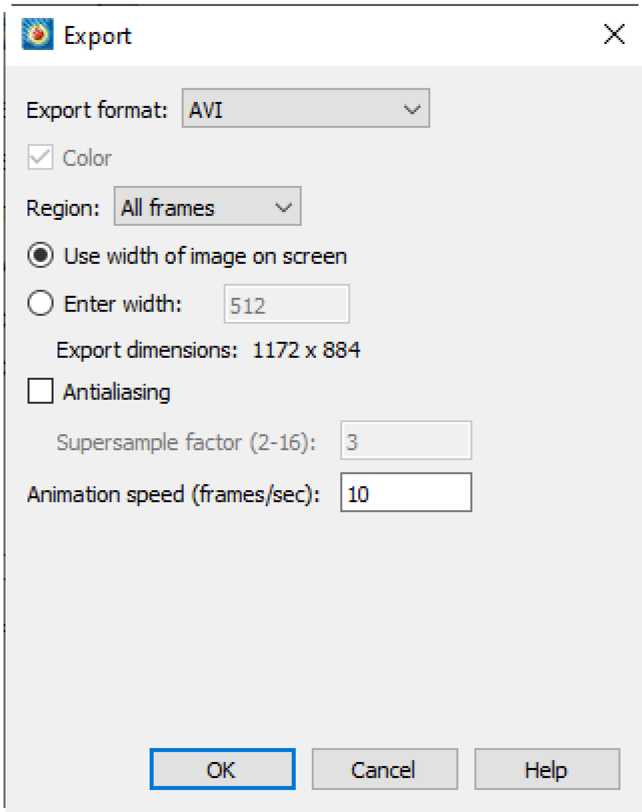
VideoLan Client (VLC)

Free player that supports many video formats, including AVI, on Windows, Mac, Linux, and several other platforms. www.videolan.org/vlc/

Adobe® Premier®

A powerful tool for professional digital video editing. More information is available at www.adobe.com/products/premiere.html.

AVI Export Options



For options common to all supported export formats, see Animation Export.

Flash Files

Adobe® Flash® is a popular tool for creating interactive Web content, including animations and movies. The Flash product is not required to create Flash-format movies using Tecplot 360, nor to view them. Flash files have a filename extension of `.swf` and are therefore also widely referred to as SWF files.

Playback

- Flash movies can be played in several freely distributed Flash players. For example, Swiff Player is a stand-alone player for Windows that enables Flash users to easily play their Flash movies.
- You can play Flash movies using the QuickTime software included with Mac.
- There are several tools at download.com that can help manage, browse, convert, and display all kinds of Flash files on your computer.

Flash in PowerPoint

The easiest way to insert and SWF into Microsoft® PowerPoint® presentations is to download ShowRoom, a Microsoft PowerPoint add-in that incorporates a free Flash player formerly called Swiff Point Player. It's from the same developer as the standalone Swiff Player described under Playback, above. (Some other features of ShowRoom are not free of charge, but these need be paid for only if you want to use them.)

A secondary option is to play it in a PowerPoint presentation using a Microsoft ActiveX control and the Macromedia Flash Player. To run the Flash file, you add an ActiveX control to the PowerPoint slide and create a link from it to the Flash file. You also have the option of embedding the Flash file in the presentation; see:

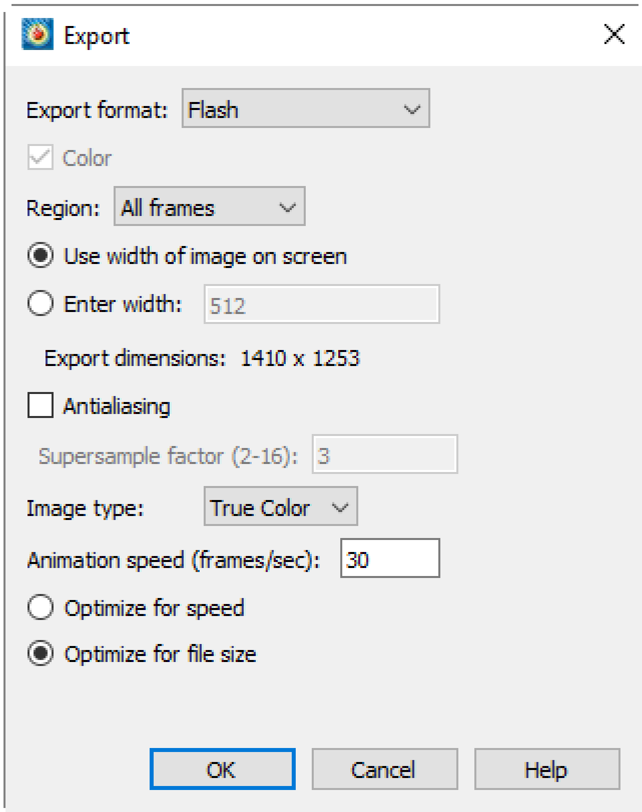
https://support.office.com/en-us/article/Play-an-Adobe-Macromedia-Flash-animation-in-a-presentation-3b681b8e-6ca9-413f-b41a-eb748babc588

Flash on the Web

Flash files can be inserted into Web documents using Web design tools such as Adobe® Dreamweaver® (although this is only one such tool). You can also create Web documents by writing HTML directly in a text editor.

Once inserted in a Web page, Flash movies play directly within your browser, assuming the Flash plug-in is installed. (Most browsers on today's desktop operating systems come with the Flash plug-in pre-installed; if your browser did not, you can easily install it.) An outside media player is not needed to launch the animation.

Flash Export Options



The following options are unique to Flash files. For options common to all supported export formats, see Animation Export.

Image Type

Choose an image type. Your options are:

True Color

Select this option to create twenty-four-bit images with lossless (ZLIB) compression.

JPEG

Select this option to create twenty-four-bit images with lossy compression. This produces smaller files than True Color, and the images will be of lower quality.

256 Colors

Select this option to reduce each image to 256 colors and compress it with ZLIB.

Optimize for Speed

For True Color or 256 Colors image types, select this option to create the output as quickly as possible. This reduces the compression level and results in larger files. It does not affect playback speed.

Optimize for File Size

For True Color or 256 Colors image types, select this option to produce the smallest possible files. This setting does not affect playback speed.

MPEG-4 Files

MPEG-4 is the modern ISO standard for video playback. MPEG-4 files can be played by a wide variety of computer software, portable devices, and set-top boxes. We suggest:

QuickTime

Included with Mac, Apple's QuickTime can play MPEG-4 files and export them to other formats as necessary.

Windows Media Player

A standard movie viewer included with Windows distribution. More information is available at support.microsoft.com/en-us/windows/windows-media-technologies-ebf04fdc-dc47-9944-c160-ae22d8413116.

VideoLan Client (VLC)

Free player that supports many video formats, including MPEG-4, on Windows, Mac, Linux, and several other platforms. www.videolan.org/vlc/

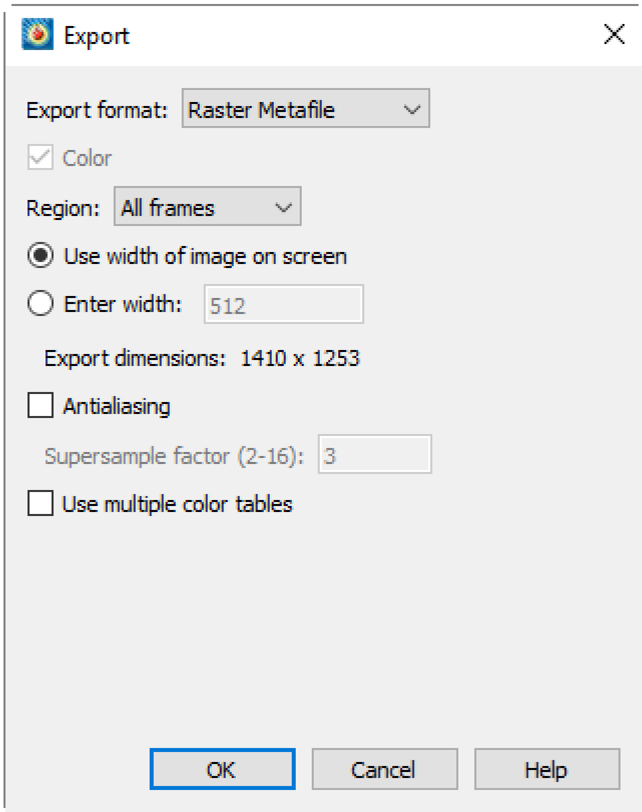
MPEG-4 export does not have any options besides the ones that apply to all animation export formats; see Animation Export.

Windows Media (WMV) Files

Windows Media Video (WMV) files are widely supported by Windows applications, including Windows Media Player and Microsoft Office (PowerPoint, etc.). They may also be played on Macintosh computers using Microsoft's free Windows Media Components for QuickTime.

WMV export does not have any options besides the ones that apply to all animation export formats; see Animation Export.

Raster Metafiles



Raster Metafile is a NASA-defined standard format for storing bitmap images and may contain one or more images, allowing it to be used for animations. The 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.

Raster Metafiles support only 256-color images (Tecplot 360 selects the best colors). Plots with many colors may have poor results when exported as Raster Metafiles when compared to exports as true-color formats. Using Raster Metafiles for animations with transparency or smooth color gradients may result in poor image quality.

When you select Raster Metafile in the **Export** dialog, you have the following additional options:

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 360 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.

Sequenced Image Files

With sequenced image formats, Tecplot will produce separate image files (e.g. png, tiff, etc.) with a sequence number appended to the body of the filename. For example, if you specify to output to "mymovie.png", the following files will be generated:

```
mymovie_000001.png
mymovie_000002.png
... and so on.
```

Note that every time an animation is exported with sequenced image files the files will be numbered starting at 1 and incrementing by 1. Even if you are doing something like a J-Planes animation and are starting at a J-Plane other than 1.

The following image file formats are available when generating animations: BMP, JPEG, PNG, and TIFF. All settings for these formats when making animations is the same as are available when exporting a single image (via the File/Export dialog). See the corresponding section for each format:

BMP: BMP Export
JPEG: JPEG Export
PNG: PNG Export
TIFF: TIFF Export

Customization

This chapter discusses how custom files are loaded on startup, how to manually open those files via the command line, the customizations that can be made by editing the Tecplot 360 configuration file, and customizations that can be made using the **Options** menu.

Custom Files loaded on Startup

On startup, Tecplot 360 will load certain configuration files. These files are editable and can be placed in different locations depending on preference. The different files types are as follows:

- tecplot.add (Add-on Loading)
- tecplot.cfg (Configuration Files)
- tecplot.fnt (Custom Character and Symbol Definition)
- tecplot.mcr (Quick Macro Panel)
- tecplot_latex.mcr (LaTeX Expressions)
- variable_aliases.txt (Creating and Using a Variable Alias File)

When Tecplot 360 first launches, it looks for the above tecplot* files in the following locations, in the order listed below. Tecplot* specifies any of the above filenames.

- The file tecplot* specified on the command line. (See Loading custom files via the Command Line.)
- The file tecplot* in the current working directory.
- The file tecplot* in your home directory when using -h flag on the command to set the home directory.

- The file tecplot* in the Users home directory. On Mac and Linux, the tecplot* file must be preceded by a period in the users home directory. For example, .tecplot.add.
- The file tecplot* in the Tecplot 360 installation directory.

The first file found in the above search order is used; others are ignored even if they exist. To see what tecplot* files are being loaded, in 360 go to **Help**→**About Tecplot 360** (on Mac, **Tecplot 360**→**About Tecplot 360**) and look for the files in the Runtime environment.

Loading custom files via the Command Line

You can specify a different custom file by adding an option flag in front of the custom file name to the command line when launching Tecplot 360.

The following command starts Tecplot 360 and installs the custom file specified at startup.

tec360 *-flag tecplot**

Where *-flag* is the flag of the certain file you want to load. Refer to the table below for the correct flag to be used.

-addonfile *filename*	Supply a custom list of addons via the *tecplot.add* file.
-c *cfgfile*	Use *cfgfile* for the configuration instead of the default configuration file, *tecplot.cfg*.
-f *fontfile*	Use *fontfile* instead of the default font file, *tecplot.fnt*.
-p *scriptfile*	Use *scriptfile* like tecplot_latex.mcr instead of the default, *tecplot_latex.mcr*.
-qm *quickpanelfile*	Place the macro functions in *quickpanelfile* in the Quick Macro Panel, instead of using the macros from the default file, *tecplot.mcr*.

To see an example of any of these files, open the file in the your Tecplot 360 installation.

Configuration Files

A Tecplot 360 configuration file is a special type of macro file that Tecplot 360 reads on start up. Use customized configuration files to override any or all of Tecplot 360’s factory default settings.

You can create a configuration file from scratch using any plain text editor, or by editing a copy of an existing configuration file.

A configuration file should include only those options for which you want to override defaults.

See Custom Files loaded on Startup for information on how it is loaded.

Editing Configuration File

You are not limited to merely customizing those settings that appear in an existing configuration file. Most settings that can be modified by the `$!Field`, `$!LineMap`,or `$!Interface` macro commands can be changed in the configuration file directly. The `$!LIMITS` macro command can be used in the configuration file only.

The simplest way to do this is to create a layout or macro with the settings you want, then copy and paste the appropriate commands into your configuration file. See the Scripting Guide for complete details on macro commands.

SetValue Commands

SetValue Commands are macro commands used to specify the value of a given plot attribute. You may add SetValue commands to your `tecplot.cfg` file to override any of Tecplot 360's default settings. For example, suppose you want your 2D axes to appear cyan. You can add this preference to your configuration file as follows:

1. Using the Tecplot 360 interface, create a 2D plot with cyan axes while either recording your steps as a macro, or else save the resulting plot as a Tecplot layout.
2. Edit the resulting macro or layout, scanning for the lines that set the 2D axis colors. The following example shows the commands that specify the X- and Y-axis details in a layout of a 2D plot with cyan axes:

```
$!TWODAXIS XDETAIL{RANGEMIN = -3}
$!TWODAXIS XDETAIL{RANGEMAX = 15}
$!TWODAXIS XDETAIL{GRIDLINES{SHOW=YES}}
$!TWODAXIS XDETAIL{AUTOGRID=NO}
$!TWODAXIS XDETAIL{GRSPACING = 5}
$!TWODAXIS XDETAIL{GRIDLINES{COLOR = CYAN}}
$!TWODAXIS YDETAIL{GRIDLINES{SHOW = YES}}
$!TWODAXIS YDETAIL{GRIDLINES{COLOR = CYAN}}
```

3. Discard everything but the lines that actually set the color:

```
$!TWODAXIS XDETAIL{GRIDLINES{COLOR = CYAN}}
$!TWODAXIS YDETAIL{GRIDLINES{COLOR = CYAN}}
```

4. Paste the resulting lines into your configuration file.

Plot Default Setting - FIELDMAP and LINEMAP

A single `$!FIELDMAP` command can be included to set plot defaults. The zone cannot be specified in the configuration file, and the command is not effective for values set dynamically by Tecplot 360, 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.

```
$!FIELDMAP CONTOUR{CONTOURTYPE = FLOOD}
$!FIELDLAYERS SHOWSCATTER = YES
$!FIELDMAP SCATTER{SYMBOLSHAPE{GEOMSHAPE = DEL}}
$!FIELDMAP SCATTER{FRAMESIZE = 1.8}
```

In the same way as above, 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}
$!LINEPLOTLAYERS SHOWSYMBOLS = YES
$!LINEMAP SYMBOLS{SYMBOLSHAPE{GEOMSHAPE = CIRCLE}}
$!LINEMAP SYMBOLS{FILLMODE = USELINECOLOR}
```

Override Automatic View→Fit

When loading a 3D plot, Tecplot 360 automatically fits the plot to the frame. To revert to the old Tecplot 360 behavior, which used a fixed zoom factor, remove the `#` from the following line in your tecplot.cfg file:

```
#$!FrameSetup Initial3DScale = 0.7
```

Interface Configuration

The many members of the `$!INTERFACE` macro help you configure Tecplot 360's user interface and graphics drawing capabilities. Although some of these commands can be executed in any macro, the best place to put these is in the Tecplot 360 configuration file: `tecplot.cfg`. Below are a few examples. Refer to the Scripting Guide for a complete listing.

General Interface Configuration Options

`$!INTERFACE` followed by:

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 Scripting Guide. These commands can only be executed in the Tecplot 360 configuration file.

USESTROKEFONTSONSCREEN = (YES, NO)

If set to `YES`, all text drawn in the work area will be drawn using Tecplot 360's internal stroke fonts. If set to `NO`, the native True Type fonts will be used instead.

USESTROKEFONTSFOR3DTEXT = (YES, NO)

If set to `YES`, all 3D text drawn in the work area will be drawn using Tecplot 360's internal stroke fonts. 3D text consists of ASCII scatter symbols and node and cell labels when the active plot type is 3D Cartesian. For 3D text, this setting overrides the setting of `USESTROKEFONTSONSCREEN`. If set to `NO`, TrueType fonts will be used instead.

OpenGL-Specific Configuration Options

Several options are available to further tune Tecplot 360 to operate with the OpenGL capabilities of your platform. To assign values to these parameters you must use the `$!INTERFACE OPENGLCONFIG` command. A complete list of these options is given in the Scripting Guide.

`$!INTERFACE OPENGLCONFIG` followed by:

{ALLOWHWACCELERATION = (YES, NO)}

In some cases, bugs in OpenGL drivers cause problems in Tecplot 360. In these situations, Tecplot 360 will typically behave better if this options is set to `NO`. However, Tecplot 360 will also be slower.

{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.

{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.

{SCREENRENDERING {MAXMULTISAMPLES = <*integer*>}}

Specifies the number of multisamples to be used for antialiasing displayed images. The default is 4. A value of 0 may be used for faster rendering (with some roughness in the display). Higher values may be used at some cost in performance.

{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 360.

{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 to exporting images from Tecplot 360.

{IMAGERENDERING {MAXMULTISAMPLES = <*integer*>}}

Specifies the number of multisamples to be used for antialiasing exported images. The default is 4.

{PRETRANSLATEDATA = (AUTO, ON, OFF)}

Applies a translation to data before sending it to OpenGL for rendering, reducing jitter when data is far from the origin but has small differences between data points. The default is `AUTO`, which applies the translation automatically when appropriate.

Default Temporary Directory

Tecplot 360 writes out a number of temporary files. To tell Tecplot 360 where to place these files, put the following macro command in the `tecplot.cfg` file

