Getting Started Manual

Release 1

Tecplot, Inc. Bellevue, WA 2013

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)
email: sales@tecplot.com, support@tecplot.com
Questions, comments or concerns regarding this document: documentation@tecplot.com
For more information, visit http://www.tecplot.com

13-360-02-1

Rev 1/2013

Table of Contents

1

Introduction

Tecplot 360 allows you to interactively explore, visualize, and analyze your data, and then communicate your results through high-quality plots for reports, papers, presentations, or websites.

The user documentation for Tecplot 360 is divided into these nine books:

- Getting Started Manual (this document) - New Tecplot 360 users are encouraged to work through the tutorials provided in the Getting Started Manual. The tutorials highlight how to work with key features in Tecplot 360.
- User's Manual - This manual provides a complete description of working with Tecplot 360 features.
- Scripting Guide - This guide provides Macro and Python command syntax and information on working with Macro and Python files and commands.
- Quick Reference Guide - This guide provides syntax for zone header files, macro variables, keyboard shortcuts, and more.
- Data Format Guide - This guide provides information on outputting simulator data to Tecplot 360 file format.
- Add-on Developer's Kit - User's Manual - This manual provides instructions and examples for creating add-ons for Tecplot 360.
- Add-on Developer's Kit - Reference Manual - This manual provides the syntax for the functions included in the add-on kit.
- Installation Instructions - These instructions give a detailed description of how to install Tecplot 360 on your machine.
- Release Notes - These notes provide information about new and/or updated Tecplot 360 features.
- Tecplot Talk - A user-supported forum discussing Tecplot 360, Tecplot Focus, Python scripting, Add-on development, TecIO and more. Visit www.tecplottalk.com for details.

This manual includes an overview of Tecplot 360, as well as three tutorials. Each tutorial takes approximately 20-40 minutes to complete.

- External Flow Tutorial - This tutorial includes calculating a pressure coefficient, using a contour layer, extracting a slice, and plotting data from multiple files in one frame.

- Internal Flow Tutorial - This tutorial includes value blanking and streamtrace animation.
- Fluid Structure Interaction Tutorial - This tutorial includes contour groups, manipulation of multiple data files, and transient animation.

For in-depth information on any of the topics covered in the Getting Started Manual, please refer to the User's Manual.

2

Overview

Tecplot 360 features allow you to visualize complex numerical and computational fluid dynamics and vital engineering plot relationships by:

- Animating XY, Polar, 2D, and 3D plots in one unified environment
- Customizing the independent axes for specialty plots (for example, Lift vs. Angle of Attack or Cp vs. Cord)
- Using the interactive slice, iso-surface, and streamtrace tools
- Detecting vortex cores, shock surfaces, and separation and reattachment lines
- Calculating critical flow functions and performing integrations
- Integrating particle trajectories with support for massed particles and drag effects
- Validating numerical models with test data in the same window
- Comparing multiple models simultaneously, including visualization of fluid structure interaction
- Supporting 30 CFD, FEA, structural analysis, and industry-standard data formats

After creating your plots, you can communicate your results clearly and effectively by:

- Using the "Copy Plot to Clipboard" command to paste a plot directly into Microsoft® PowerPoint®, Microsoft Word®, and other Microsoft Office® applications
- Animating to a file for use in a PowerPoint presentation, a webpage, or a Framer (AVI, Flash®, or Raster Metafile)
- Using the **Publish** command to share results directly on the Web
- Exporting presentation-quality vector and raster formats (JPEG, PNG, TIFF, BMP, WMF, Adobe® PostScript®, or EPS)

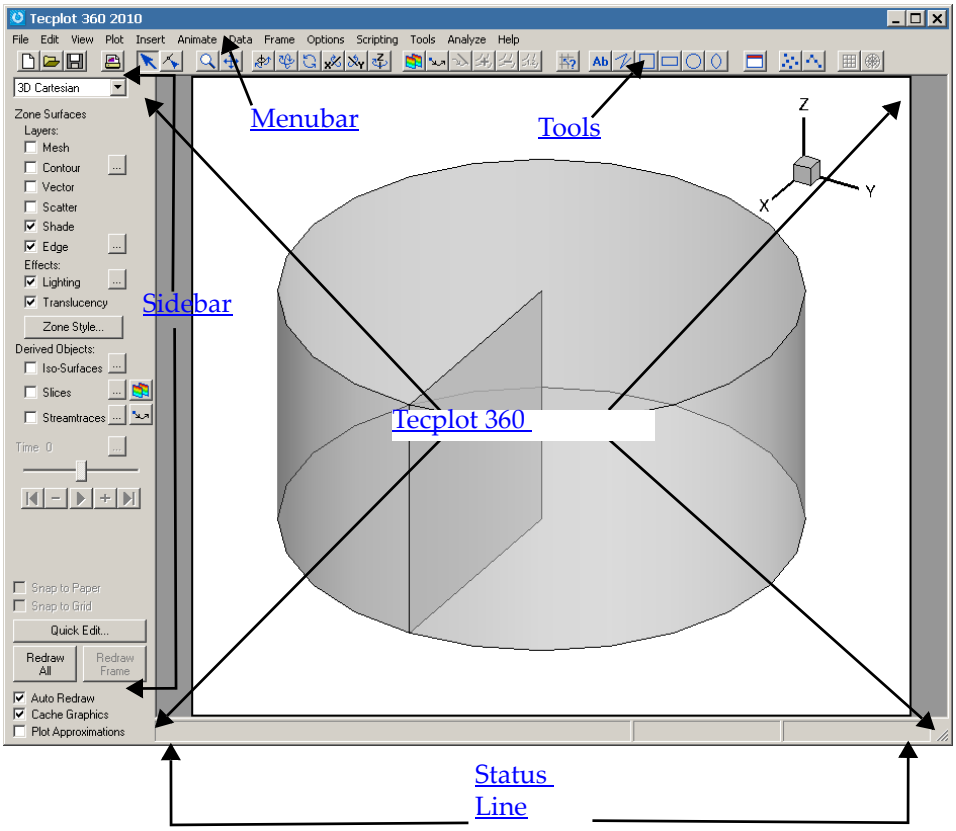
You may also save time and effort by automating routine analyses and plotting operations through these methods:

- Customizing the interface to your workflow
- Creating macros by recording or writing scripts
- Using the **Quick Macro Panel** dialog for one-click macro access
- Batch process plotting and printing

- Extending Tecplot 360 functionality with the Add-on Developer's Kit

2 - 1 Interface

Five major sections make up the Tecplot 360 interface:



2 - 2 Menubar

The menubar offers rapid access to most of Tecplot 360's 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, print and export plots, and set configuration preferences.
- **Edit** - Use the **Edit** menu to select, undo, cut, copy, paste, and clear objects, open the **Quick Edit** dialog, and change the draw order for selected items (push or pop).

Cut, **Copy**, and **Paste** work only within Tecplot 360. To place a graphic image of your layout into another program, use **Copy Plot to Clipboard**. This option is available on Windows® and Macintosh® platforms.

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

  The **View** menu includes the following convenient sizing options:

- **Fit Everything (3D Only)** - This options resizes plots so that all data points, text, and geometries are included in the frame.
- **Fit Surfaces (3D Only)** - This option resizes plots so that all surfaces are included in the frame, excluding any volume zones.
- **Fit to Full Size** - This option fits the entire plot into the frame. This option does not affect the axis ranges.
- **Nice Fit to Full Size** - This option sets the axis range to begin and end on major axis increments (if axes are dependent, the vertical axis length is adjusted to accommodate a major tick mark).
- **Data Fit** - This option fits the data points to the frame.
- **Make Current View Nice** - This option modifies the range on a specified axis to fit the minimum and maximum of the variable assigned to that axis, and then snaps the major tick marks to the ends of the axis. (If axis dependency is not set as independent, this may affect the range on another axis.)
- **Center** - This option moves the plot image so that the data points are centered within the frame. (Only the data is centered; text, geometries, and the 3D axes are not considered.)

- **Plot -** Use the **Plot** menu to control the style of your plots. The menu items available are dependent upon the active plot type (chosen in the Sidebar).
- **Insert** - Use the **Insert** menu to add text, geometries (polylines, squares, rectangles, circles, and ellipses), or image files. 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 animate IJK Planes, IJK Blanking, iso-surfaces, mappings, slices, streamtraces, time, and zones.
- **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 the attributes of your workspace, including the color map, paper grid, display options, and rulers.
- **Scripting** - Use the **Scripting** menu to play or record macros, and to access the **Quick Macros Panel** dialog.
- **Tools** - Use the **Tools** menu to launch the **Quick Edit** dialog or 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** - Choose “Tecplot 360 Help” from the **Help** menu to get specific, complete help on features or operations within Tecplot 360. By choosing “About Tecplot 360” from this menu, you can obtain specific information about your license.

2 - 2.1 Sidebar

The Sidebar provides easy access for frequently used plot controls. The functions available in the Sidebar depend on the plot type of the active frame. For 2D or 3D Cartesian plot types, you can add or subtract zone layers, zone effects (3D only), and derived objects from your plot using the Sidebar. For line plots (XY and Polar), you can add or subtract mapping layers using the Sidebar.

To customize your plot, simply:

- Choose the desired type of plot from the Plot Types menu.
- Use the toggle switches to add and subtract Zone Layers/Map Layers, Zone Effects, or Derived Objects. Use the **Zone Style/Mapping Style** dialogs to further customize your plot by adding

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

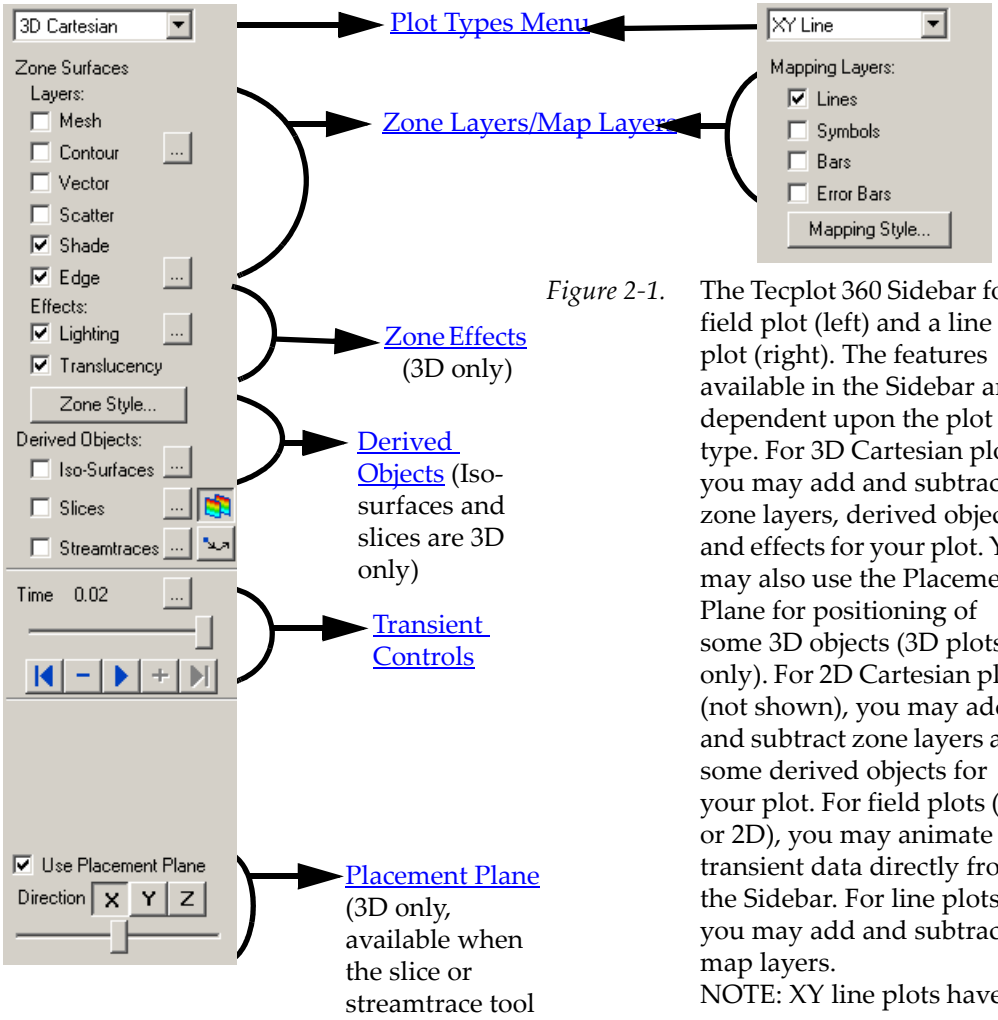


Figure 2-1. The Tecplot 360 Sidebar for a field plot (left) and a line plot (right). The features available in the Sidebar are dependent upon the plot type. For 3D Cartesian plots, you may add and subtract zone layers, derived objects, and effects for your plot. You may also use the Placement Plane for positioning of some 3D objects (3D plots only). For 2D Cartesian plots (not shown), you may add and subtract zone layers and some derived objects for your plot. For field plots (3D or 2D), you may animate transient data directly from the Sidebar. For line plots, you may add and subtract map layers.
NOTE: XY line plots have

Plot Types

The plot type, combined with a frame's dataset, active layers, and each layer's 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 axes 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 with the horizontal axis **(x)** as the independent variable and the vertical axis (**y = f(x)**) as a dependent variable
- **Polar Line** - Line plots of independent and dependent variables on a polar grid
- **Sketch** - Plots without data, such as drawings, flow charts, and viewgraphs

Zone Layers/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 and plotted in the layers. There are six zone layers for 2D and 3D Cartesian, four map layers for XY Line, two map layers for Polar Line, and no layers for Sketch.

The following six zone layers are options for 2D and 3D Cartesian plot types:

- **Mesh** - Lines connecting the data points within each zone
- **Contour** - Lines having a constant value, the region between these lines (contour flooding), or both
- **Vector** - Arrows indicating the direction and magnitude of physical quantities
- **Scatter** - Symbols at the location of each data point
- **Shade** - The effect used to tint each zone with a specified solid color, or to add light-source shading to a 3D surface plot. When using this effect in conjunction with the Lighting zone effect, you may set Paneled or Gouraud shading. When using the shade effect in conjunction with the Translucency zone effect, you may create a translucent surface for your plot.
- **Edge** - Zone edges and creases for ordered data and creases for finite element data

The four XY Line map layers are:

- **Lines** - Lines that plot 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) that are individual data points and are represented by a symbol that you specify
- **Bars** - A pair of variables (X and Y) as a horizontal or vertical bar chart
- **Error Bars** - An option that allows you to add error bars to your plot

The two Polar Line map layers are:

- **Lines** - A pair of variables (R and Theta) that are a set of line segments or a fitted curve
- **Symbols** - A pair of variables (R and Theta) that are individual data points represented by a symbol that you specify

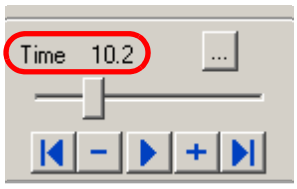
Zone Effects

For 3D Cartesian plot types, use the Sidebar to turn Lighting and Translucency on or off. Only shaded and flooded contour surface plot types are affected.

Derived Objects

For Cartesian plot types (2D and 3D), toggle-on "Iso-Surfaces", "Slices", or "Streamtraces" from the Sidebar to explore these elements. Their corresponding **Details** dialogs can be accessed with the Details button in the Sidebar. (In 2D Cartesian plots, only streamtraces are available.)

Transient Controls



When working with transient data, simply click the Play  button in the Sidebar to animate the data over time. The active frame will be animated from the Current Solution Time (left, circled in red) to the last time step. You may also drag the slider left or right to change the Current Solution Time of your plot.

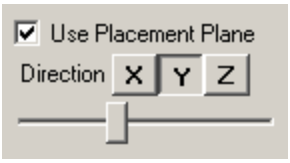
The Transient Controls have the following functions:

-  - This button jumps to the Starting Value.
-  - This button jumps toward the Starting Value by one step.
-  - This button runs animation as specified by the Operation field of the **Time Details** dialog. This Play button becomes a Pause button while the animation is playing.
-  - This button jumps toward the Ending Value by one step.
-  - This button jumps to the Ending Value.

Use the Details button in the Transient Controls region of the Sidebar to launch the **Time Details** dialog.

See Section 7 - 2 "Time Aware" in the User's Manual for more information on time controls and the **Time Details** dialog.

Placement Plane



When using certain tools to add objects to your plot, toggle-on "Use Placement Plane" in the Sidebar to place the objects along a given plane (3D Plots only). Use the X, Y, and Z buttons to choose the plane to use, and use the slider to reposition the Placement Plane. The Placement Plane will appear as a gray slice in your plot.

The Placement Plane is available for:

- Placing streamtraces (using the Add Streamtrace tool )
- Placing slices (using the Slice tool )
- Adding Contour Levels (using the Add Contour Level tool )
- Deleting Contour Levels (using the Remove Contour Level tool )
- Probing (using the Probing tool )

Snap Modes

Snap to Grid - Constrains object movement to whole steps on the axis grid. This can be useful for aligning text and geometries with specific plot features.

Snap to Paper - Constrains object movement to whole steps on the paper's ruler grid. This can be useful for positioning frames precisely for printing, or for absolute positioning of text, geometries, and other plot elements.

Redraw Buttons

The redraw buttons allow you to keep your plot up to date: Redraw All-CTRL-D redraws all frames, SHIFT-Redraw All causes Tecplot 360 to completely regenerate the workspace, and Redraw-CTRL-R redraws only the current frame.

Auto Redraw

Use Auto Redraw - When toggled-on, Tecplot 360 will automatically redraw the plot whenever style or data changes. Some users prefer to turn this option off while setting multiple style settings and then manually clicking Tecplot 360's Redraw or Redraw All button on the Sidebar to see a full plot.

An auto redraw can be interrupted at any time with a mouse click or key press.

Cache Graphics

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 than having Tecplot 360 send the instructions again. This is particularly true for the interactive manipulation of a plot. However, this performance potential comes at the cost of using more memory. If the memory need is too high, the overall performance could be less. Tecplot 360 has three graphics cache modes: cache all graphics, cache only lightweight graphics objects, and do not cache graphics.

When “Cache Graphics” is toggled-on in the Sidebar, Tecplot 360 assumes there is enough memory to generate the graphics cache. Assuming this is true, Tecplot 360's rendering performance will be optimal for the interactive manipulation of a plot.

When memory constraints are very limited, consider toggling-off “Cache Graphics”. If you intend to interact with the plot under limited memory constraints, also consider setting the “Plot Approximation” mode to “All Frames Always Approximated”.

See Section “Graphics Cache” in the User’s Manual for more information.

Plot Approximations

If “Plot Approximation” is toggled-on and the number of data points is above the point 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.

Tools

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

Double-click on a tool to launch the **Details** dialog associated with that tool.

Selector Tool

Use the Selector tool to select objects in your workspace. The objects can be modified using the **Quick Edit** dialog when the Quick Edit button is chosen in the Sidebar before an object is selected.

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

- 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, either double-click on the desired object or drag the cursor to select a group of objects to call up the **Group Select** dialog. Click OK, and then click the **Object Details** button in the Sidebar.

Adjustor Tool

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

- Change the location of individual or groups of data points in the grid
- Modify the values of the dataset variables at a particular point
- Change the length or placement of individual axes (2D Cartesian and XY Line plot types only)
- Change the spacing between an axis label and its associated axis (2D Cartesian and XY line plot types only)
- Change the shape of a polyline

Except for the above actions, the behavior of the Adjustor tool is identical to that of the Selector tool.

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

To select multiple points, you can either SHIFT-click after selecting your initial point to select additional points, or you can draw a group select band to select the points within the band. (In Line plots, you can select points from only one mapping at a time.)

Once you have selected all desired points, move the Adjustor over the selection handles of one of the points, then drag to the desired location of the chosen data point. Other selected points will move as a unit with the chosen data point, maintaining their relative positions.

For XY Line plots: If several mappings are using the same data for one of the variables, adjusting one of the mappings will result in simultaneous adjustments to the others. You can avoid this by pressing the H or V keys on your keyboard while adjusting the selected point. The H and V keys restrict the adjustment to the horizontal and vertical directions, respectively.

Group Select

The **Group Select** dialog is opened when you select a group of objects with the Selector or Adjustor tools. Drag to create a rectangle around the objects you want to select. Use the **Group Select** dialog to specify which types of objects within the specified selection region should be selected.

The **Group Select** dialog allows you to specify the following object types to be selected (if the selection rectangle does not include a specific object, its associated check box is inactive):

- Text
- Geometries
- Frames
- Zones
- Axis Grid Area
- Contour Labels
- Streamtraces

The **Group Select** dialog offers the following attribute filters:

- **Geoms of Type** - Choose geometries of a particular type from the menu.
- **Geoms with Line Pattern** - Choose all geometries having a particular line pattern.
- **Text with Font** - Choose all text displayed in a particular font.
- **Objects with Color** - Choose all objects of a particular color. (Choose the appropriate color from the **Select Color** dialog.)

Zoom Tool 

This tool enables zooming into or away from a plot.

With the zoom tool selected as the mouse mode, 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 about the region that you want to fit into the frame. The box may be larger than the frame. Drawing a 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** is toggled-on in the Sidebar, you cannot make the zoom box larger than the grid area.

To return to the previous view, choose “Last” from the **View** menu (CTRL-L). To restore the original 2D view, choose “Fit to Full Size” (CTRL-F) from the **View** menu.

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** - The 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 the 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 such that the longest dimension of the zoom box fits into the workspace. You can fit one or all frames to the workspace by using the "Fit Selected Frames to Workspace" or "Fit All Frames to Workspace" options from the **View>Workspace** menu. To return to the default paper view, choose "Fit Paper to Workspace" from the **View>Workspace** menu.

Use the center mouse button to zoom smoothly into or out of the plot.

Clicking anywhere in your plot while the zoom tool is active zooms in on the plot and centers that zoom around your click.

Translate Tool 

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

While in Translate/Magnify 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.

Use the right mouse button to translate objects within a frame interactively.

Rescale image: You can rescale your image by choosing the translate tool and pressing "+" to magnify or "-" to shrink.

Rescale paper: To rescale the paper, first SHIFT-drag to move the paper, and then use the rescale buttons "+" or "-" to magnify or shrink the paper, respectively.

Three-dimensional Rotation

Tecplot 360 allows you to rotate your data in a variety of ways. Choose one of the six possible 3D rotation mouse modes, then drag the pointer in the workspace to rotate your 3D image. The six rotation mouse modes can be engaged by selecting one of these six tools:

- **Spherical**  - Drag the mouse horizontally to rotate about the Z-axis; drag the mouse vertically to control the tilt of the Z-axis.
- **Rollerball**  - Drag the mouse in the direction you want to move the plot 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 other rotation capabilities or rotation modes 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	Smooth zoom in and out of the data.

Right-click	Translate the data.
O	Move rotation origin to probed point of data. This shortcut can be used without first selecting a rotation mouse mode. Hover over your intended point of origin, type O, and then CTRL-right-click and drag to rotate the image.
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.

Slice Tool 

Use the Slice tool to control your slice rendering 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 add a slice. Primary Slices active [ONLY] - Turn on Start/End Slices and add a slice. Start/End Slices active [ONLY] - Turn on Start/End Slices and add a slice.
-	Primary Slices, Start/End Slices active - Remove start and end slices. Primary Slices active [ONLY] - Remove the primary slice. Start/End Slices active [ONLY] - Remove the Start and End Slices.
Click-drag	Update the position of the primary slice (if active). If only start and end slices are visible, click to update the position of the slice closest to the click.
ALT-click/ALT-drag	Determine the XYZ-location by ignoring zones and looking only at derived volume objects (streamtraces, slices, iso-surfaces).
SHIFT-click	Place the start or end slice (whichever is closest to the initial click location). Show Start/End Slices as activated, if necessary.
SHIFT-drag	Move the start or end slice (whichever is closest to the initial click location). Show Start/End Slices as 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 

Choose 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.

Refer to the Chapter 15: "Streamtraces" in the User's Manual for more information about using streamtraces.

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 added when placing a rake of streamtraces

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

Streamtrace Termination Line 

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

To draw a Streamtrace Termination Line:

1. Move the cursor into the data region.
2. Click once at the desired starting point for the line.
3. Click again at each desired break point.
4. When the polyline is complete, double-click on the last point of the polyline, press the ESC key on your keyboard, or right-click.

The polyline will end any streamtrace(s) that pass through it.

Add Contour Level 

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

You can use the following keyboard and mouse shortcuts when the Add Contour Level tool is selected:

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

Delete Contour Level

Choose the **Delete Contour Level** tool to delete a contour level interactively by clicking anywhere in the current 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

Choose the Add Contour Label tool to switch to the Contour Label mode, which enables you to add a contour label interactively (by clicking anywhere in the current 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 slightly offset from the lines that they label.

The Contour Type must be "Lines" or "Both Lines & 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

Choose the Probe 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, choose any point in the data region. To obtain exact values for the data point nearest the specified location, CTRL-click at the desired location.

For XY plots - When you move into the axis grid area, the cursor crosshair 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 be probed by pressing X to probe the X-axis or Y to probe the Y-axis.

Insert Text

To add text to any frame, click the Add Text tool and draw a text box in the selected frame. The **Text Details** dialog will be launched automatically. Use it to assign and modify text.

Insert Geometries

Use the corresponding geometry buttons in the toolbar to insert geometries into your plot:

Polylines

Squares

Rectangles

 Circles

 Ellipses

Select a geometry shape for insertion, and then drag in the workspace to create the shape of desired size.

Create New Frame 

Select the Create New Frame tool to create a new frame.

To add a frame, drag in the workspace to create a frame of desired size and shape.

If you have data loaded into Tecplot 360 and you create a new frame, you can attach the existing dataset to the new frame by changing the plot type to match that of the existing dataset.

Extract Discrete Points 

Choose the Extract Discrete Points tool to extract selected points to a data file or a new zone.

To select points:

1. Left-click at each location where you would like to extract a point.
2. To end extraction, either double-click on the last point, press the ESC key, or right-click.
3. The **Extract Data Points** dialog appears; use it to specify how many points to extract and how to save the data.

Extract Points along Polyline 

Choose the Extract Line tool to extract points along a specified polyline to a data file or a new zone.

To select points:

1. Click your left-hand mouse button at each location where you would like to extract a point along a polyline.
2. To end extraction, either double-click on the last point, press the ESC key, or right-click.
3. The **Extract Data Points** dialog appears; use it to specify how many points to extract and how to save the data.

Create Rectangular Zone (2D Only) 

Choose the Create Rectangular Zone tool to add new 2D rectangular zones to the current Tecplot 360 dataset.

To create a rectangular zone:

1. Click once in the current data region to anchor one corner of the zone.
2. Drag the diagonal corner until the zone is the desired size and shape. The new zone created is IJ-ordered.

To specify the maximum I-index and J-index, use the **Create Rectangular Zone** dialog (accessed via **Data>Create Zone>Rectangular**).

The current frame must have a dataset attached to it in order for this tool to be active. (This option is only available in 2D Cartesian plots.)

Create Circular Zone (2D Only) 

Select the Create Circular Zone tool to add new 2D circular zones to the current Tecplot 360 dataset.

To create a circular zone:

1. Click once in the current data region to specify the center of the zone.
2. Drag until the zone has the desired radius. The new zone created is IJ-ordered.

To specify the maximum I-index and J-index, use the **Create Circular Zone** dialog (accessed via **Data>Create Zone>Circular**).

The current frame must have a dataset attached to it in order for this tool to be active. (This option is only available in 2D Cartesian plots.)

2 - 2.2 Status Line

The status line, running along the bottom of the Tecplot 360 window, gives “hover help”. When you move the pointer over a tool in the toolbar, a button on the **Quick Edit** dialog, or a menu item, a description of the control appears. It also provides a progress bar and information during long calculations.

2 - 2.3 Tecplot 360 Workspace

The workspace is the portion of your screen in which you create sketches and plots. Each sketch or plot is created within a subwindow called a frame. The current state of the workspace, including the sizing and positioning of frames, the location of the data files used by each frame, and all current attributes for all frames, makes up a layout. By default, the workspace displays a representation of the paper Tecplot 360 is set up to draw on, as well as a reference grid and rulers. The active frame in which you are currently working is on top. All modifications are made to the active frame.

2 - 3 Data Hierarchy

Tecplot 360 structures data on two levels: datasets and zones. Datasets are contained within frames. Each dataset is composed of a zone or group of zones, and each zone contains a variable or group of variables. All zones in a dataset contain the same set of variables.

2 - 3.1 Frames

You can create multiple plots simultaneously in Tecplot 360 using subwindows called “frames”. By default, one frame is open when you launch Tecplot 360. You can add frames to the workspace using the **Frame** menu. Datasets can be unique to the frame or shared between frames. Linking data between frames allows you to generate unique plots of the same data. For more information on working with frames, please refer to Chapter 2: “Using the Workspace” in the User’s Manual.

2 - 3.2 Datasets

A dataset is defined as "all of the data in a frame". Starting with an empty frame, a dataset is created and assigned to the active frame when you read one or more data files into Tecplot 360, or when you create a zone within Tecplot 360.

2 - 3.3 Zones

Zones are subsets of datasets. A dataset can be composed of a single zone or several zones. Zones are either defined in the data file or created directly in Tecplot 360. The number of zones in a concatenated dataset is the sum of the number of zones in each of the data files that are loaded.

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, and so forth. Alternatively, zones may be defined based on material. For example, a dataset of a fluid tank may have a zone for the tank itself and additional zones for each fluid therein.

2 - 4 Data Structure

Tecplot 360 accommodates two different types of data: ordered and finite element. The data structure is defined within the data file. Each zone is composed of one data type.

2 - 4.1 Ordered Data

Ordered data is a set of points logically stored in a 1D, 2D, or 3D 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.

- **1D array (I-ordered, J-ordered, or K-ordered)** - A 1D array is a one-dimensional array of data points, where one dimension (I, J, or K) is greater than or equal to one, and the other dimensions are equal to one. In a one-dimensional array, the total number of data points is equal to the length of the single-ordered array. For example, an I-ordered dataset with I=5, J=K=1 has five data points.
- **2D array (IJ-ordered, JK-ordered, IK-ordered)** - In a 2D array of data points, two of the three dimensions (I, J, or K) are greater than one, and the other dimension is equal to one. The number of data points in a 2D ordered dataset is the product of the all of the dimensions. For example, in an IJ-ordered dataset, the number of data points is equal to I x J (where K=1).
- **3D array (IJK-ordered)** - In a 3D array of data points, all three of the dimensions (I, J, and K) are greater than one. The number of data points is the product of the I, J, and K dimensions.

2 - 4.2 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.

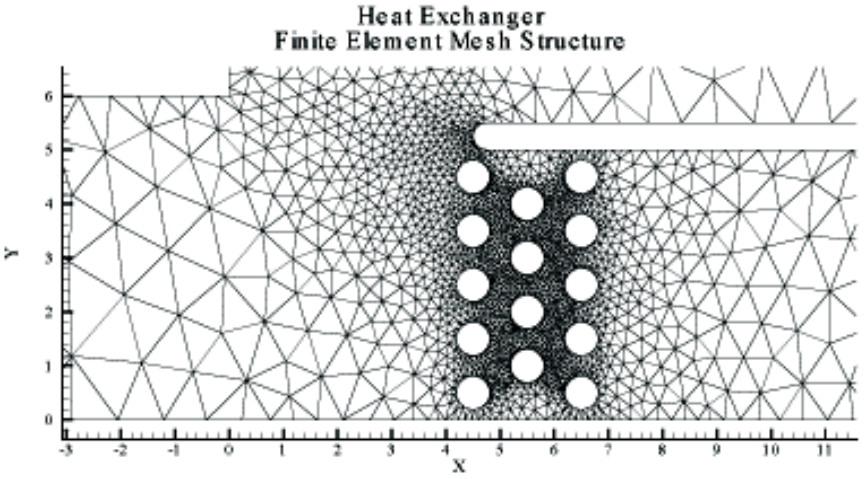
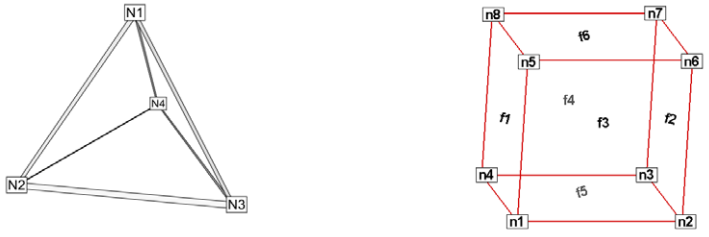


Figure 2-2. This figure shows finite element data used to model a complex boundary. This plot file, *feexchng.plt*, is located in your Tecplot 360 distribution under the *examples/2D* subdirectory.

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 (by 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 variable 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.



Tetrahedral connectivity arrangement *Brick connectivity arrangement*

Figure 2-3. Connectivity arrangements for FE-volume datasets

In the brick format, points may be repeated to achieve 4, 5, 6, or 7 point elements. For example, a connectivity list of "`n1 n1 n1 n1 n5 n6 n7 n8`" (where `n1` is repeated four times) results in a quadrilateral-based pyramid element.

Section 4 - 5 "Finite Element Data" in the Data Format Guide provides detailed information about how to format your FE data in Tecplot's data file format.

2 - 5 Creating Plots

The basic steps for creating a plot in Tecplot 360 are the following:

1. Define your dataset using one of the following methods:
   a. Using the "Load Data File(s)" command from the **File** menu to load any type of data file
   b. Using the "Open Layout" command from the **File** menu to load linked layout or layout package files
   c. Using 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
2. Choose the Plot Type (3D, 2D, XY line, Polar line, or Sketch) from the Sidebar.
3. Toggle-on any mapping or zone layers from the Sidebar (for example, contour zone layer or symbols mapping layer). Use the Details button to customize zone layers.
4. **OPTIONAL (3D only)** - Toggle-on zone effects (translucency and lighting).
5. **OPTIONAL** - Use the **Zone Style** or **Mapping Style** dialogs to opt zones in and out of plot layers or the entire plot.
6. **OPTIONAL (2D or 3D only)** - Add derived objects (slices, streamtraces, or iso-surfaces). Use the Details button to customize any derived objects.

You are not limited to working with only one plot at a time in Tecplot 360. You can create multiple files at once using frames and frame linking.

Once you have loaded your data, you can use the options in the **Plot** menu (such as "Blanking" or "Axis Details") to customize how your data is displayed. You can also use the options in the **Data** menu (such as "Specify Equations" or "Interpolation") to alter the dataset.

2 - 6 Output Formats

Once you have completed your plot(s), you can use any of the following media to distribute or publish your plot(s) outside of Tecplot 360:

- **Printing** - Use the “Print” option from the **File** menu to print your plots.
- **Exporting to an image file** - Use the “Export” option from the **File** menu and select the desired image format in the **Export** dialog.
- **Exporting to an animation file** - Access this export option via any of the **Animation** dialogs by selecting “To File” in the Animate field, or by selecting a movie file format from the **Export** dialog (accessed via the **File** menu).
- **Publishing** - Use the “Publish” option from the **File** menu to save your plots in HTML format.
- **Copying the plot to a clipboard** (Windows and Macintosh operating systems only) - Use the “Copy Plot to Clipboard” option from the **Edit** menu to paste your plot into word processing software.

3

External Flow Tutorial

3 - 1 Introduction

The following tutorial demonstrates how to display pressure on an airplane wing, and how to compare measured and simulated data on a two-dimensional projection plane. We will implement several Tecplot 360 features to create this plot, including the PLOT3D loader, the Calculate Variables option, contour legends, a 2D projected plane, and a line legend.

3 - 1.1 Background Information

In 1972, the ONERA Aerodynamics Department designed a swept, semi-span wing with no twist[1]. The wing was instrumented to be used as an experimental support for basic studies of 3D flows at high Reynolds numbers from low to transonic speeds (that is, local supersonic flow, shocks, and turbulent boundary layer separation).

The simulated dataset is a PLOT3D file with freestream conditions M = 0.8395, Re = 11.72 x 10^6, Angle of Attack = 6 degrees, and angle of side-slip = 0.0 degrees. The measured dataset was obtained in the ONERA S2MA wind tunnel by Schmitt and Charpin at Mach numbers of 0.7, 0.84, 0.88, and 0.92 for angles of attack up to 6 degrees and a Reynolds number of approximately 12 x 10^6. The measured dataset is an ASCII file. The tutorial compares the datasets at M = 0.84 and y/b = 0.44 (where b is the wingspan).

3 - 1.2 Tutorial Summary

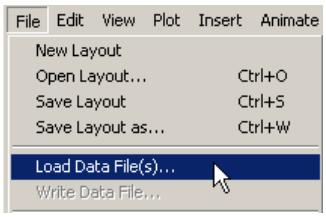
The tutorial should take approximately 30-40 minutes to complete. All supporting data files for this tutorial can be found in: *$TEC_360_2013R1\tutorials\external_flow*[2].

1. Schmitt, V. and F. Charpin, "Pressure Distributions on the ONERA-M6-Wing at Transonic Mach Numbers," Experimental Data Base for Computer Program Assessment. Report of the Fluid Dynamics Panel Working Group 04, AGARD AR 138, May 1979.
2. *$TEC_360_2013R1* is the installation directory for Tecplot 360. For Windows users, this is typically C:\Program Files\Tecplot\Tec360 2013R1.

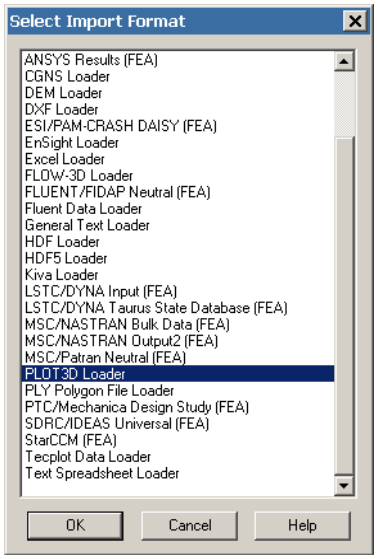
3 - 2 Getting Started

Step 1 Load a Plot3D File (Including a Boundary File)

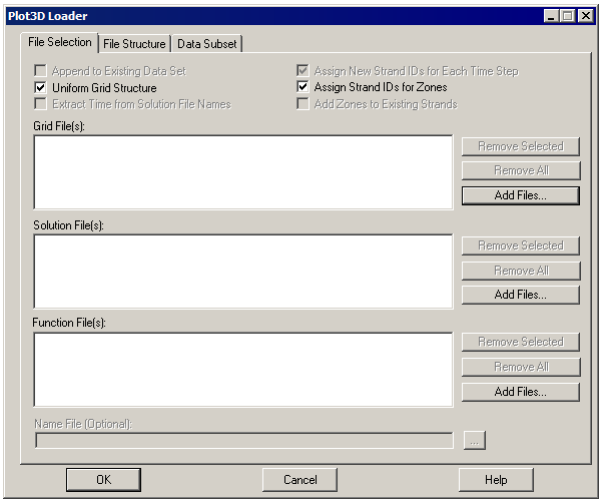
A. Choose “Load Data File(s)” from the **File** menu.



B. Choose “PLOT3D Loader” in the **Select Import Format** dialog.



C. In the **Plot3D Loader** dialog:



i. Click the “Load Grid and Solution/Function File(s)” radio button.

ii. Select the Add Files button to the right of “Grid File(s)”.

D. In the **Read Grid File** dialog, navigate to: *$TEC_360_2013R1\tutorials\external_flow\data* and select *m6wing.xyz*.

E. Click the Add to List button, then the Open Files button.

F. In the **Plot3D Loader** dialog, click the Browse button next to "Solution File(s)".

G. In the **Read Solution File** dialog:

   i. Navigate to: *$TEC_360_2013R1\tutorials\external_flow\data*.

   ii. Select *m6wing.q*.

   iii. Click the Add To List button.

   iv. Click the Open Files button.

H. Click OK after returning to the **Plot3D Loader** dialog.

You have now successfully loaded a PLOT3D data file into the Tecplot 360 workspace.

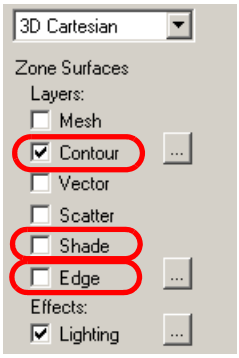
Tecplot 360 will automatically load the boundary file, *m6wing.xyz.fvbnd*, since its filename prefix (*m6wing.xyz*) matches the grid file, and the files are located in the same folder.

Step 2 Manipulate Layers

To quickly view a more pleasing image, remove the edge layer and display the contour layer on the plot.

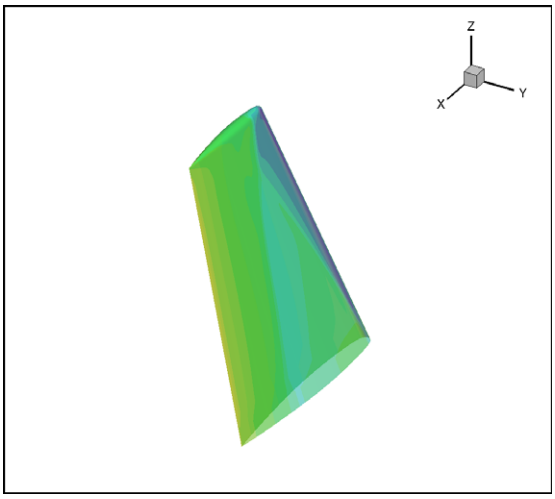
A. Toggle-off the "Edge" layer in the Sidebar. Toggle-off the "Shade" layer.