```
$!FILECONFIG TEMPFILEPATH = "tempfilepath"
```

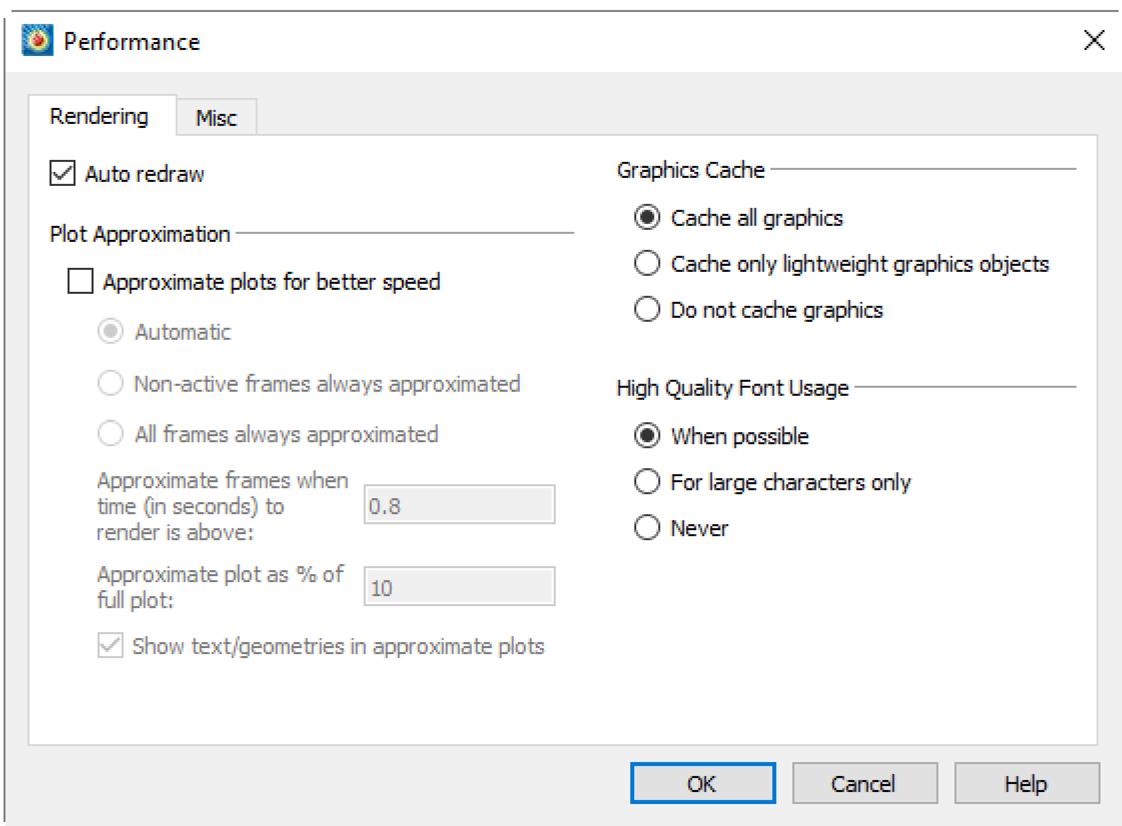
where `tempfilepath` is the new path. The default path is system dependent.

Performance Dialog

Use the Rendering Settings page of the **Performance** dialog (accessed via the **Options** menu) to adjust the Plot Approximation and Graphics Cache. Use the Miscellaneous Settings page of the **Performance** dialog (accessed via the **Options** menu) to adjust Data I/O, Load On Demand, Variable Derivation, and Status Information.

Rendering Settings

The Rendering page of the **Performance** dialog has the following options:



Auto Redraw

When selected, Tecplot 360 will automatically redraw the plot whenever style or data changes. Some users prefer to turn this option off while changing multiple style settings, and then manually pressing Tecplot 360’s [Redraw] button on the Plot sidebar to see the full plot.

Auto-redraw can be interrupted with a mouse click or key press.

Plot Approximation

Approximate Plots for Better Speed

An approximate plot may be used when manipulating the plot to improve interactive performance. This feature is most useful on older, slower hardware and defaults to off. Approximate plots may also be used for non-active frames. The degree of detail of the approximation is controlled by the following settings:

Automatic

When the time to render (in seconds) is above the set threshold, Tecplot 360 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-Active Frames Always Approximated

When only one frame exists, this option is equivalent to automatic mode. If more than one frame exists, the active frame is set to automatic mode while the other frames are approximated.

All Frames Always Approximated

When the number of data points is above the point threshold, Tecplot 360 will render the approximate plot in any frame. To see the full representation press the **Redraw** button on the Plot sidebar.

Approximate Frames when Time (in seconds) to Render is Above

Use this value to determine when to automatically turn on plot approximation for all frames. All frames will be approximated when the time to render the plot (all frames) while doing interactive view changes (rotation, translation, scaling) is greater than the supplied value. This setting does not apply when using the "All Frames Always Approximated" mode.

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. However, the interactive performance is reduced. This number should be adjusted until there is a balance between good interactive performance and sufficient detail. Typically, 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.

Graphics Cache

Tecplot 360 uses OpenGL to render plots. OpenGL provides the ability to cache graphic instructions for rendering and can re-render the cached graphics much faster. 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.

Use one of the following Graphics Cache modes to optimize your computer’s performance:

Cache All Graphics

When selected, Tecplot 360 assumes that there is enough memory to generate the graphics cache. If this is valid, Tecplot 360’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. This is a good setting for memory constrained problems. Consider using this option in conjunction with the "Plot Approximation" mode set to "All Frames Always Approximated".

Do Not Cache Graphics

Consider using this option when memory is very limited. If you intend to interact with the plot, also consider setting the "Plot Approximation" mode set to "All Frames Always Approximated".

> In order to optimize animation, graphics caching is temporarily disabled during animations that include zones, line mappings, time, or blanking elements. Graphics caching is not altered during animations that include slices, streamtraces, or iso-surfaces.

High Quality Font Usage

Tecplot 360 supports high quality TrueType fonts and can use any TrueType font installed on your system. (See Font Folders and Fallback for more information on how fonts work with Tecplot 360.)

Tecplot 360 has three high quality font modes:

When Possible (Default setting for Windows)

Tecplot 360 uses available TrueType fonts for any size text. This produces the best rendering quality; however, performance is slower for large amounts of text.

For Large Characters Only

Tecplot 360 uses the TrueType fonts for large characters only. Small characters will use Tecplot 360’s built-in stroke fonts. This is a good blend of quality and performance. However, small characters may not have the same appearance as large characters.

Never

Tecplot 360 never uses TrueType fonts.

Best Practices For Rendering Performance

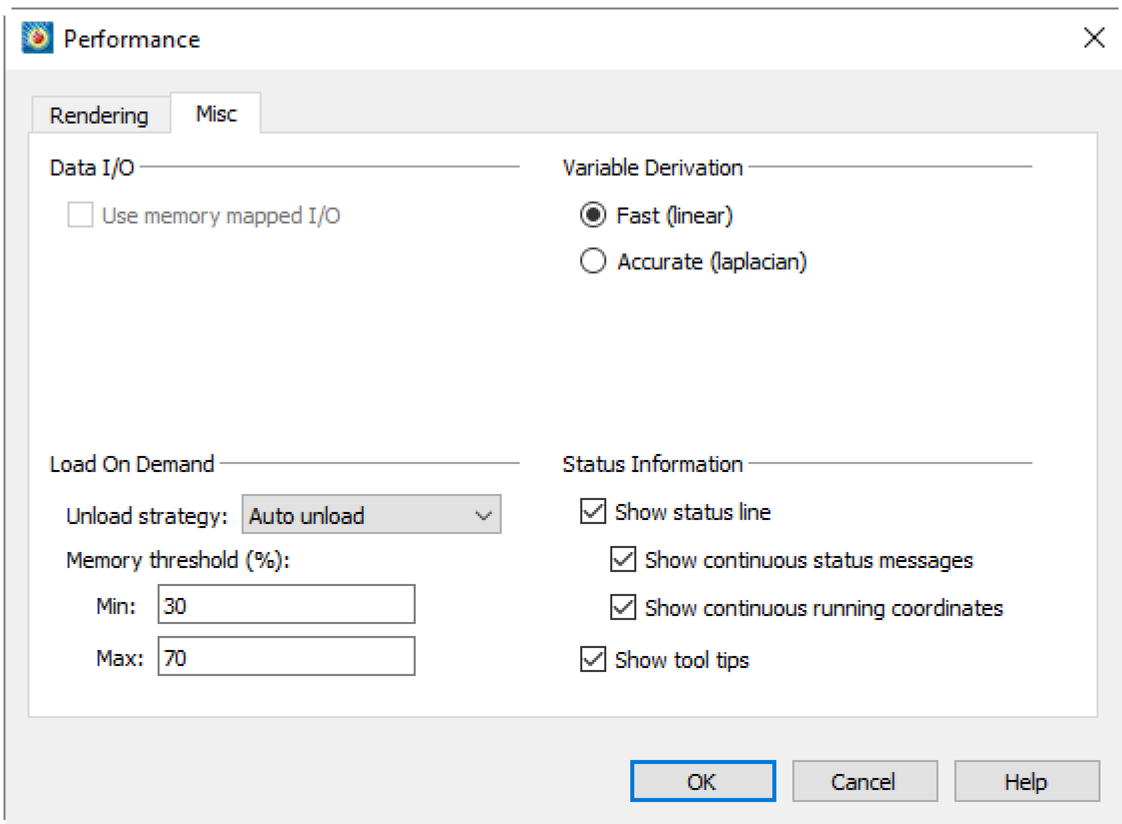
The factory settings in the **Performance** dialog are designed for moderately-sized data and occasionally may need to be adjusted to optimize Tecplot 360's rendering performance. For example, if you are doing a lot of interactive work with your data, you may wish to turn on Plot Approximation to make these operations more responsive. If you do not have much spare memory, you may wish to turn off the Graphics Cache, or set it to "Cache only Lightweight Graphics Objects."

> The size of the data isn't the only factor when rendering in Tecplot 360. If your plot includes slices or iso-surfaces, you may also need to adjust your plot approximation mode and graphics cache settings.

If you are using Plot Approximation, adjust the "Approximate Plot as % of Full Plot" value to give an acceptable balance between interactive performance and plot detail.

Miscellaneous Settings

The Miscellaneous page of the **Performance** dialog has the following functions:



Data I/O

Use Memory Mapped I/O

When toggled-on, Tecplot 360 will use system level memory mapping functions to map Tecplot 360 variables directly over block data in a binary data file or layout package file. Doing this at the system level generally provides the best available performance for loading the data.

The advantage of mapping variable data is that Tecplot 360 will only load the variable when it is needed, not when initially opening the file (but see also Load On Demand in this dialog). In addition, the mapped variable data can be shared between other Tecplot 360 sessions running on the same machine, saving memory. Memory mapped I/O is most useful when there are a large number of data points to load from a file and they are not all being used by Tecplot 360 at the same time. Only variable data that is in a binary block format (the default for `.plt` files generated by Tecplot 360) can be memory mapped.

Load On Demand

With load-on-demand activated, Tecplot 360 generates plots faster and using less memory by only loading data that is needed for the plot. If changes to the plot style require additional variables to be loaded, Tecplot 360 will automatically load them, and if necessary, unload variables that are no longer used. Tecplot 360's ability to automatically load and unload variables on demand allows you to examine data that is much larger than the physical or virtual memory of your computer.

> This setting does not affect reading data in Tecplot Subzone `(.szplt)` format, also known as SZL. SZL data is always loaded on demand, and in the smallest possible amount, which may be less than a full zone's worth of a variable.

For large datasets, only the zones and variables currently in use will be loaded. However, for small datasets, some other zones and variables may be loaded for you (based on the Memory Threshold).

Unload Strategy

Specifies how to manage unloading variables and other load-on-demand resources.

Auto Unload

This strategy attempts to keep Tecplot 360's memory use within the defined Min and Max Memory Thresholds. Tecplot 360 uses these values to determine when and how much it should unload. **This is the best option for exploring data** as Tecplot 360 **only unloads if and when the memory threshold has been exceeded**; if the threshold is not exceeded, data loaded on-demand remains available in memory if you need it again.

Minimize Memory Use

This strategy is used if more aggressive unloading of variables and other load-on-demand resources is required. **This option is best suited for animating through a very large number of time steps, where each time step consumes a significant part of the computer's available physical and virtual memory**.

Never Unload

This strategy disables the unloading capability of load-on-demand while still preserving the ability to load variables on demand.

> Most users should select either the "Auto Unload" or "Minimize Memory Use" options.

Memory Threshold (%) [Auto Unload ONLY]

When Tecplot 360 uses at least the maximum percentage of the available physical and virtual memory, it will attempt to unload variables and other load-on-demand resources until the available physical and virtual memory is at or below the specified minimum percentage.

Variable Derivation

When Tecplot 360 needs to create a nodal variable from a cell centered variable, it uses a prescribed derivation method. Tecplot 360 provides two such derivation methods: fast and accurate.

Fast (Linear)

When selected, Tecplot 360 uses simple averaging to derive a nodal variable from a cell centered one.

Accurate (Laplacian)

When selected, Tecplot 360 uses Laplacian interpolation to derive a nodal variable from a cell centered variable.

Status Information

Use the following controls in the Status Line region of the **Performance** dialog to customize what is displayed in the status line:

Show Status Line

Turn this preference on/off to control the display of status messages.

If you are remotely displaying Tecplot 360 on an X terminal, updating the status line can slow down processing. If this is the case, turn off the Show Status Line control.

Show Continuous State Messages

Turn on this control to receive context-sensitive commentary in the status line.

Show Continuous Running Coordinates

Turn on this control to display the coordinates of your mouse cursor in the status line.

Show Tool Tips

Use this option to turn tool tips on or off.

Custom Character and Symbol Definition

Generally, you will use TrueType fonts for displaying text on your screen and for printing. However, you can configure Tecplot 360 to use stroke fonts instead using the settings on the Rendering page of the Performance dialog (see High Quality Font Usage) for adding the `USESTROKEFONTSONSCREEN` and `USESTROKEFONTSFOR3DTETXT` macro commands to your configuration file (see Interface Configuration).

Stroke fonts define characters to be displayed on the screen using a set of straight lines (called strokes, naturally). Stroke fonts, if enabled, are also used in exported images (however, vector printing and file formats may use TrueType or PostScript fonts).

The stroke fonts are faster to draw than TrueType fonts, which can be important if you create plots with a lot of complex text. Another advantage of the stroke fonts is that you can redefine or modify the strokes that make up the characters, as they are stored in a plain text file called `tecplot.fnt`, which can be modified using an editor. For information on how to load a custom font file, see Custom Files loaded on Startup.

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" refers to Font File). CharCellHeight is the interline spacing (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 360 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 that ranges from 32 to 127 and 160 to 255 for each font (see Character Indices in Tecplot 360 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. 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*

Where:

- 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 75 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.

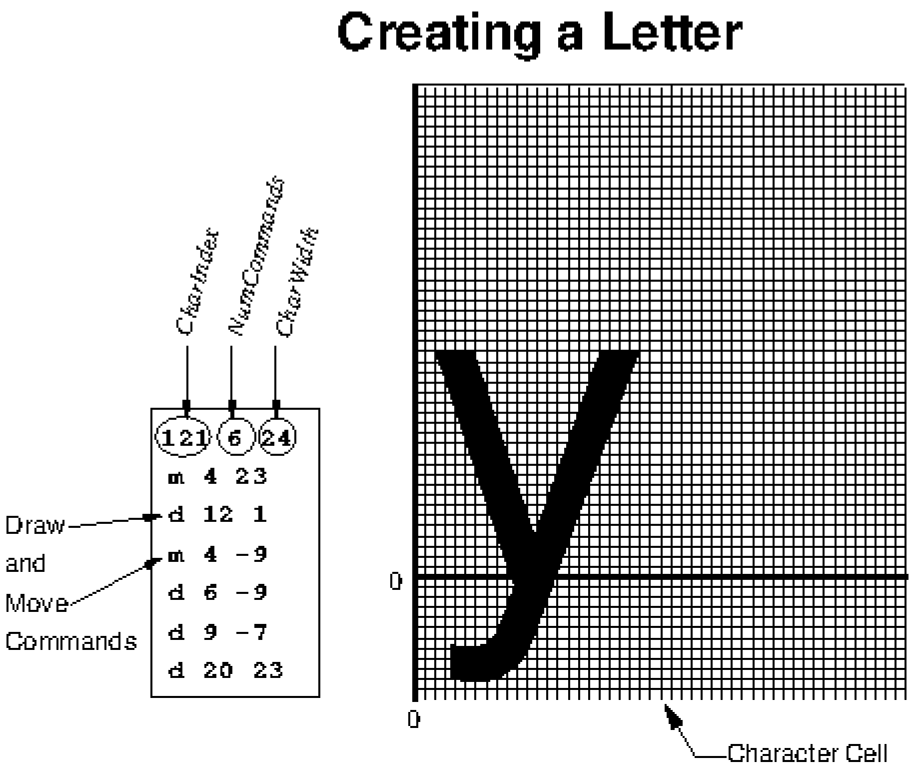


Figure 75. Defining a user-defined character.

Figure 76 shows a symbol being defined. Symbols should generally be centered about (0, 0) so that they are drawn centered on the point they mark (an exception might be an arrow symbol or similar, where you want the arrow's tip to indicate the marked point). The font file included with Tecplot 360 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.

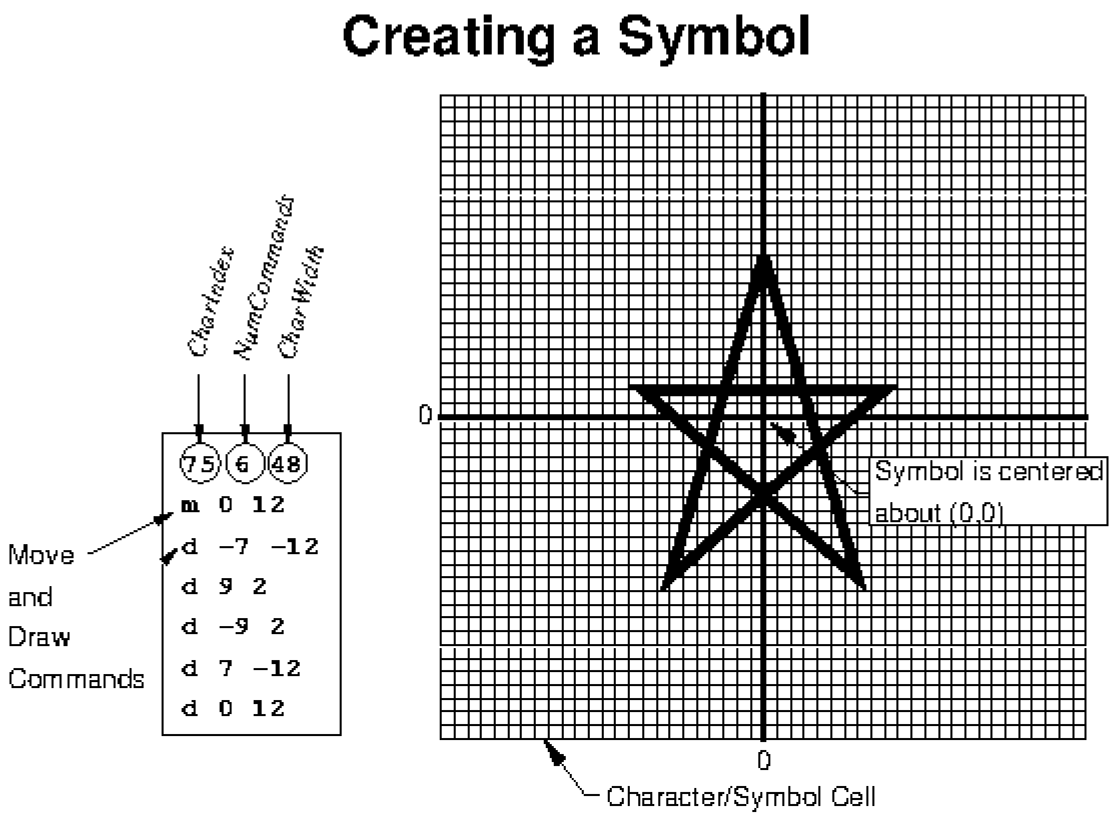


Figure 76. Defining a user-defined plotting symbol.

Add-Ons

Add-ons are a way to extend the basic functionality of Tecplot 360. They are executable modules designed to perform specific tasks. A number of add-ons are available that load data in a variety of formats, allow advanced editing, or extend Tecplot 360's capabilities.

Add-ons are external programs that attach themselves to Tecplot 360 and are accessed through the Tecplot 360 interface. When Tecplot 360 launches, it goes through various initialization phases, including the processing of the `tecplot.cfg` file, the loading of the Tecplot 360 stroke font file (`tecplot.fnt`), and the initialization of the graphics. After completing this, Tecplot 360 looks for add-ons.

Add-on Loading

You can load add-ons by several different methods: using the Drag-and-Drop Method, editing the Tecplot.add File, using the Specifying Add-Ons on the Command Line, or by Specifying a Secondary Add-On Load File.

Add-ons created for versions of Tecplot 360 prior to Tecplot 360 2014 R1 will not work with Tecplot 360 2025 R1.

Drag-and-Drop Method

The easiest method for loading an add-on for a single use is the drag-and-drop method. This method loads the add-on for the current session of Tecplot 360 (rather than every subsequent launch of Tecplot 360).

To use this method, find the library file of the add-on you wish to load in the `bin` subdirectory of your Tecplot 360 installation. Select the library, and, while holding down the mouse button, drag the library into the Tecplot 360 window. Then release the mouse button to "drop" the add-on into Tecplot 360. The add-on will be loaded and made available immediately.

This action will load the add-on only for the rest of the current Tecplot 360 session. The next time you close and reopen Tecplot 360, you will need to load the add-on again if you wish to in the new session.

Tecplot.add File

Every time Tecplot 360 launches, it reads a file called `tecplot.add` from the Tecplot 360 home directory and loads all add-ons indicated in that `tecplot.add` file.

The `tecplot.add` file contains multiple `$!LoadAddOn` commands that load add-ons into Tecplot 360. This special macro file accepts only `$!LoadAddOn` macro commands, which require this syntax:

```
$!LoadAddOn "libname"
```

In this example, *libname* represents the name of an add-on's shared object library file. The libname must be enclosed in quotes.

To unload an add-on (that is, to prevent Tecplot 360 from loading that add-on when launching), comment out the appropriate line in `tecplot.add` by adding a pound ("#") symbol in front of the load command. For example, the pound ("#") symbol at the beginning of the following line will prevent the Auxiliary Data Editor add-on (libname "tecutiltools_editauxdata") from loading:

```
# $!LoadAddOn "tecutiltools_editauxdata"
```

To load an add-on that does not load automatically, remove the pound ("#") symbol from in front of the `$!LoadAddOn` command that includes the libname of the add-on you wish to load. If the tecplot.add file does not list the libname of the add-on you wish to load, add a `$!LoadAddOn` command followed by the libname of that add-on.

For information on loading the tecplot.add file, see Custom Files loaded on Startup.

Libname

Special rules govern how the *libname* name is specified in the `tecplot.add` file. In all cases, the filename extension is omitted. If you assign *libname* to the basename of the shared object library, then Tecplot

360 will do the following:

Linux

The shared library to load will come from the file specified by *Install-Directory* `/lib/lib+`*basename.so*.

Windows

Tecplot 360 will search for the add-on *basename*`.dll` in the following directories (in this order):

- The directory where the Tecplot 360 executable resides.
- The Windows system directories.
- The directories in your `PATH` environment variable.

When using `V7ActiveX` style add-on libraries on Windows, Tecplot 360 connects to the add-on via the *libname* entry in the registry.

If an absolute path name is used in *libname*, then on Windows platforms, `.dll` is appended and on Mac and Linux platforms, `.so` is appended

Specifying Add-Ons on the Command Line

You can also instruct Tecplot 360 to load a particular add-on by including the following option when running Tecplot 360 from the command line:

```
-loadaddon <libname>
```

where <libname> is the full name (including path and extension) of an addon and `-loadaddon` can be omitted if the filename given uses the standard library extension for your operating system (`.dll` for windows and `.so` for Mac and Linux).

You may also load more than one addon from the command line.

After add-ons are loaded, Tecplot 360 re-processes all command line arguments not processed earlier (for graphics and add-on initialization). This allows a data loader add-on to be used to load data specified on the command line.

Specifying a Secondary Add-On Load File

You may also instruct Tecplot 360 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.
- Set the environment variable `TECADDONFILE`.

Both of these methods tell Tecplot 360 the name of an additional add-on load file to process (which itself contains `$!LoadAddon` commands). This secondary file will be processed after the main `tecplot.add` file.

Add-ons included in the Tecplot 360 distribution

Your Tecplot 360 installation includes the add-ons listed following (alphabetically) and described in Working with Add-ons. Some of these load automatically with your installation; others require you to load the add-on by uncommenting the appropriate line from your `tecplot.add` file (located in your Tecplot 360 home directory), or by any of the other methods described in Add-on Loading.

- Advanced Quick Edit
- Auxiliary Data Editor
- CFD Analysis
- Create Multiple Frames
- Extend Macro
- Extend Time Macro
- Extract Over Time
- Key Frame Animator
- Measure Distance
- Multi-Frame 3D
- Solution Time and Strand Editor
- Tensor Eigensystem
- Time Series
- Write Data as Formatted Text

Working with Add-ons

The default Tecplot 360 installation includes the add-ons discussed in this section. To load or unload them, use the methods described in Add-on Loading.

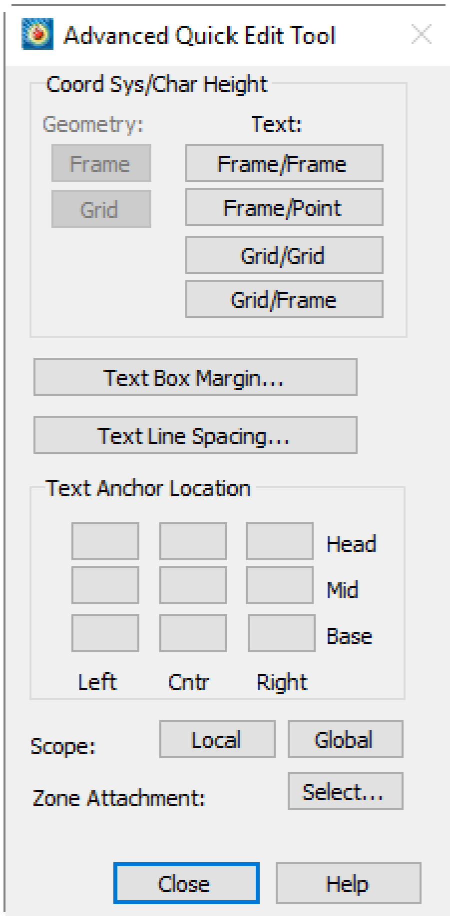
Advanced Quick Edit

The Advanced Quick Edit add-on, with the libname `tecutiltools_advqet`, loads automatically, and causes the "Advanced Quick Edit Tool" to appear in the **Tools** menu. This add-on allows you to make rapid changes to text and geometries selected in the active frame.

To enable this add-on, uncomment the line with `$!LoadAddon "tecutiltools_advqet"` in your Tecplot.add File or use one of the other methods to load addons described in Add-on Loading.

To use this add-on, select "Advanced Quick Edit Tool" in the **Tools** menu to open the **Advanced Quick Edit Tool** dialog.

Controls in the **Advanced Quick Edit Tool** dialog are active only when you have one or more text and/or geometries 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 affect the specific 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.



The **Advanced Quick Edit Tool** dialog includes the following options:

Geometry Coordinate System

Change selected geometries to the Frame or Grid coordinate system by selecting the appropriate button in the **Coord Sys/Char Height** region of the dialog. Changing the coordinate system in this dialog will modify each geometry's anchor position and size so that it appears visually unchanged in your plot.

Text Coordinate System and Character Height Units

Change the position coordinate system and character height units of all selected text by selecting the appropriate button in the Coord Sys/Char Height region of the dialog. There are four valid combinations: [Frame/Frame], [Frame/Point], [Grid/Grid], and [Grid/Frame]. Changing a coordinate system in this dialog will modify each text object's anchor position and character height so that it appears visually unchanged in your plot.

Text Box Margin

Change the text box margin of all selected text with the [Text Box Margin] button.

Text Line Spacing

Change the line spacing of all selected text by using the [Text Line Spacing] button.

Text Anchor Location

Change the text anchor location for all selected text by selecting one of the nine possible anchor points from the button grid located in the Text Anchor Location region of the dialog.

Text and Geometry

Change the scope of all selected text and geometries by selecting either [Local] or [Global] scope. Objects with local scope appear only in the frame in which they were originally created. If the objects are defined as having global scope they will appear in all "like" frames, that is, those frames using the same data set as the one in which the objects were originally created.

Text and Geometry Zone or Map Attachment

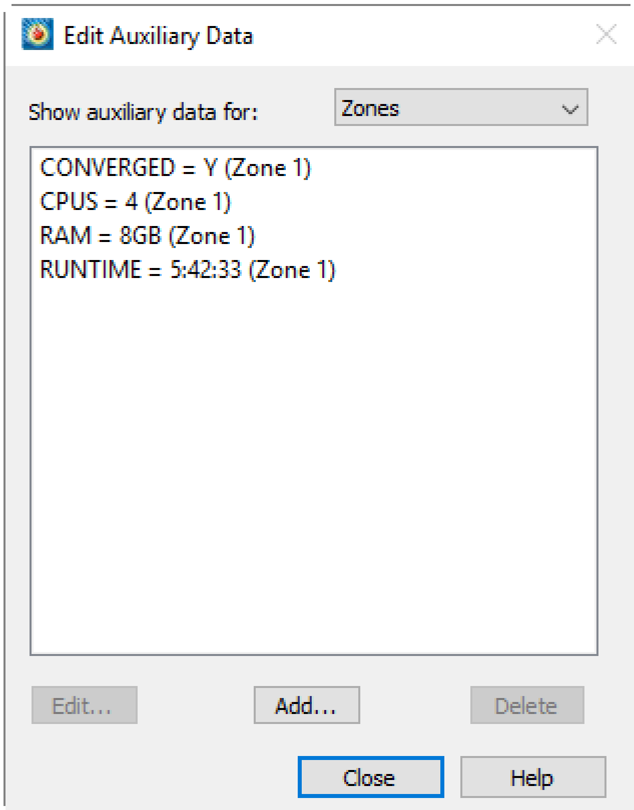
Change the zone or map with which the selected text or geometries are associated by selecting Zone Attachment [Select]. This calls up the **Attachment Selection** dialog. The **Attachment Selection** dialog lists zone names or numbers when Tecplot 360 is in the 2D or 3D Cartesian or Sketch plot types, and mappings when Tecplot 360 is in the XY Line plot type. The "<Unattach Object>" entry dissociates each selected text or geometry from its zone or map.

Auxiliary Data Editor

The Auxiliary Data Editor add-on loads automatically with Tecplot 360. This add-on enables you to edit auxiliary data within Tecplot 360 and journal the results into the saved layout file. To enable this add-on, uncomment the line with `$!LoadAddon "tecutiltools_editauxdata"` in your Tecplot.add File or use one of the other methods to load addons described in Add-on Loading.

> Auxiliary data are name-value pairs associated with a data set, frame, line map, page, variable, or zone, which Tecplot 360 stores as text strings. You can view all auxiliary data included in your layout in the **Data Set Information** dialog (by going to **Data**→**Data Set Info**).

To use this add-on, select "Auxiliary Data" from the **Data** menu to open the **Edit Auxiliary Data** dialog.



The **Edit Auxiliary Data** dialog contains the following controls:

Show auxiliary data

In the drop-down menu at the top of the dialog, select the level at which you wish to view, add or delete auxiliary data. Choose from Zones, Variables, Line Maps, Active Frame, Active Page, Data Set, Layout, or Everything.

Auxiliary data box

In the box in the middle of the dialog, all auxiliary data for the selected level will display. You can select any item in this box to edit it or delete it, with the [Edit] button or [Delete] button, respectively.

Add auxiliary data

With the [Add] button, you can add auxiliary data at any of the available locations (zone, variable, etc.).

Edit auxiliary data

With the [Edit] button, you can edit any existing auxiliary data. First select the item you wish to edit in the Auxiliary data box, and then select the [Edit] button.

Delete auxiliary data

Use the [Delete] button to delete any existing auxiliary data. Select the item you wish to delete in the Auxiliary data box, and then select the [Delete] button.

You can view auxiliary data on the Aux Data page of the **Data Set Information** dialog (accessible by going to **Data→Data Set Info**).

Tecplot 360 will journal all added or edited auxiliary data to layout and packaged layout files when you save your layout or packaged layout file after editing the auxiliary data.

Macro Commands

You can add auxiliary data using the `$!EXTENDEDCOMMAND` macro, with the following syntax:

```
$!EXTENDEDCOMMAND
  COMMANDPROCESSORID = 'Aux Data Editor'
  COMMAND = '<operator> Name = <string> Value = <string> Location = <location>
LocationIndex = <int>'
```

Where:

Command Entity	Notes
<operator>	Choose one of `Add` or `Delete`
Name = <string>	For <string> use an alphabetic character followed by zero or additional alphanumeric characters and underscores (no spaces)
Value = <string>	Only used with the `Add` operator. For <string> use the value to assign to the named data piece. Surround the value with quotation marks if it has spaces
Location = <location>	For <location> use one of: Zone, Var, Frame, Page, DataSet, LineMap, or Layout.
LocationIndex = <int>	For <int> provide the integer value representing which Zone, Var, Frame, or LineMap to which Tecplot 360 attaches the auxiliary data. If adding auxiliary data to a Page or DataSet, do not indicate a LocationIndex.

Following is an example of a macro command adding an auxiliary data piece named "TestID" with the value of "Sequence 23, test 2" to the first zone.