B. Toggle-on the "Contour" layer.



Any combination of layers can be viewed at the same time by turning them on and off in the Sidebar.

The image will appear as follows:

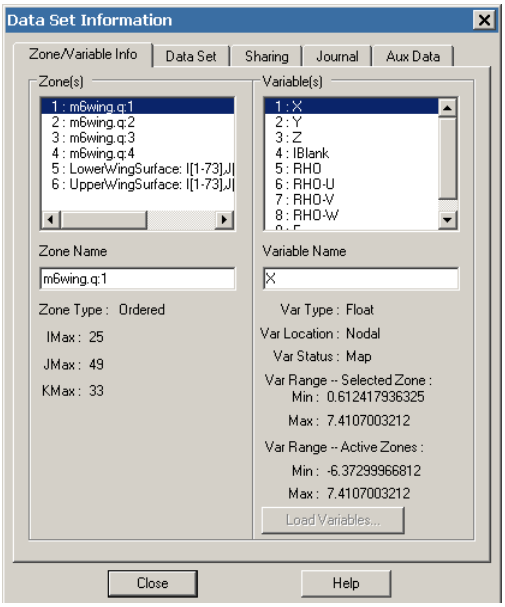


Step 3 Determine the Number of Zones and Variables

In Tecplot 360, the data can be subdivided into smaller regions called "zones". The zones can be defined in the data file itself or manually created by going to **Data>Create Zone**. In this case, the zones were defined within the data file using the grid blocks established by the preprocessor. See also Section 2 - 3.3 "Zones".

A. Go to **Data>Data Set Info** to explore the dialog.

In this dataset, there are six zones, each of which contain nine variables.



B. Close the **Data Set Information** dialog.

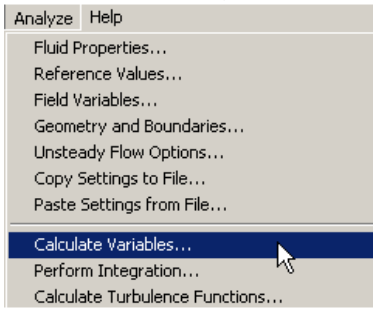
3 - 3 Displaying a Contour Map of the Pressure Coefficient

In order to have a more useful visualization of the wing, we would like to use the Contour layer (which we toggled-on in Step 2) to view the pressure on the wing. In this case, the pressure data was not provided in the data file. However, Tecplot 360 can calculate the Pressure Coefficient for us using the helpful Calculate Variables feature!

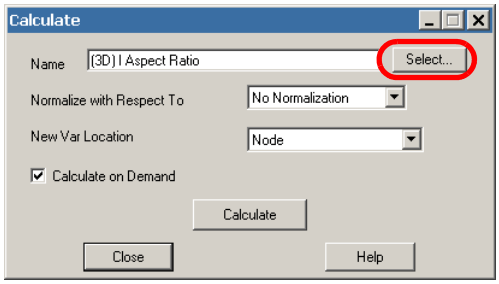
The following steps teach the calculation of the Pressure Coefficient using Tecplot 360's Calculate on Demand method. This method adds the variable to the dataset, but does not calculate the variable until it is needed, which can optimize operation speed when manipulating unsteady solutions. For more information about calculating on demand, see Section 22 - 6.2 "Calculate-on-demand Variables" in the User's Manual.

Step 4 Calculate the Pressure Coefficient

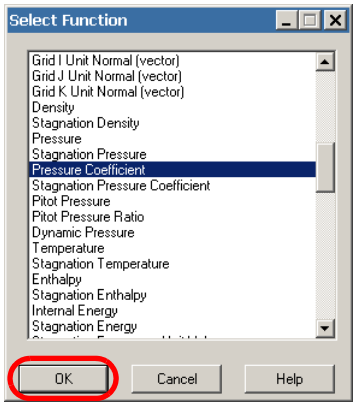
A. Choose "Calculate Variables" from the **Analyze** menu.



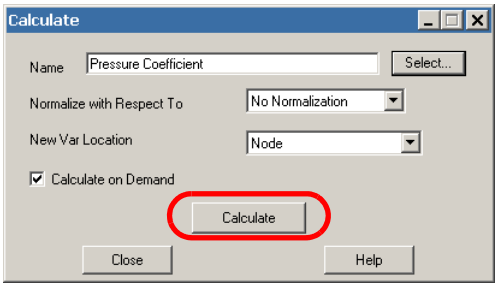
B. In the **Calculate** dialog, click the Select button to choose the variable to calculate.



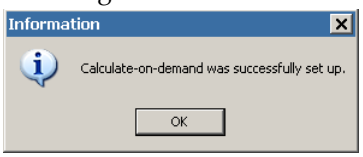
C. In the **Select Function** dialog, highlight "Pressure Coefficient" and click OK.



D. Click the Calculate button.



E. Click OK in the **Information** dialog.

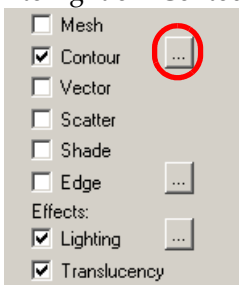


F. Close the **Calculate** dialog.

While calculating variables, you may use any units. However, you must be consistent within your variable set. For information on specifying variables' units, see Section 22 - 6 "Calculating Variables" in the User's Manual.

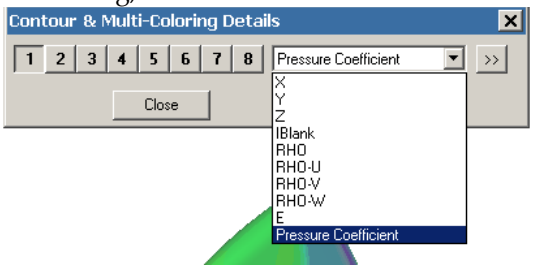
G. Open the **Contour Details** dialog by either of the following methods.

i. Clicking the Details button to right of "Contour" in the Sidebar



ii. Choosing "Contour/Multi-Coloring" in the **Plot** menu

H. In the **Contour Details** dialog, select the "Pressure Coefficient" as the contour variable.



3 - 4 Modifying the Contour Layer

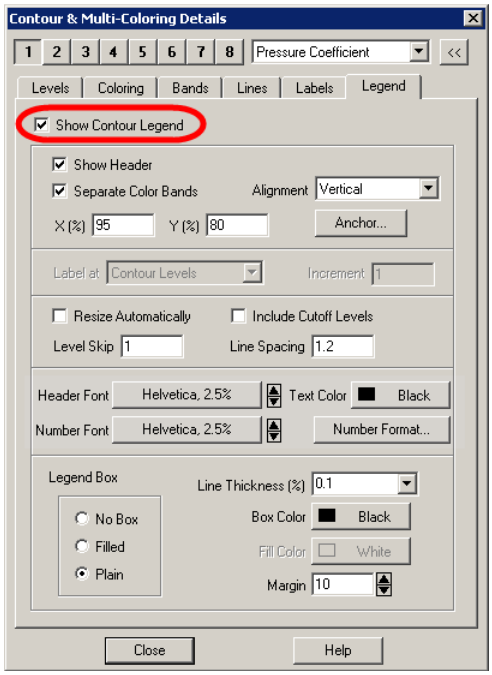
Using the **Contour and Multi-Coloring Details** dialog (which we will refer to as the **Contour Details** dialog), you can customize contour levels, coloring, and more.

Step 5 Add a Contour Legend

A. Expand the **Contour Details** dialog by clicking the >> button.

B. Switch to the Legend page of the dialog.

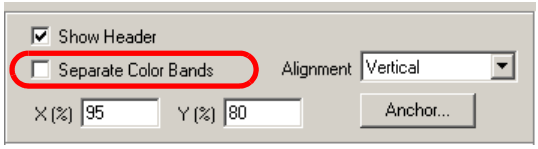
C. Toggle-on "Show Contour Legend".



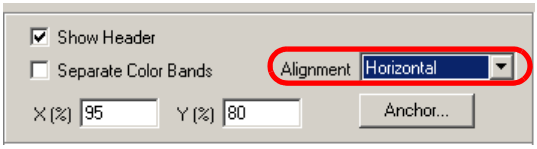
Step 6 Modify the Contour Legend

A. With the Legend page of the **Contour Details** dialog still open, go to the Alignment region of the page and perform the following steps:

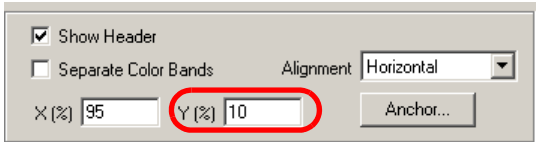
i. Toggle-off "Separate Color Bands" to remove the level lines from the legend.



ii. Change the Alignment to "Horizontal".

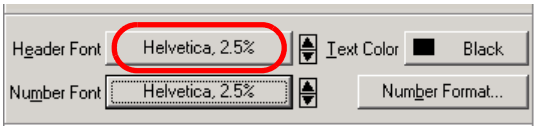


iii. Reposition the legend to "10" for Y%.

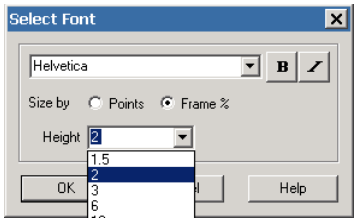


B. In the Font region of the page, perform the following steps:

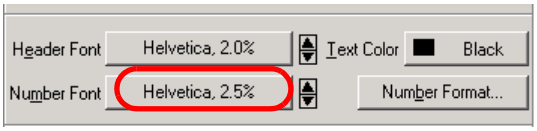
i. Click the Header Font button.



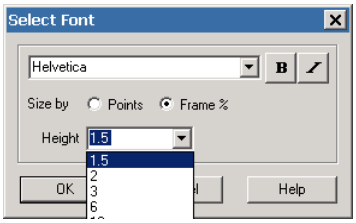
ii. In the **Select Font** dialog, change the header font size (Height) to "2%" and click OK to close the dialog.



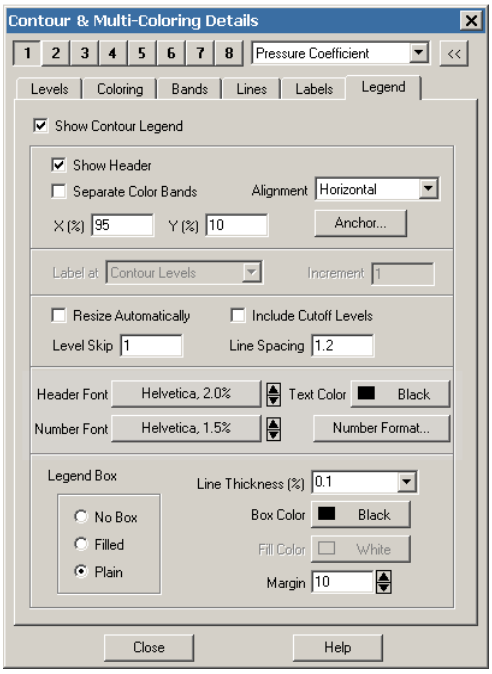
iii. Click the Number Font button.



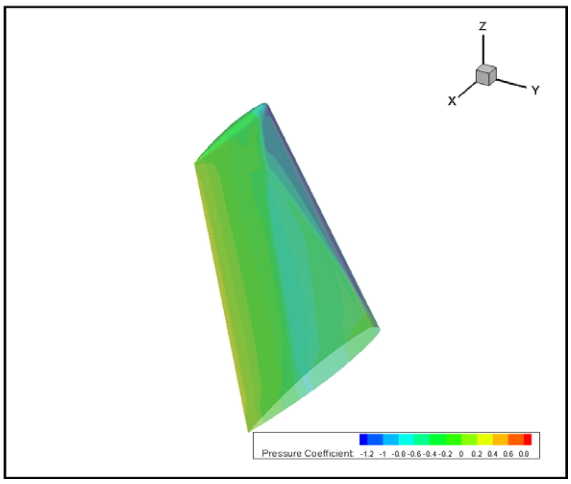
iv. Change the number font size to "1.5%" and click OK to close the dialog.



After completing this step, the Legend page of the **Contour Details** dialog will look as follows:



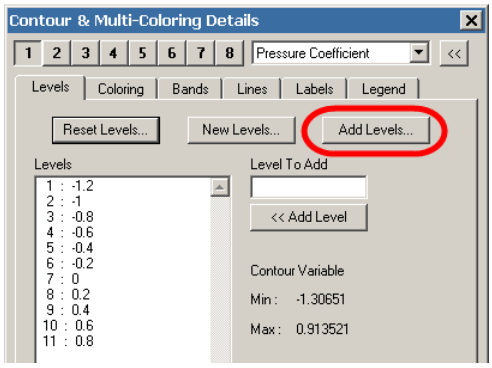
The plot will look as follows:



Step 7 Add Contour Levels

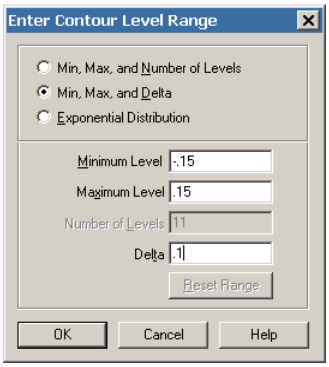
Judging from the legend, the bulk of the data values are between 0.2 and -0.4. We would like to include more levels in that range to add definition to the plot.

A. Switch to the Levels page of the **Contour Details** dialog.



B. Click the Add Levels button.

C. In the **Enter Contour Level Range** dialog, make the following changes:



i. Click the “Min, Max, and Delta” radio button.

ii. Enter “-.15” in the Minimum Level field.

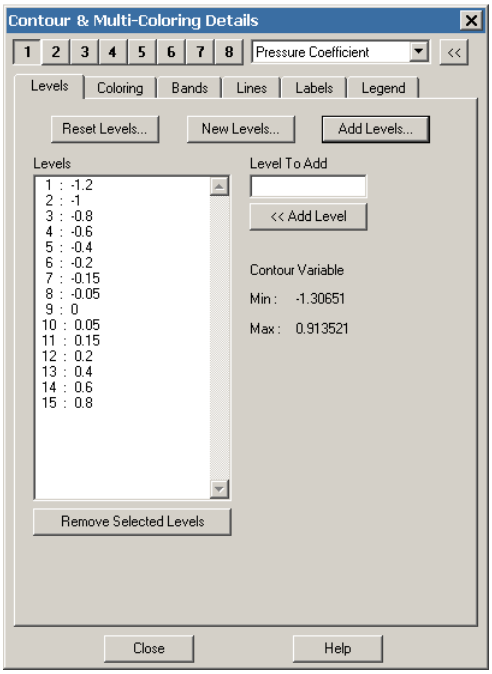
iii. Enter ".15" in the Maximum Level field.

iv. Enter ".1" in the Delta field.

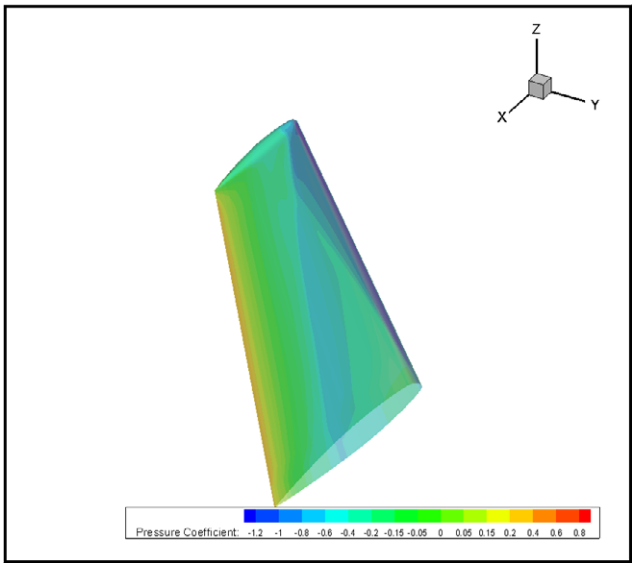
v. Click OK to apply the changes and close the dialog.

Alternatively, you can add contour levels to your plot using the Add Contour Level tool, or using the Level To Add field on the Levels page and adding the following levels: ".15", "-.15", "-.5", and ".5" (one at a time).

The Levels page will look as follows:



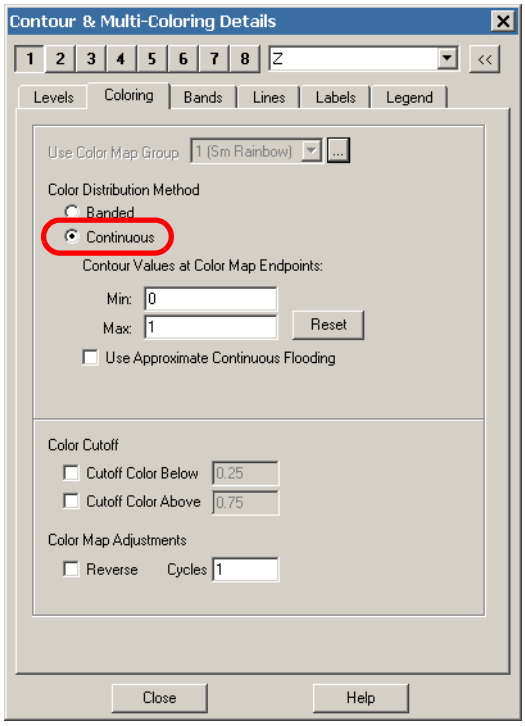
The plot containing a legend will look as follows:



Step 8 Modify the Contour Coloring

A. Switch to the Coloring page of the **Contour Details** dialog.

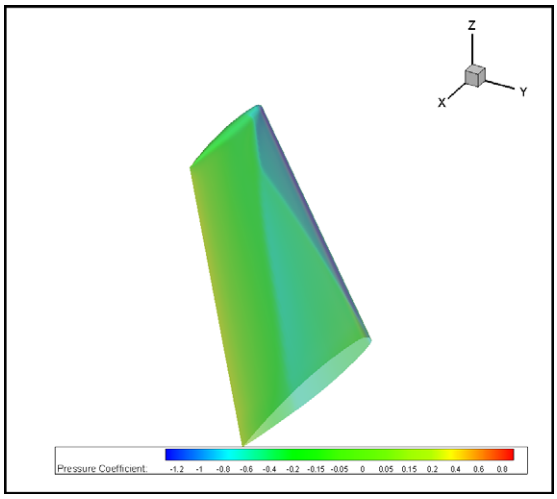
B. Click the "Continuous" radio button.



C. Close the dialog.

You can adjust the color map and its control points by choosing "Color Map" from the **Options** menu.

The plot will look as follows:



Step 9 Isolate the Wing Using the Sidebar

The list of zones represented by the current plot is also viewable by clicking the Zone Style button in the Sidebar. For the purposes of this tutorial, we wish to isolate Zones 5 and 6, the two zones that define the lower wing surface and the upper wing surface ("LowerWingSurface" and "UpperWingSurface").

A. Click the Zone Style button in the Sidebar.

The **Zone Style** dialog allows you to customize display settings for each of the zones in your data file. In this case, we would like to limit the number of zones that are displayed. The first four columns of the **Zone Style** dialog are independent of the active page. Changes made in any of those columns will be applied to the entire plot.

B. In the **Zone Style** dialog, select Zones 1-4 by any of the following methods.

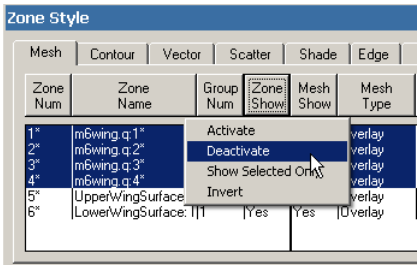
i. Click on Zone 1, holding down the SHIFT key, and click on Zone 4.

ii. Click Zone 1 and drag to Zone 4.

iii. Select "Select Range" from the Zone Num button, and enter "1" in the Begin field and "4" in the End field of the **Enter Range** dialog.

iv. Choose "Select by Name" from the Zone Name button and enter "m6*" in the **Enter Name** dialog.

C. Click the Zone Show button and select "Deactivate".



Alternatively, you can select Zones 5-6 and choose "Show Selected Only" from the Zone Show button.

D. Close the dialog.

The specific zone(s) of the data in which the wing is located is a characteristic of the data file. If you have a data file that is not arranged in this manner, you may find the following methods helpful:

- For many viscous CFD datasets, you can use **Blanking** to isolate regions where the absolute value of the velocity vector is zero.

- For preliminary visualization, you may be able to isolate the region you are looking for by toggling-on "Contour".

Experiment with turning zones on and off and the options on the Surfaces page of the **Zone Style** dialog. In this particular example, if the boundary plot is not loaded, you could view the wing by selecting "J-Planes" from the Surfaces to Plot button on the Surfaces page of the **Zone Style** dialog.

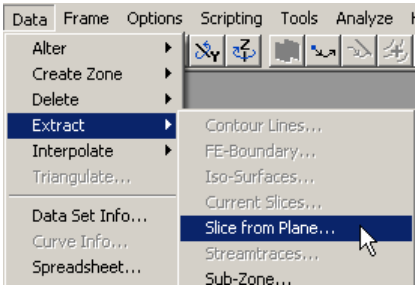
3 - 5 Creating a 2D Projection Plane

During the original ONERA wing study, the researchers compared simulation and measured data at y/b equal to 0.2, 0.44, 0.65, 0.8, 0.9, 0.95, and 0.99, where b is the wingspan. The following steps will recreate the comparison plots for y/b equal to 0.44.

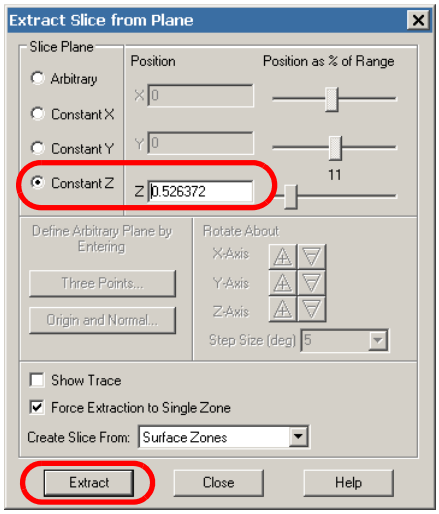
Step 10 Extract a Slice

The y-axis from the data is equivalent to the z-axis in Tecplot 360. From the ONERA study, we know that b (wingspan) is 1.1963 m. As such, we would like to extract a slice at z=0.44*b, or z=0.526372.

A. Go to **Data>Extract>Slice from Plane**.



B. Make the following adjustments in the **Extract Slice from Plane** dialog:

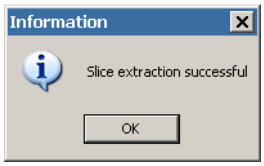


i. Choose the “Constant Z” radio button.

ii. Set “Position Z” equal to “0.526372”.

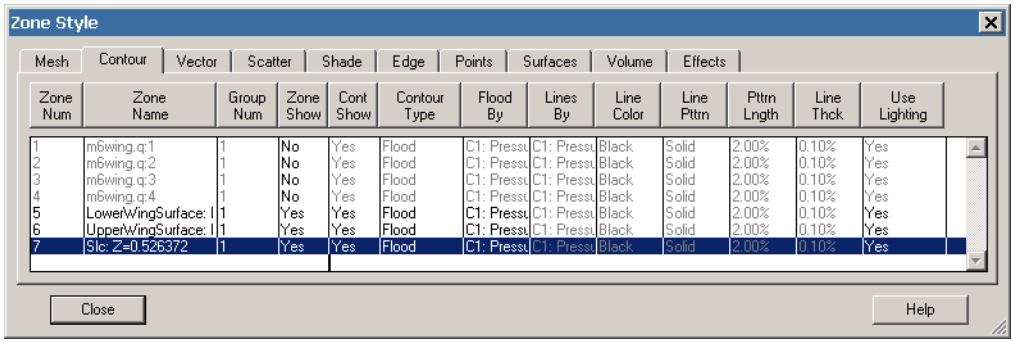
iii. Click the Extract button.

C. Click OK in the **Information** dialog.



D. Close the **Extract Slice from Plane** dialog.

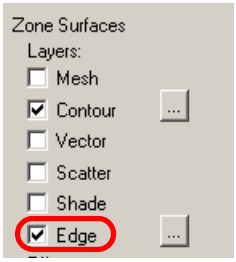
E. Go to the **Zone Style** dialog and verify that a new zone has been created. The new zone is named “SLC Z=0.526372”.



Step 11 Activate the Edge Layer for Zone 7 Only

We would like to show the slice in the Edge layer, and also have the slice as the only zone visible in the Edge layer. We can accomplish this by using the **Zone Style** dialog.

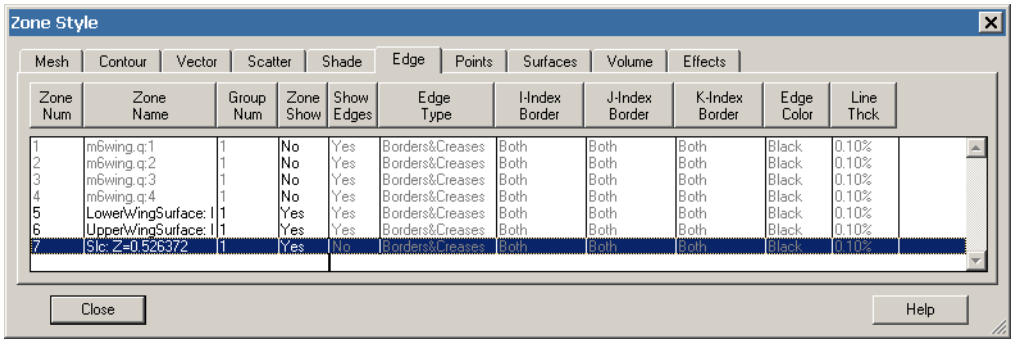
A. Toggle-on “Edge” in the Sidebar.



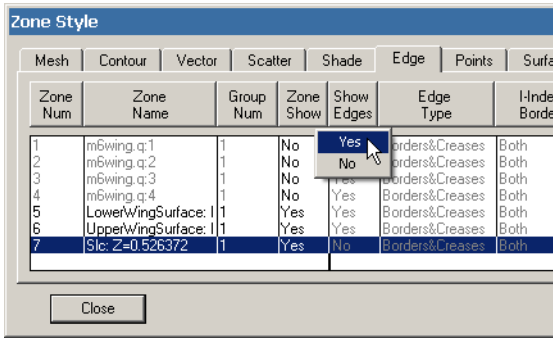
B. Reopen the **Zone Style** dialog, if you closed the dialog at the end of Step 10.

C. Switch to the Edge page of the **Zone Style** dialog and perform the following steps:

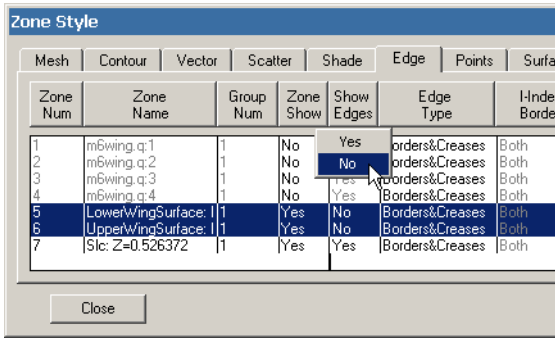
i. Choose Zone 7 (the extracted slice).



ii. Select "Yes" from the Show Edges button.

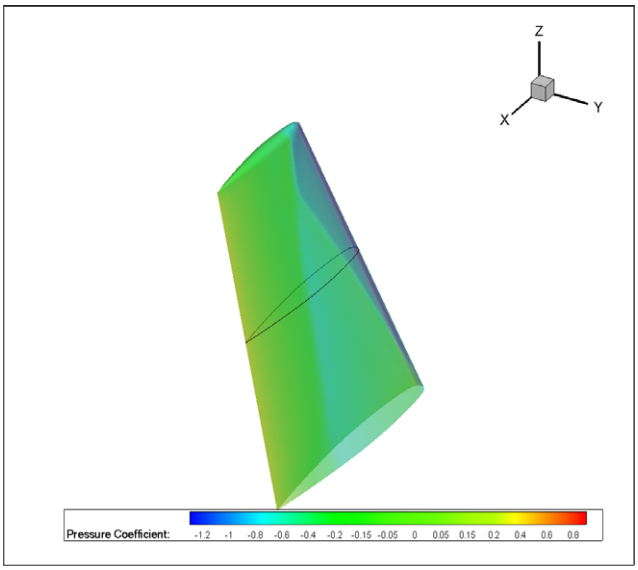


iii. Select Zones 5-6 and select "No" from the Show Edges button.



iv. Close the **Zone Style** dialog.

The plot will look as follows:



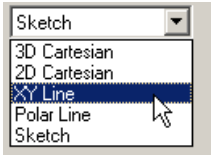
When the wingspan is not recorded, you can approximate the wingspan using the Probe tool and a distance equation.

Step 12 Create a New Frame

A. Click the Create New Frame tool in the toolbar  .

B. Drag in the workspace to create a new frame.

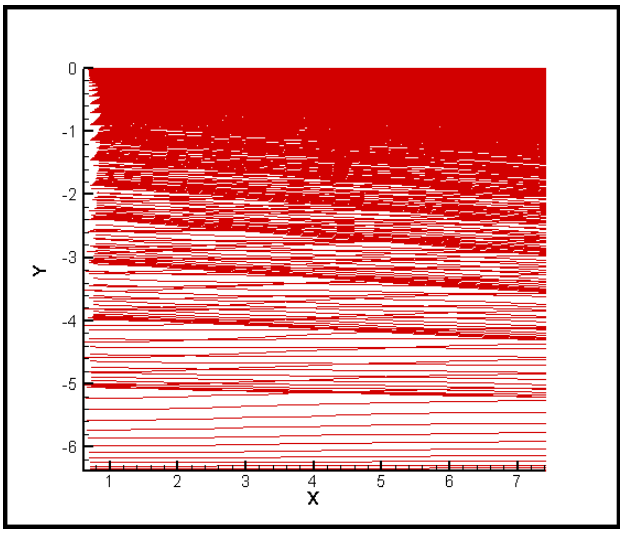
C. To associate the new frame with the existing dataset, go to the Sidebar and switch the Plot Type to "XY line"



Prior to this point, the newly created frame did not have an associated dataset. Switching the plot type for a new frame to a plot type that requires a dataset will automatically attach the dataset from another frame to the new frame.

When working with multiple frames, click within a frame to make that the active frame. By default, the active frame displays with a thicker black border.

The plot will look as follows:



Step 13 Delete a Group of Mappings

The new plot currently displays an XY line plot of the first zone in the original dataset. Ultimately, we would like to plot Cp versus x/c.

A. With your new frame as the active frame, open the **Mapping Style** dialog by either:

   i. Clicking the Mapping Style button in the Sidebar

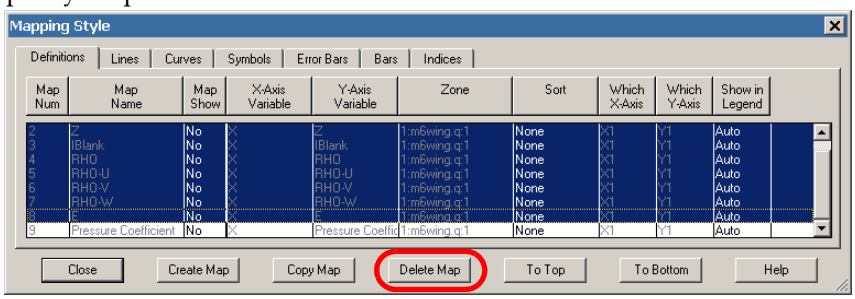
   ii. Selecting **Plot>Mapping Style**

B. In the **Mapping Style** dialog, select Maps 1-8 by any of the following methods:

i. Click on Map 1, hold down the SHIFT key and click on Map 8

ii. Click on Map 1, and drag to Map 8

iii. Select "Select Range" from the Map Num button and use the **Enter Range** dialog to specify Maps 1-8



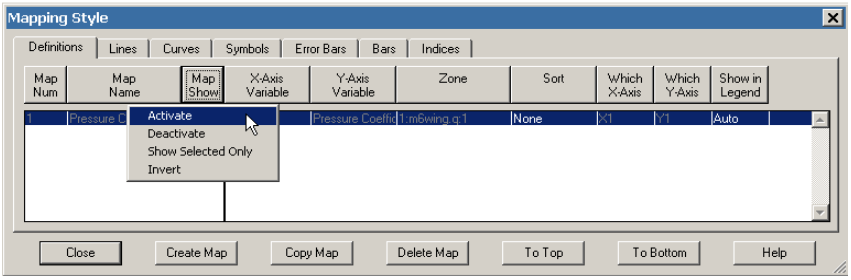
C. With Maps 1-8 highlighted, click the Delete Map button.

D. Click OK to confirm deletion of the eight zones.

Step 14 Activate the Mapping

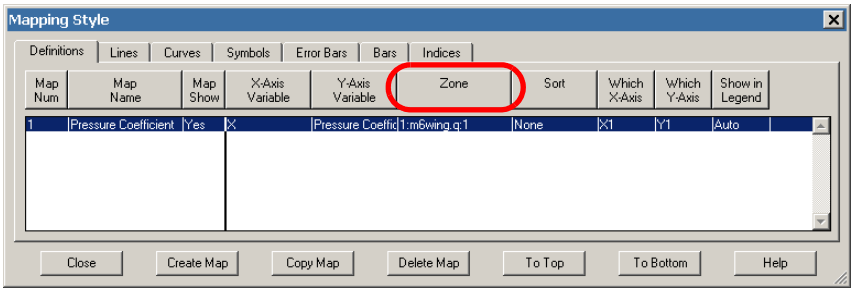
A. With the **Mapping Style** dialog still open, the remaining mapping will be highlighted.

B. Select "Activate" from the Map Show button.

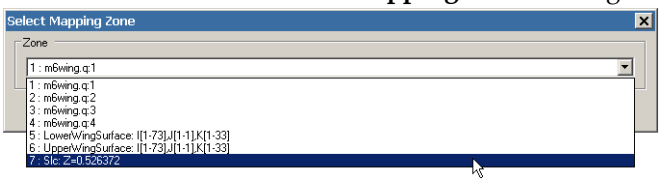


Step 15 Change the Zone Associated with the Mapping

A. In the **Mapping Style** dialog, click the Zone button.



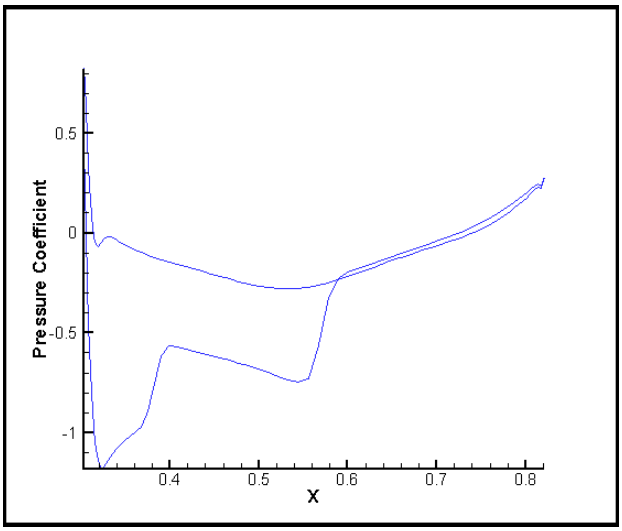
B. Select "7: SLC Z=0.526372" from the **Select Mapping Zones** dialog.



C. Click the OK button to close the **Select Mapping Zone** dialog. Close the **Mapping Style** dialog.

Step 16 Adjust the View

Still with the XY Line plot as the active frame, go to **View>Fit to Full Size**. The plot will look as follows:



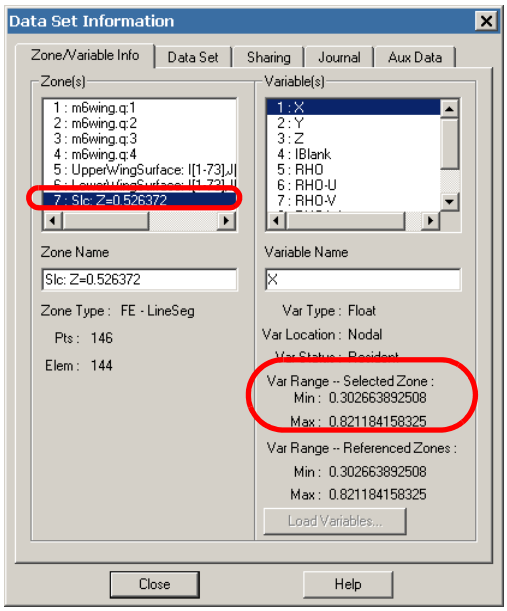
3 - 6 Manipulating Data

The simulated data (Pressure Coefficient) is currently plotted against the x variable. The measured data that will be loaded in the next section uses x/c, where c is the chord length. In order to compare the datasets, we will create a new variable {x/c} that shifts the current x variable to 0 and scales the value by c.

Step 17 Determine the Max and Min Values of the X Variable

A. Go to **Data>Data Set Info.**

B. Select "Zone 7" in the Zone(s) area of the **Data Set Information** dialog.



C. Make note of the Min & Max values listed in the Var Range - Selected Zone region of the dialog. We will use these values in the next step.

i. Min = "0.3027"

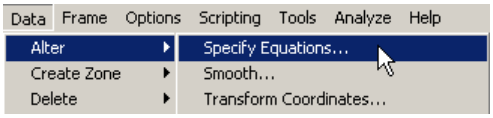
ii. Max = "0.8212"

iii. Range = "0.5185" (not listed)

D. Close the dialog.

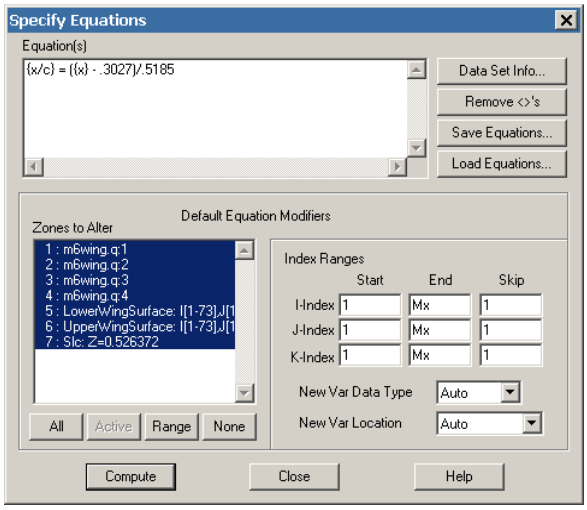
Step 18 Create a New Variable

A. Go to **Data>Alter>Specify Equations**.



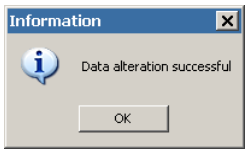
B. Enter the following equation in the Equations(s) field of the **Specify Equations** dialog:

{x/c} = ({x} - .3027)/.5185



C. Click the Compute button.

D. Click the OK button in the **Information** dialog.



Dataset variables that are identified by name are denoted with curly braces, {}, in the Specify Equations dialog. New variables can be created simply by placing the new variable name within curly braces at the left-hand side of the equation. New variables must be applied to all zones.

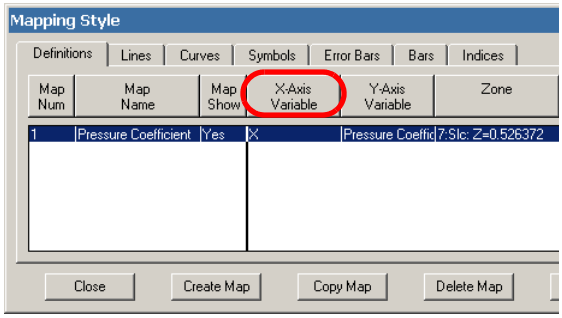
In the above dialog, a new variable {x/c} was created using shifted and scaled values of the existing {x} variable.

E. Close the dialog.

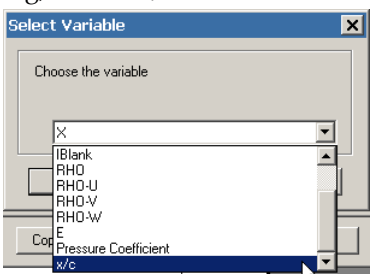
Step 19 Change the X-Axis Variable

A. Open the **Mapping Style** dialog. The remaining mapping will be highlighted.

B. Click the X-Axis Variable button on the Definitions page.

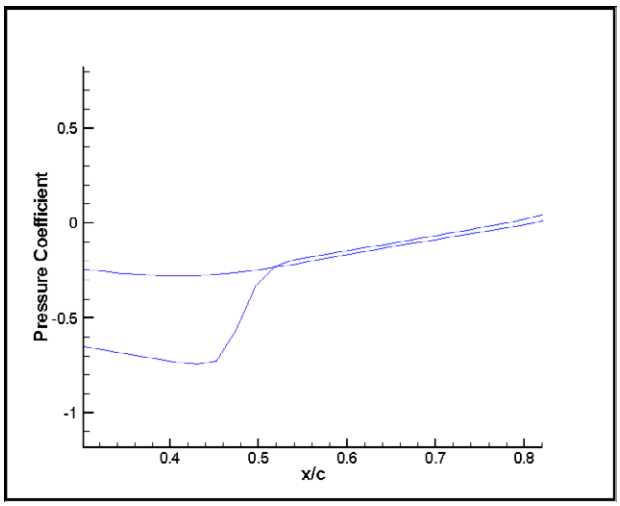


C. In the **Select Variable** dialog, select "x/c" from the menu.



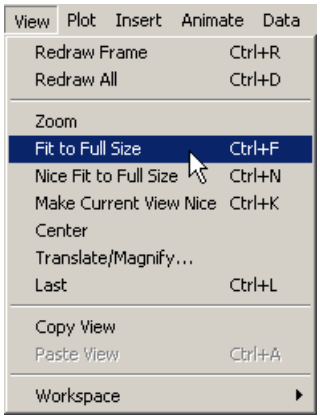
D. Click OK to assign the variable and close the dialog.