```
$!EXTENDEDCOMMAND
  COMMANDPROCESSORID = 'Aux Data Editor'
  COMMAND = 'Add Name = TestID Value = "Sequence 23, test 2" Location = Zone
LocationIndex = 1'
```

Adding Auxiliary Data

The **Add Auxiliary Data** dialog enables you to add auxiliary data at any of several levels. To reach this dialog, select the [Add] button in the **Edit Auxiliary Data** dialog. Data added in the **Add Auxiliary Data** dialog will subsequently appear in the **Edit Auxiliary Data** dialog, in the Auxiliary Data display box. The **Add Auxiliary Data** dialog has the following controls:

Location

Choose the level at which to add the data from the Location drop-down menu. You can choose to add it at the zone, variable, line map, active frame, active page, layout, or dataset level.

Number

When the Location indicates "Zone", "Variable", or "Line Map", the Number box will activate for you to specify the Zone Number, Variable Number, or Line Map number at which to add the auxiliary data.

Name

Enter the name of your data in the Name box.

Value

Enter the value of your data in the Value box.

Select the **OK** button in the **Add Auxiliary Data** dialog to finish adding a piece of auxiliary data to the selected location.

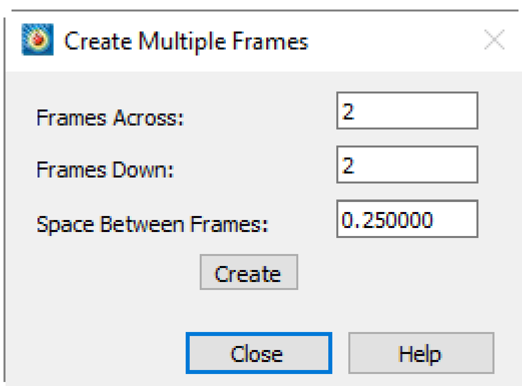
Create Multiple Frames

Use the Create Multiple Frames add-on to make a set of new frames with uniform size and spacing within the active frame.

To enable this add-on, uncomment the line with `$!LoadAddon "tecutiltools_mulframe"` in your Tecplot.add File or use one of the other methods to load addons described in Add-on Loading.

To use this add-on, select **Create Multiple Frames** in the **Tools** menu. This will open the **Create Multiple Frames** dialog.

The **Create Multiple Frames** dialog has the following options:



Frames Across

Enter the number of frames to be displayed in each row of frames.

Frames Down

Enter the number of rows of frames.

Space Between Frames

Enter the amount of space to display between each frame in each direction, in paper ruler units.

Select the [Create] button to create the multiple frames as specified.

See Frame Creation for information on creating a single frame.

Extend Macro

The Extend Macro add-on, libname "tecutilscript_extendmcr", extends the Tecplot macro language to include additional commands. To enable this add-on, uncomment the line with `$!LoadAddon "tecutilscript_extendmcr"` in your Tecplot.add File or use one of the other methods to load addons described in Add-on Loading.

You may use the Extend Macro add-on by adding the following function call to your macro file:

$!EXTENDEDCOMMAND COMMANDPROCESSORID='extendmcr' COMMAND='*command option*'

The commands supported by the add-on are listed in the following table

When specifying a macro variable name in an Extend Macro command, you should not surround the variable name with vertical bars. For example, ZONENUM rather than |ZONENUM|.

Table 10. Command Options for Extend Macro

Command	Notes
QUERY.ZONENAMEBYNUM *nnn VVV*	Get the string for zone *nnn* and assign to variable *VVV*.
QUERY.VARNAMEBYNUM *nnn VVV*	Get the string for variable *nnn* and assign to variable *VVV*
QUERY.ZONENUMBYNAME "zonename" *VVV*	Get the number of zone named *zonename* and assign to variable *VVV*

Command	Notes
QUERY.VARNUMBYASSIGNMENT assignment *VVV*	Get the number of variable by assignment and assign to variable *VVV*. *VVV* may have any of the following values: • `X`, `Y` or `Z` - Variable assigned to the X, Y or Z-axis. • `U`, `V` or `W` - Variable assigned to be the U, V or W-vector component. • `C` Variable assigned to contours. • `S` Variable assigned to scatter sizing. • `B` Variable assigned to the first constraint for value-blanking.
QUERY.DATASETTITLE *VVV*	Get the string for the dataset title and assign to variable *VVV*
STRING.LENGTH StrSource *VVV*	Get the length of string `StrSource` and assign to variable *VVV*.
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.
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*.
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."
QUERY.MAPNAMEBYNUM *nnn VVV*	Returns a string (the name of the map) for map number *nnn* and places it in variable *VVV*. The active plot must be XY-Line or Polar-Line.
QUERY.ISADDONLOADED *commandprocessorid VVV*	Return "YES" in *VVV* if Add-on *commandprocessorid* is loaded, otherwise return "NO"
QUERY.FILEEXISTS "filename" *VVV*	If the file exists, *VVV* will be "YES" otherwise *VVV* will be "NO"
QUERY.ISZONEACTIVE *ZZZ VVV*	Test to see if zone *ZZZ* is active. If so, *VVV* is set to "YES"," otherwise it is set to "NO"

Command	Notes
QUERY.LINEMAPZONEASSIGNMENT *nnn VVV*	Get the zone number assigned to line map *nnn* and put the result in *VVV*.
QUERY.ACTIVELINEMAPS *VVV*	Get the set of active line maps and put the result in *VVV*. Note: The set string does not include any blank spaces. If linemaps 2, 4, 6, 7, and 8 are active, VVV would have the string "2,4,6-8."
QUERY.ISLINEMAPACTIVE *nnn VVV*	Test to see if line map nnn is active. If so, *VVV* is set to "YES", otherwise it is set to "NO." *VVV* is set to "INVALID" if *nnn* ⇐0.
FILE.APPENDSTRING "FileName" "String"	Appends the given string to the indicated file, creating the file if necessary. FileName may be a relative path from Tecplot's working directory or an absolute path.
FILE.APPENDLASTERROR "FileName"	Appends the last error message, if any, to the indicated file, creating the file if necessary. FileName may be a relative path from Tecplot's working directory or an absolute path. If no Tecplot errors have occurred, this command will have no effect.

If you have declared macro variables and would like to use them with the Extend Macro add-on, you can do so by surrounding the command call with single quotes and the macro variable with double-quotes.

For example:

```
$!VarSet |ZoneName| = "Unknown"
$!EXTENDEDCOMMAND
  COMMANDPROCESSORID = "extendmcr"
  Command = 'query.zonenamebynum 1 "ZoneName"'
$!RemoveVar |ZoneName|
```

Refer to the Scripting Guide for additional information on working with Tecplot 360's macro language.

QUERY.DATASETTITLE

The following example, uses the `QUERY.DATASETITLE` command to place the title of the dataset at a specific position on the plot.

```
$!VARSET |ZNUM| = "blank"
```

```
$!EXTENDEDCOMMAND COMMANDPROCESSORID='extendmcr'
  COMMAND='QUERY.DATASETTITLE ZNUM'

$!ATTACHTEXT
  XYPOS
  {
    X = 5
    Y = 90
  }

  TEXT = "Title is: |ZNUM|"
```

QUERY.VARNAMEBYNUM

The following example uses QUERY.VARNAMEBYNUM to place the name of variable 2 at a specific position on the plot.

```
$!VARSET |VNAME| = "X"

$!EXTENDEDCOMMAND COMMANDPROCESSORID = 'extendmcr'
  COMMAND='QUERY.VARNAMEBYNUM 2 VNAME'

$!ATTACHTEXT
  XYPOS
  {
    X = 5
    Y = 85
  }

  TEXT = "Var 2 is: |VNAME|"
```

QUERY.ZONENAMEBYNUM

The following example uses QUERY.ZONENAMEBYNUM to place the title of Zone 1 at a specific position on the plot.QUERY.ACTIVEZONES

```
$!VARSET |ZNAME| = "HELLO"

$!EXTENDEDCOMMAND COMMANDPROCESSORID = 'extendmcr'
  COMMAND='QUERY.ZONENAMEBYNUM 1 ZNAME'

$!ATTACHTEXT
  XYPOS
  {
```

```
    X = 5
    Y = 80
  }
  TEXT = "Zone is: |ZNAME|"
```

QUERY.ACTIVEZONES

The follow example uses QUERY.ACTIVEZONES to display a list of the active zones.

```
$!VARSET |ZNUMS| = "blank"

$!EXTENDEDCOMMAND COMMANDPROCESSORID='extendmcr'
  COMMAND='QUERY.ACTIVEZONES ZNUMS'

$!ATTACHTEXT
  XYPOS
  {
    X = 5
    Y = 70
  }
  TEXT = "Active zones are: |ZNUMS|"
```

Refer to the Scripting Guide for additional information on working with Tecplot 360's macro language.

Extend Time Macro

The Extend Time Macro add-on simplifies the macro interface by allowing you to use a simple loop to query the number of solution times in the dataset and advance the time step. This differs from the native Tecplot macro language as it does not require that you know the solution time of your data.

This add-on uses a different algorithm than Tecplot 360 for sorting the solution times. Because Tecplot 360 combines time steps that are sufficiently close together, the number of time steps reported by this add-on may differ from the number of time steps reported by Tecplot 360.

To enable this add-on, uncomment the line with `$!LoadAddon "tecutilscript_extendtimemcr"` in your Tecplot.add File or use one of the other methods to load addons described in Add-on Loading.

Macro Processing

The Extend Time Macro add-on can be invoked from the macro language by using the following commands:

This command retrieves the number of time steps. This number is returned in the macro variable VVV.

```
$!EXTENDEDCOMMAND COMMANDPROCESSORID='extend time mcr'
```

```
  COMMAND='QUERY.NUMTIMESTEPS VVV'
```

This command sets the solution time at time step nnn. The acceptable value range number for nnn is: 1 - NumTimeSteps.

```
$!EXTENDEDCOMMAND COMMANDPROCESSORID='extend time mcr'
  COMMAND='SET.CURTIMESTEP nnn'
```

This command retrieves the solution time at time step nnn. This number is returned in the macro variable VVV.

```
$!EXTENDEDCOMMAND COMMANDPROCESSORID='extend time mcr'
  COMMAND='QUERY.TIMEATSTEP nnn VVV'
```

The following is a sample loop that uses the Extend Time Macro add-on:

```
$!EXTENDEDCOMMAND COMMANDPROCESSORID='extend time mcr'
  COMMAND='QUERY.NUMTIMESTEPS NUMTIMESTEPS'
$!LOOP |NUMTIMESTEPS|
  $!EXTENDEDCOMMAND COMMANDPROCESSORID='extend time mcr'
    COMMAND='SET.CURTIMESTEP |LOOP|'
  $!EXTENDEDCOMMAND COMMANDPROCESSORID='extend time mcr'
    COMMAND='QUERY.TIMEATSTEP |LOOP| CURTIME'
  $!PAUSE "Current time is: |CURTIME|"
$!ENDLOOP
```

See also: Extract Over Time, Solution Time and Strand Editor, Extend Macro, Time Series, and Time Aware.

Extract Over Time

The Extract Over Time add-on, extracts iso-surfaces, streamtraces, and points from polyline geometries in transient data. The resulting zones are assembled into a new strand with proper solution times set for each zone.

> To save a layout file, you must also save a data file when using this add-on.

To enable this add-on, uncomment the line with `$!LoadAddon "tecutiltools_extractovertime"` in your Tecplot.add File or use one of the other methods to load addons described in Add-on Loading.

To use this add-on, go to **Data**→**Extract** and select one of the following menu options.

Extract Slices Over Time

This option has moved to the Extract Slices dialog. See Extracting Slices to Zones. The macro, below, can still be used.

Extract Polyline Over Time

Extracts a polyline geometry to a zone for each solution time. It allows you to define the number of points along the polyline to extract. It has the following requirements:

- The frame must be in 2D or 3D.
- Only a single polyline can be selected.
- The data must be transient.
- The polyline must be positioned over an existing zone.

Extract Iso-Surfaces Over Time

Extracts an iso-surface to a zone for each solution time. It has the following requirements:

- The frame must be in 3D.
- The frame must contain one or more iso-surfaces.
- The data must be transient.

Extract Streamtraces Over Time

Extracts a set of streamtraces to a zone for each solution time. It has the following requirements:

- The frame must be in 2D or 3D.
- The frame contains one or more streamtraces
- The data must be transient.

After extracting, the new strand is available and may be animated or saved to a file.

Macro Processing

The Extract Over Time add-on can be invoked from the macro language by using the following commands:

Extract What	Command
Slices	$!EXTENDEDCOMMAND COMMANDPROCESSORID='Extract Over Time' COMMAND='ExtractSliceOverTime' Note: Prefer using the $!EXTRACTSLICES macro command instead. See the Scripting Guide for more details on this command.
Iso-Surfaces	$!EXTENDEDCOMMAND COMMANDPROCESSORID='Extract Over Time' COMMAND='ExtractIsoSurfaceOverTime'

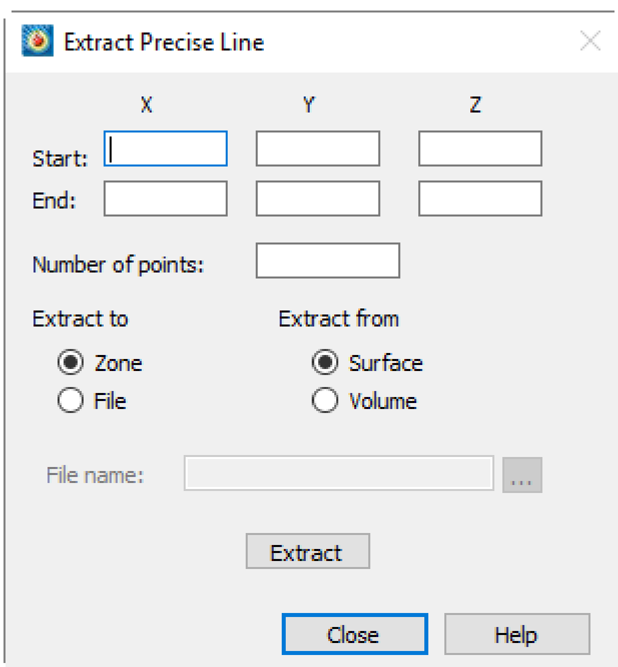
Extract What	Command
Streamtraces	$!EXTENDEDCOMMAND COMMANDPROCESSORID='Extract Over Time' COMMAND='ExtractStreamOverTime'
Geometries	$!EXTENDEDCOMMAND COMMANDPROCESSORID='Extract Over Time' COMMAND='ExtractGeomOverTime' Optionally, you may specify the number of points to extract by using the following format: $!EXTENDEDCOMMAND COMMANDPROCESSORID='Extract Over Time' COMMAND='ExtractGeomOverTime:nnn' Where nnn >= 2. If this condition is not met when the macro is played back, the action will silently fail and your macro will continue processing.

See also: Extend Time Macro, Solution Time and Strand Editor, Extend Macro, Time Series, and Time Aware.

Extract Precise Line

The Extract Precise Line add-on (libname: `tecutiltools_extractpreciseline)` allows you to extract points along a line to a new zone or to a Tecplot ASCII data file. You specify the precise X, Y, and Z coordinates of the start and the end points of the line, along with the number of points along this line to extract.

To use the Extract Precise Line add-on, choose **Data → Extract → Extract Precise Line**.



Start, End X/Y/Z

Enter the X, Y, and Z coordinates of the start and end points of the line.

Number of Points

The number of points between the start and end points to be extracted. (The start and end points themselves are always extracted and this number does not include them.)

Extract To

Whether to extract to a zone or a file. If extracting to a file, you must also specify an output file path in the File Name field.

Extract From

Choose whether to extract from the surface the line passes across or the volume it passes through.

When extracting from a surface in 3D, extraction is view-dependent: the specified line is projected from the plane of the screen onto the surface being extracted. Changing the view (e.g. rotating) will change the points extracted.

File Name

The path of the file to extract to. You may type the path, paste it, or click the **...** button to open the Write Tecplot ASCII File dialog and navigate interactively.

Click **Extract** to perform the extraction.

Macro Processing

There is no macro command syntax specific to this add-on. Instead, it records an `$!EXTRACTFROMPOLYLINE` command. See the Scripting Guide for more details on this command.

Extract Blanked Zones

The Extract Blanked Zones add-on (libname: *tecutiltools_extractblankedzones*) allows you to extract subsets of existing zones based on the current blanking conditions. When activated, the Extract Blanked Zones capability will be available below the Data/Extract menu. To use simply set up blanking conditions (Value Blanking or IJK Blanking) and then choose the **Data→Extract→Extract Blanked Zone...** menu option. You may then select one or more zones to operate on. One new zone will be created for each zone selected (unless blanking dictates that all cells and points in the entire zone are blanked).

Special handling for FE surface and volume zones that contain orphaned nodes.

Orphaned nodes are nodes that are not connected to a cell in Finite-Element data. When value blanking is turned on interactively these nodes are also processed. If you elect to plot "All Nodes" in the "Points to Plot" column in the "Points" tab of the zone style dialog you can see the orphaned nodes.

When using the extract blanked zones capability, these nodes, for surface and volume FE data are not processed and thus they will not be part of the generated zone.

Special handling for "linear" zones (i.e. I-Ordered, J-Ordered, K-Ordered, or FE-Line Segment zones).

Linear zones are handled in a special manner whereby blanking is applied to points and not cells. If the current plot is 2D or 3D this also means ignoring the current value blanking "Cell Blanking Condition" setting (the option below the "Blank entire cells when" option in the dialog). Cells in FE Line Segment zones are only retained if both end points for the cell are not blanked. For I, J, and K-Ordered zones the resulting zone will continue to be the same "order" but keep in mind that all of the resulting data points are joined together thus resulting in new "cells" that skip over points that were blanked out. Points that are orphaned in linear FE-line segment zones, unlike Finite-Element surface zones, will be retained in the resulting blanked zone.

Cell-centered variables for I-ordered, J-ordered and K-ordered zones will be interpolated to the nodes before extracting the blanked regions. FE-line segment zones will retain the cell-centered values for cells that are not blanked.

Macro Processing

The Extract Blanked Zone add-on can be invoked from the macro language by using the following command:

```
$!EXTENDEDCOMMAND COMMANDPROCESSORID='EXTRACTBLANKEDZONES'
COMMAND='<commandoption>'
```

The command options are listed below:

Table 11. Command Options for Extend Macro

Command	Notes
ZONES = <set>	The ZONES option is the set of zones to operate on.

Example:

Extract blanked zones from zones 1 and zones 5-9:

```
$!ExtendedCommand
  CommandProcessorID = 'EXTRACTBLANKEDZONES'
  Command = 'ZONES = [1,5-9]'
```

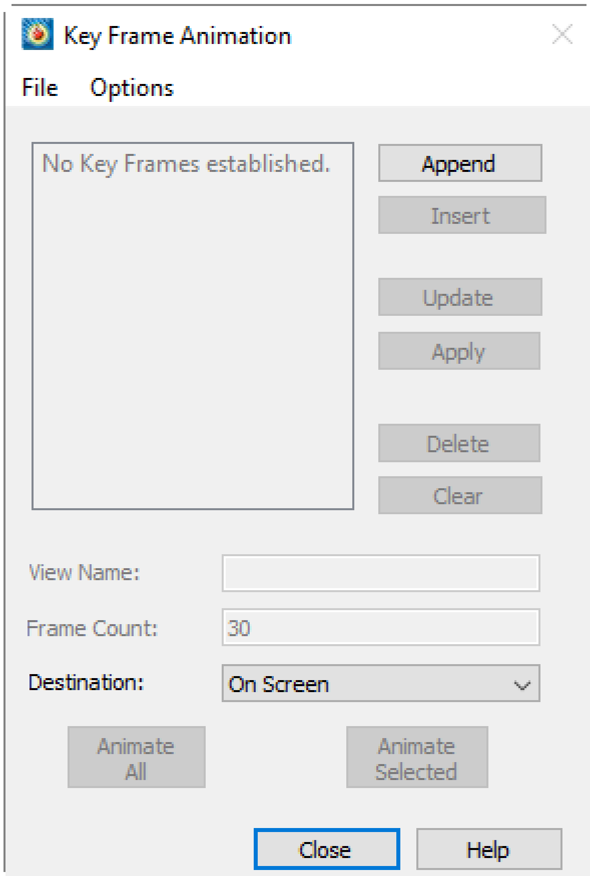
Key Frame Animator

With this add-on, you can easily animate a smooth progression through two or more specified views (key frames) and export them as an AVI, Flash, or Raster Metafile animation.

To enable this add-on, uncomment the line with `$!LoadAddon "tecutiltools_keyframe"` in your Tecplot.add File or use one of the other methods to load addons described in Add-on Loading.

To use this add-on, first create a layout and use the 3D Cartesian mode to view your plot. Then select "Key Frame Animation" in the **Animate** menu to open the **Key Frame Animation** dialog.

> You must be in 3D Cartesian mode to use the Key Frame Animator add-on.



To create a key frame animation using this dialog, follow these general steps:

1. Rotate and/or zoom the plot to the initial view for the animation, and select the [Append] button in the dialog. If you wish, you may change the name of the view (key frame) by selecting the frame ("View 1") in the Key Frame box and editing the name in the View Name field of the dialog (for example, "End of Airfoil").
2. Rotate and/or zoom the plot to the next view you wish your animation to include, and select the [Append] button to add that view to the end of the animation. Or you can select the [Insert] button to insert a view above the selected one in the list. These key frames are snapshots of the view in Tecplot 360, so limit rotations to 180 degrees or less between key frames, or the animation may not rotate as you expect.
3. Continue to rotate and/or zoom and select [Append] or [Insert] to add additional key frames to your animation, if desired.
4. When finished adding key frames, select [Animate All] to perform a complete animation of your views, or [Animate Selected] to animate only the views selected in the Key Frame box. Tecplot 360 will interpolate all views in between your views to create an animation.
5. To export an animation, change the Destination from "On Screen" to "To File". Then select the

[Animate All] or [Animate Selected] to open the **Export Options** dialog. You can choose to export the file as AVI, Flash, or Raster Metafile. Refer to Exporting Plots for details on exporting your file.

The **Key Frame Animation** dialog includes these controls:

Append

The [Append] button adds the current view in the active frame to the list.

Insert

Use the [Insert] button to insert the current view above the currently selected view in the key frame list.

Update

The [Update] button replaces the view selected in the list with the current view in Tecplot 360. This also re-captures the current solution time.

Apply

Use the [Apply] button to display the selected view as the current view of your plot.

Delete

Select the [Delete] button to delete the selected key frame from the list.

Clear

Use the [Clear] button will delete all key frames from the list.

View Name

Use the View Name field to rename the view selected in the key frame list. By default, Tecplot 360 names the views as "View 1", "View 2", etc.

Frame Count

The Frame Count box sets the number of frames drawn by Tecplot 360 between the selected view and the next view in the animation.

Destination

In the Destination drop-down menu, choose whether to animate the key frames On Screen or To File.

Animate All

Select [Animate All] to animate through all the key frames in the list.

Animate Selected

Select [Animate Selected] to animate only the selected views. This allows you to refine a subset of animation without performing the complete animation.

Animation Customizations

The Key Frame Animator add-on includes additional capabilities in customized **File** and **Options** menus at the top of the Key Frame Animation dialog. These menus give you the following additional options:

File→Open/Save Animation

In the **File** menu, you can save an animation that you have created, or you can open an animation that you have previously saved. An animation file stores settings and all key frames, and defaults to the name `animation.keyframe`. The ability to save and load animation files enables you to keep track of created animations without having to recreate them.

Options

In the **Options** menu, each choice is a toggle except for "Time Animation Options" (which opens the Time Animation Options dialog). Select an option from this menu to turn on that preference. When an option is toggled-on, a checkmark will appear next to that option in the **Options** menu.

Animate Time Concurrently

Select this option to animate time when animating the key frames, as specified in the Time Animation Options dialog.

Animate Streamtraces Concurrently

If streamtraces display as markers or dashes in your plot, enable "Animate Streamtraces Concurrently" to animate streamtrace markers or dashes concurrently with key frame animation.

Fade In/Fade Out

Toggle-on "Fade In" and/or "Fade Out" to fade lighting intensity and background lighting during the start and/or end of the key frame animation. The fade will last either 30 frames or the first ten percent of all frames, whichever is shorter. Fade In and Fade Out work best with a black background and without any plot features that do not have light sensitivity (mesh lines, edge lines, etc.).

Time Animation Options

With the **Time Animation Options** dialog (opened by selecting "Time Animation Options" in the **Options** menu), you can specify additional options for animating your key frames. Choose to animate the frames by any of the following methods.

Forward

With "Forward" selected, Tecplot 360 will display the earliest solution time in the first key frame, and will progress solution time throughout key frame animation so that the latest solution time displays with the final frame of the animation.

Loop

With "Loop" selected, the solution time will progress as in the "Forward" option, except that the complete cycle of solution times displays multiple times over one key frame animation cycle. Set the number of solution time cycles to display in the Num Cycles box.

Bounce

With "Bounce" selected, the solution time will progress forward during animation, until the solution time cycle completes. The solution time will then decrement in equal intervals until the initial solution time displays. Specify the number of times to repeat this bounce in the Num Cycles box.

As Captured

Select "As Captured" to link each key frame with the solution time that the frame displayed when that frame was added to the key frame list. During the key frame animation, the solution time will vary in equal intervals between successive pairs of key frames so that each key frame displays the linked solution time.

See Also: Key Frame Animator.

Export Animation Options

In the **Key Frame Animation** dialog, when animating a set of key frames, you can export your animation by selecting "To File" from the Destination drop-down menu. Then select either [Animate All] or [Animate Selected] to launch the **Export Options** dialog. In this dialog, you can customize the export settings for your animation. Then select **OK** to select file location and export your animation.

The **Export Options** dialog includes the following export settings for your animation:

Format

Select the format of your exported file.

Region

Select the portion of your workspace that you wish to export. You can export the Active Frame, All Frames, or the entire Work Area.

Width

Choose the width of the exported animation. You can use the default width of image on screen, or enter your own dimensions.

Animation Speed

Enter a number to indicate how quickly you would like your animation to progress, in units of frames per second.

Antialiasing

Toggle-on "Antialiasing" to use antialiasing in your exported animation. For more information on antialiasing, refer to Antialiasing Images.

Image Type

If exporting as Flash, choose the type of image on which to base your animation.

Optimize

With the two Optimize choices at the bottom of the dialog, you can choose to optimize your animation for either faster speed or smaller file size.

Macro Processing

To invoke the Key Frame Animator from the macro language, use the following commands:

Save Animation File:

```
$!EXTENDEDCOMMAND
  COMMANDPROCESSORID = 'Key Frame Animation'
  COMMAND = 'Animate AnimationDestination=<animationdest>'
```

In this command, use either OnScreen or ToFile for the <animationdest>.

Open Animation Settings File:

```
$!EXTENDEDCOMMAND
  COMMANDPROCESSORID = 'Key Frame Animation'
  COMMAND = 'OpenSettingsFile FileName = <string>'
```

Save Animation Settings File:

```
$!EXTENDEDCOMMAND
  COMMANDPROCESSORID = 'Key Frame Animation'
  COMMAND = 'SaveSettingsFile FileName = <string>'
```

Use the `$!ExportSetup` commands to set up the necessary parameters for exporting to file (i.e., to set format, filename, etc.)

Note that the `$!ExportSetup` parameter `EXPORTFNAME`, which controls the export file name, can accept intrinsic macro variables such as `SOLUTIONTIME`, `FRAMENAME`, and `LAYOUTFNAME`. For example, if you wish to save a `.png` image, that tracks the frame name and solution time it came from, the macro command below is valid if the supplied intrinsic variables exist:

```
$!EXPORTSETUP EXPORTFNAME = 'path/to/output/directory/|FRAMENAME|-|SOLUTIONTIME|.png'
```

See Also: Key Frame Animator.

Measure Distance

The Measure Distance add-on provides two menu items on the Tools pull-down menu, which are available only in 3D and 2D cartesian plots.

To enable this add-on, uncomment the line with `$!LoadAddon "tecutiltools_measuredistance"` in your Tecplot.add File or use one of the other methods to load addons described in Add-on Loading.

The Tools menu items are:

Measure Distance

Choose this command from the Tools menu, then click two points on your plot. After the second click, a dialog appears indicating the coordinates of the two points, the differences in their coordinates, and the linear distance between the two points.

In three dimensions, the distance measured is the shortest distance between the different points selected and does not follow the surfaces. In two dimensions, the distance is in the 2D plane.

Measure Angle

Choose this command from the Tools menu, then click three points on your plot. After the third click, a dialog appears indicating the distance from point 1 to point 2, the distance from point 2 to point 3, and the angle in radians and degrees.

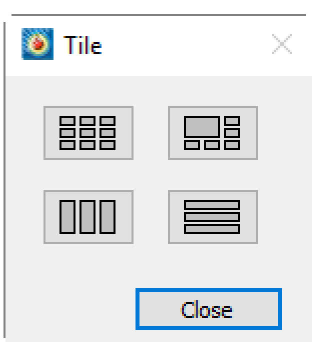
In three dimensions, the angle measured is the angle between vectors 1-2 and 2-3 in the plane defined by points 1,2, and 3. In two dimensions the angle is in the 2D plane.

Multi-Frame 3D

The Multi-Frame 3D add-on adds two menu items to the Frame pull-down menu: Tile Frames and 3D Multi-Frames. Each command displays a small dialog giving you options for managing the frames in your plot.

To enable this add-on, uncomment the line with `$!LoadAddon "tecutiltools_multiframe3d"` in your Tecplot.add File or use one of the other methods to load addons described in Add-on Loading.

Tile Frames



The Tile Frames dialog lets you quickly arrange multiple frames on a page in a variety of layouts. Clockwise from upper left, these are:

Square

Frames are made as close to square as possible and arranged in a grid with approximately equal numbers of rows and columns.

Wrap

The first frame is displayed in a larger size and the rest of the frames are wrapped around the right and bottom edge. The size of the main frame depends on the number of additional frames that need to be wrapped around the page.

Horizontal

Equal-size horizontal frames are stacked vertically.

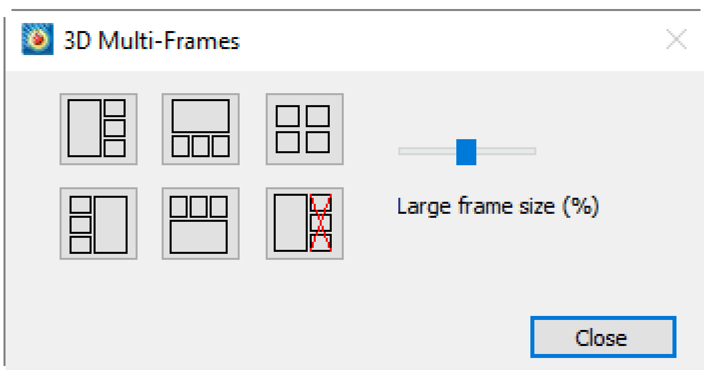
Vertical

Equal-sized vertical frames are arranged horizontally.

When frames are resized, no attempt is made to re-fit the data to them. If a frame's aspect ratio changes, data may no longer fit the frame as it did before tiling.

3D Multi-Frames

The 3D Multi-frames dialog, available only when the active frame is in 3D Cartesian mode, creates three additional frames displaying top (XY), front (XZ), and side (YZ) views of the 3D frame.



You may choose from five options for arranging the frames. Four of these options use a large frame for the original 3D frame and place the top, front, and side views along the top, bottom, or side of the large frame. For these views, the slider chooses the size of the large frame from 25% to 75% of the paper area. A grid arrangement giving equal prominence to all four frames is also provided. The button with the red X is for deleting the auxiliary frames (they may also be deleted manually).

When the new frames are created, the data and styles are shared with the original frame, and the new views are fit to their frames. The program also links field layer toggles, such as Mesh, Contour, Lighting, and so forth, so that a change in one frame is automatically reflected in the others.

Macro Processing

To tile frames using the Tecplot macro language, use the following command.

```
$!EXTENDEDCOMMAND
  COMMANDPROCESSORID = 'Multi Frame Manager'
  COMMAND = 'TILEFRAMESSQUARE'
```

The command can be TILEFRAMESSQUARE, TILEFRAMESVERT, TILEFRAMEZHORIZ, or TILEFRAMESWRAP.

To generate the top/front/side frames from a 3D view, use the following command.

```
$!EXTENDEDCOMMAND
  COMMANDPROCESSORID = 'Multi Frame Manager'
  COMMAND = 'MAKEFRAMES3D ARRANGE=LEFT SIZE=25'
```

ARRANGE may be LEFT, TOP, RIGHT, BOTTOM, or TILE, and the first four accept the SIZE parameter, which indicates a percentage of the paper size and can be 25-75.

To delete the top/front/side frames, use the following command.

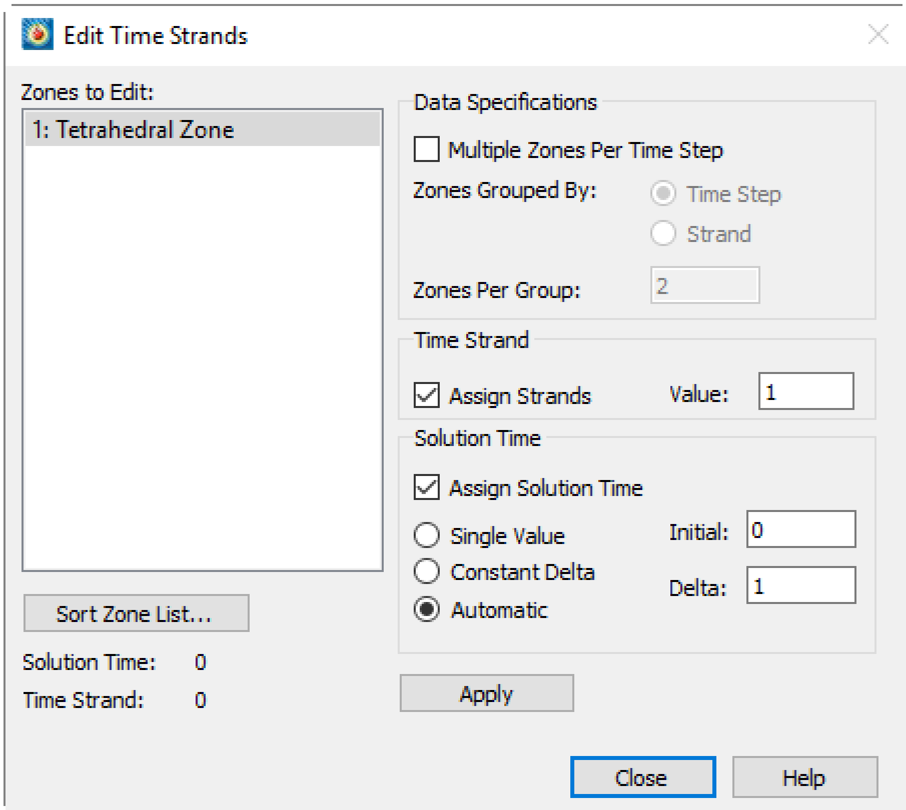
```
$!EXTENDEDCOMMAND
  COMMANDPROCESSORID = 'Multi Frame Manager'
  COMMAND = 'DELETEFRAMES3D'
```

Solution Time and Strand Editor

The Solution Time and Strand Editor add-on allows you to modify the strand and solution time values from the user interface. This helps when loading a series of data files that do not have a solution time or strand ID. Adding the solution time simplifies both animating and setting component styles.

To enable this add-on, uncomment the line with `$!LoadAddon "tecutiltools_strandeditor"` in your Tecplot.add File or use one of the other methods to load addons described in Add-on Loading.

To access this add-on, select **Edit Time Strands** from the **Data** menu. The dialog has the following options:



Zones to Edit

In the Zones to Edit box, select a set of zones to edit.

Data Specifications

Toggle-on "Multiple Zones Per Time Step" in order to select grouping by time step or strand. Use this Data Specifications region of the dialog to specify how the add-on groups zones.

Strand ID

In the Time Strand region of the dialog, toggle-on "Assign Strands" to assign strand IDs.

Solution Time

In the Solution Time region of the dialog, toggle-on "Assign Solution Time" to assign the solution time using one of the following options:

Single Value

The Single Value option assigns the specified solution time to all selected zones.

Constant Delta

The Constant Delta option applies a constant delta between zones (or groups of zones, depending on the Data Specification settings).

Automatic

The Automatic option attempts to determine the solution time for each zone in this order:

- Examines the Common.Time auxiliary data attached to the zone.

- Tries to read a number from the zone name.
- Tries to find a dataset variable that contains the time value. If found, uses the minimum value of this variable for that zone.
- If an existing solution time is defined for the zone, the Automatic option uses it.
- If all previous efforts fail, the Automatic option will fall back to use the Constant Delta option.

Tensor Eigensystem

The Tensor Eigensystem enables you to calculate the eigenvalues and eigenvectors of a 3-by-3 symmetric tensor whose components are stored in your data set. The add-on calculates for each node in the data set and stores the results as new data set variables.

To enable this add-on, uncomment the line with `$!LoadAddon "tecutiltools_tensoregn"` in your Tecplot.add File or use one of the other methods to load addons described in Add-on Loading.

To use this add-on, select "Tensor Eigensystem" from the **Tools** menu to open the **Tensor Tools** dialog.

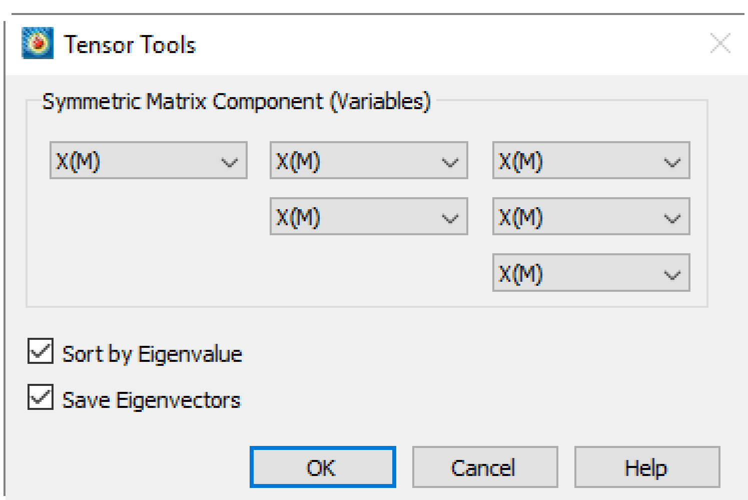
In the **Tensor Tools** dialog, select variables from your data set to represent the 6 components of the tensor. Toggle-on "Sort by Eigenvalue" to sort the results by the eigenvalues, and/or toggle-on "Save Eigenvectors" to store the calculated eigenvectors in addition to the calculated eigenvalues, which the calculation always saves. Then select **OK** to perform the calculation.

Tecplot 360 will store the eigenvalues of the tensor as variables EgnVal1, EgnVal2, and EgnVal3 in your data set. If you choose to save the eigenvectors as well, they are stored as variables EgnVec11 through EgnVec33. If you toggled-on "Sort by Eigenvalue", the eigenvalues and their corresponding eigenvectors will be sorted largest-to-smallest at each grid node. The association of EigenValue to EigenVector is as follows:

EigVal1 → EgnVec11, EgnVec21, EgnVec31

EigVal2 → EgnVec12, EgnVec22, EgnVec32

EigVal3 → EgnVec13, EgnVec23, EgnVec33



A common use of Tensor Eigensystem is to visualize vortex cores in a flow solution. You can do this by the following method:

1. Calculate Velocity Gradient in **Analyze→Calculate Variables**
2. Calculate the symmetric tensor S2 + Ohm-2 with **Data→Alter→Specify Equations** (see below for an equation file to perform this)
3. Use the **Tensor Eigensystem** dialog to calculate the sorted eigenvalues
4. Display an iso-surface of Lambda-2, the middle of the three eigenvalues, which is negative in the vicinity of a vortex core

The following equation file calculates the required symmetric tensor components of S2 + Ohm-2 from the velocity gradient for step 2 above. Save the below to a text file, then load it into the **Specify Equations** dialog by selecting "Load Equations" and selecting the file.

```
#!MC 1100
$!ALTERDATA EQUATION = '{s11} = {dUdX}'
$!ALTERDATA EQUATION = '{s12} = 0.5*({dUdY}+{dVdX})'
$!ALTERDATA EQUATION = '{s13} = 0.5*({dUdZ}+{dWdX})'
$!ALTERDATA EQUATION = '{s22} = {dVdY}'
$!ALTERDATA EQUATION = '{s23} = 0.5*({dVdZ}+{dWdY})'
$!ALTERDATA EQUATION = '{s33} = {dWdZ}'
$!ALTERDATA EQUATION = '{Omga12} = 0.5*({dUdY}-{dVdX})'
$!ALTERDATA EQUATION = '{Omga13} = 0.5*({dUdZ}-{dWdX})'
$!ALTERDATA EQUATION = '{Omga23} = 0.5*({dVdZ}-{dWdY})'
$!ALTERDATA EQUATION = '{s2o2_11} = {s11}**2 + {s12}**2 + {s13}**2 - {Omga12}**2 -
{Omga13}**2'
$!ALTERDATA EQUATION = '{s2o2_12} = {s11}*{s12} + {s12}*{s22} + {s13}*{s23} -
{Omga13}*{Omga23}'
$!ALTERDATA EQUATION = '{s2o2_13} = {s11}*{s13} + {s12}*{s23} + {s13}*{s33} +
{Omga12}*{Omga23}'
$!ALTERDATA EQUATION = '{s2o2_22} = {s12}**2 + {s22}**2 + {s23}**2 - {Omga12}**2 -
{Omga23}**2'
$!ALTERDATA EQUATION = '{s2o2_23} = {s12}*{s13} + {s22}*{s23} + {s23}*{s33} -
{Omga12}*{Omga13}'
$!ALTERDATA EQUATION = '{s2o2_33} = {s13}**2 + {s23}**2 + {s33}**2 - {Omga13}**2 -
{Omga23}**2'
```

Remember that the Tensor Eigensystem add-on can only analyze 3-by-3 symmetric tensors (not 2-by-2, anti-symmetric, or non-symmetric tensors).

Macro Processing

To invoke the tensor eigensystem add-on with a macro, use the following syntax.

```
$!EXTENDEDCOMMAND
  COMMANDPROCESSORID = 'Tensor Eigensystem'
  COMMAND = 'T11VarNum = <varref>\nT12VarNum = <varref>\nT13VarNum =
             <varref>\nT22VarNum = <varref>\nT23VarNum = <varref>\nT33VarNum =
             <varref>\nSortEgnV = <Boolean>\nSaveEgnVect = <Boolean>'
```

Where <varref> can be an integer or the variable name contained within double quotes.

Time Series

The Time Series add-on, extracts a single point over time and plots the result in a new frame as an XY Line Plot. The solution time of the time series plot's frame is linked to that of its parent, and a marker gridline is added to show the current solution time. If a nearest-point probe is performed (using Control-click), you may follow a particular node through time, or through an XY(Z) location.

To enable this add-on, uncomment the line with `$!LoadAddon "tecutiltools_timeseries"` in your Tecplot.add File or use one of the other methods to load addons described in Add-on Loading.

> When tracking a node through time, each zone must have the same node map (or a shared node map)
>
> This add-on may not work as expected when there are multiple zones in the same strand at the same solution time.

To use the Time Series add-on, choose one of the following menu options from the **Tools** menu:

Probe To Create Time Series Plot

This option sets the mouse mode to probe. You can use either the mouse or the **Probe At** dialog to probe a point. The probed location will be sampled over time (for transient data) and a resulting XY Line Plot will be created. If a nearest point probe is done (Control-click), a dialog appears to ask if you want to track the node or the XY(Z) location through time. If you track the node, it is important that your data has the same node map for each zone through time.

Send Time Series Data To New Frame

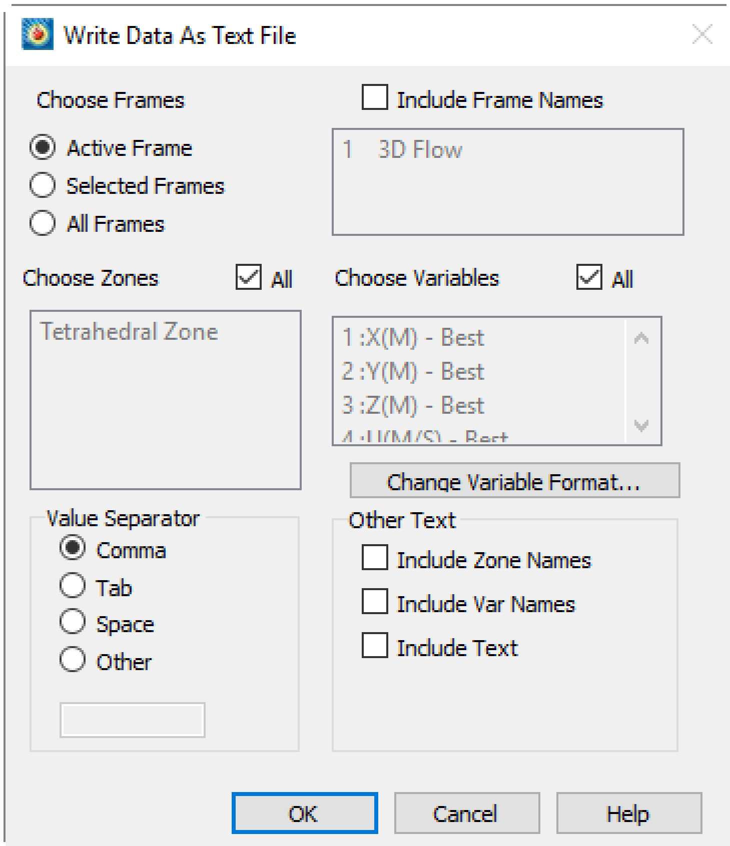
This option creates a new frame for each point extracted. By default, the frame used for the time series data is reused to avoid creating an excess of frames.

See also: Extract Over Time, Extend Time Macro, Solution Time and Strand Editor, Extend Macro, and Time Aware.

Write Data as Formatted Text

The Write Data as Formatted Text add-on enables you to write data from an XY Line Plot to a text file in `*.csv` format (for comma separated data) or in `*.txt` format (for any other data separators).

To enable this add-on, uncomment the line with `$!LoadAddon "tecutildataio_excsv"` in your Tecplot.add File or use one of the other methods to load addons described in Add-on Loading. To use it, select "Write Data As Formatted Text" from the **Tools** menu. This will open the **Write Data as Text File** dialog.



The **Write Data as Text File** dialog includes the following options:

Choose Frames

Use the Choose Frames region of the dialog to specify which frames to include in the text file. Choose from the following options:

Active Frame

Select to include the active frame only.

Selected Frames

When "Selected Frames" is active, the box to the right will become sensitive. Use the Shift and Control keys to select multiple items in the list.

All Frames

Select "All Frames" to include data from all frames in the text file.

Choose Zones

Toggle-off the "All" option to select a subset of zones to include the in text file. Because different frames may have different zones, this option is available only when "Active Frame" is selected in the

Choose Frames region of the dialog.

Choose Variables

Toggle-off the "All" option to select a subset of variables to include in the text file. Because different frames may have different variables, this option is available only when "Current Frame" is selected in the Choose Frames region of the dialog.

Value Separator

Use the Value Separator options to specify the delimiter to use in the text file.

Change Variable Format

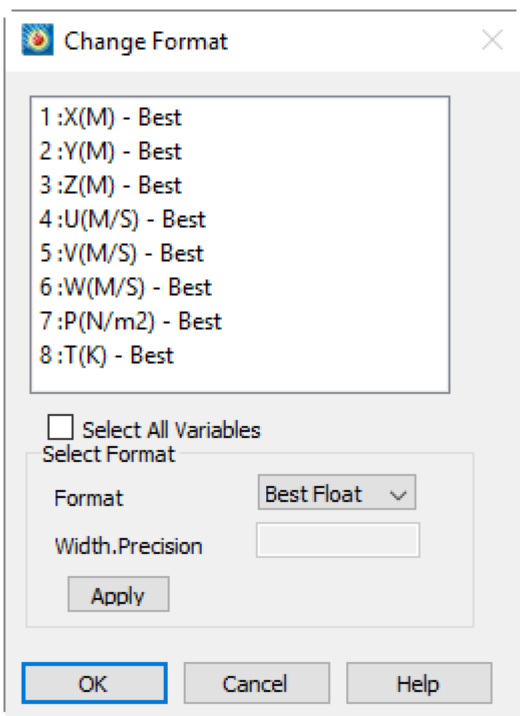
Click to launch the Change Format dialog. If you choose not to change the variable format, values will be written in the "best" format, except variables named "Time" or "Date."

Other Text

Toggle-on any of the options in the Other Text region of the dialog to include the desired values in the text file.

Change Format

Use the **Change Format** dialog (accessible using the Change Variable Format button in the Write Data as Text File dialog) to change the format for one or more variables. To change the format for all variables simultaneously, toggle-on **Select All Variables**.



Choose a new format with the Select Format section of the dialog. Available formats are Best Float, Integer, Float, and Exponent. "Best Float" automatically selects either regular floating point or scientific notation depending on the value and uses as many digits as are necessary to represent the value. "Float" and "Exponent" use standard and scientific notation, respectively.

When you have chosen "Float" or "Exponent," you may use the Width.Precision box to specify the width and precision of your new variable format using a C-style numeric formatting specifier (see examples following). Width refers to the total number of characters (including the decimal point; numbers are space-padded on the left if necessary), while Precision refers to the number of places beyond the decimal point. Precision is a hard limit, while Width is a soft limit: the specified number of digits of precision are always used, but numbers are not cut off if they do not fit in the specified width (see second example following). Leading zeroes are not displayed, except for one immediately preceding a decimal point (that is, for values between -1 and 1).

For example, a `Width.Precision` of `10.2` yields:

```
Month,Seattle Rainfall,Dallas Rainfall,Miami Rainfall,Error 1,
1.00, 4.30, 4.00, 4.00, 0.20,
2.00, 4.50, 4.00, 4.10, 0.22,
3.00, 4.00, 3.50, 4.50, 0.24,
4.00, 4.20, 3.40, 4.20, 0.24,
```

While a `Width.Precision` of `3.2` yields:

```
Month,Seattle Rainfall,Dallas Rainfall,Miami Rainfall,Error 1,
1.00,4.30,4.00,4.00,0.20,
2.00,4.50,4.00,4.10,0.22,
3.00,4.00,3.50,4.50,0.24,
4.00,4.20,3.40,4.20,0.24,
```

Part 8: Appendices

Command Line Options

Tecplot 360 Command Line

The general form of the Tecplot 360 command line is:

```
tec360 [options] [layoutfile] [datafiles] [macrofile]
```

Most Tecplot 360 command line options are available for Windows operating systems, not just Linux and Mac. To use them, start Tecplot 360 from the Run command or at the command prompt (`cmd.exe`).

Under Windows, invoking Tecplot 360 from the command line or in a `.bat` file returns immediately rather than waiting for the application to terminate. If you are using Tecplot 360 to convert files or run a macro, you probably want to wait until Tecplot 360 has finished one operation before proceeding with the next. Otherwise, the next step may be working with partially-written data. To do this, invoke it with `start /wait tec360` followed by the desired options.

Where:

layoutfile

File with extension *`.lay` or *`.lpk`. See also Layout Files, Layout Package Files, Stylesheets.

datafiles

One or more data files. If both a layout file (*`.lay` only) and data files appear on the command line, Tecplot 360 substitutes the data files referenced in the layout file with the data files listed in the command line.

macrofile

Macro file name. See also Macros.

options

one or more of the following:

-addonfile *filename*	Supply a custom list of addons via the tecplot.add file.
-b macrofile	Run in batch mode.
-c *cfgfile*	Use *cfgfile* for the configuration instead of the default configuration file, *tecplot.cfg*.

-convert	Loads all the specified data files, and runs specified macros in a restricted batch mode in which the only permitted options are reading data files (in any supported format) and writing SZL files. Does not use a license key. See also -o.
-display *computername*	Display on computer *computername* (Linux only). The target system must have X-server capability with the GLX extension.
-datafile *filename*	Load data file *filename*.
-debug *dbugfile*	Send debug information to the file *dbugfile*. Information is displayed to aid in debugging a new 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" (Mac/Linux only).
-f *fontfile*	Use *fontfile* instead of the default font file, *tecplot.fnt*.
-h *homedir*	Use *homedir* for the home directory instead of the default home directory.
--help	Open help message for command-line flags.
-loadaddon " *addonname*"	Load an add-on named *addonname*.
-m *colormapfile*	Use *colormapfile* as the initial color map file.
--max-available -processors *numprocs*	Restrict the number of processors (processor cores) employed by Tecplot 360 to the *numprocs* specified. Some tasks can be performed in parallel, so using all available processors greatly increases performance of those tasks. By default, Tecplot 360 uses all processors available on the machine to provide the best performance in most cases. Assign a value less than the total number of available processors to limit the number of processors used by Tecplot 360 to the assigned number.
--mesa -mesa (deprecated)	Linux Only – Use a version of the OpenGL rendering library which uses software rendering (using CPU) rather than hardware rendering (using the GPU and installed/native drivers). Use this flag in situations when a graphics card and/or driver do not exist or it is not working. For assistance troubleshooting see the Troubleshooting Appendix in the User's Manual. Mesa libraries are used for both on- and off-screen rendering, if you are only working in batch use --osmesa. Mesa may be slower and lower quality on-screen rendering than hardware rendering (using the GPU) but will be reliable. For Windows - copy the opengll32.dll from the bin\mesa directory to the bin directory in order to use mesa rendering.
--mesa-swrast --osmesa-swrast	Linux Only – Use when the standard versions of mesa and osmesa libraries conflict with default libraries. Use these options only as a last resort. They cannot be used with regular --mesa or --osmesa commands.
-nobanner	Do not show the opening banner (i.e. splash screen).

-nobatchlog	Suppress creation of the file *batch.log* during batch mode operation.
-nostdaddons	Do not load the add-ons listed in the *tecplot.add* file.
-notoolbar	Run with the toolbar deactivated.
-nowelcomescreen	Do not display the Welcome Screen at startup (it may still be opened after launch from the View menu)
-o *outputfile*.szplt	Writes the specified SZL file from the data loaded from other files (including macros) specified on the command line, then exit. Does not require a license key. See also the *-convert* command.
--osmesa	Linux Only – Use a version of the OpenGL rendering library which is designed for off screen export using software rendering (using CPU) where there is no X server connection. OSMesa is automatically used when running in batch mode without an X server connection. Also use this flag if 360 or PyTecplot crash in batch mode.
-p *scriptfile*	Play the macro commands in the file *scriptfile.mcr*.
-q	Use quick playback mode. Ignores delay and pause commands.
-qm *quickpanelfile*	Place the macro functions in *quickpanelfile* in the Quick Macro Panel, instead of using the macros from the default file, *tecplot.mcr*.
-quiet	Turns off all standard-out messages (Linux only).
-r *printfile*	Set the filename for routing Print Files to *printfile*.
-s *stylefile*	Use *stylefile* as a stylesheet for the first frame (**.sty*).
-showpanel	Open the Quick Macro Panel upon startup.
--use-openssl	Linux Only - Force the use of supplied OpenSSL libraries instead of installed system libraries. For systems with unusual OpenSSL configurations, this may enable the Load Remote Data option in the file menu (the SZL Client add-on).
--use-sys	Use the libraries from the bin/sys sub-folder below the tecplot home directory when running Tecplot. The sys sub-folder contains the version of libstdc++ that was used to build Tecplot and may be required when running older Linux systems. (Linux only).
-v	Display the version number.
-y *exportfile*	Set the filename for export files to *exportfile*.
-z	Display macro commands in the Macro Viewer. This allows you to see macro commands prior to their launch.

Command Line Input	Result
`tec360`	Run Tecplot 360 without pre-loading any data files

Command Line Input	Result
`tec360 ex1.plt`	Run Tecplot 360 loading the data file `ex1.plt` as the first dataset
`tec360 ex1.plt ex2.plt ex3.plt`	Run Tecplot 360 loading the data files `ex1.plt`, `ex2.plt`, and `ex3.plt` as the first dataset
`tec360 -h /usr/myhome -c /usr/myhome/myset.cfg`	Run Tecplot 360 using `/usr/myhome` as the Tecplot 360 home directory and loading the Tecplot 360 configuration file `/usr/myhome/myset.cfg`
`tec360 sumtr1.lay`	Run Tecplot 360 using layout file `sumtr1.lay`
`tec360 calc.lay temp.plt`	Read a Tecplot 360 layout file `calc.lay` and replace the first dataset referenced in the layout file with the data file `temp.plt`. Overriding data files from the command line is only supported for layout files using the Tecplot Data Loader and/or the Tecplot Subzone Loader.
`tec360 -p a.mcr`	Run Tecplot 360 and play a macro called `a.mcr`
`tec360 infile.dat -o outfile.szplt`	Reads `infile.dat`, writes `outfile.szplt` in batch mode. Does not require a license.
`tec360 -convert conversion.mcr`	In batch mode, run the macro file `conversion.mcr` which may contain instructions only to read files and write them in `.szl` format. Does not require a license.

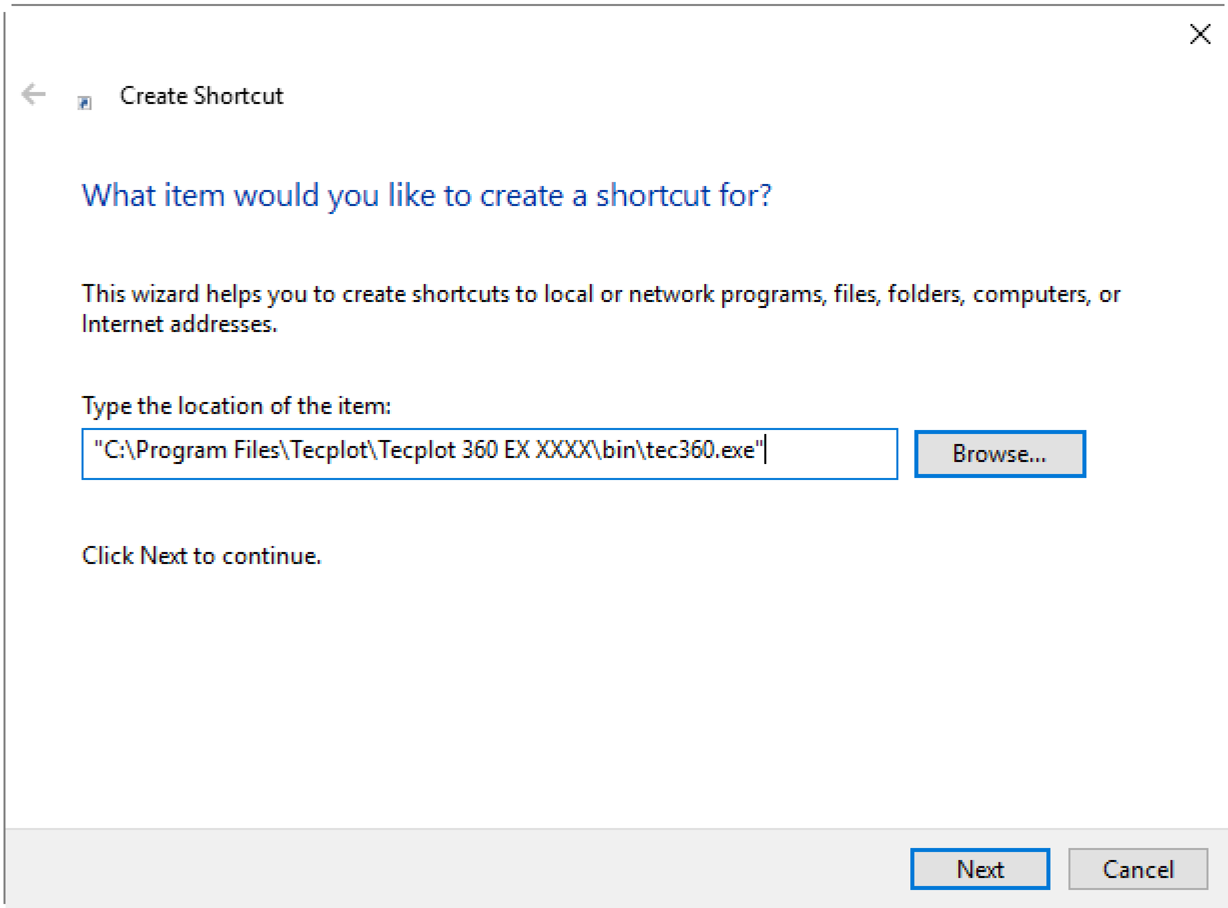
Using Command Line Options in Windows Shortcuts

All of the command line options that can be entered at the DOS or WCommand prompt by using the Run command can also be used in a Windows shortcut.

If you frequently run Tecplot 360 using the same command line flags, it may be useful to create a shortcut on your Windows desktop that launches Tecplot 360 with the desired command line flags. Here’s how this can be done:

> This process may vary slightly in different supported versions of Windows. The Windows 7 dialogs are shown here.

1. Right click any blank space on your Windows desktop.
2. Select **New**→**Shortcut** from the resulting Menu.



3. In the **Create Shortcut** dialog, type the location of the Tecplot 360 executable, along with any command flags you want to specify. An example command line is:

```
"C:\Program Files\Tecplot 360 EX 2025 R1\bin\tec360.exe" -p C:\Me\mymacro.mcr
```

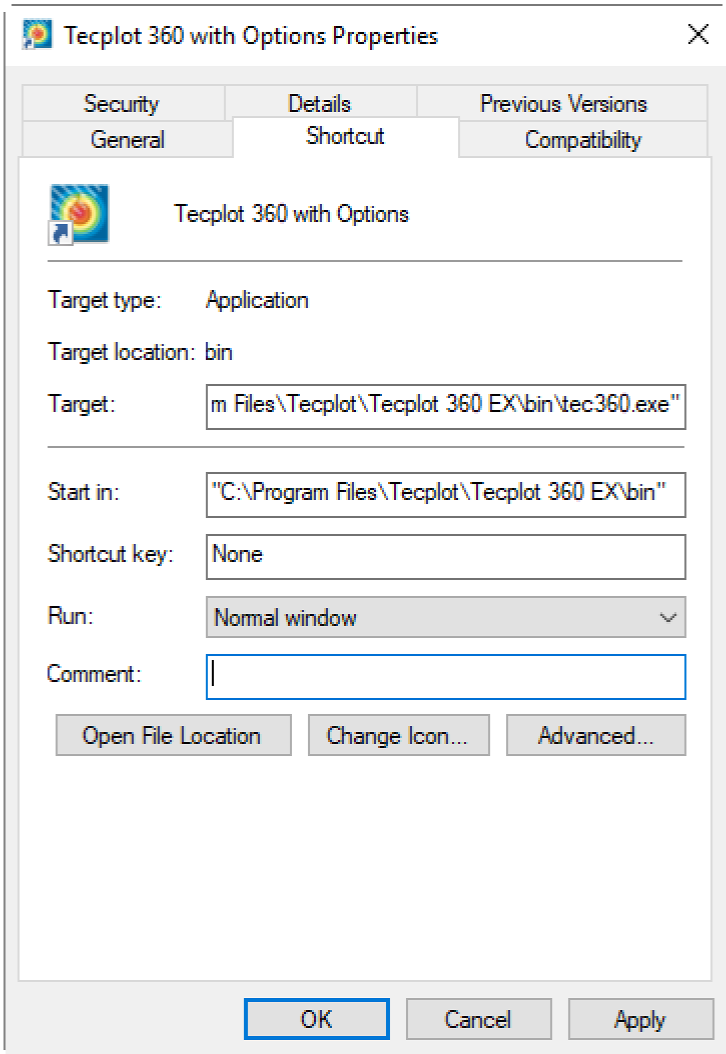
   You may also click the Browse button to choose the executable using an Open dialog, instead of typing the path.

4. Click **Next**.
5. Select a name for your shortcut, then click **Finish**.
6. A new shortcut icon will be placed on your Windows desktop.

Changing Shortcuts

You can alter an existing shortcut by doing the following:

1. Right-click on the shortcut icon you want to change and select **Properties**.
2. On the Shortcut page, modify the command line by changing the setting for Target. To change the working directory that Tecplot 360 runs under, change the Start in location.



Exit Codes

Upon completing execution, Tecplot 360 returns an exit code, which can be used in scripts to determine whether a problem occurred. This exit code is 0 (success) if execution completed without error, or 1 (error) if execution was terminated by an error.

Tecplot 360 Utilities

The following utilities are included with the Tecplot 360 distribution:

- Excel Add-In - Allows you to load Excel spreadsheet data directly into Tecplot 360 from inside Microsoft Excel.
- LPK View - A utility to catalog, preview or unpack a layout package file into its component data and layout files.
- Preplot - A utility to convert an ASCII data file into a Tecplot binary file.
- Pltview - A utility to view the header information for a Tecplot binary file.

Excel Add-In

The **Excel Add-In** provides a convenient way to load data directly from your Microsoft Excel spreadsheet into Tecplot 360. When loaded, it adds a Tecplot section to the Add-Ins ribbon, which contains a button that lets you open the highlighted region of the spreadsheet in Tecplot 360. (In older versions of Excel, previous to the ribbon user interface, this command appears on the Tools menu and as a button in the toolbar.)

> The Excel Add-In is available on Windows platforms only and requires Microsoft Excel be installed. Visual Basic for Applications, a component of Microsoft Office, must also be installed; if you receive an error about the workbook having lost its VBA project, make sure that Visual Basic for Applications is installed by using the Programs and Features dialog (Add/Remove Programs in older versions of Windows) to modify your Microsoft Office installation. Visual Basic for Applications can be found under the Office Shared Features heading in the Office installer.

The Excel add-in features:

Highlight and Plot

It's easy to plot just a portion of your data. Click in the upper left cell of the region or highlight the entire region, and then click on the Tecplot button in the Add-Ins ribbon or toolbar or on the Tecplot option in Excel's **Tools** menu.

Multiple Zones

The **Excel Add-In** makes loading multiple zones much easier. Highlight the entire region and then click on the Tecplot button in the Add-Ins ribbon or toolbar or on the Tecplot option in Excel's **Tools** menu. If your zones are separated by blank rows or columns, the add-in will automatically detect them and load them into Tecplot 360.

Formulas

The highlighted region of the spreadsheet can contain formulas, or can be created entirely with formulas.

The first time you use the Excel Add-In, you will be prompted to choose the Tecplot 360 executable to be launched from Excel. This will be the `tec360.exe` file in the `bin` folder of your Tecplot 360 installation. Your choice will be remembered and you will not be asked again unless the Excel Add-In cannot find the previously-chosen executable. A button in the Tecplot section of the Excel ribbon (or the Tools menu), Find Tecplot, will allow you to choose a different Tecplot 360 executable if you have upgraded or wish to use the Excel Add-In with more than one installed version of Tecplot 360.

A Read Me file, located in the Util/Excel directory, further describes installation and use of this add-in.

As an example, let's say you have 3D data obtained by drilling a number of wells and measuring contaminant concentrations of various chemicals at different depths. Your data is in an Excel spreadsheet, and you want to load the data into Tecplot 360 to get a visual representation of the contamination. The data has nine variables and twenty-seven zones. A small part is shown in Figure 77.

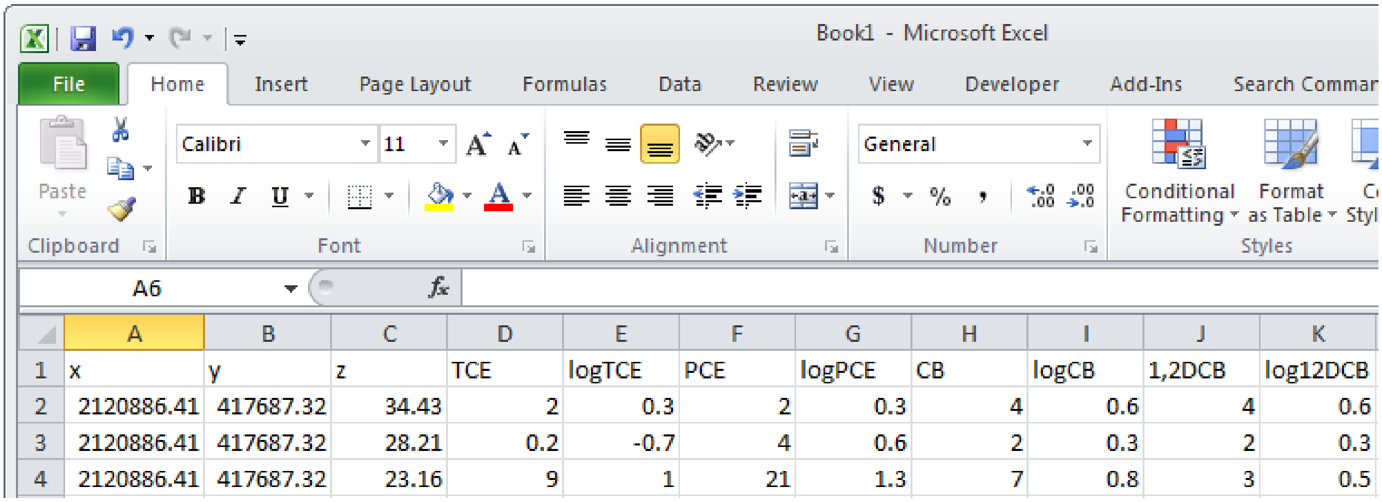


Figure 77. The beginning well data displayed in Excel.

Make sure you have a blank row separating the zones in your Excel spreadsheet.

Perform the following steps to import your data and visualize the contaminant plumes:

1. Starting with the top left-hand cell, highlight all twenty-seven zones and nine variables.
2. Click on Tecplot in Excel's Add-Ins ribbon. The menu option launches Tecplot 360 with the selected data loaded.
3. Switch to 3D Cartesian plot mode to see the location and measurement depths of the well samples. The resulting plot is shown in Figure 78.

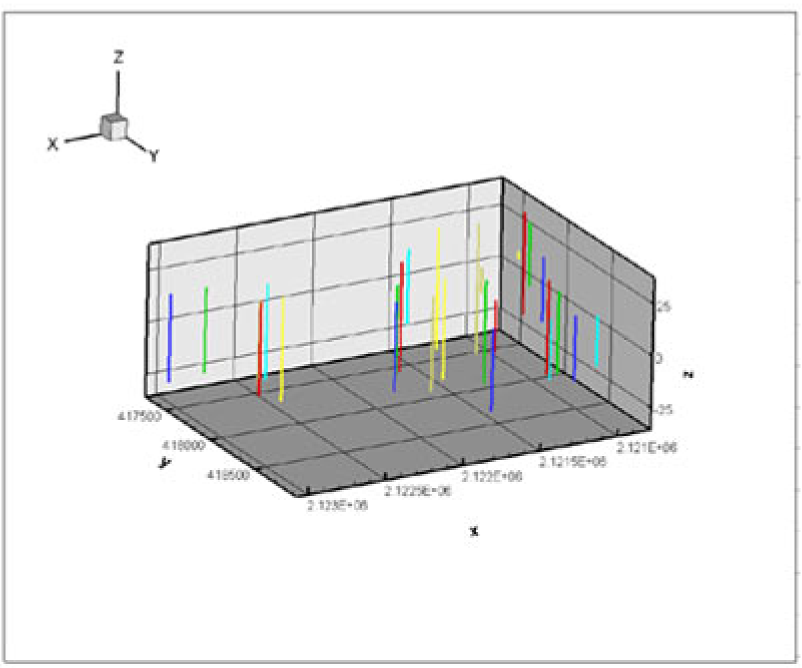


Figure 78. The Excel well data plotted in Tecplot 360.

Your wells have different depths, so the number of measurements are not the same for each well (there are only three measurements at well five).

LPK View

As a convenience a command line utility, lpkview, is provided to catalog and unpack layout packages. In its simplest form (when no options were included), 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
myplot_1.plt
myplot_2.plt
myplot_3.plt
```

Tecplot 360 determines the names for unpacked files when the package is created. Tecplot 360 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.

The utility's syntax is as follows:

```
lpkview [[-t] | [-ild] | [[-c <preview command>] -p]] filename
```

Brackets (`[]`) surround optional parameters and the vertical bar (`|`) separates one mutually exclusive set of options from another. 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	Specify preview command. The executable must be in a directory specified in the `PATH` or else a complete pathname. In the specified command, `%s` is substituted with the path of the temporary preview image exported by `lpkview`. (Mac/Linux only)
-p	Preview image using the command specified with -c. (Mac/Linux only)

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.

Preplot

The Preplot executable included in the standard Tecplot 360 installation converts Tecplot format ASCII data files to binary data files. The following options are available:

`-d`	Turn on debug echo. Use `-d2`, `-d3`, `-d4` for more detailed debug information.
`-foreignplt`	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 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.

Pltview

Pltview is a command line utility to examine the header information for binary Tecplot data files. It is included in your Tecplot 360 distribution and installed in the `bin` directory. To run pltview:

1. Launch the Command Prompt
2. Navigate to the `bin` directory of your Tecplot 360 installation.
3. in the command prompt, type:

   ```
   pltview "fullpath/filename.plt"
   ```

   You must enter the full path for your file.

4. The command prompt will display:
    - File Name
    - File Version
    - File Type
    - Data Set Title
    - Number of Zones
    - Number of Variables
    - Variable Names
    - Auxiliary Data
    - I, J and K Max for each ordered zone
    - Total number of Nodes, Elements, Faces, as well as the element type, for each finite element zone

An example session using the pltview utility is shown here.

```
C:\Temp>pltview chem.plt

FileName     : chem.plt
File Version: 7.0
DataSetTitle: Time versus Concentration
NumZones     : 2
NumVars      : 6
Var Names    : Time,Concentration,V3,V4,V5,V6

ZoneName                  IMax      JMax      KMax      Node        Elmt        EType
-------------------------------------------------------------------------------------
y=x**3.21                  20         1         1        ---         ---         ---
y=1.7**x                   20         1         1        ---         ---         ---


C:\Temp>
```

Shortcuts

Keyboard Shortcuts

On some platforms, Num Lock may interfere with some keyboard shortcuts. If shortcuts don't work, try toggling Num Lock.

Rotate Tools

Alt-Click-and-drag	Rotate about the viewer position using the active Rotate tool.
Middle-click-and-drag	Smoothly zoom in and out of the data.

Right-click-and-drag	Translate the data.
Control-right-click-and-drag (Mac: Command-right-click-and-drag)	Rotate about the rotation origin (any tool may be active).
C	Move the rotation origin to the probed point, ignoring zones.
O	Set the center of rotation.
R	Switch to Rollerball rotation.
S	Switch to Spherical rotation.
T	Switch to Twist rotation.
X	Switch to X-axis rotation.
Y	Switch to Y-axis rotation.
Z	Switch to 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.
Control-click	Replace the nearest contour line with a new line.
Click-and-drag	Move the new contour line.
-	Switch to the Contour Remove tool.

Contour Remove Tool

Click	Remove the contour line nearest to the probed location.
+	Switch to the **Contour Add** tool.

Geometry Polyline Tool

A	Allow translation of polyline segments in all directions.
H	Restrict translation of the current polyline segment to horizontal.
U	End the current polyline at last clicked point and start a new one.
V	Restrict translation of current polyline segment to vertical.

Probe Tool

Click	If the pointer is over a single valid cell, the interpolated field values from all nodes in the cell are returned. If multiple cells are candidates, the action is dependent upon the plot type: • For 2D, the cell from the highest number zone is used. • For 3D, the cell closest to the viewer is used.
Control-click	If the pointer is over a single valid cell, the field values from the nearest node in the cell are returned. If multiple cells are candidates, the action is dependent upon the plot type: • For 2D, the cell from the highest number zone is used. • For 3D, 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-Control-click	The field values from the nearest point on the screen are returned (ignoring surfaces, zone number, and depth of the point). This is useful in 3D for probing on data points that are on the back side of a closed surface without having to rotate the object. In 2D, this is useful for probing on data points for zones that may be underneath other zones.
Alt-click	Probe only on streamtraces, iso-surfaces, or slices. If multiple cells are candidates, the action is dependent upon the plot type: • For 2D, the cell from the highest number zone is used. • For 3D, the cell closest to the viewer is used.
Alt-Control-click	Probe only on streamtraces, iso-surfaces, or slices. If multiple cells are candidates, the action is dependent upon the plot type: • For 2D, the cell from the highest number zone is used. • For 3D, 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.
Alt-Control-Shift-click	Probe only on streamtraces, iso-surfaces, or slices. The field values from the nearest point on the screen are returned.
X, Y T, R	When probing, press X or Y in XY Line to switch dependencies, or R or T in Polar Line.

Slice Tool

+	Turn on start/end slices, or increment the number of intermediate slices.
-	Turn off start/end slices, or decrement the number of intermediate slices.
Click	If no slices are displayed for the current slice group, place the primary slice. Otherwise, move the closest displayed start, end, and primary slice from its current position to the clicked position.
Alt-click	Place the start, end, or primary slice (whichever is closer to the click position) on the nearest derived object (streamtrace, slice or iso-surface).
Control-click	Place the start, end, or primary slice (whichever is closer to the click position) on the nearest data point.
I, J, K	Switch to slicing constant I, J, or K-planes, respectively. Available for ordered zones only.
X, Y, Z	Switch to slicing constant X-, Y-, or Z-planes, respectively.
A	Switch to arbitrary slice mode.
1-8	Switch between slice groups.

Streamtrace Placement tools (3D Cartesian plots only)

D	Change the streamtrace style to streamrods.
R	Change the streamtrace style to streamribbons.
S	Change the streamtrace style to surface lines.
V	Change the streamtrace style to volume lines.
1-9	Change the number of streamtraces to be added when placing a rake of streamtraces.

Translate Tool

-	Reduce the magnification of the data.
+	Increase the magnification of the data.
Drag	Translate the data.
Shift-drag	Translate the paper.
Shift - -	Reduce the magnification of the paper.
Shift - +	Increase the magnification of the paper.

Zoom Tool

Click	Center the zoom around the location of your click.

Control-click	Center the zoom around the location of your click and zoom out.
Drag	Draw a box to set the frame view.

Selector Tool

Click	Select the frontmost object at the clicked location.
Control-click	Select the next object behind the currently selected object ("dig").
Shift-click	Multiple selection. Click the first object, then shift-click subsequent objects to add them to the selection
Alt-click	Ignore zone objects when selecting.

Selected 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.
Delete	Delete selected object(s).
Control-C	Copy selected object(s) to the clipboard.
Control-V	Paste selected object(s) from the clipboard.
Control-X	Cut selected object(s) to the clipboard, deleting them from the plot.

Time Navigation

Home	Jump to start.
Left Arrow	Step backward.
Space bar	Play or pause animation.
Right Arrow	Step forward.
End	Jump to end.

Other Keyboard Operations

Control-F	Fit Surfaces (3D Only) - Resize plot so that all surfaces are included in the frame, excluding any volume zones. Fit to Full Size (2D, XY, Polar, Sketch) - Fit the entire plot into the frame (including data, text and geometries).
Control-E	Fit Everything (3D Only) - Resizes plot so that all data points, text, and geometries are included in the frame.

Control-L	Last - Restore the last frame view.
Control-O	Open a layout file.
Control-P	Print.
Control-Q	Exit.
Control-R	Redraw the current frame.
Control-S	Save the current layout to its file.
Control-W	Save the current layout to a specified file.

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 Toolbar. 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 Control 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, when the active mouse tool is the "Selector"  or the "Adjustor". 

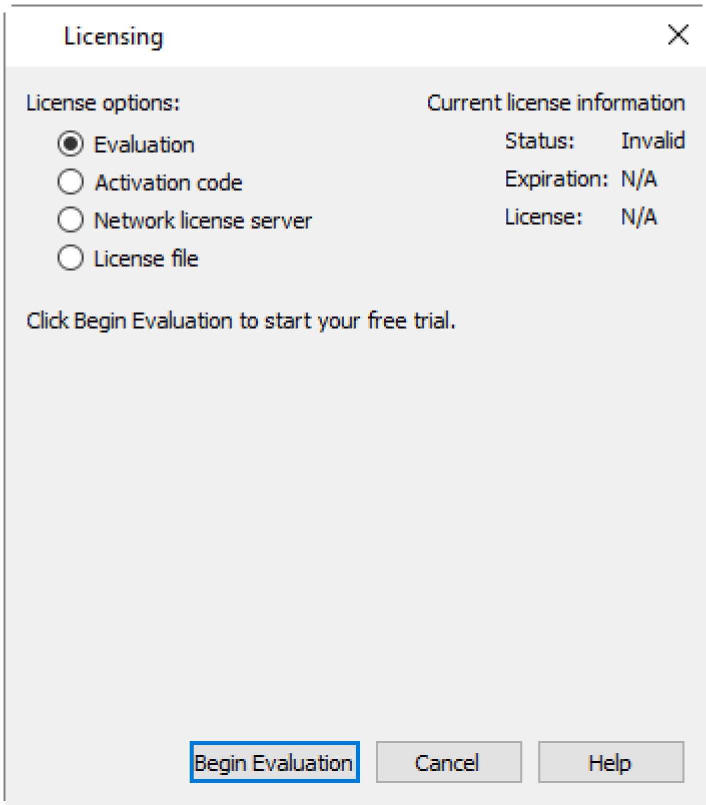
Action	**Middle Button**[13]	**Right Button**
Click	Redraw. If the pointer is in the active frame, Tecplot 360 will redraw the active frame. Otherwise, redraw all frames.	Switch from the current tool to the Selector.
Drag	Smoothly zoom in or out. An upward motion zooms out. A downward motion zooms in.	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	Smoothly zoom in or out. An upward motion zooms out. A downward motion zooms in.

The above actions (and other mouse gestures) are configurable by editing your `tecplot.cfg` file. Find the `$!INTERFACE MOUSEACTIONS` command and look for the `MIDDLEBUTTON` or `RIGHTBUTTON` section within that command.

License Management

Entering Your License

The first time you launch Tecplot 360 after a fresh installation, you will be prompted to enter your license information:



You will also see this dialog if Tecplot 360 cannot validate your license information (for example, because your evaluation license has expired, or because the network license server is not available). Additionally, you may change your license information at any time from the **Help** menu.

Tecplot products store their license configuration file in a platform-specific location. Each Tecplot product has its own license configuration file.

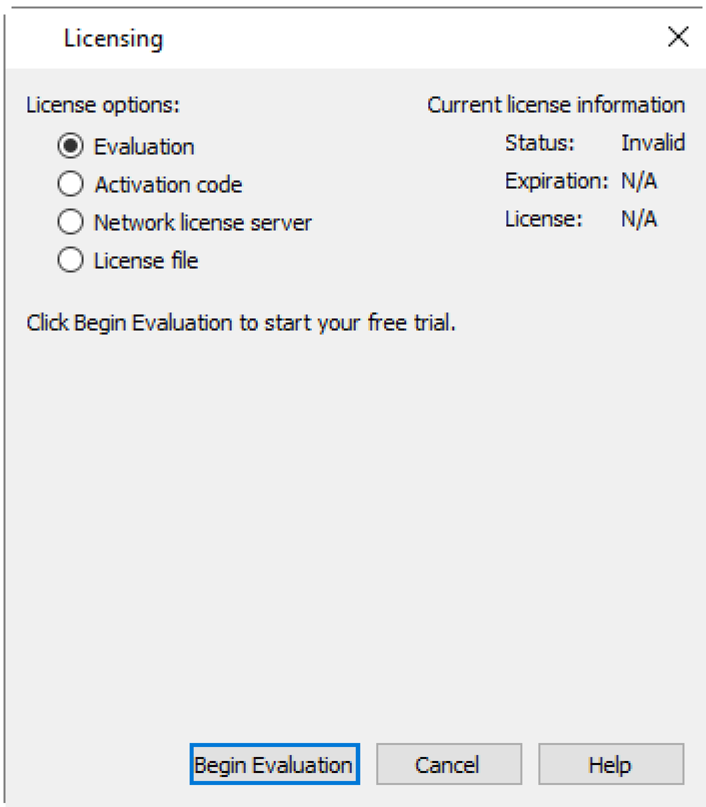
```
Windows: C:\Users\<username>\AppData\Local\Tecplot
Linux: ~/.local/share/data/tecplot
Mac: ~/Library/Application Support/Tecplot
```

The above directories are specific to the individual workstation users. If you want all users of the computer to use the same license configuration for a given Tecplot product, you may move the license configuration file to the product installation directory after initial license setup.
To prevent users from editing the license configuration, you may then change the permissions on the license file to be writable only by an administrator or root user.

How you install your license information depends on what type of license you have.

- If you wish to evaluate the product before purchasing, you may obtain an **Evaluation** license with a single click. See Evaluation License Setup.
- If you have an Activation Code for a **Single-User** license and have Internet access, you enter the activation code. See Single-User License Setup Using An Activation Code.
- If you have a License File for a **Single-User** license, you select the license file. See Single-User License Setup Using A License File.
- If you have a **Network** license, you specify the server name and port number of the RLM license server on your network. Network License Setup.
- If you have a License File for a **Single-User** license, you select the license file. See Single-User License Setup Using A License File.

Evaluation License Setup

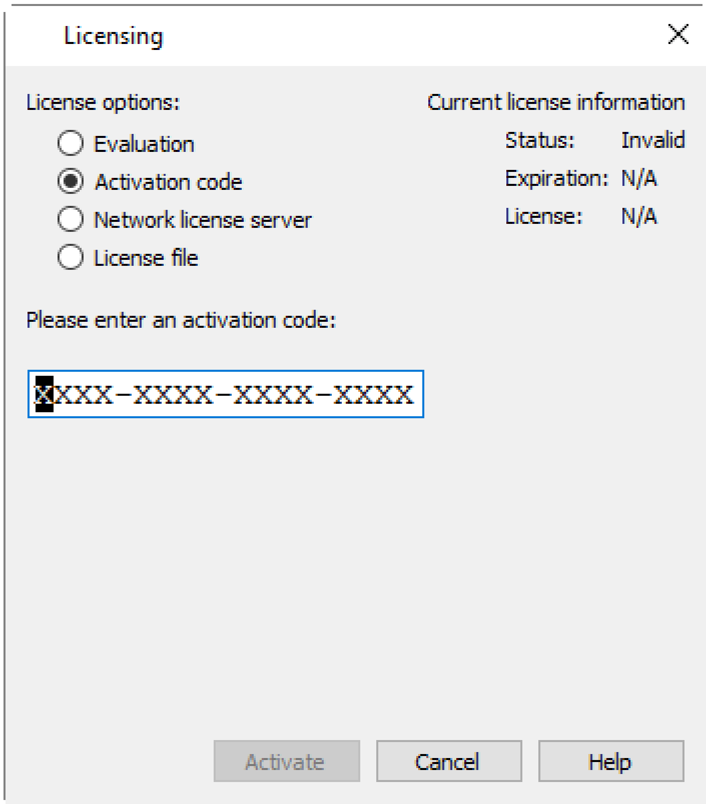


To obtain a time-limited evaluation license key so you can try the product before purchasing, make sure the "Evaluation" license type is selected in the Licensing dialog, then click **Begin Evaluation**. The product evaluation period begins. You will see a notice on the product's Welcome Screen indicating when your evaluation license will expire.

This procedure requires an active Internet connection. If for some reason you cannot fully evaluate Tecplot 360 on an Internet-connected computer, or if you need additional time to complete your evaluation, contact Tecplot Support for a time-limited single-user license file, which may then be installed using the instructions in Single-User License Setup Using A License File.

Single-User License Setup Using An Activation Code

If you received an activation code for your product, allowing it to be activated over the Internet, follow the instructions below. If you received a license key file, or if the workstation you have installed the product on does not have Internet access, see Single-User License Setup Using A License File.



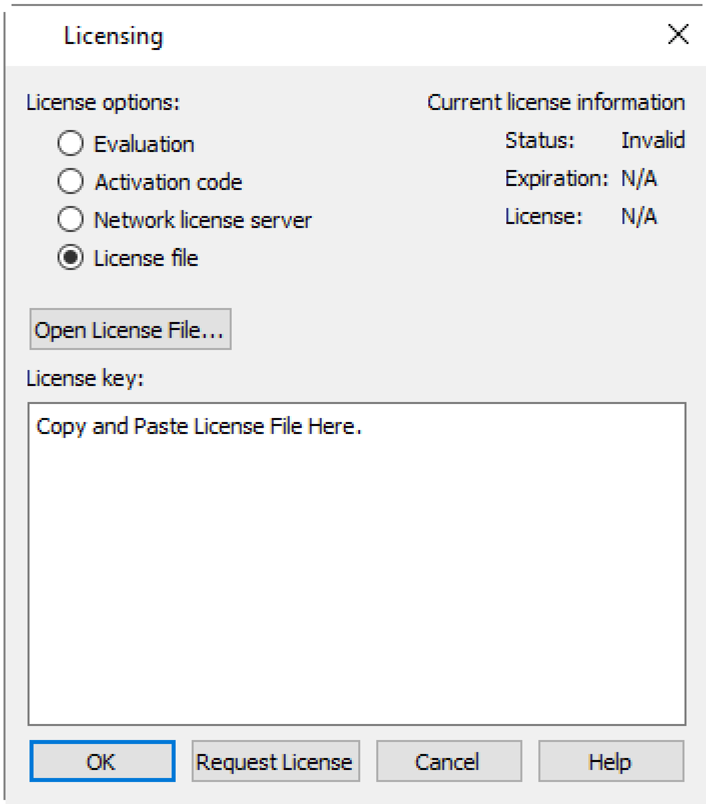
To activate the product, simply enter or paste your activation code in the field provided. Then click **Activate**.

The license key corresponding to your activation code is downloaded to your computer, and Tecplot 360 uses that license.

Single-User License Setup Using A License File

You will need the license file sent to you by Tecplot as an e-mail attachment. Please save it to a file on your computer in an easy-to-find location, such as the Desktop.

> If you do not have your license file, or have lost it, click **Request License** in the Licensing dialog for instructions on how to obtain it.

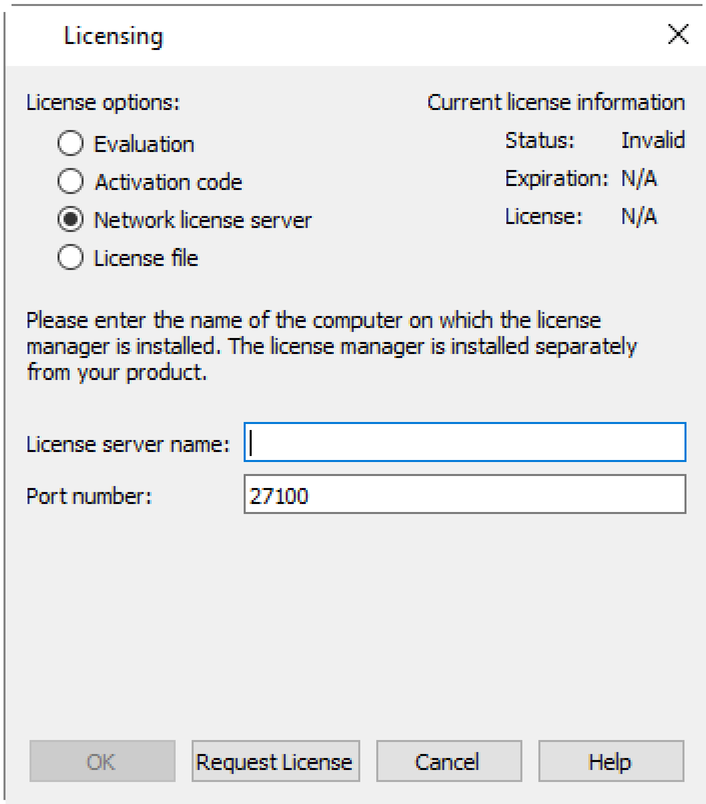


1. Make sure the "License File" option is selected in the Licensing dialog.
2. Click **Open License File**. A file browser appears. Select the license file e-mailed to you, then click **Open**.
   Alternatively, you can Copy and Paste the contents of the license file to the available field.
   The message "Status: Valid" appears in the Current License Information panel at the upper right.
3. Click **OK** to save the license information.

Tecplot 360 now uses the chosen license.

If your license requires a USB license security key ("dongle"), make sure it is connected to your computer before launching Tecplot 360. It must remain connected while the product is running.

Network License Setup



To use a network license, you or your system administrator must first install the Reprise License Manager (RLM) version 11 or newer and the license key for your product on a network server. (See the installation instructions included with the RLM download.) Once this has been accomplished, follow these steps:

1. Make sure the "Network" license type is selected in the Licensing dialog.
2. Enter the server name (you may use a hostname or IP address) and port number of the license server in the fields provided.
3. Click **OK** to save the license information.

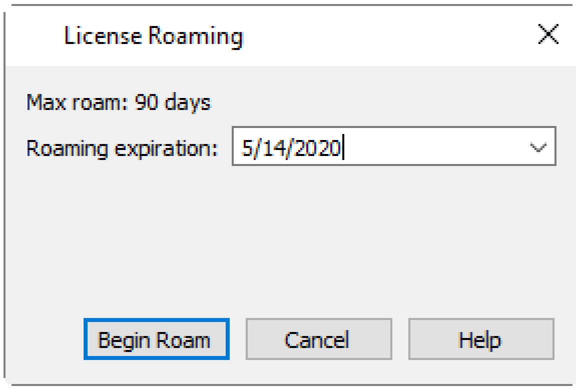
Tecplot 360 will now use a license obtained from the license server.

If the product is unable to obtain a license key, an error message will appear, and you should verify that you have specified the correct network license information.

License Roaming

Network license users who need to use Tecplot 360 while not connected to the same network as their license server (for example, while traveling) can use license roaming. License roaming allows you to use the product without access to the license server for a limited period of time. The roaming license is considered "in use" by the server continuously until it expires or is returned.

To manage license roaming, choose **License Roaming** from the **Help** menu. The License Roaming dialog appears.



Starting Roaming

If you are on a mac then there is one more requirement before you can begin roaming. That being the folder "/Library/Application Support/Reprise" must be writable by all users. If this is not the case you must do the following using a login that has sudo privileges:

```
sudo mkdir -p "/Library/Application Support/Reprise"
sudo chmod 777 "/Library/Application Support/Reprise"
```

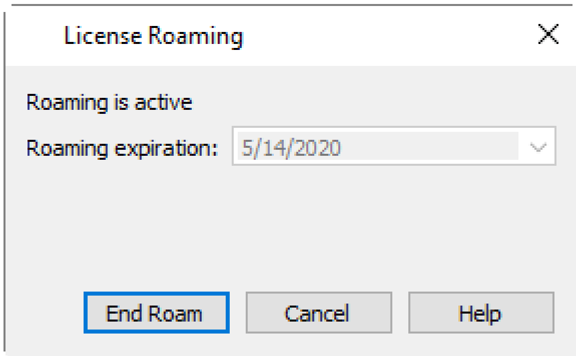
To start roaming, choose the expiration date for the roaming license using the pop-out calendar that appears when you click the roaming expiration date. Your license will expire at the end of that day, at midnight.

To account for time zones, we suggest roaming for a day longer than you think you need when you will be traveling.

Click **Begin Roam** to obtain the roaming license and begin roaming.

The license server administrator and the license key may restrict the maximum amount of time users may roam and/or the total number of licenses that may roam at a given time. Additionally, only one instance of Tecplot 360 may be run while roaming.

While roaming, the license status is noted as Roaming in the Licensing dialog, and the expiration date of the roaming session is also shown. You may not change license information while roaming.



When you are getting close to your roaming license's expiration date, a notification appears in the Welcome Screen at startup.

Do not upgrade Tecplot 360 while roaming. If you do, your license may no longer work correctly. Set the `RLM_ROAM` environment variable to -100 before launching to rectify this situation (see RLM_ROAM).

Ending Roaming

Your roaming license expires automatically at the end of the day (that is, midnight) on the day you selected when you began roaming. After this date, your computer must be able to connect to the network license server to run Tecplot 360.

You may also end roaming early and make your license available for other users. To do this, you must first connect your computer to a network that allows you to reach the license server. Then choose **License Roaming** from the **Help** menu and click the **End Roam** button in the License Roaming dialog.

RLM_ROAM

You can initiate roaming from the command line by setting the `RLM_ROAM` environment variable to the number of days you wish to roam, then starting Tecplot 360 from that command line session. The roaming session expires at the end of the day after the number of days specified (for example, if it is noon Tuesday, and you set `RLM_ROAM` to `2`, the roaming license expires at the end of the day Thursday). `RLM_ROAM` may also be set to the special value `today` to have the roaming license expire at the end of the current day (the value `0` means that no changes to roaming will be made).

If `RLM_ROAM` is set permanently (for example, using the System Properties in Windows), the license roaming will be refreshed each time you run Tecplot 360 while connected to the network. If RLM_ROAM is `1`, for example, each time you start Tecplot 360, the roaming license is refreshed to expire at midnight the following day. This allows you to easily take your computer home with you any night and continue using Tecplot 360 while disconnected from the license server, without needing to explicitly roam each day.

You may end a roaming session early by setting `RLM_ROAM` to `1` before launching Tecplot 360.

If you are having trouble ending your roaming early (a common cause is upgrading

> Tecplot 360 while roaming), try setting `RLM_ROAM` to `-100` before launching Tecplot 360. The roaming license will be forcibly removed from your workstation, and Tecplot 360 will begin requesting a regular license from a license server at each startup. At this point you can obtain a fresh roaming license, if desired. The reservation for your previous roaming license will, however, continue to be held on the license server that issued it until it expires.

LaTeX Setup

A few requirements are needed before LaTeX outputs can be generated. The first is to have a LaTeX engine installed. LaTeX distributions are largely free and available on many different operating systems. For installation recommendations, see the LaTeX Project page.

The second requirement is to have the **tecplot_latex.mcr** file correctly configured in your installation directory. Each LaTeX engine will come configured with different ways to parse commands. Details of how to configure it can be found in the **tecplot_latex.mcr** file. The default settings have been tested with MiKTeX and TeXLive engines. If using the default install options for MiKTeX, the amscls and zhmetrics packages will also need to be installed.

For information on how to load a custom tecplot_latex.mcr file, see Custom Files loaded on Startup.

Troubleshooting

Rendering and Export Troubleshooting

As a software with high graphical and rendering requirements, Tecplot 360 has a few alternative libraries and configurations which allow use on machines which have limited graphics hardware. Some example cases where these configurations may be needed include use on HPC systems, X-Display or other remote desktop solutions.

Sample Interactive Rendering and Export Test

Try these steps. If you have any problems check the OS specific section below.

1. Start 360
2. sample data:
   a. Create data from: Data → Create → Create Rectangular Zone.
   b. Select defaults to create the zone
   c. Turn on contour checkbox on the plot sidebar.
3. Try interacting with the plot:
   a. <Ctrl> + right mouse drag to rotate the plot
   b. click to display the context menu

c. click and drag to select multiple objects in the plot.

4. Go to File → Export, use default PNG settings and select a place to save a sample image. Once export is complete, ensure that the image was created as expected.

Linux Troubleshooting for Interactive Rendering and Export

The following is a set of tests to determine what are the best set of rendering command line options for a Linux system. Run each test in order and STOP when a test was successful.

1. Run `tec360` without any additional command line arguments and step through the sample test above. If everything is successful and performant continue to use Tecplot 360 this way.
2. If any of the sample steps failed:
   a. Ensure your graphics drivers are up to date. If you are using native Linux drivers (usually nouveau) try installing drivers supplied by your graphics card manufacturer.
   b. If you are on the most up to date drivers, try `tec360 --mesa`, then repeat the sample test above. Mesa uses software rendering and will be slower than using graphics hardware but will be the most reliable software rendering option. See the --mesa option in Command Line Options for additional information.

If you struggled following these steps or none of these solutions worked, contact support@tecplot.com.

Linux Batch Mode Troubleshooting

On some systems with limited graphics resources Tecplot 360 may crash in batch mode. If this occurs run **tec360 –b --osmesa**.

If you are running PyTecplot scripts in batch mode you will also need to use the osmesa libraries. For example:

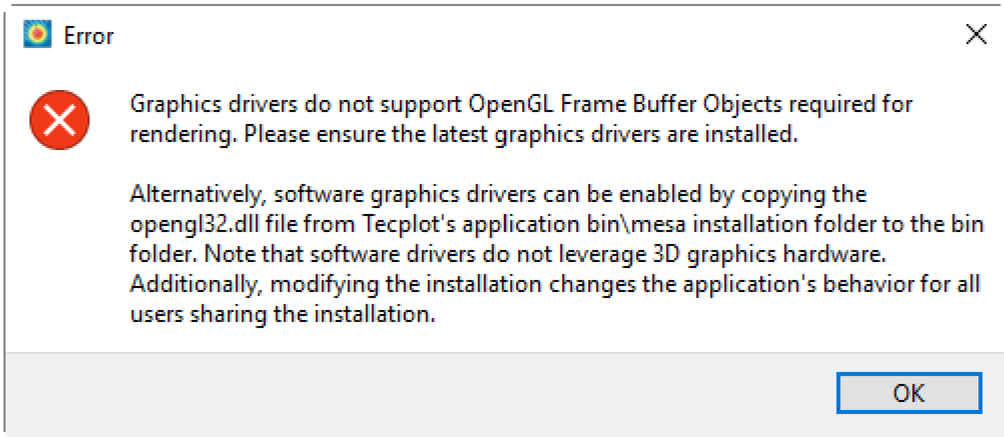
```
tec360-env --osmesa -- python myscript.py
```

To run these with PyTecplot see the environment setup section of the pytecplot installation notes.

tecplot.azureedge.net/products/pytecplot/docs/install.html#environment-setup-batch-only

Windows Troubleshooting

When remoting to or running on some windows systems, either in rendering the work area or during export, you may encounter an error similar to:



This error indicated that the graphics drivers on the machine are not sufficient. In this case, find the mesa version of the OpenGL library in *C:\Program Files\Tecplot\Tecplot 360 EX 2022 R2\bin\mesa\opengl32.dll* and copy it to the bin directory *C:\Program Files\Tecplot\Tecplot 360 EX 2022 R2\bin*. This will enable software rendering for the Windows version of Tecplot 360.

Note that in some cases this same error is presented when exporting images with larger extents. If you are seeing this error exporting images know that in most cases super sample factors larger than 3 show no increase in render quality. As such, try with smaller width and do not go over 3 for a super sample factor.

Mac Troubleshooting

Most, if not all Mac systems, have stricter hardware and software requirements for running MacOS. As such Tecplot 360 does not currently supply any alternative rendering libraries or options today. Contact support@tecplot.com if you need assistance.

Glossary

The following terms are used throughout the Tecplot 360 User’s Manual and are included here for your reference.

2D	Plotting in two dimensions. Line plots of one or more variables (XY and Polar Line plots) are not considered.
2D Cartesian Plot	A plot displaying a 2D scattering of points, surfaces, or volumes using two orthogonal axes.
3D	Plotting in three dimensions. Three-dimensional plotting can be subdivided into 3D surface and 3D volume.
3D Cartesian Plot	A plot displaying a 3D scattering of points, surfaces, or volumes using three orthogonal axes.

3D Sorting	The process that Tecplot 360 uses to determine which surface to plot first. In this process, the cells are sorted relative to the viewer and plotted beginning with the farthest point away and ending with the closest. Sorting is used when printing 3D plots or rendering translucent 3D objects on the screen.
3D Surface	Three-dimensional plotting confined to a surface. For example, the surface of a wing.
3D 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 displayed in the active plot, as determined in the Zone Style dialog.
Add-on	Any component or program which provides additional functions to Tecplot 360.
ADK	Add-on Developer's Kit designed to create add-ons for use with Tecplot 360.
Anchor point	The fixed point of an object on the plot.
Antialiasing	The process of removing or reducing the jagged distortions in curves and diagonal lines.
ASCII Data File	A data file composed of code in a human-readable and editable format, using the ASCII character set encoding.
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, datasets, and frames.
Banded Contour Flooding	A field plot where the region between contour lines is filled with a constant color that corresponds to each variable.
Bars Mapping Layer	Mapping Layer (XY line plots only) where bars are used to depict the relationship between the dependent and independent variables.
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 360 that excludes certain cells and points from a plot.
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 Cell Faces	A set of un-blanked cell faces in a 3D 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-planes, and the first and last K-planes. For a finite element 3D 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 3D volume plotting.
Carpet Plot	A 3D surface plot where a variable is plotted in the third dimension and is singular-valued with respect to the independent variables.
Cartesian Plot	A 2D or 3D plot of some variable by location on a single plane using two axes (or in a volume using three axes).
Cell	Either an element of finite element data, or the space contained by one increment of each index of IJK or IJK-ordered data.
Cell-centered Values	Values located at the center of the cell (assumed to be the centroid).
Code Generator	Add-on that generates style value code for any style changes made in the Tecplot 360 user interface.
Coordinate	Any of a set of numbers used in specifying the location of a point on a plot.
Configuration File	File (tecplot.cfg) containing Tecplot 360 defaults. See Configuration Files.
Color Map	A color spectrum used to plot contour flooding and multi-colored objects.
Connectivity List	The portion of a finite element data file which defines the elements or cells by listing the relationships between points. The number of 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 3D volume plot where a portion of a 3D volume zone is cut-away by blanking to reveal the interior.
Cutting Plane	A planar surface used to slice 3D volume or surface zones.
Data File	A file that contains data used for plotting in Tecplot 360.
Data Format	The type of zone data as specified by the format parameter in a Tecplot data file, such as BLOCK or POINT.
Data Loader	A Tecplot 360 add-on which allows you to read non-Tecplot data files.
Data Point	An XYZ-point at which field variables are defined.
Data Smoothing	A process that shifts the value of a variable at a data point towards an average of the values at its neighboring data points to reduce "noise" and lessen discontinuities in data.
Dataset	A set of one or more zones. A dataset may be plotted in one or more frames. However, a single frame may only plot one dataset. A dataset 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.
Depth Blanking	A blanking option which excludes cells in a 3D 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, i.e. an Iso-Surface, a Slice, or a Streamtrace.
Display List	A group of OpenGL commands that have been stored for subsequent execution. Using a display list 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.
Dynamic Text	Special placeholders added to text that change with the data or the display environment.
Edge	A 2D or 3D field plot option. Plotting the edge of a zone plots the connection of all outer lines (IJ-ordered zones), finite element surface zones, or planes (IJK-ordered zones).