E. Close the **Mapping Style** dialog. The plot will look as follows:

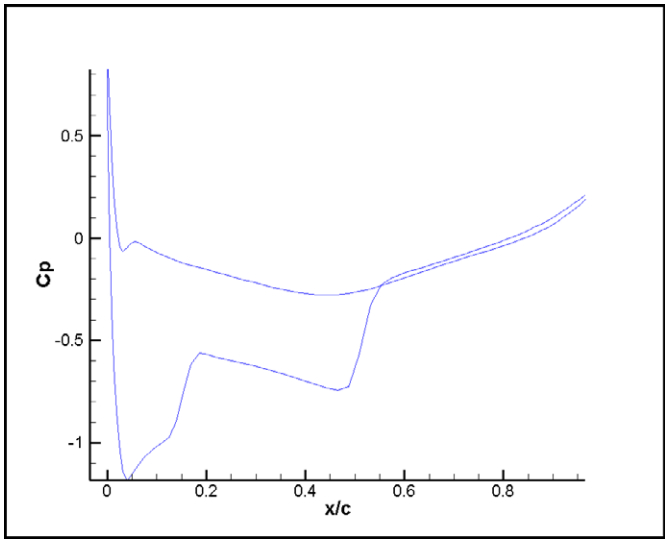


Step 20 Adjust the View

Go to **View>Fit to Full Size**.



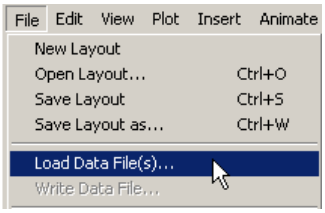
The plot will look as follows:

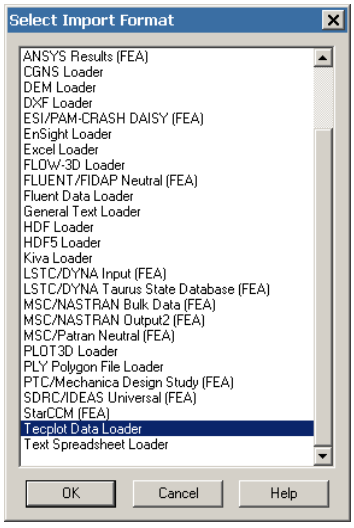


3 - 7 Comparing Simulation Data with Measured Data

Step 21 Load the Measured Data File

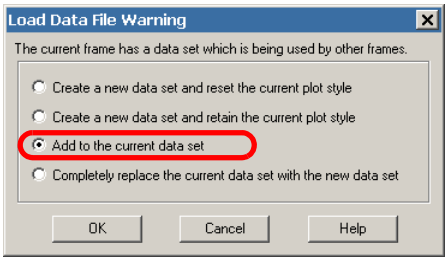
A. Go to **File>Load Data File**.



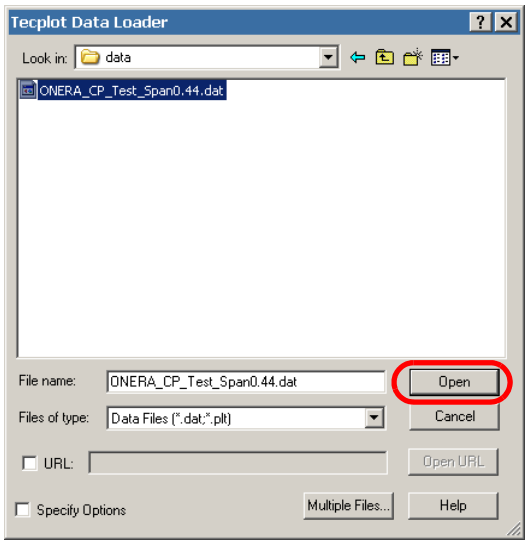


B. In the **Select Import Format** dialog, select the "Tecplot Data Loader".

C. In the **Load Data File Warning** dialog, choose the "Add to the current data set" radio button and click the OK button.

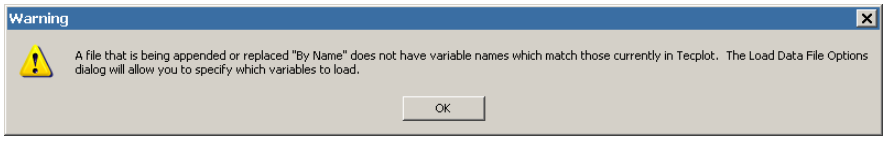


D. In the **Tecplot Data Loader** dialog, navigate to: $TEC_360_2013R1*tutorials**external_flow**data* and select "*ONERA_CP_Test_Span0.44.dat*".

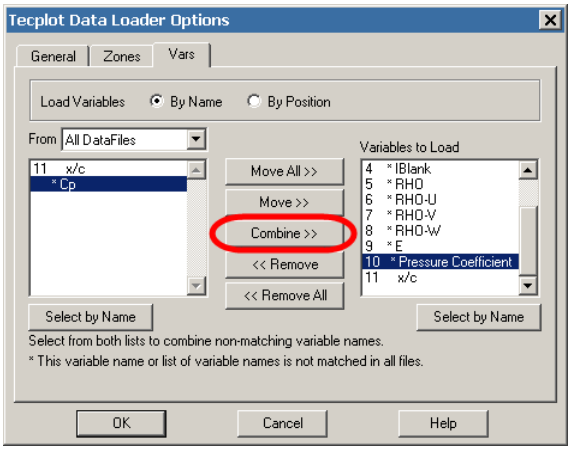


E. Click the Open button to load the file.

F. Click OK in the **Warning** dialog.



G. Go to the Vars page of the **Tecplot Data Loader Options** dialog.



Variables listed on the left-hand side of the dialog are from the new data file. Variables listed on the right-hand side of the dialog are already in the dataset. If a variable has an '*' before its name, it is not present in all files.

Because the Pressure Coefficient has a different variable name in the experimental and measured datasets, we need to combine the two pressure variables.

i. Choose the "Cp" variable in the left-hand column.

ii. Choose the “Pressure Coefficient” variable in the right-hand column.

iii. Click Combine from the center column.

iv. Click the OK button to apply your changes and close the dialog.

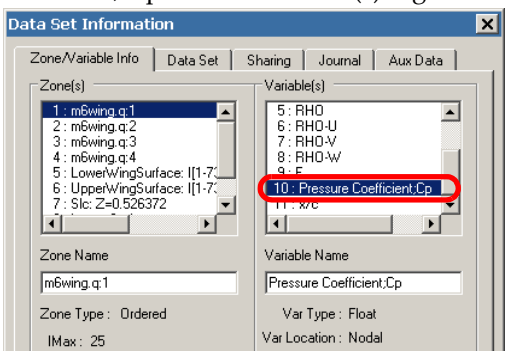
The combined variable name will be “Pressure Coefficient;Cp”.

Step 22 Change the Variable Name

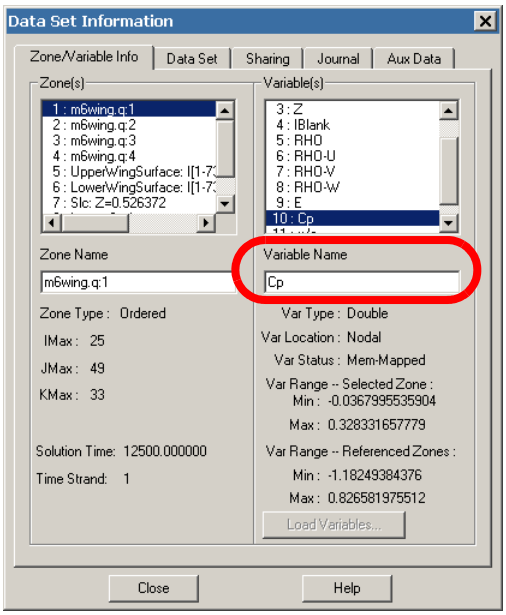
To simplify our axes labels, we will change the combined variable name.

A. Go to **Data>Data Set Info**.

B. Choose “Pressure Coefficient; Cp” in the Variable(s) region of the dialog.



C. Enter “Cp” in the Variable Name field.



The name change will apply to all zones.

D. Close the dialog.

Step 23 Create New Mappings

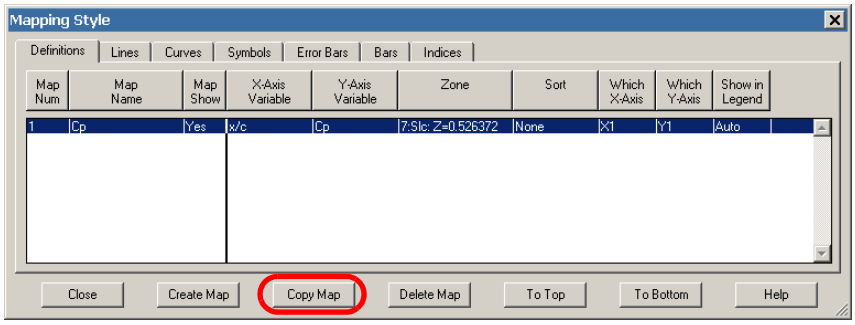
We now need to add mappings of the measured dataset to our plot.

A. Open the **Mapping Style** dialog by either of the following options:

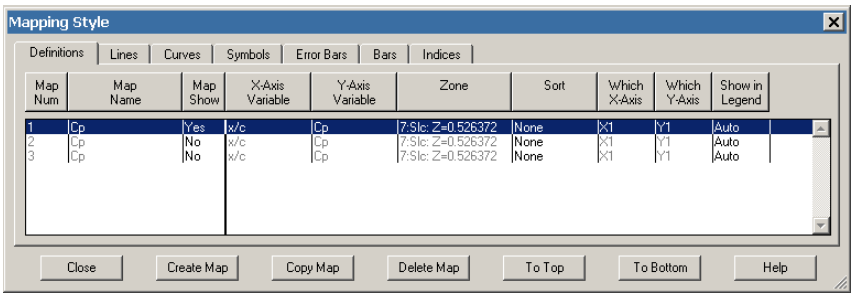
i. Clicking the Mapping Style button in the Sidebar

ii. Choosing “Mapping Style” from the **Plot** menu

B. With the remaining mapping selected, click the Copy Map button twice.



The **Mapping Style** dialog will look as follows:



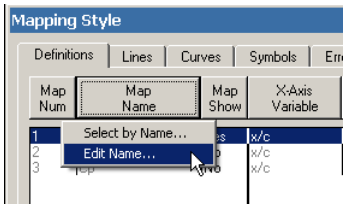
Step 24 Rename the Mappings

To avoid confusion, we will rename all of the mappings in the plot.

A. Rename Map 1.

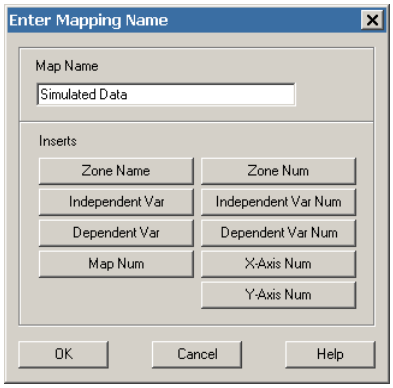
i. Select Map 1.

ii. Choose “Edit Name” from the Map Name button.



iii. Enter “Simulated Data” in the **Enter Mapping Name** dialog.

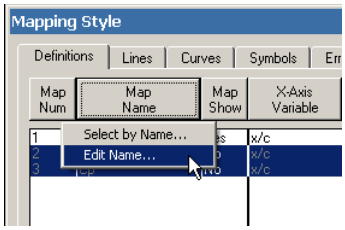
iv. Click the OK button to apply the changes and close the dialog.



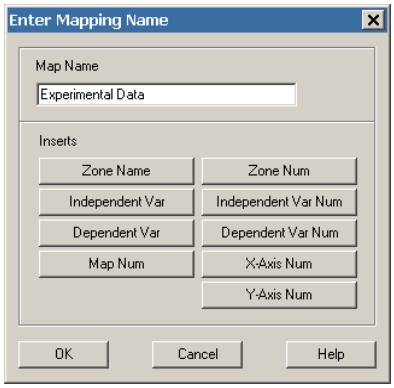
B. Rename Maps 2 & 3.

i. Select Maps 2 & 3.

ii. Choose “Edit Name” from the Map Name button.



iii. Enter "Experimental Data" in the **Enter Mapping Name** dialog.



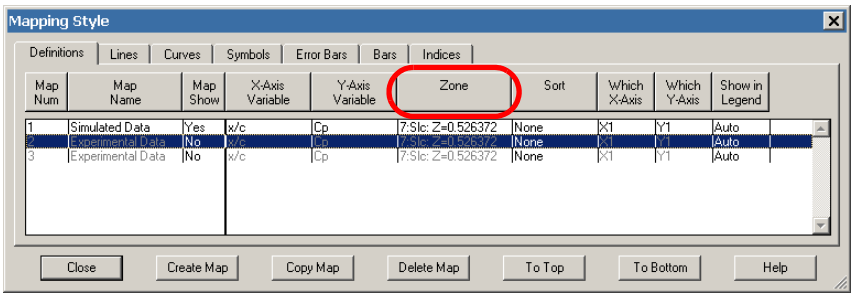
iv. Click the OK button to apply the changes and close the dialog.

Step 25 Change Zone Assignments

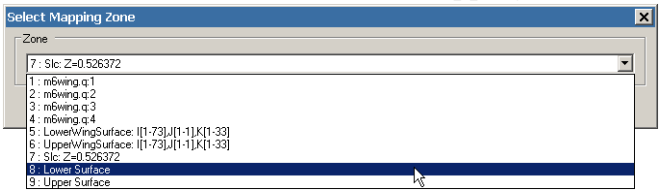
A. Map Num 2:

i. In the **Mapping Style** dialog, choose "Map Num 2".

ii. Click the Zone button.



iii. Choose "8: Lower Surface" in the **Select Mapping Zone** dialog.

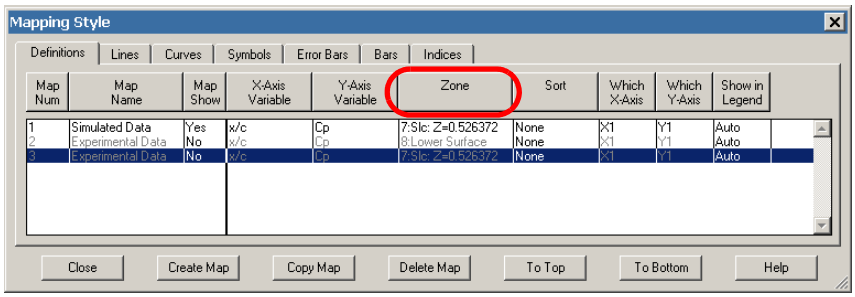


iv. Click the OK button to apply the changes and close the dialog.

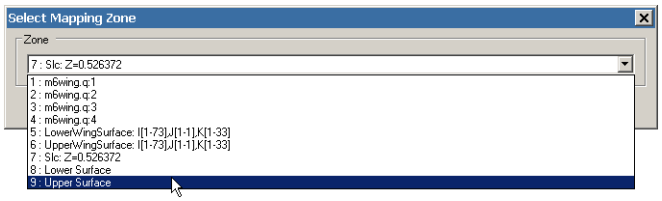
B. Map Num 3:

i. In the **Mapping Style** dialog, choose "Map Num 3".

ii. Click the Zone button.



iii. Choose "9: Upper Surface" from the **Select Mapping Zone** dialog.



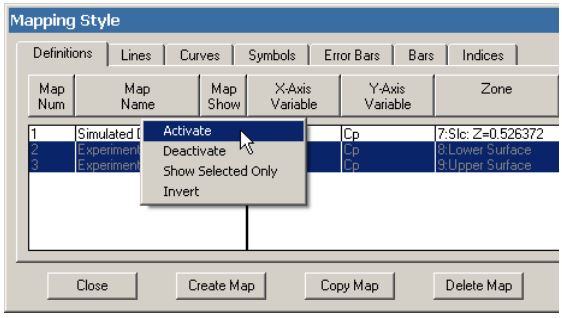
iv. Click the OK button to apply the changes and close the dialog.

Step 26 Activate the New Mappings

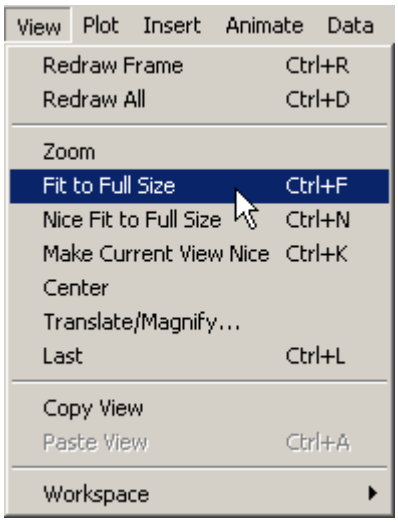
The new mappings have not appeared in the plot yet because they have not been activated.

A. Choose Maps 2 & 3 in the **Mapping Style** dialog.

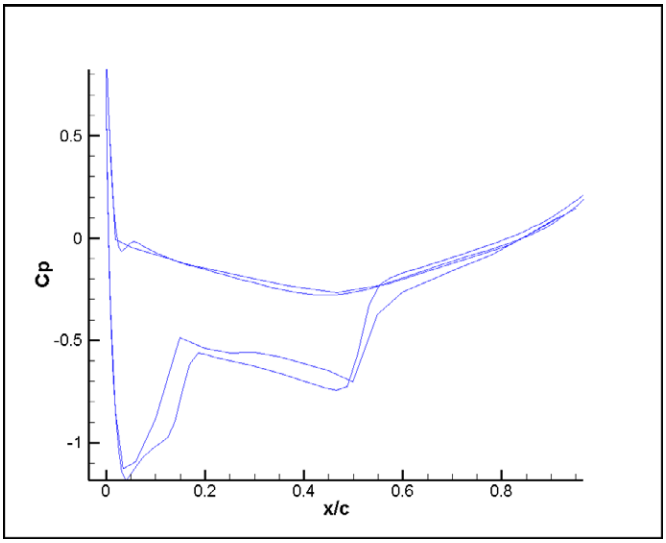
B. Choose "Activate" from the Map Show button.



C. Go to **View>Fit to Full Size**.



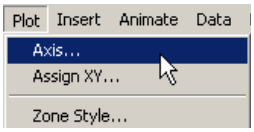
The plot will look as follows:



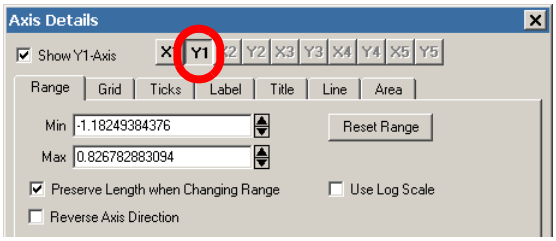
Step 27 Reverse the Y-Axes

In aerodynamics, it is customary to display the negative pressure increasing upwards, so we will reverse the vertical axis.

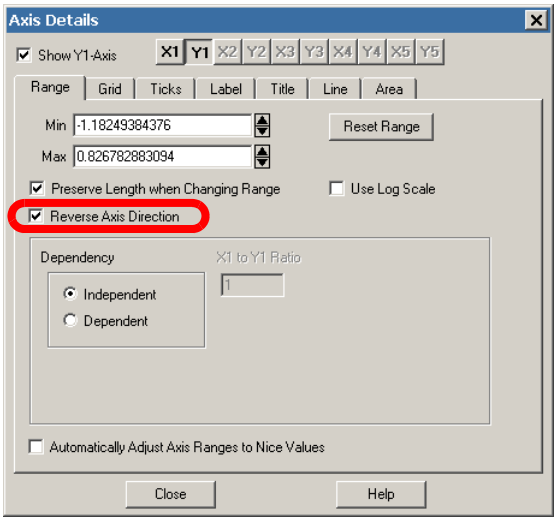
A. Go to **Plot>Axis**.