Element Type	The form of individual elements in a finite element zone. There are four types of cell-based finite element zones: Triangle and Quadrilateral (finite element surface types), and Tetrahedron and Brick (finite element volume types). For cell-based finite elements, the element type of a zone determines the number of nodes per element and their orientation within an element. There are two types of face-based finite element zones: polygonal (2D) and polyhedral (3D). For face-based elements, the number of nodes per element is variable.
Error Bars Mapping Layer	XY Line mapping layer where a second dependent variable (error) is used to show the accuracy of the first dependent variable, typically used in conjunction with the Bars Mapping Layer.
Exposed Cell Faces	The set of those cell faces in 3D 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 360 add-on which extends Tecplot 360's XY-plot curve-fitting capabilities.
Eye Coordinate System	Coordinate system aligned with the Grid coordinate system so that objects that are drawn move with the data as you zoom and translate, but remain fixed when you rotate the plot.
Extra 3D Sorting	Perform extra work to resolve hidden surface problems encountered during 3D sorting.
Face Neighbor	A neighboring cell whose faces share all nodes in common with the selected cell.
FE	An abbreviation for finite element (a common means of arranging data for calculations, often referred to as "unordered" or "unstructured").
FEA	An abbreviation for finite element analysis. See FEA Loader for information and supported formats.
Fence Plot	A plot of planes of a 3D data field.
FE Surface	A finite element zone of the element type Triangle or Quadrilateral, or Polygon . These zones are used for 2D and 3D surface plots.
FE Volume	A finite element zone of the element type Tetrahedron or Brick, or Polyhedron . These zones are used for 3D volume plots.
Field Map	A collection of zones for 2D and 3D field plots. A common style can be easily applied to all zones in the selection.

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	A way to specify the directory for Tecplot 360 to locate a given file. For instance, a linked layout saved with an absolute file path contains the complete directory structure to load the associated file starting with the root directory or (on Windows) the drive letter.
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 six different element types supported by Tecplot 360: Triangle, Quadrilateral, Brick, Polygonal, and Polyhedral. See also Connectivity List and node].
Font Modifier	The modifier used to embed Greek, Math, or User-Defined characters in a text string.
Frame	Area within the workspace where sketches and plots are created. The workspace shows a page at a time, and each page can contain multiple frames.
Frame Coordinate System	Coordinate system fixed to the frame that does not change when the plot is zoomed, translated, or rotated.
Gouraud Shading	Shading used to achieve smooth lighting on low-polygon surfaces by linearly interpolating a color or shade across a polygon.
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.
Grid Coordinate System	The grid coordinate system consists of two dimensional physical coordinates that are aligned with the coordinate system used by the plot axes.
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 2D, the intersection of gridlines.
Hidden Line	Mesh type where mesh lines that appear behind other plot layers are not drawn.

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 2D or 3D, this type of data point ordering is sometimes called irregular, and is only useful for scatter plots, or for interpolating or triangulating into 2D, 3D surface, or 3D volume zones. (This type of data can also be used for 2D or 3D vector plots if streamtraces are not required.)
IJ-Ordered	A type of data point ordering where the points are arranged in an array used for 2D and 3D 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 3D array. Used for 3D volume plotting as well as 2D and 3D surface plotting.
Image Format	Any of the raster or bit-mapped graphic formats supported by Tecplot 360.
Inactive Zone	A zone loaded into Tecplot 360 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 any 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).
Interpolate Mode	When probing is activated using a single mouse click, the value returned is linearly interpolated from all nodes in the cell. See also: Nearest Point Mode.
Internal Macro Variable	A read-only macro variable which allows you to access certain key values in Tecplot 360. 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 IJK or IJK-ordering.
Iso-Surface	A surface within a 3D zone where the contour variable has a constant value at all locations.
Journal	Log of data manipulation/creation/deletion instructions.
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.
Kriging	A technique to interpolate the value of a random field at an unobserved location from observations of its value at nearby locations.
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 with extension .lay which preserves a plot created within Tecplot 360. When the layout is opened, it restores Tecplot 360 to the state it was in when the layout file was saved.
Layout Package File	A binary layout file with the .lpk extension which has the data embedded in the file.
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 360.
Macro Command	An instruction given to Tecplot 360 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 Play option of the Scripting menu.
Macro Function	A self-contained macro sub-routine.
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 360 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 3D, the grid axis which when scaled is not the shortest nor the longest axis.
Menu Bar	The top bar of the Tecplot 360 screen used to select menu options.
Mesh	A 2D or 3D field plot type which plots connections between data points.
Multi-Colored	Any Tecplot 360 object which is colored by the value of the contouring variable. Multi-colored objects may include mesh, scatter symbols, vectors, contour lines, and streamtraces.
Nearest Point Mode	When probing is activated using a Control+click, the value returned is the precise value of the closest data point. See also: Interpolate Mode.
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 3D graphics. It commonly takes advantage of hardware acceleration for 3D 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.

Orthographic Projection	A plot view in which the shape of the object is independent of distance.This is used for displaying physical objects when preserving the true lengths is important.
Overlay	Mesh type where mesh lines are drawn over all field-plot layers, except for vector and scatter layers.
Pick	Select an item in the plot area by clicking on it with the mouse.
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.
Polar Line Plot	A 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).
Polygonal	A 2D, face-based finite element type. The number of nodes per element is variable. That is, a single polygonal zone may contain triangular, quadrilateral, hexagonal, …, etc. elements.
Polyhedral	A 3D, face-based finite element type. The number of nodes per element is variable. That is, a single polyhedral zone may contain tetrahedral and brick (and other) elements.
Polylines	A shape composed of one or more joined line segments. May be closed (i.e., a polygon) or open.
Polytope	The generalization to any dimension of "polygon" in two dimensions, "polyhedron" in three dimensions, and "polychoron" in four dimensions
Pop	Bring selected geometries or text to the top of the viewstack.
Precise Dot Grid	The points of intersection of the imaginary lines extending from the X- and Y-axes' tick marks.
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.
Probe	To obtain interpolated values of the dataset variables at a specified location by clicking on a point in the data region.
Push	Send selected geometries or text to the bottom of the viewstack.
Quadrilateral	An element type of finite element surface data which is composed of four node points arranged in a quadrilateral. Used in 2D and 3D surface plotting.
Quick Macro Panel	A user-defined panel accessed from the Scripting menu which allows quick access to your macro functions.
Rake	A line along which a number of streamtraces may be seeded.

RGB Color Flooding	The assignment of color based on Red, Green, and Blue components defined at field data locations.
Ribbon	(See Streamribbon.)
Richardson Extrapolation	A sequence acceleration method, used to improve the rate of convergence of a sequence.
Rod	(See Streamrod.)
Scatter	A 2D or 3D field plot type which plots a symbol at each data point.
SDK	A Tecplot software development kit, providing a library that integrates visualization capabilities into an application.
Shade Plot	A 2D or 3D 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 360 until the number of parties accessing the data is reduced to zero. Variables and connectivity information may be shared.
Sidebar	A movable, dockable panel (by default located on the left side of the Tecplot 360 window) which provides quick access to frequently-used functionality. Tecplot 360 includes multiple sidebars: Plot, Frames, Pages, Probe, and Quick Macro.
Sketch Plot	A plot which displays only text and geometries, not any data-derived elements.
Slice	A set of data created by the intersection of a plane with 3D 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.
Sorting	The process of resolving hidden-surface problems during display of a 3D plot.
Specular Highlights	Highlights with qualities similar to those of a smooth reflecting surface such as metal, where light is reflected so intensely from some points that the plot appears white (or the color of the light) at those points.
Status Line	The bar located at the bottom of the Tecplot 360 screen which provides "hover" help for tools and acts as a progress bar during calculations.
Strand	A series of transient zones that represent the same part of a dataset at different times.
Stream	Particle traces through a vector field.

Stream Format	The type of streamtraces being placed in Tecplot 360. For example, Surface Line, Volume Line, Volume Ribbon, or Volume Rod.
Streamline	A 2D or 3D line which is parallel to the vector field 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 Termination Line	A polyline that terminates any streamtraces that cross it.
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 360 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 3D streamline which is confined to remain on a 3D surface. Also used to refer to streamlines.
Symbols Mapping Layer	Line plot where symbols are used to depict the relationship between the dependent and independent variables.
Tecplot Toolbox	The Tecplot Toolbox is a convenience library that provides alternate method for communicating with the Tecplot Engine.
Time/Date	A type of axis label format with which you can label axes by using a multitude of codes that can display data in years, months, days, hours, minutes, and seconds.
Toolbar	The bar at the top of the Tecplot 360 workspace.
Tetrahedron	An element type of finite element volume data which is composed of four node points arranged in a tetrahedron. (Used in 3D volume plotting.)
Transient Data	Data that has a time component in addition to a spatial coordinate.

Translucency	A property allowing you to see through an object to areas within or beyond it. In Tecplot 360 you may vary the amount of translucency, controlling the extent that an object closer to you obscures an object it overlays.
Triangle	An element type of finite element surface data which is composed of three node points. (Used in 2D and 3D surface plotting.)
Unordered or Unorganized Data	(See Irregular Data.)
Value-Blanking	A feature of Tecplot 360 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 dataset.
Vector Layer	A field plot showing the direction and or the magnitude of vector quantities.
Volume Line	A type of 3D streamline which is not confined to remain on a surface and may travel through 3D 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.")
Wire Frame	Mesh type where mesh lines are drawn behind all other plot layers.
Workspace	The portion of your screen where you can create Tecplot 360 frames. This includes but is not limited to the region covered by the displayed paper.
XY-Dependent	A 3D 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, and curve fitted lines are all examples of XY Line plots.
Zone	A subset of a dataset 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 2D or 3D plot’s dataset. The plot is the sum of the active zone layers, which may include mesh, contour, vector, shade, scatter and edge.

Calculate Variables Reference

This chapter details the functions available in the Calculate dialog (accessed via the **Calculate Variables** command on the **Analyze** menu). Formulae, where not trivial, are given for each function. For functions that have equivalent PLOT3D function numbers, the numbers are listed as well. Refer to Selecting a Function for a description of how to use these numbers.

Symbols

The following symbols are used in formulae below. Other symbols are defined in context.

Table 12. Analyze Symbology

Symbol	Description
$()_\infty$	Reference or free-stream quantity.
g	Ratio of specific heats, $\frac{C_p}{C_v}$
r	Density, mass per unit volume (area in 2D).
ξ	Generalized curvilinear coordinate in the I-direction.
η	Generalized curvilinear coordinate in the J-direction.
ζ	Generalized curvilinear coordinate in the K-direction.
ω	Vorticity
a	Speed of sound.
c_p	Specific heat at constant pressure.
c_v	Specific heat at constant volume.
M	Mach number.
m	Mass.
p	Pressure.
R	Specific gas constant $p = \rho RT$
T	Temperature.
U	Velocity vector.
u	X-velocity component.
v	Y-velocity component.
w	Z-velocity component.

Scalar Grid Quality Functions

I, J, K-aspect Ratio

The ratio of maximum edge length squared to face area:

$$AR = \frac{(Max\, edge\, length)^2}{Area}$$

For a rectangle or square, this simplifies to: $AR = \frac{height}{width}$

For collapsed faces where the area is zero, the aspect ratio is set to zero.

For Polyhedral zones, the maximum aspect ratio is calculated for all faces of a given cell.

I, J, K-stretch Ratio

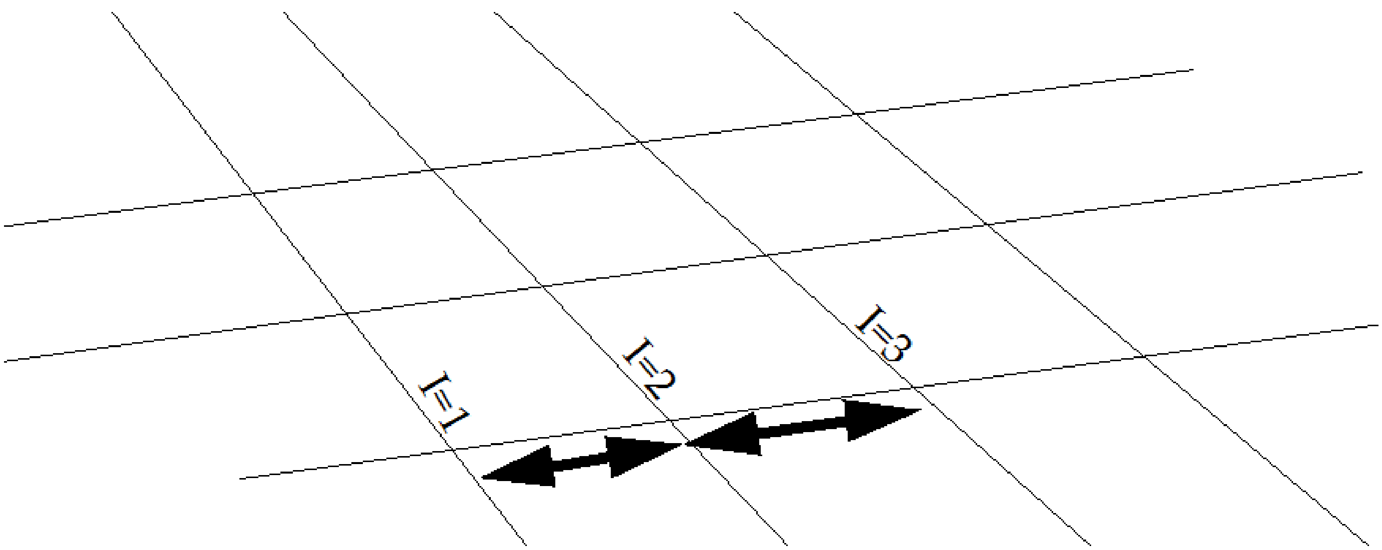
The ratio of the length of line segment I2-I3 to segment I1-I2 (or J or K):

$$stretch\, ratio = \frac{length\, of\ segment\ I2 - I3}{length\, of\ segment\ I1 - I2}$$

or

$$\frac{length\, of\ segment\ I1 - I2}{length\, of\ segment\ I2 - I3}$$

such that the stretch ratio is always > 1.



If either segment has zero length, the stretch ratio is set to one.

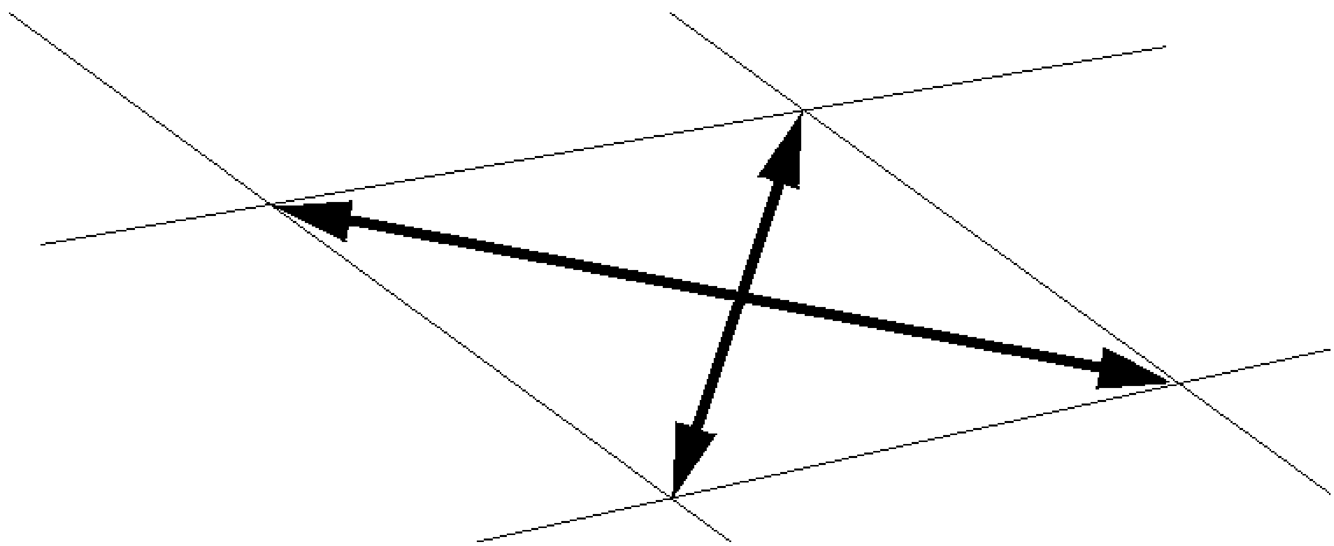
If you have specified on the **Geometry and Boundaries** dialog that adjacent zones are connected, these stretch ratios will be made continuous across connected zone boundaries provided that the index directions are aligned.

I, J, K-face Skewness

The ratio of the two face diagonal lengths subtracted from one (the diagonals are ratioed so that this number is always non-negative):

$$\text{face skewness} = 1 - \frac{length\,of\;shorter\;face\,diagonal}{length\,of\;longer\;face\,diagonal}$$

For polyhedral zones, the cell value is the maximum skewness over all cell faces.



Cell Diagonal1 or Diagonal2 Skewness

The ratio of the lengths of two body diagonals subtracted from one (always non-negative). There are four body diagonals. We choose pairs which would be coplanar in an unskewed cell, that is, (i,j,k) → (i+1,j+1,k+1) and (i,j,k+1) → (i+1,j+1,k).

$$\text{cell skewness} = 1 - \frac{length\,of\;shorter\;body\,diagonal}{length\,of\;longer\;body\,diagonal}$$

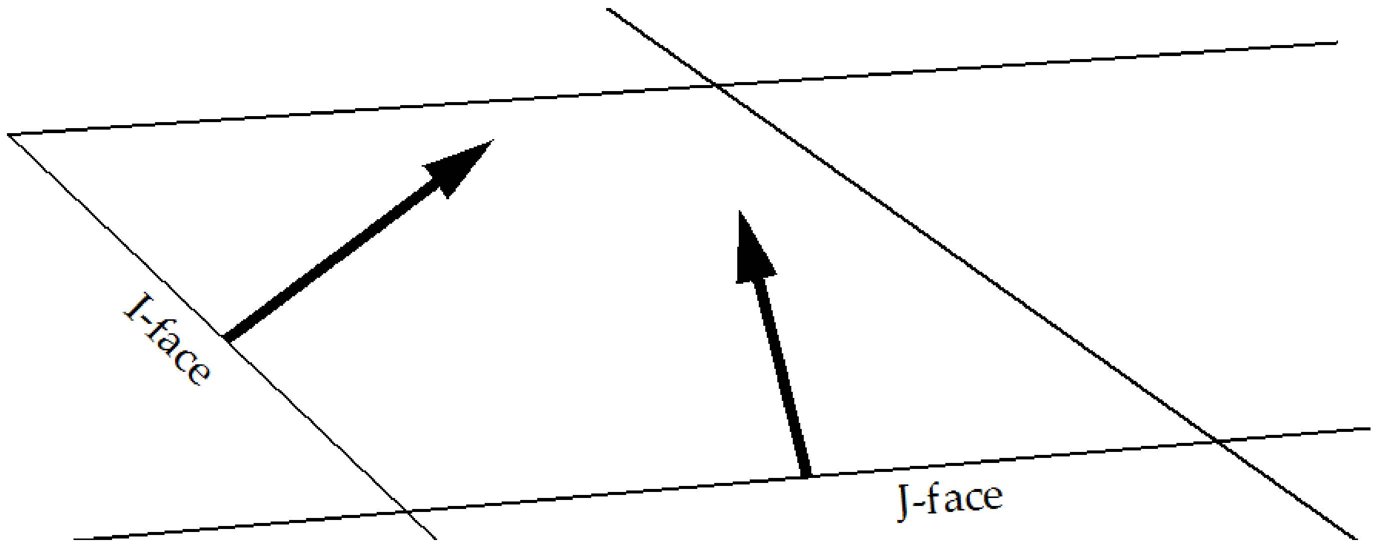
For polygonal zones, the ratio of the lengths will be zero.

IJ, JK, KI, or Max Normals Skewness

The dot product of face unit normals for the two given faces.

$$\text{IJ-skewness: } S_{IJ} = |\hat{n}_I \cdot \hat{n}_J|$$

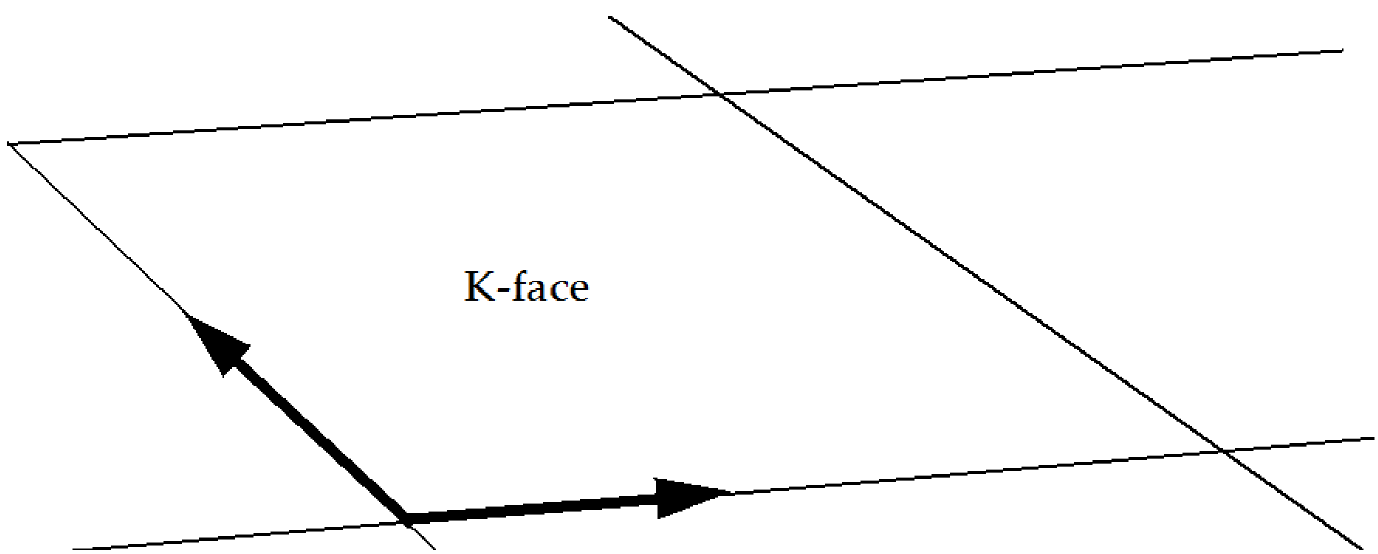
The following figure illustrates this for IJ-skewness.



I, J, K, or Min Orthogonality

One minus the absolute value of the dot product of two unit vectors which point in the direction of two adjacent edges of the given face.

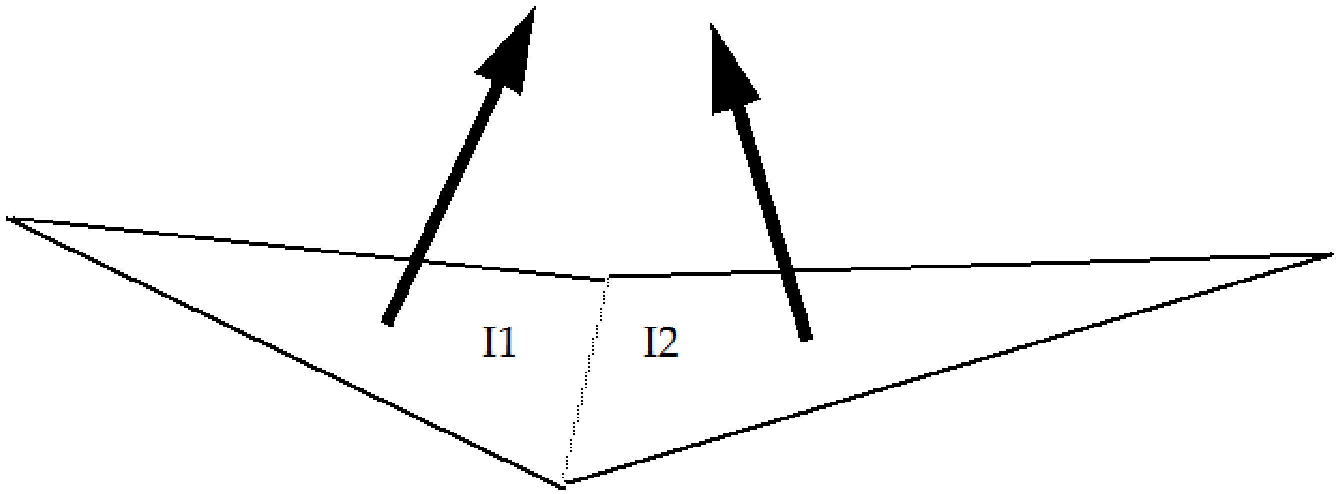
$$\text{For the K-face:} orthogonality = 1 - |\hat{t}_I \cdot \hat{t}_J|$$



I, J, K, or Min Nonplanarity

Two triangles are formed with the four nodes of the face, and the dot product of the two unit normals of those triangles is subtracted from one.

$$\text{Non-planarity of the four-node face shown below} = 1 - |\hat{n}_{I1} \cdot \hat{n}_{I2}|$$



For polyhedral zones, the max over all cell faces is found.

Jacobian

For ordered zones, the Jacobian is calculated with the standard formula.

$$J = \frac{1}{x_{\xi}(y_{\eta}z_{\zeta} - y_{\zeta}z_{\eta}) - x_{\eta}(y_{\xi}z_{\zeta} - y_{\zeta}z_{\xi}) + x_{\zeta}(y_{\xi}z_{\eta} - y_{\eta}z_{\xi})}$$

The subscripts above represent partial derivatives, which are approximated with finite differences.

For finite element zones, Tecplot 360 approximates the Jacobian by inverting the average areas or volumes of the grid cells surrounding each node, 1/A or 1/V.

If the denominator of the above formula is zero (ordered zones), or all cells surrounding a node have zero area (finite element zones), the Jacobian is set to zero.