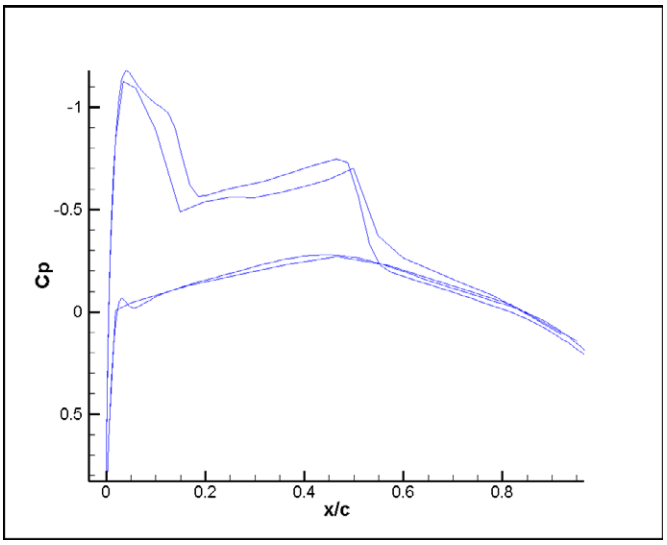
B. In the **Axis Details** dialog, click the Y1 button.



C. On the Range page of the **Axis Details** dialog, toggle-on "Reverse Axis Direction".



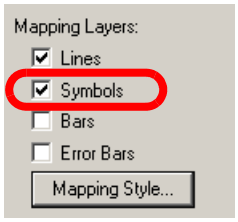
D. Close the **Axis Details** dialog. The plot will look as follows:



Step 28 Adjust the Mapping Style

The current style of the plot makes it difficult to discriminate between the simulated and measured data. As such, we will change the measured data to a symbol plot.

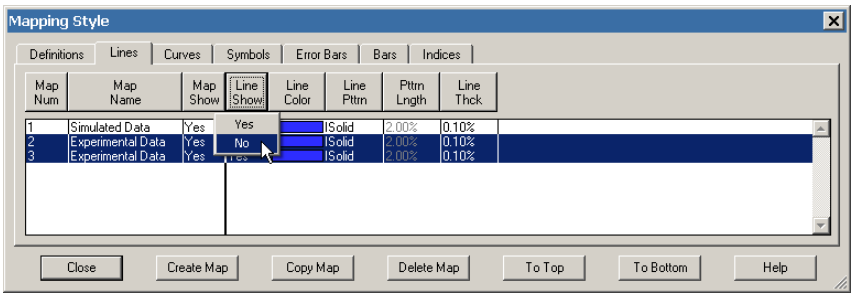
A. Toggle-on the "Symbols" layer in the Sidebar.



B. Open the **Mapping Style** dialog and go to the Lines page.

i. Select Maps 2 & 3.

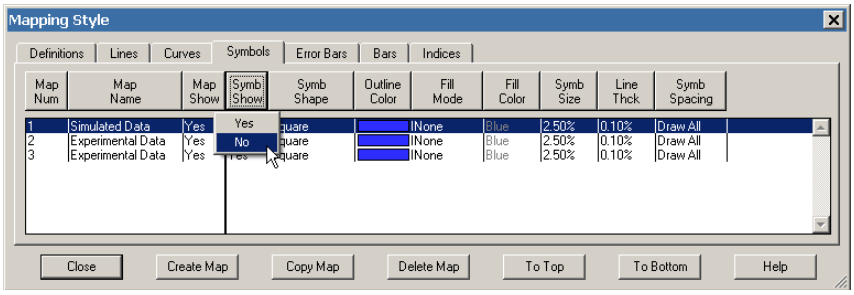
ii. Choose “No” from the Line Show button.



C. On the Symbols page of the **Mapping Style** dialog:

i. Select Map 1.

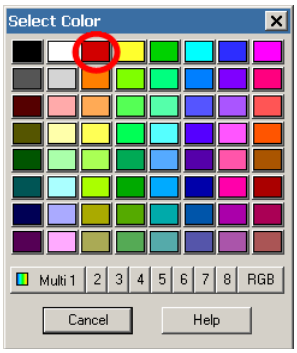
ii. Choose “No” from the Symb Show button.



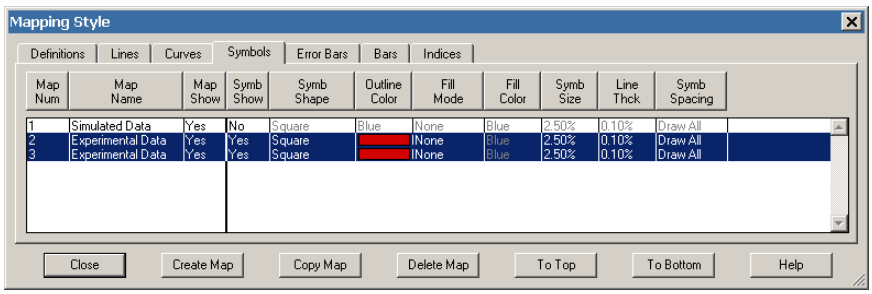
iii. Select Maps 2 & 3.

iv. Click the Outline Color button.

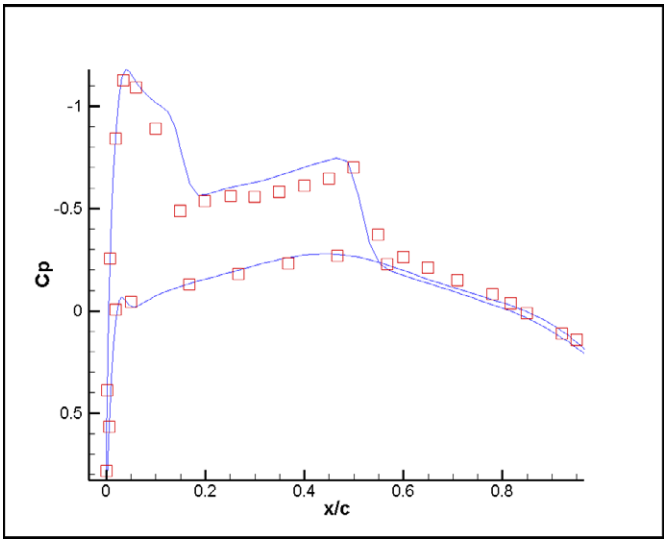
v. Select Red in the **Select Color** dialog.



The Symbols page of the **Mapping Style** dialog will look as follows:

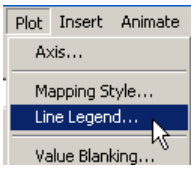


The plot will look as follows:

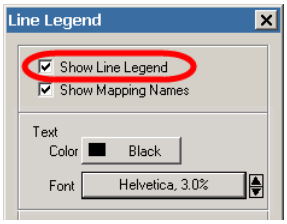


Step 29 Add a Line Legend

A. Go to **Plot>Line Legend**.

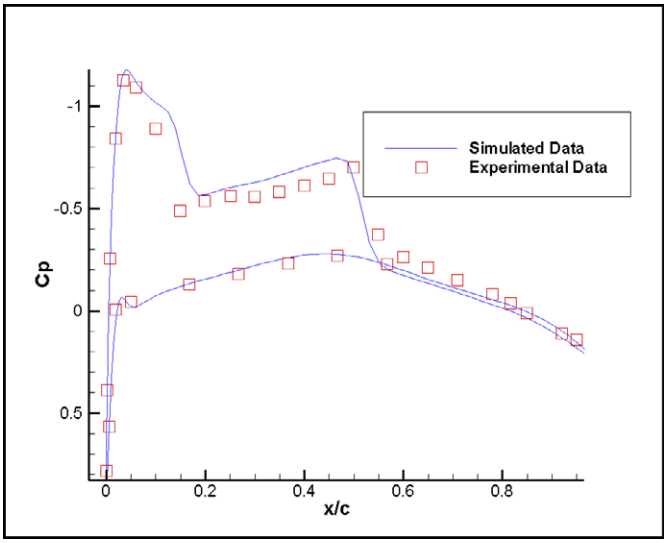


B. In the **Line Legend** dialog, toggle-on "Show Line Legend".



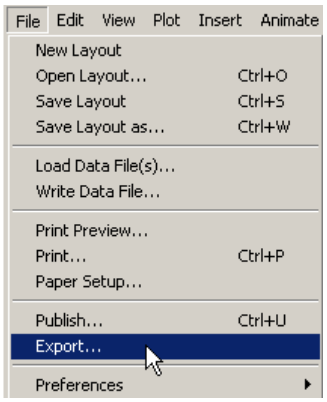
C. Accept the remaining default settings by closing the dialog.

The final plot will look as follows:

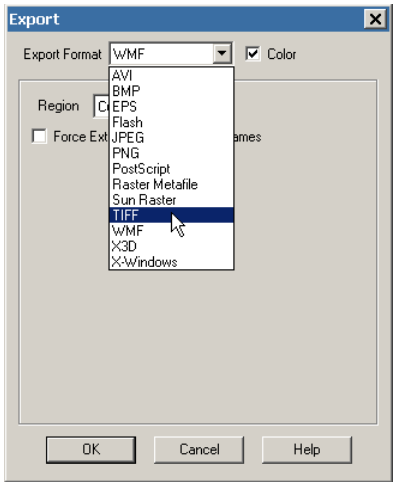


Step 30 Export the Frames to a File

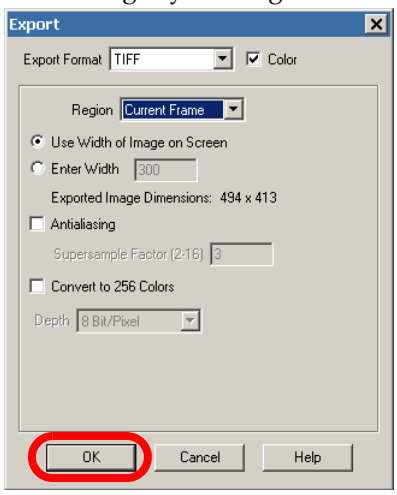
A. Go to **File>Export**.



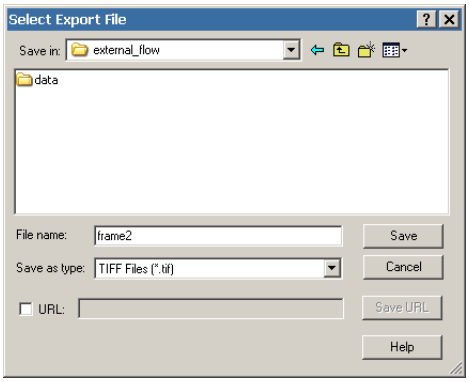
B. In the **Export** dialog, choose "TIFF" from the **Export Format** menu.



C. Accept the remaining default settings by clicking OK.



D. In the **Select Export File** dialog, navigate to the desired directory and specify a filename.



E. Click Save to export the image.

The image file is now ready to be imported into your presentation software.

3 - 8 Conclusion

This concludes the External Flow tutorial. Having completed this tutorial you should now be familiar with calculating and displaying contour variables such as pressure, as well as comparing measured and simulated data on a 2D projection plane. Refer to the User's Manual for details regarding any of the features discussed in this tutorial.

4

Internal Flow Tutorial

4 - 1 Introduction

This tutorial demonstrates how to visualize the internal flow during the mixing process of soap or detergent. We will use several Tecplot 360 features to create our plot: 3D rotation, surface clipping, lighting effects, and streamtraces.

4 - 1.1 Background Information

There are four primary steps for manufacturing soap and detergent: mixing, heat transfer, drying, and separation. In both liquid and powder detergent manufacturing, dry and liquid ingredients are blended into a uniform mixture using static mixers called crutchers. Location of the dry and liquid inlets, internal temperature, and the velocity of the particles leaving the crutcher all play an important role in optimizing the manufacturing process.

This tutorial will model and examine some of these properties with a crutcher model.

4 - 1.2 Tutorial Summary

The tutorial takes an average of 20 minutes to complete. All supporting data files for this tutorial can be found in *$TEC_360_2013R1\tutorials\internal_flow,* where $TEC_360_2013R1 represents the default installation folder for Tecplot 360. For Windows users, Tecplot 360 installs by default to C:\Program Files\Tecplot\Tec360 2013R1.

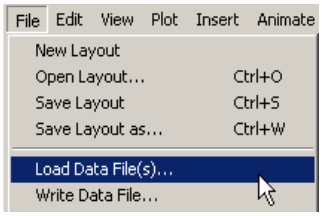
4 - 2 Getting Started

Step 1 Load a Fluent File

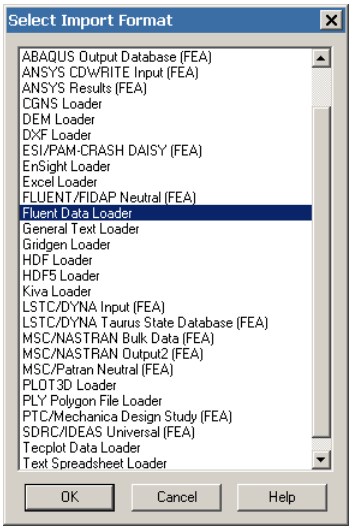
Tecplot 360's **Fluent Data Loader** allows you to load FLUENT® case *.gas* and/or data *.dat* files (as well as compressed case *.cas.gz* and/or compressed data *.dat.gz* files). For this tutorial, we will load one set of case and data files.

The "Fluid Structure Interaction Tutorial" demonstrates loading multiple Fluent data files.

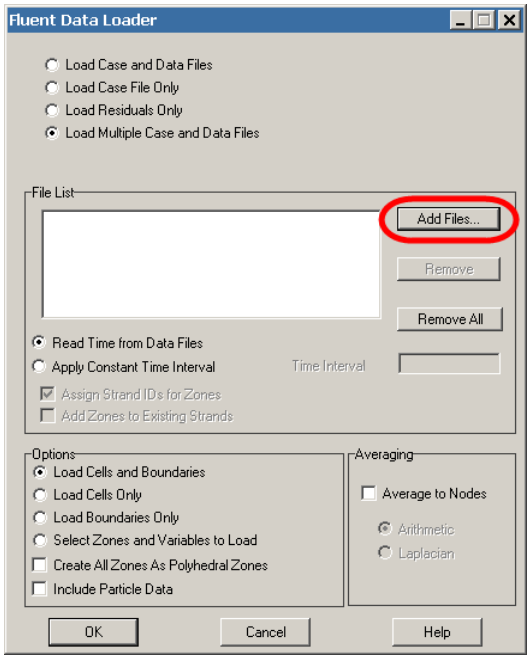
A. To start loading the files, choose "Load Data File(s)" in the **File** menu.



B. Select "Fluent Data Loader" in the **Select Import Format** dialog.



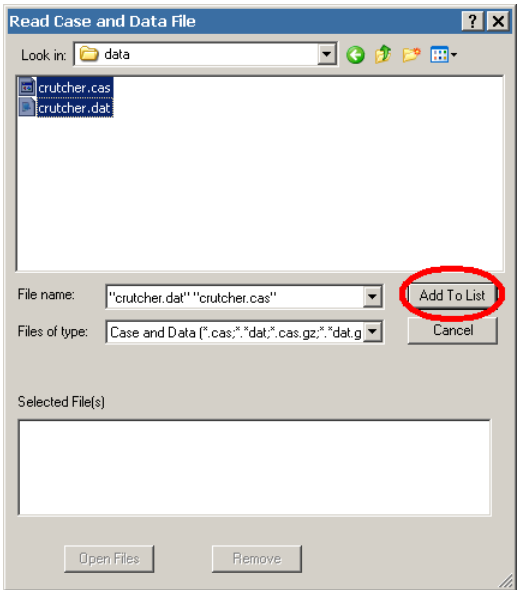
C. In the **Fluent Data Loader** dialog, click the Add Files button.



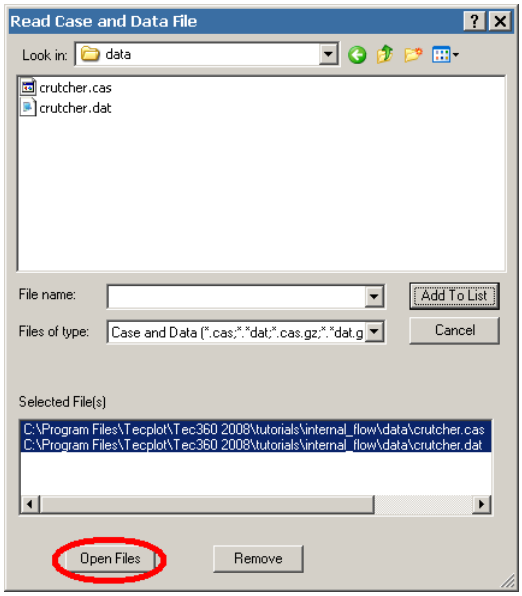
D. In the **Read Case and Data File** dialog:

i. Navigate to: *$TEC_360_2013R1\tutorials\internal_flow\data.*

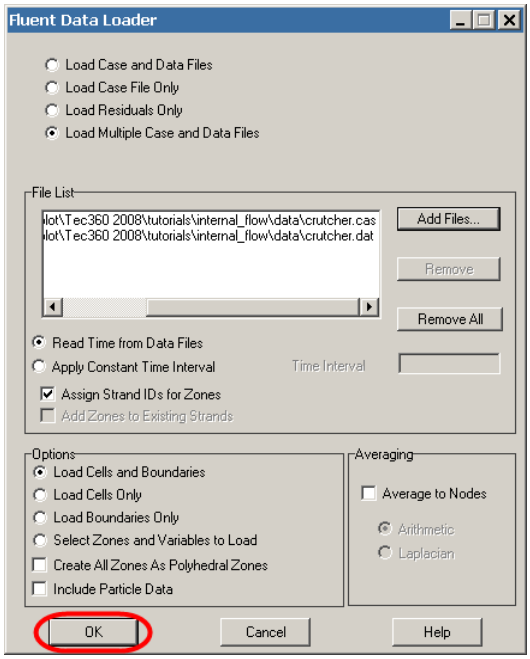
ii. Highlight both files (*crutcher.dat* and *crutcher.cas*) and click the Add To List button.



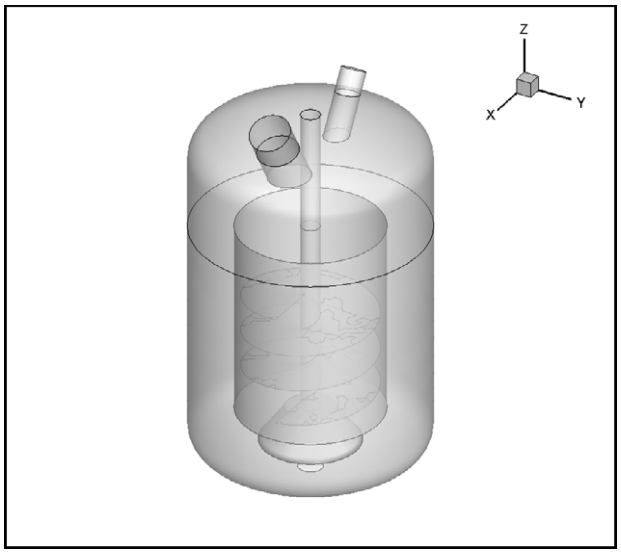
iii. Click the Open Files button.



E. In the **Fluent Data Loader** dialog, accept the remaining default settings and click OK to close the dialog.

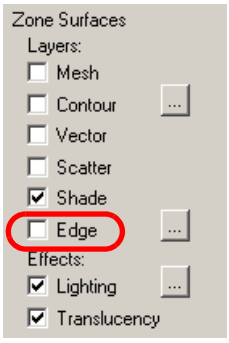


The initial view of the plot will look as follows:



Step 2 Use the Sidebar to Deactivate Plot Layers

By default, the Edge layer is toggled-on when a data file is first loaded into Tecplot 360. In this case, the number of zones in the data make it difficult to discern the objects in the plot.



Toggle-off the Edge layer in the Sidebar.

4 - 3 Adjusting the Plot View

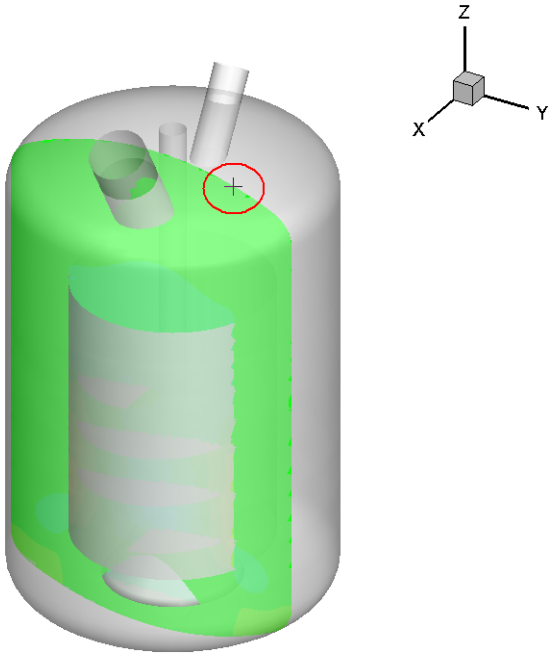
Step 3 Slice and Clip to View a Cross Section

We would like to view the mixer's interior without eliminating any zones, which we can accomplish by using surface clipping.

To use surface clipping, we will insert a slice, adjust the slice position, and "clip off" the display of all zones in front of the slice.