Cell Volume

For ordered zones, the cell volume for a particular node (I, J, K) is the volume of the cell between nodes (I, J, K) and (I+1, J+1, K+1). In 2D, this function becomes cell area. Nodes on the IMax, JMax, and KMax boundaries are assigned the same value as the nodes at IMax-1, JMax-1, and KMax-1 respectively.

For finite element zones, the cell volume for a node is the average volume (area in 2D) of all cells of which that node is a part.

Vector Grid Quality Functions

Grid I, J, or K-unit Normal

Vectors of unit length normal to I=, J=, or K=constant grid planes.

unit normal for $I = \hat{n}_I$

For polyhedral zones, the result will be set to (1, 0, 0).

Scalar Flow Variables

Density

The mass per unit volume of the fluid:

$$\rho = \frac{m}{V}$$

PLOT3D function numbers: 100 (not normalized), or 101 (normalized).

Stagnation Density

$$\rho^0 = \rho(1 + \frac{\gamma - 1}{2}M^2)^{\frac{1}{\gamma - 1}} \text{ (compressible)}$$

$$\rho^0 = \rho \text{ (incompressible)}$$

PLOT3D function numbers: 102 (not normalized), or 103 (normalized).

Pressure

$$p = \rho RT \text{ (compressible)}$$

PLOT3D function numbers: 110 (not normalized), or 111 (normalized).

Stagnation Pressure

$$p^0 = p(1 + \frac{\gamma - 1}{2}M^2)^{\frac{\gamma}{\gamma - 1}} \text{ (compressible)}$$

$$p^0 = p + \frac{1}{2}\rho\|U\|^2 \text{ (incompressible)}$$

PLOT3D function numbers: 112 (not normalized) or 113 (normalized).

Pressure Coefficient

$$C_p = \frac{p - p_\infty}{\frac{1}{2}\rho_\infty u_\infty^2}$$

PLOT3D function number: 114 (not normalized). There is no function number for normalized pressure coefficient, since reference value normalization is not possible (the free-stream pressure coefficient is zero).

Stagnation Pressure Coefficient

$$C_{p^0} = \frac{p^0 - p_\infty}{\frac{1}{2}\rho_\infty u_\infty^2}$$

PLOT3D function number: 115 (not normalized). As above, there is no function number for normalized stagnation pressure coefficient.

Pitot Pressure

Equals stagnation pressure for subsonic/incompressible flow. For supersonic flow:

$$p_{02} = p\frac{(\frac{\gamma+1}{2}M^2)^{\frac{\gamma}{\gamma+1}}}{(\frac{2\gamma}{\gamma+1}M^2 - \frac{\gamma-1}{\gamma+1})^{\frac{1}{\gamma-1}}}$$

PLOT3D function number: 116 (not normalized).

Pitot Pressure Ratio

The pitot pressure divided by the free-stream pressure. PLOT3D function number: 117 (not normalized).

Dynamic Pressure

$$q = \frac{1}{2}\rho\|U\|^2$$

PLOT3D function number: 118 (not normalized).

Temperature

$$T = \frac{p}{\rho R} \ \ \text{(compressible)}$$

PLOT3D function numbers: 120 (not normalized), or 121 (normalized).

Stagnation Temperature

$$T^0 = T(1 + \frac{\gamma-1}{2}M^2) \ \ \text{(compressible)}$$

$$T^0 = T \ \ \text{(incompressible)}$$

PLOT3D function numbers: 122 (not normalized), or 123 (normalized).

Enthalpy

per unit mass:

$$h = c_pT$$

$$c_p = \frac{\gamma R}{\gamma - 1} \text{ (compressible\:only)}$$

PLOT3D function numbers: 130 (not normalized), or 131 (normalized).

Stagnation Enthalpy

per unit mass:

$$h^0 = c_pT + \frac{1}{2}\|U\|^2$$

PLOT3D function numbers: 132 (not normalized), or 133 (normalized).

Internal Energy

per unit mass:

$$e = c_vT$$

$$c_v = \frac{R}{\gamma - 1}\ \text{(compressible\:only)}$$

PLOT3D function numbers: 140 (not normalized), or 141 (normalized).

Stagnation Energy

per unit mass:

$$e^0 = c_vT + \frac{1}{2}\|U\|^2$$

PLOT3D function numbers: 142 (not normalized), or 143 (normalized).

Stagnation Energy per Unit Volume

Stagnation energy multiplied by density. PLOT3D function number: 163 (not normalized).

Kinetic Energy

Per unit mass, one-half the square of the velocity magnitude.

$$KE = \frac{1}{2}\|U\|^2$$

PLOT3D function numbers: 144 (not normalized), or 145 (normalized).

Velocity Components *U*, *V*, or *W*

The scalar velocity components. PLOT3D function numbers: 150 (u, not normalized), 151 (v, not normalized), or 152 (w, not normalized).

Velocity Magnitude

The 2-norm of the velocity vector components:

$$\|U\| = \sqrt{u^2 + v^2 + w^2}$$

PLOT3D function number: 153 (not normalized).

Mach Number

The flow speed divided by the local speed of sound, for compressible flow:

$$M = \frac{\|U\|}{a}$$

PLOT3D function number: 154 (not normalized).

Speed of Sound

$$a = \sqrt{\gamma RT} = \sqrt{\frac{\gamma p}{\rho}} = \sqrt{\gamma(\gamma - 1)(\frac{e}{\rho} - \frac{1}{2}\|U\|^2)} \text{ (compressible)}$$

PLOT3D function number: 155 (not normalized).

Cross Flow Velocity

This presumes that free-stream velocity is purely in the X-direction:

$$v_{cf} = \sqrt{v^2 + w^2}$$

PLOT3D function number: 156 (not normalized).

Equivalent Potential Velocity Ratio

The ratio of velocity magnitude to the potential velocity, as calculated with the incompressible Bernoulli equation. Refer to previous sections for definitions of $\|U\|$ and p^0.

$$\frac{\|U\|}{\sqrt{\frac{p^0_\infty - p}{0.5\rho}}}$$

PLOT3D function number: 159 (not normalized).

X, Y, Z-momentum Component

Per unit volume, the product of density and the scalar velocity components.

$$momentum_x = \rho u$$

PLOT3D function numbers: 160 (X-momentum, not normalized), 161 (Y-momentum, not normalized), 162 (Z-momentum, not normalized).

Entropy

$$s = c_v ln(\frac{p}{p_\infty}) + c_p ln(\frac{\rho_\infty}{\rho})$$

PLOT3D function number: 170 (not normalized).

Entropy Measure S1

$$s_1 = \frac{p}{p_\infty}(\frac{\rho}{\rho_\infty})^{-\gamma} - 1$$

PLOT3D function number: 171 (not normalized).

X-, Y-, Z-Vorticity

$$\begin{bmatrix} \omega_x \\ \omega_y \\ \omega_z \end{bmatrix} = \begin{bmatrix} \frac{\partial w}{\partial y} - \frac{\partial v}{\partial z} \\ \frac{\partial u}{\partial z} - \frac{\partial w}{\partial x} \\ \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} \end{bmatrix}$$

PLOT3D function numbers: 180 (X-Vorticity, not normalized), 181 (Y-vorticity, not normalized), 182 (Z-vorticity, not normalized).

Vorticity Magnitude

$$\|\omega\| = \sqrt{\omega_x^2 + \omega_y^2 + \omega_z^2}$$

PLOT3D function number: 183 (not normalized).

Q Criterion

$$Q = \frac{1}{2}(|\Omega|^2 + |S|^2)$$

Where Ω and S are the anti-symmetric and symmetric components of the velocity gradient tensor:

$$\Omega = \frac{1}{2}(\nabla \vec{u} - (\nabla \vec{u})^T)$$

$$S = \frac{1}{2}(\nabla \vec{u} + (\nabla \vec{u})^T)$$

and the norms for the tensors are the Frobenius norms—the square root of the sum of the squares of all tensor elements.

Swirl

$$Swirl = \frac{\omega \cdot U}{\rho \|U\|^2}$$

PLOT3D function number: 184 (not normalized).

Velocity Cross Vorticity Magnitude

$$|U \times \omega|$$

PLOT3D function number: 185 (not normalized).

Helicity

$$H = U \cdot \omega$$

PLOT3D function number: 186 (not normalized).

Relative Helicity

$$H_r = \frac{U \cdot \omega}{\|U\| \, \|\omega\|}$$

PLOT3D function number: 187 (not normalized).

Filtered Relative Helicity

H_r as calculated above, but set to zero when: $|U \cdot \omega| < 0.1 U_\infty^2$

PLOT3D function number: 188 (not normalized).

Shock

For compressible flow:

$$\frac{U}{a} \cdot \frac{\nabla p}{\|\nabla p\|}$$

PLOT3D function number: 190 (not normalized).

Filtered Shock

Shock, as shown above, but set to zero when the magnitude of the pressure gradient $\|\nabla p\| < 0.1 \gamma p_\infty$

PLOT3D function number: 191 (not normalized).

Pressure Gradient Magnitude

$$\|\nabla p\| = \sqrt{p_x^2 + p_y^2 + p_z^2}$$

PLOT3D function number: 192 (not normalized).

Density Gradient Magnitude

$$\|\nabla \rho\| = \sqrt{\rho_x^2 + \rho_y^2 + \rho_z^2}$$

PLOT3D function number: 193 (not normalized).

X, Y, Z-density Gradient

$$\begin{bmatrix} \rho_x \\ \rho_y \\ \rho_z \end{bmatrix} = \begin{bmatrix} \frac{\partial \rho}{\partial x} \\ \frac{\partial \rho}{\partial y} \\ \frac{\partial \rho}{\partial z} \end{bmatrix}$$

PLOT3D function numbers: 194 (X-density Gradient, not normalized), 195 (Y-density Gradient, not normalized), 196 (Z-density Gradient, not normalized).

Shadowgraph

The Laplacian of density, $\nabla^2 \rho$.

PLOT3D function number: 197 (not normalized).

Divergence of Velocity

$$\nabla \cdot U = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z}$$

PLOT3D function number: 158 (not normalized).

Sutherland's Law

Sutherland's Law is a method of estimating the viscosity of a fluid from its temperature. The formula is:

$$\mu = C_1 \frac{T^{\frac{3}{2}}}{T + C_2}$$

For the constants, Tecplot 360 uses the meters/kilograms/seconds values for air,

$$C_1 = 1.458 \times 10^{-6} \frac{kg}{m\, s \sqrt{K}}$$

and C_2 = 110.4 K. Unlike other functions, this function is units-specific. Tecplot 360 uses the meters/kilograms/seconds units for this calculation, so the input temperature (data set variable) must be in Kelvin. The resulting viscosity will be in units of: kg /m s.

Isentropic Density Ratio

$$\frac{\rho^0}{\rho} = (1 + \frac{\gamma - 1}{2} M^2)^{\frac{1}{\gamma - 1}}$$

Isentropic Pressure Ratio

$$\frac{p^0}{p} = (1 + \frac{\gamma - 1}{2} M^2)^{\frac{\gamma}{\gamma - 1}}$$

Isentropic Temperature Ratio

$$\frac{T^0}{T} = 1 + \frac{\gamma - 1}{2} M^2$$

Vector Flow Variables

Velocity

The velocity vector, U. PLOT3D function number: 200 (not normalized).

Vorticity

See above for vorticity components. PLOT3D function number: 201 (not normalized).

Momentum

Per unit volume, density multiplied by the velocity vector. PLOT3D function number: 202 (not normalized).

Perturbation Velocity

$$U' = U - U_\infty$$

PLOT3D function number: 203 (not normalized).

Velocity Cross Vorticity

$$U \times \omega$$

PLOT3D function number: 204 (not normalized).

Pressure Gradient

The vector of pressure partial derivatives in space:

$$\nabla p = \begin{bmatrix} \frac{\partial p}{\partial x} \\ \frac{\partial p}{\partial y} \\ \frac{\partial p}{\partial z} \end{bmatrix}$$

PLOT3D function number: 210 (not normalized).

Density Gradient

The vector of density partial derivatives in space:

$$\nabla \rho = \begin{bmatrix} \frac{\partial \rho}{\partial x} \\ \frac{\partial \rho}{\partial y} \\ \frac{\partial \rho}{\partial z} \end{bmatrix}$$

PLOT3D function number: 211 (not normalized).

The Velocity Gradient Tensor

In addition to the scalar and vector variables listed in the previous sections, Tecplot 360 can calculate one tensor variable, the velocity gradient:

$$\nabla U = \begin{bmatrix} \frac{\partial u}{\partial x} & \frac{\partial u}{\partial y} & \frac{\partial u}{\partial z} \\ \frac{\partial v}{\partial x} & \frac{\partial v}{\partial y} & \frac{\partial v}{\partial z} \\ \frac{\partial w}{\partial x} & \frac{\partial w}{\partial y} & \frac{\partial w}{\partial z} \end{bmatrix}$$

Each component in the tensor is stored as a separate variable in the dataset. The names indicate which component they represent, such as dUdX, dUdY and so on.

Functional Limits

Hard Limits

The following hard limits are designed into Tecplot 360. Your workstation's memory may, however, not always be sufficient to allow the product to reach these limits.

Table 13. Hard limits for Tecplot 360

Item	**Limit**
Maximum number of data points per data set	Quintillions (limited by memory)
Maximum number of data points per zone	Quintillions (limited by memory)
Largest floating point absolute value	10^{150}
Smallest non-zero floating point absolute value	10^{-150}
Maximum number of datasets	2048
Maximum number of zones	More than 2 billion
Maximum number of variables	More than 2 billion
Maximum number of frames	2048
Maximum number of pages	2048
Maximum number of macro variables	400

Item	Limit
Maximum number of value blank constraints	8
Maximum number of contour groups	8
Maximum number of geometries	Limited only by memory
Maximum number of polylines per line geometry[14]	50
Maximum number of points per circle or ellipse	720
Maximum number of custom label sets	Limited only by memory
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	32,000
Maximum number of streamtrace steps	10,000
Maximum number of color map overrides	16
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 dataset title	256
Maximum number of views per view stack	16
Maximum number of characters in an auxiliary data string	32,000
Line length limit in ASCII `.dat` files (character limit per line)	32,000

Table 14. Hard Limits for plot styles

Item	Limit
Printing Gouraud shaded plots with continuous flooding	On screen or exported bitmap image only
Printing plots with translucency	On screen or exported bitmap image only

Soft Limits

The following soft limits may be changed by editing your Tecplot 360 configuration file. The configuration file, `tecplot.cfg`, is installed in your Tecplot 360 home directory.

Table 15. Soft limits for Tecplot 360

Limit	Soft Limit	Hard Limit	Macro Parameter
Points per line[15]	20000	Quintillions	MAXPTSINALINE
Contour levels	150	5,000	MAXNUMCONTOUR
Characters per text label	16,000	100,000	MAXCHRSINTEXTLABELS
Picked objects	1500	More than 2 billion	MAXNUMPICKOBJECTS
Load on demand - min threshold	0.3	0[16]	LODTHRESHOLDMINFRACT
Load on demand - max threshold	0.7	1	LODTHRESHOLDFRACT
Available processors	0[17]	64	MAXAVAILABLEPROCESSORS

Tecplot 360 User’s Manual is for use with Tecplot 360 2025 R1.

Tecplot, Inc.
Post Office Box 52708
Bellevue, WA 98015-2708 U.S.A.

Tel:1.800.763.7005 (within the U.S. or Canada), 00 1 (425) 653-1200 (internationally)

E-mail: sales@tecplot.com, support@tecplot.com
Questions, comments or concerns regarding this document: support@tecplot.com

For more information, visit www.tecplot.com

Part Number: 22-360-01-2 Build Revision {CI_PIPELINE_ID}

Released: 06/2025

Third-Party Trademarks

Trademarks not owned by Tecplot, Inc. are used in this software and documentation to indicate compatibility with, or to describe use of Tecplot products with, particular products not owned by Tecplot, Inc. No endorsement by any third party of any Tecplot product (or vice versa) is intended.

3D Systems is a registered trademark or trademark of 3D Systems Corporation in the U.S. and/or other countries.

Mac, Macintosh, Mac OS, and Mac OS X are registered trademarks or trademarks of Apple, Incorporated in the U.S. and/or other countries.

Reflection-X is a registered trademark or trademark of Attachmate Corporation in the U.S. and/or other countries.

EnSight is a registered trademark or trademark of Computation Engineering Internation (CEI), Incorporated in the U.S. and/or other countries.

EDEM is a registered trademark or trademark of DEM Solutions Ltd in the U.S. and/or other countries.

Exceed, Exceed 3D, and Hummingbird, are registered trademarks or trademarks of OpenText Corporation in the U.S. and/or other countries.

Konqueror is a registered trademark or trademark of KDE e.V. in the U.S. and/or other countries.

VIP and VDB are registered trademarks or trademarks of Halliburton in the U.S. and/or other countries.

ECLIPSE FrontSim is a registered trademark or trademark of Schlumberger Information Solutions (SIS) in the U.S. and/or other countries.

Debian is a registered trademark or trademark of Software in the Public Interest, Incorporated in the U.S. and/or other countries.

X3D is a registered trademark or trademark of Web3D Consortium in the U.S. and/or other countries.

X Window System is a registered trademark or trademark of X Consortium, Incorporated in the U.S. and/or other countries.

ANSYS, Fluent and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS Incorporated or its subsidiaries in the U.S. and/or other countries.

LS-DYNA is a registered trademark or trademark of Livermore Software Technology Corporation in the U.S. and/or other countries.

MSC/NASTRAN is a registered trademark or trademark of MSC.Software Corporation in the U.S. and/or other countries.

NASTRAN is a registered trademark or trademark of National Aeronautics Space Administration in the U.S. and/or other countries.

3DSL is a registered trademark or trademark of StreamSim Technologies, Incorporated in the U.S. and/or other countries.

SDRC/IDEAS Universal is a registered trademark or trademark of UGS PLM Solutions Incorporated or its subsidiaries in the U.S. and/or other countries.

Star-CCM+ is a registered trademark or trademark of CD-adapco in the U.S. and/or other countries.

Reprise License Manager is a registered trademark or trademark of Reprise Software, Inc. in the U.S. and/or other countries.

Python is a registered trademark or trademark of Python Software Foundation in the U.S. and/or other countries.

Abaqus, the 3DS logo, SIMULIA and CATIA are registered trademarks or trademarks of Dassault Systèmes or its subsidiaries in the U.S. and/or other countries. The Abaqus runtime libraries are a product of Dassault Systèmes Simulia Corp., Providence, RI, USA. © Dassault Systèmes, 2023

FLOW-3D is a registered trademark or trademark of Flow Science, Incorporated in the U.S. and/or other countries.

Adobe, Flash, Flash Player, Premier, and PostScript are registered trademarks or trademarks of Adobe Systems, Incorporated in the U.S. and/or other countries.

AutoCAD and DXF are registered trademarks or trademarks of Autodesk, Incorporated in the U.S. and/or other countries.

Ubuntu is a registered trademark or trademark of Canonical Limited in the U.S. and/or other countries.

Helvetica Font Family and Times Font Family are registered trademarks or trademarks of Linotype GmbH in the U.S. and/or other countries.

Linux is a registered trademark or trademark of Linus Torvalds in the U.S. and/or other countries.

ActiveX, Excel, Microsoft, Visual C++, Visual Studio, Windows, Windows Metafile, Windows XP, Windows Vista, Windows 2000 and PowerPoint are registered trademarks or trademarks of Microsoft Corporation in the U.S. and/or other countries.

Firefox is a registered trademark or trademark of The Mozilla Foundation in the U.S. and/or other countries.

and/or other materials provided with the distribution. The name of the contributors may not be used to endorse or promote products derived from this software without specific prior written permission. THIS SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE COPYRIGHT HOLDER OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

libssh2 Copyright © 2004-2007 Sara Golemon <sarag@libssh2.org→. Copyright © 2005,2006 Mikhail Gusarov <dottedmag@dottedmag.net→. Copyright © 2006-2007 The Written Word, Inc. Copyright © 2007 Eli Fant <elifantu@mail.ru→. Copyright © 2009-2014 Daniel Stenberg. Copyright © 2008, 2009 Simon Josefsson. All rights reserved. Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met: Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer. Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution. Neither the name of the copyright holder nor the names of any other contributors may be used to endorse or promote products derived from this software without specific prior written permission. THIS SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE COPYRIGHT OWNER OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

NetCDF Library Copyright 1993-2017 University Corporation for Atmospheric Research/Unidata

Portions of this software were developed by the Unidata Program at the University Corporation for Atmospheric Research. Access and use of this software shall impose the following obligations and understandings on the user. The user is granted the right, without any fee or cost, to use, copy, modify, alter, enhance and distribute this software, and any derivative works thereof, and its supporting documentation for any purpose whatsoever, provided that this entire notice appears in all copies of the software, derivative works and supporting documentation. Further, UCAR requests that the user credit UCAR/Unidata in any publications that result from the use of this software or in any product that includes this software, although this is not an obligation. The names UCAR and/or Unidata, however, may not be used in any advertising or publicity to endorse or promote any products or commercial entity unless specific written permission is obtained from UCAR/Unidata. The user also understands that UCAR/Unidata is not obligated to provide the user with any support, consulting, training or assistance of any kind with regard to the use, operation and performance of this software nor to provide the user with any updates, revisions, new versions or "bug fixes." THIS SOFTWARE IS PROVIDED BY UCAR/UNIDATA "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL UCAR/UNIDATA BE LIABLE FOR ANY SPECIAL, INDIRECT OR CONSEQUENTIAL DAMAGES OR ANY DAMAGES WHATSOEVER RESULTING FROM LOSS OF USE, DATA OR PROFITS, WHETHER IN AN ACTION OF CONTRACT, NEGLIGENCE OR OTHER TORTIOUS ACTION, ARISING OUT OF OR IN CONNECTION WITH THE ACCESS, USE OR PERFORMANCE OF THIS SOFTWARE.

Portions of this software are copyright © 2023 The FreeType Project (www.freetype.org). All rights reserved.

FTGL - OpenGL font library
Copyright © 2001-2004 Henry Maddocks <ftgl@opengl.geek.nz>
Copyright © 2008 Ãric Beets <ericbeets@free.fr>
Copyright © 2008 Sam Hocevar <sam@zoy.org>

Permission is hereby granted, free of charge, to any person obtaining a copy of this software and associated documentation files (the "Software"), to deal in the Software without restriction, including without limitation the rights to use, copy, modify, merge, publish, distribute, sublicense, and/or sell copies of the Software, and to permit persons to whom the Software is furnished to do so, subject to the following conditions:

The above copyright notice and this permission notice shall be included in all copies or substantial portions of the Software.

THE SOFTWARE IS PROVIDED "AS IS", WITHOUT WARRANTY OF ANY KIND, EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO THE WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE AND NONINFRINGEMENT. IN NO EVENT SHALL THE AUTHORS OR COPYRIGHT HOLDERS BE LIABLE FOR ANY CLAIM, DAMAGES OR OTHER LIABILITY, WHETHER IN AN ACTION OF CONTRACT, TORT OR OTHERWISE, ARISING FROM, OUT OF OR IN CONNECTION WITH THE SOFTWARE OR THE USE OR OTHER DEALINGS IN THE SOFTWARE.

ZeroMQ Copyright © 2007-2021 ZeroMQ Project

OpenSSL library version 1.1.1q 5 Jul 2022
Copyright © 1998-2022 The OpenSSL Project
Copyright © 1995-1998 Eric A. Young, Tim J. Hudson

OpenGL Mathematics (GLM) Copyright © 2005 - G-Truc Creation. Refer to pub/glm/copying.txt for license information.

FFTW Copyright Notice. The software [or "Portions of the software"] incorporated herein is Copyright ©2011, Massachusetts Institute of Technology ("MIT"). All Rights Reserved. The name "MIT" (alone or as part of another name) or any logos, seals, insignia or other words, names, symbols or devices that identify MIT or any MIT school, unit, division or affiliate may not be used to endorse or promote products derived from this software without specific prior written permission.

Asciidoctor
Copyright © 2012-present Dan Allen, Sarah White, Ryan Waldron, and the individual contributors to Asciidoctor.
Refer to pub/asciidoctor/license.txt for license information

Lunr
Copyright © 2013 by Oliver Nightingale
Refer to pub/lunr/license.txt for license information

[1] The HDF Loader uses the public-domain HDF API code library from the National Center for Supercomputing Applications (NCSA),

University of Illinois, Urbana-Champaign.

[2] Auxiliary data assigned to both zones and the dataset assign the value from the last zone processed to the dataset.

[3] Overflow specific constants.

[4] Copyright for Third Party Library. This loader utilizes a modified version of a library written by Greg Turk while at Stanford University. The copyright for this library is: Copyright © 1994 The Board of Trustees of The Leland Stanford Junior University. All rights reserved. Permission to use, copy, modify and distribute this software and its documentation for any purpose is hereby granted without fee, provided that the above copyright notice and this permission notice appear in all copies of this software and that you do not sell the software. The software is provided "as is" and without warranty of any kind, express, implied or otherwise, including without limitation, any warranty of merchantability or fitness for a particular purpose.

[5] For details, see Lancaster, Peter and Salkauskas, Kestutis "Curve and Surface Fitting, An Introductin", 1986, Academic Press.

[6] For more information see Russell W. Stineman's "A Consistent Well-behaved Method of Interpolation" in the July, 1980, issue of Creative Computing.

[7] With orthographic projection, the shape of the objects is independent of distance. This is sometimes an "unrealistic" view, but it is often used when preserving true dimension is important (such as in drafting).

[8] Refer to the Data Format Guide for details regarding face neighbors.

[9] A represents the theta (or angle) axis variable in Polar Line plots.

[10] Derived Objects can opt in or out of blanking. (See Blanking Settings for Derived Objects.)

[11] If you have multiple x-axes or y-axes in an XY line plot, the variables assigned to the x and y-axis in the first available mapping will be used.

[12] Use the `$!INTERFACE DATA {DERIVATIVEBOUNDARY = method}` setting the configuration file tecplot.cfg to select the method to use. Set *method* to `SIMPLE` or `COMPLEX`.

[13] Mac users can simulate a click of the middle button by holding down the Control key on the keyboard while clicking the right mouse button. Hold any additional keys (such as Alt/Option) along with Control to invoke extended mouse operations.

[14] A polyline is a continuous series of line segments, and can be a subset of a line geometry.

[15] 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.

[16] LODTHRESHOLDMINFRACT may not be less than 0 (the given hard limit is the lower bound, not the upper limit as with the other hard limits).

[17] If MAXAVAILABLEPROCESSORS is 0, Tecplot 360 queries the hardware for the number of processors and uses that value.