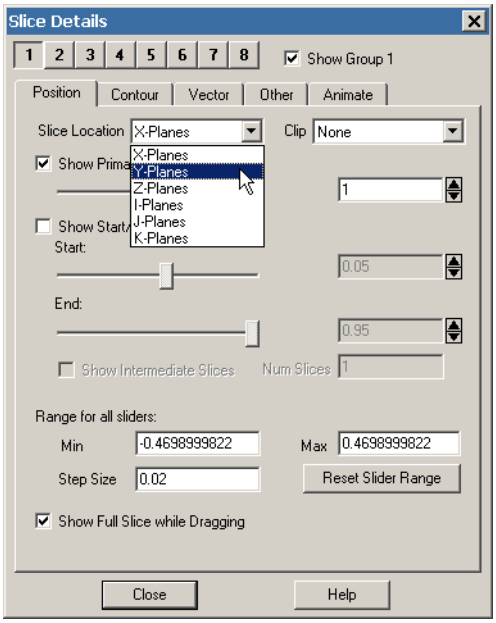
While experimenting with the position of the slice, you can rotate the mixer by pressing the Ctrl key while dragging in the workspace with the right mouse button.

A. Click the Slice tool in the Sidebar to activate the Slice tool.  Click in the mixer to insert a slice.



B. Open the **Slice Details** dialog by clicking the Details button next to Slices in the Sidebar.

C. Choose Y-Planes from the Slice Location menu to orient the slice along the Y plane rather than the default X plane. Keep this dialog open.

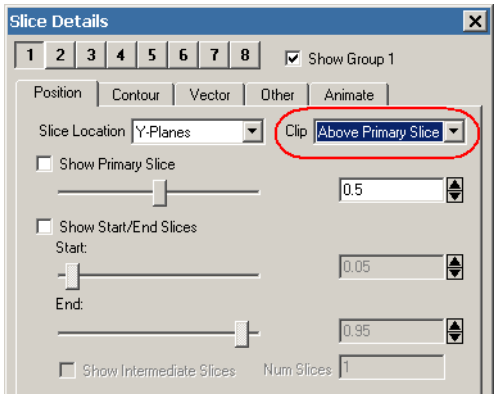


D. You can adjust the position of the slice by either of two methods:

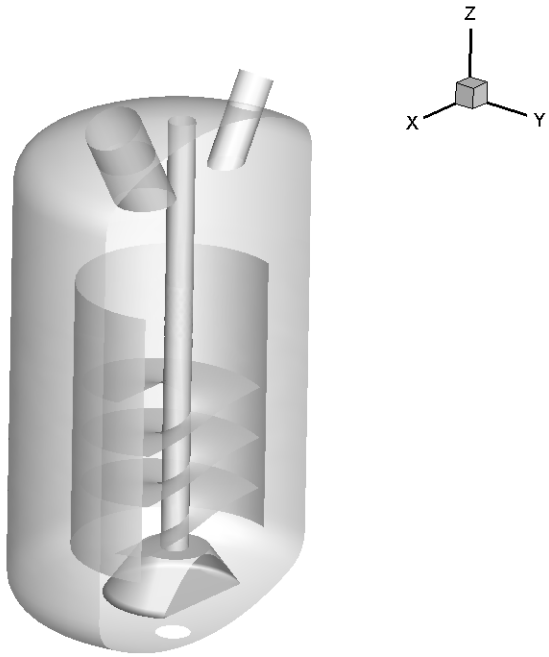
i. **Adjust Interactively** To adjust the position of the slice interactively, with the Slice tool still selected, drag the slice. You can also drag the slider in the Slice Location region of the dialog, below the Show Primary Slice toggle.

ii. **Edit Position** To adjust the position by editing coordinates, in the **Slice Details** dialog (accessible by clicking the Details button next to Slices in the Sidebar, or by choosing "Slices" from the **Plot** menu), enter a coordinate value the first coordinate field in the dialog. A good value for this cut-away is 0.1.

E. To clip off the display in front of the slice, in the **Slice Details** dialog, select "Above Primary Slice" from the Clip menu. The primary slice display will automatically turn off, revealing the inside of the mixer.



F. Close the **Slice Details** dialog. Your plot will look similar to the following:



You can still change the position of the clipping slice by choosing the Slices tool and dragging the slice in your plot.

Step 4 Adjust View Inside Mixer

Now we can see into the mixer. However, we would like to see complete views of internal structures, including the blades and the shaft. In this step we will adjust which zones obey the surface clipping.

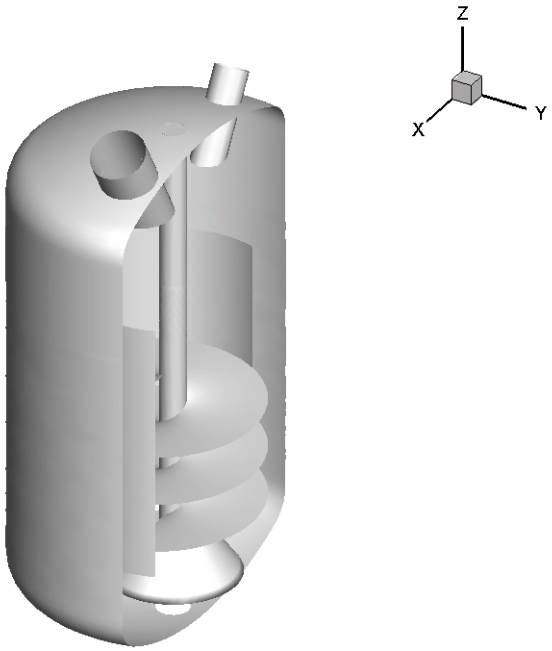
A. First, the translucency effect is no longer needed, since the surface clipping allows an inside view of the mixer. Toggle-off the Translucency layer in the Sidebar.

A. Click the Zone Style button in the Sidebar to open the **Zone Style** dialog. Notice that each structural piece of the mixer is controlled by a zone.

B. On the Effects page of the **Zone Style** dialog, select Zones 9 and 10 (using the SHIFT key or by dragging with your pointer tool). Click the Clip By button, and select "None" from the menu.

C. Select Zones 21 and 22, and select "None" from the Clip By menu. This will prevent the blades from being clipped by the clipping slice.

Your plot will look similar to the following:



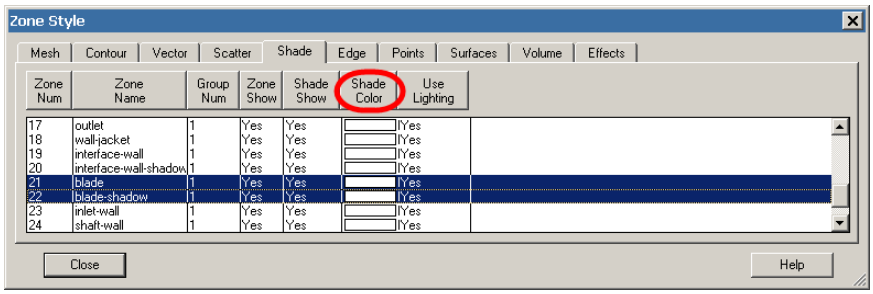
4 - 4 Using the Zone Style Dialog to Customize Settings

In the previous step of this tutorial, we had a glimpse of how to customize specific zones in the **Zone Style** dialog. Now we will begin to fine-tune the individual zone settings. Click the Zone Style button from the Sidebar to reopen the **Zone Style** dialog.

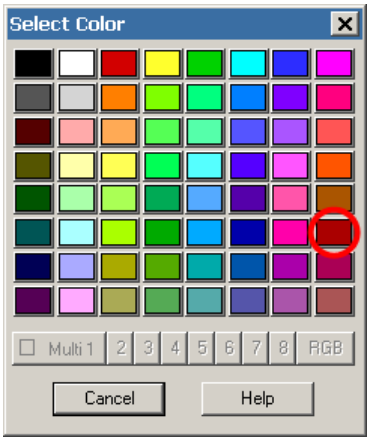
Step 5 Accentuate the Blade

A. On the Shade page of the **Zone Style** dialog, highlight Zones 21 & 22 (using the SHIFT key or by dragging with your pointer tool).

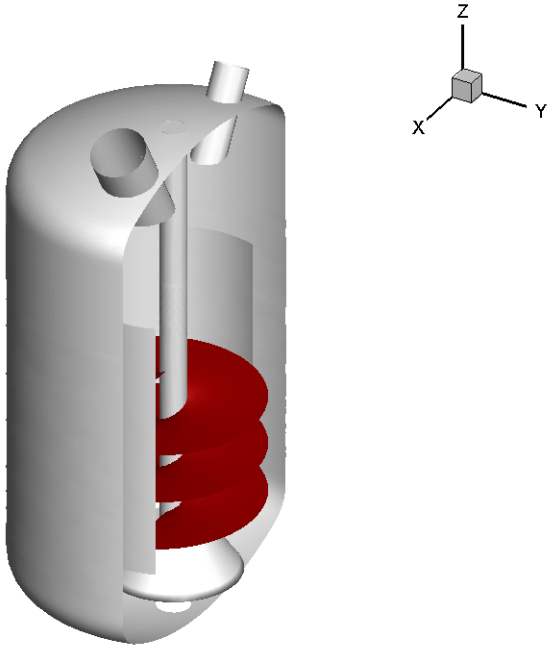
B. Click the Shade Color button.



C. Choose Dark Red in the **Select Color** dialog.



Your plot will look as follows:



Step 6 Adjust the Lighting Effects

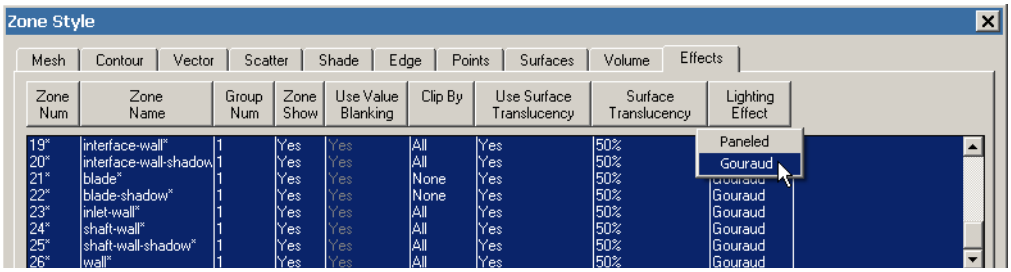
By default, Tecplot 360 sets the shading to be continuous (Gouraud), which provides a cleaner image. For better performance when working with slower processor speeds, set the shading to Paneled. The paneled shading option uses a single shading value for each cell.

To adjust the lighting effects:

A. Switch to the Effects page of the **Zone Style** dialog.

B. Click the Zone Num button and choose “Select All” from the menu.

C. Click the Lighting Effect button, and choose “Paneled” or “Gouraud” from the menu.



D. Close the **Zone Style** dialog.

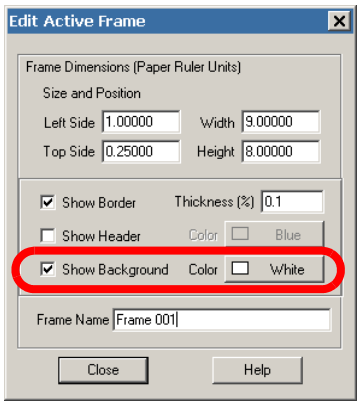
Especially if you wish to export your plot, you will probably wish to retain the default Gouraud shading for better image quality.

Step 7 Change the Frame Background Color

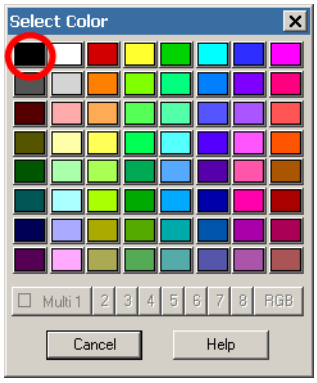
In order to contrast the bright tones of the image, we would like to change the frame background color.

A. Go to **Frame>Edit Active Frame**.

B. Toggle-on “Show Background” if it is toggled-off.

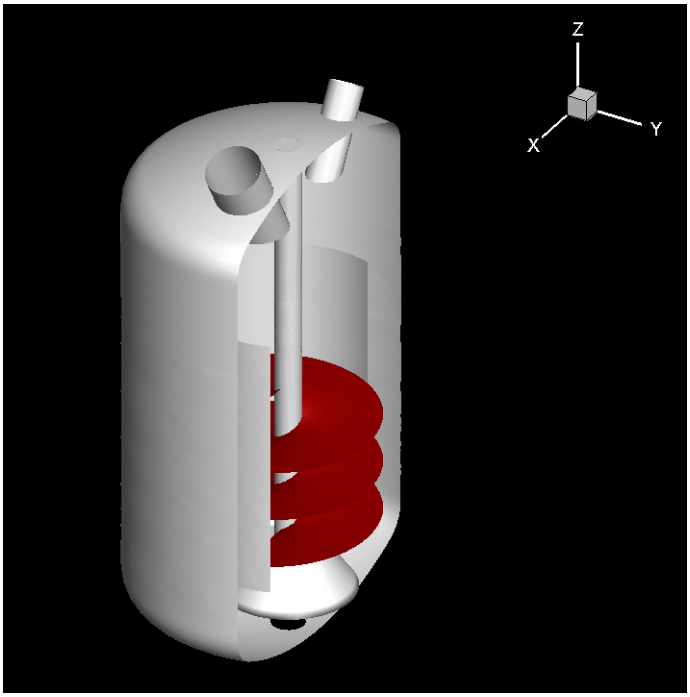


C. Click the Color box and choose Black from the color palette.



D. Click Yes in the confirmation box, and close the **Edit Active Frame** dialog.

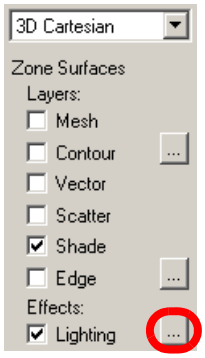
The result will look as follows:



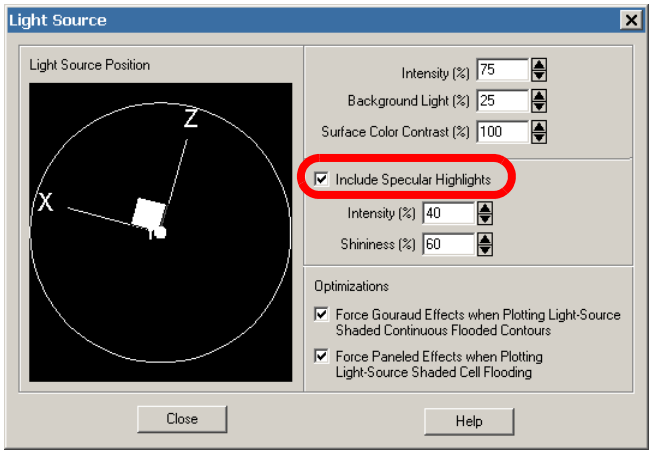
By clicking “yes” in the confirmation box when changing the background color of the frame to a dark color, you allow objects such as the 3D orientation axis to change from black to white.

Step 8 Adjust the Lighting

We can also adjust the lighting, by using the **Light Source** dialog. Open this dialog by clicking the Details button next to Lighting in the Sidebar (or you can choose “Light Source” from the **Plot** menu)



Specular Highlights are toggled-on by default and can be adjusted in the **Light Source** dialog.



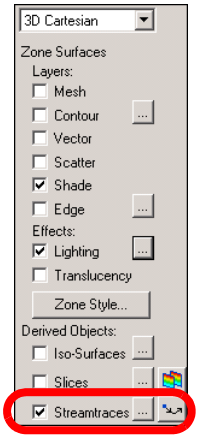
You can interactively move the light source position by dragging the circle within the Light Source Position region of the dialog.

E. Close the **Light Source** dialog.

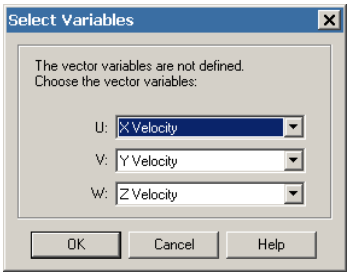
Step 9 Activate Streamtraces

We would like to visualize the internal flow of the mixer using streamtraces. You can access streamtrace controls and the **Streamtrace Details** dialog from the Sidebar (or by choosing “Streamtraces” in the **Plot** menu).

A. Toggle-on “Streamtraces” in the Sidebar.



B. Accept the default vector variables in the **Select Variables** dialog by clicking OK.



After the vector variables have been defined, you can change them by going to **Plot > Vector > Variables**.

Step 10 Position Streamtraces Using the Add Streamtraces Tool

Similarly to slices, streamtraces can be added interactively, using the Add Streamtraces tool, or by using the Position page of the **Streamtrace Details** dialog (accessible by choosing “Streamtraces” in the **Plot** menu, or clicking the Details button in the Sidebar).

A. Click the Add Streamtraces tool in the Sidebar.

When the Add Streamtraces tool is selected, the cursor will change from a pointer to a crosshair (+).

B. Click or drag within your plot. This will add streamtraces to your plot.

Step 11 Position Streamtraces Using the Streamtrace Details Dialog

[OPTIONAL] To more closely match the internal flow layout included in your installation, place streamtraces precisely using the **Streamtrace Details** dialog.

A. Open the **Streamtrace Details** dialog by clicking the Details button to the right of Streamtraces in the Sidebar.

B. Specify the following settings:

i. Select the “Enter XYZ Positions” radio button.

ii. For the Streamtrace Start Position, set the following variable values:

- X = “- 0.103”
- Y = “0”
- Z = “0.23”

C. Click the Create Stream(s) button.

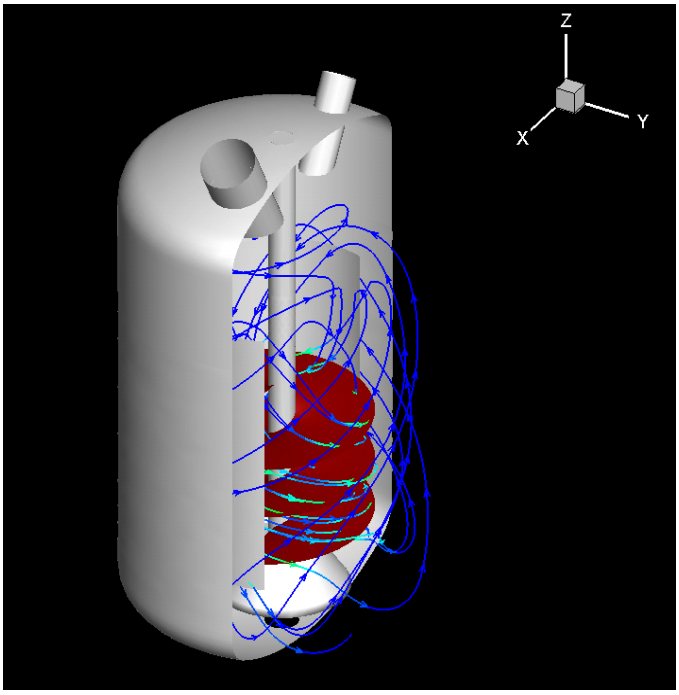
D. To contour the streamtraces according to the variable “Turbulent Kinetic Energy”, set the following:

- Switch to the Line page of the **Streamtrace Details** dialog, and choose “Multi 1” as the Line Color.
- Also on the Line page, set the Line Thickness % to 0.3 for better visibility.
- Open the **Contour...Details** dialog by clicking the Details button next to the Contour toggle in the Sidebar, and choose “Turbulent Kinetic Energy” from the menu. Close this dialog.

E. Close the **Streamtrace Details** dialog.

You can see that the clipping slice does not clip the added streamtraces. To instruct the streamtraces to obey the clipping slice(s), toggle-on “Obey Clipping Settings” on the Integration page of the **Streamtrace Details** dialog.

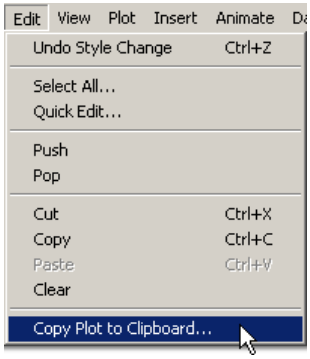
Your final plot will look as follows:



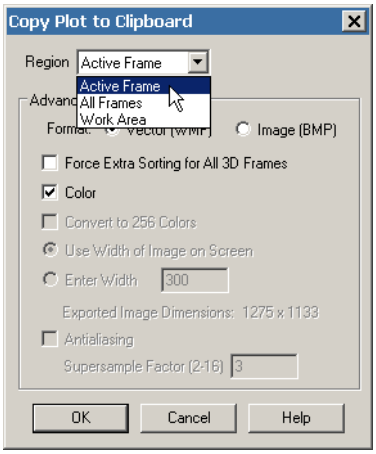
Step 12 Copy Plot to Clipboard

Now that your plot is complete, you can copy it to the clipboard and paste it into other software (such as into a PowerPoint project).

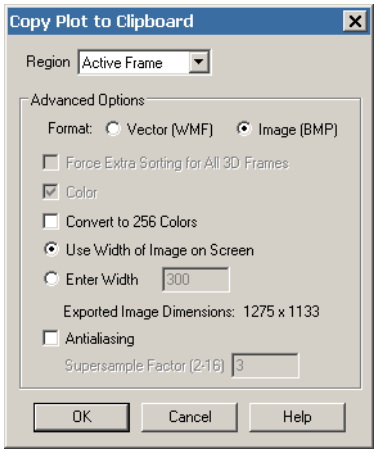
A. Choose “Copy Plot to Clipboard” from the **Edit** menu.



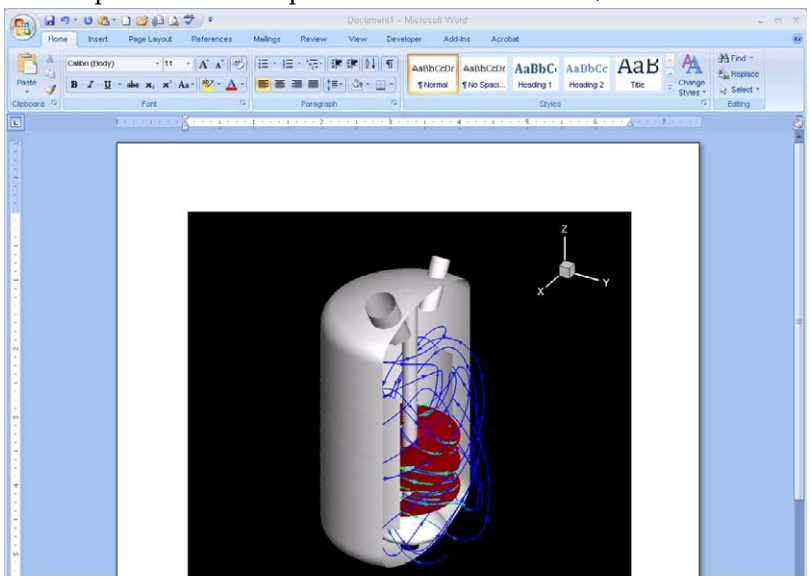
B. In the **Copy Plot to Clipboard** dialog, choose “Active Frame” from the **Region** menu.



C. Choose the “Image (BMP)” radio button.



D. Click OK. The plot can now be pasted into other software, such as Microsoft® Word®.



4 - 5 Conclusion

This concludes the Internal Flow tutorial. Having completed this tutorial, you should now be familiar with using the Tecplot 360 features of surface clipping, lighting effects, and streamtraces. Refer to the User's Manual for details regarding any of the features discussed in this tutorial.

5

Fluid Structure Interaction Tutorial

5 - 1 Introduction

This tutorial illustrates how to use Tecplot 360 to visualize a Fluid Structure Interaction (FSI). We will produce an animation that shows the movement of a nonlinear baffle as it is affected by fluid pressure. To create this plot, we will implement several Tecplot 360 features, including contours, color maps, streamtraces, and animation.

5 - 1.1 Background Information

The FSI used in this tutorial illustrates the transient deformation of a nonlinear baffle due to fluid flow within a duct. The baffle has a height of 0.1 meters and a thickness of 0.005 meters, and the duct is twice the height of the baffle. The fluid flow enters the duct from the left (inlet) and exits on the right (outlet) over time[1]. The baffle was modeled in Abaqus, and the fluid data was calculated using FLUENT.

5 - 1.2 Tutorial Summary

The tutorial should take approximately 30-40 minutes to complete.

The supporting data files for this tutorial are available for download at: http://download.tecplot.com/360/tutorials/fsi.zip. The remainder of the tutorial assumes that you have downloaded the zip file and placed its contents in: *$TEC_360_2013R1\tutorials\fsi*, where $TEC_360_2013R1 is your installation directory for Tecplot 360. For Windows users, this is typically C:\Program Files\Tecplot\Tec360 2013R1.

We strongly recommend downloading the files to your computer's local drive and not to a network drive. Depending on the speed of your network, accessing the files over the network could dramatically increase the loading time into Tecplot 360.

5 - 2 Getting Started

This portion of the tutorial illustrates how to load an ABAQUS Output Database file and a set of FLUENT files.

1. The Tecplot 360 FSI tutorial is based on a tutorial created by SIMULIA®. For more information on SIMULIA, visit www.simulia.com.

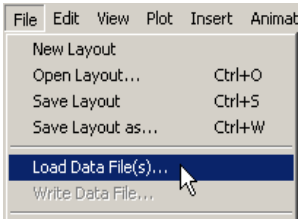
Step 1 Load an Abaqus Output Database File

The first step is to load an Abaqus file into Tecplot 360.

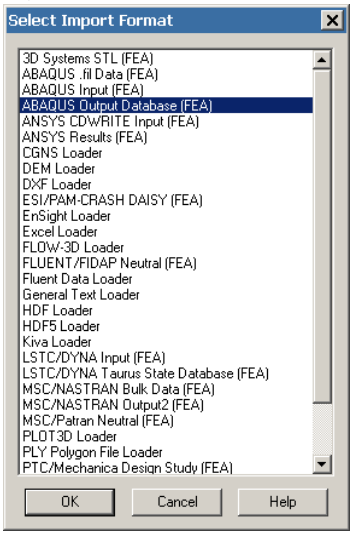
The Abaqus Output Database loader is available on Windows® platforms only. If you are on a Windows machine, follow the steps below. Otherwise, proceed to the Section "Non-Windows Platforms" on page 81 to load a Tecplot 360 *.plt* file of the same data.

For Windows Users

A. Go to **File>Load Data File(s)**.

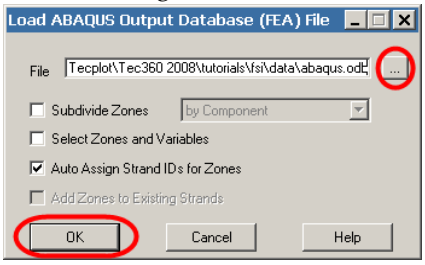


B. In the **Select Import Format** dialog, choose the "ABAQUS Output Database (FEA)" loader; then click OK.



C. In the **Load ABAQUS Output Database (FEA) File** dialog:

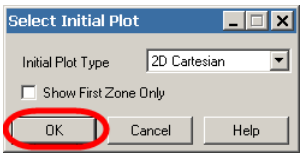
i. Click the Browse button to navigate to *$TEC_360_2013R1\tutorials\fsi*.



ii. Choose the file named: *abaqus.odb*.

iii. Accept the default options and click OK to finish the loading process.

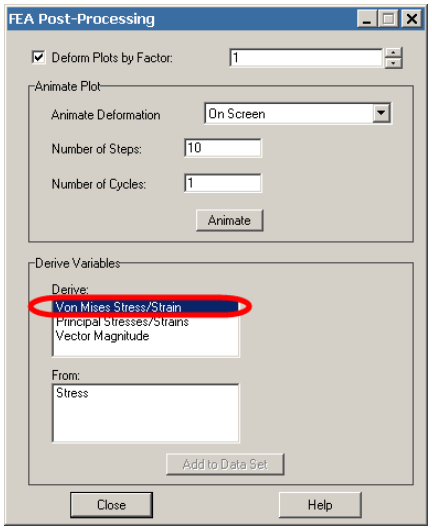
D. The **Select Initial Plot** dialog will launch automatically. Make sure "2D Cartesian" is selected in the **Initial Plot Type** menu, and click OK.



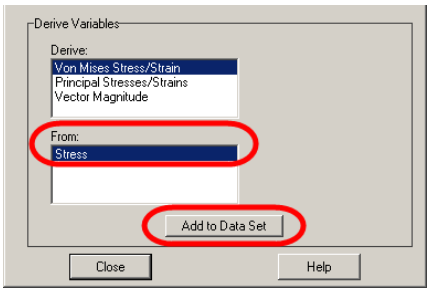
E. The **FEA Post-Processing** dialog will be launched automatically[1].

F. In this dialog:

i. Choose "Von Mises Stress/Strain" in the Derive Variables region.



ii. Select "Stress" in the From: list. This will allow you to derive the Von Mises Stress from the Stress variable already included in the data file.



iii. Click the Add to Data Set button to add the Von Mises Stress to your data.

iv. Click Close to close the dialog.

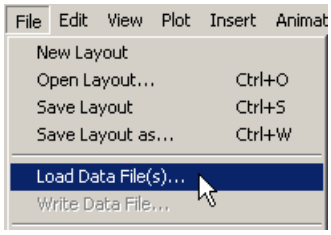
v. Proceed to Step 2 Load a Set of Transient Fluent Data Files on page 83.

Non-Windows Platforms

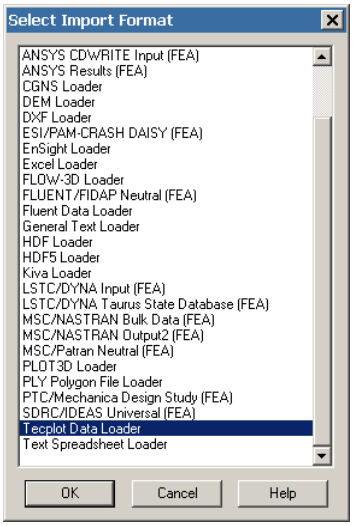
For non-Windows users, we have provided a *.plt* file of the Abaqus data file. This file was created using the "Write Data File" option in the **File** menu[2].

1. You can access this dialog at any time via the **Tools** menu.

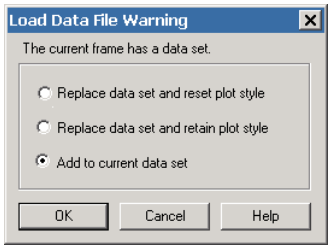
A. Go to **File>Load Data File(s)**.



B. In the **Select Import Format** dialog, select "Tecplot Data Loader" from the list, and click OK.

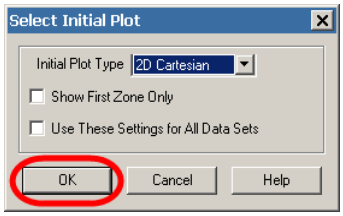


C. Choose "Add to current data set" in the **Load Data File Warning** dialog.



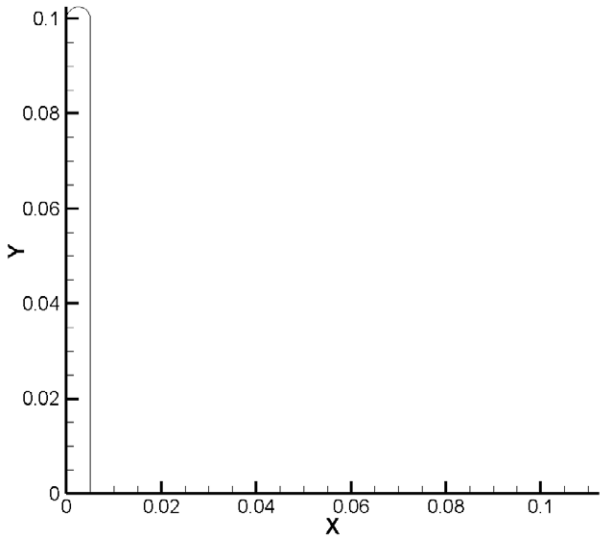
D. In the **Tecplot Data Loader** dialog, navigate to *$TEC_360_2013R1\tutorials\fsi\data* and choose the file named *abaqus_data.plt*. Click Open.

E. In the **Select Initial Plot** dialog, be sure that "2D Cartesian" is selected and click OK.



2. The Von Mises Stress that was calculated in Step 1: For Windows Users, was calculated for this data prior to writing out the data file.

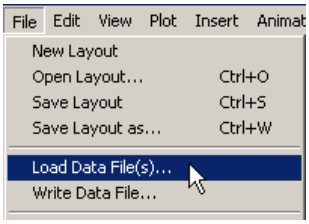
The initial plot will look as follows:



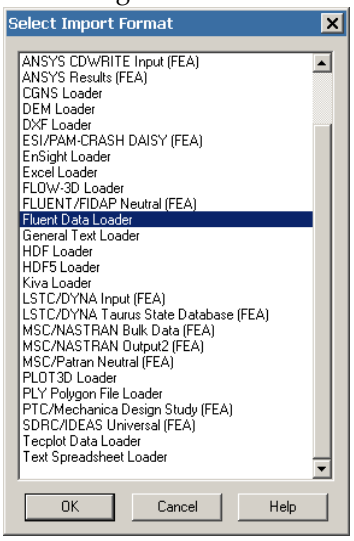
Step 2 Load a Set of Transient Fluent Data Files

Now that we have the Abaqus file loaded (which we will refer to as the “baffle”), we will append our dataset with a set of transient FLUENT data files.

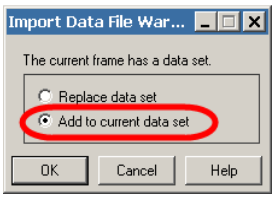
A. Go to **File>Load Data File(s)**.



B. In the **Select Import Format** dialog, select “Fluent Data Loader” from the list and click OK to launch the **Fluent Data Loader** dialog.

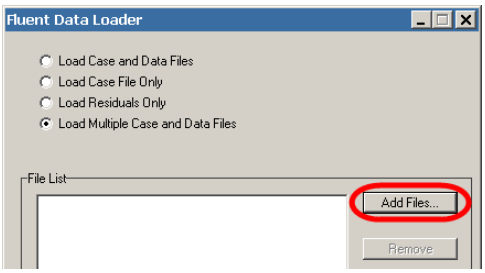


C. In the **Import Data File Warning** dialog, choose the "Add to current data set" radio button. Then click OK.



D. In the **Fluent Data Loader** dialog:

i. Click the Add Files button.



E. In the **Read Case and Data File** dialog:

i. Navigate to `$TEC_360_2013R1\tutorials\fsi\data\f`.

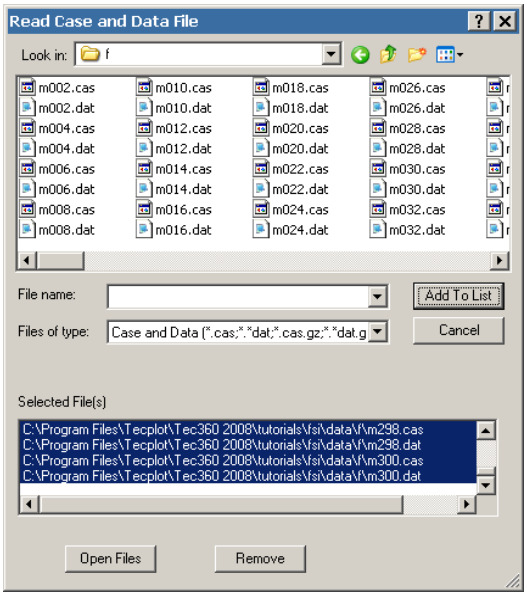
ii. Select all available files (by using the SHIFT key or pressing CTRL-A). The FLUENT data consists of 150 time steps with one case (*.cas*) file and one data (*.dat*) file for each time step.

iii. Click the Add To List/Add button (Windows/UNIX).

On certain machines, the animation process may dramatically slow if all the data files are loaded. To avoid a lengthy animation series, we recommend loading a subset of the time steps using the Remove button.

Be sure that you remove both the *.cas* file and the *.dat* file for any time step that you choose to omit.

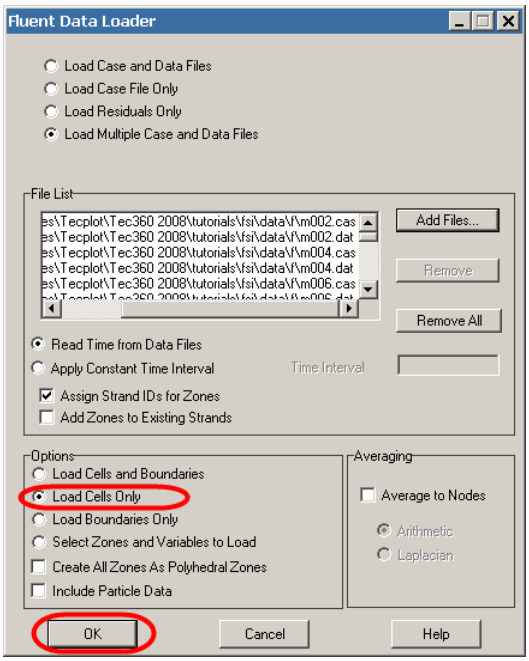
iv. Click the Open Files/OK button (Windows/UNIX).



F. In the **Fluent Data Loader** dialog:

i. Select the “Load Cells Only” radio button.

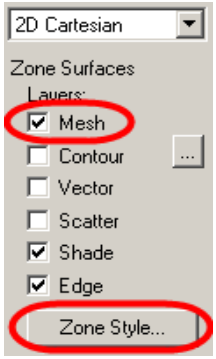
ii. Click OK in the **Fluent Data Loader** dialog to finish the loading process. Depending on the number of time steps you chose to load, it may take a few minutes to load all the data.



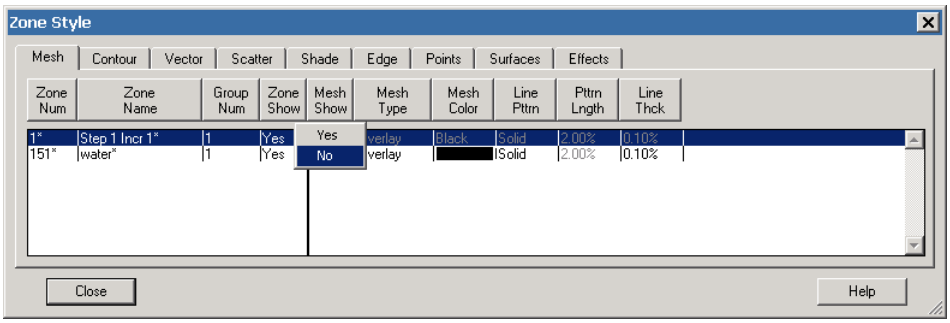
Step 3 Modify the Plot View

Now we will modify the plot view to better see the “water” zone by turning on “Mesh” and changing the zone color.

A. Toggle-on “Mesh” in the Sidebar, and then click the Zone Style button to launch the **Zone Style** dialog. (The **Zone Style** dialog can also be launched at any time by double-clicking directly on the plot.)



B. To better differentiate the zones, choose the baffle (Step 1 Incr 1) and choose “No” from the Mesh Show button.



C. Next we will change the “water” zone color by doing the following.

i. Click on the “water” zone to select it in the **Zone Style** dialog.

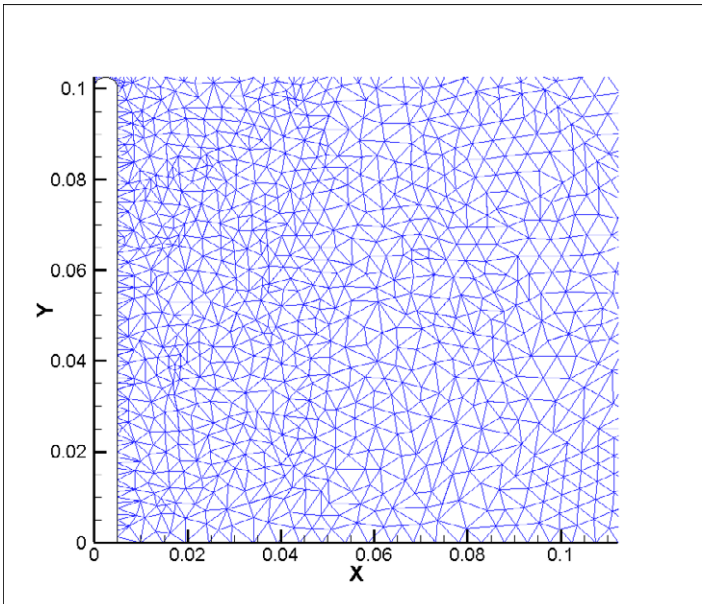
ii. Click the Mesh Color button on the Mesh page of the **Zone Style Dialog**.

An asterisk (*) next to a “Zone Num” or “Zone Name” indicates that the zone is a transient zone. The zone number is linked to the active time step. In this case, Zone Name describes the object. Press the Play button in the Sidebar and watch the zone numbers change with the time step. Refer to Section 7 - 1 “Field Plot Modification - Zone Style Dialog” in the User’s Manual for more information about transient zones.

iii. Select Blue from the **Select Color** dialog.

D. Click Close.

E. The plot will look as follows:

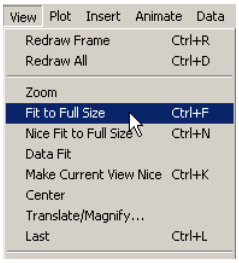


5 - 3 Adjusting the View

Notice that the FLUENT data surrounds the baffle, but that only one side is visible. Even though the baffle deforms to the right, we would like to see all the surrounding fluid flow. To accomplish this, we will adjust the view.

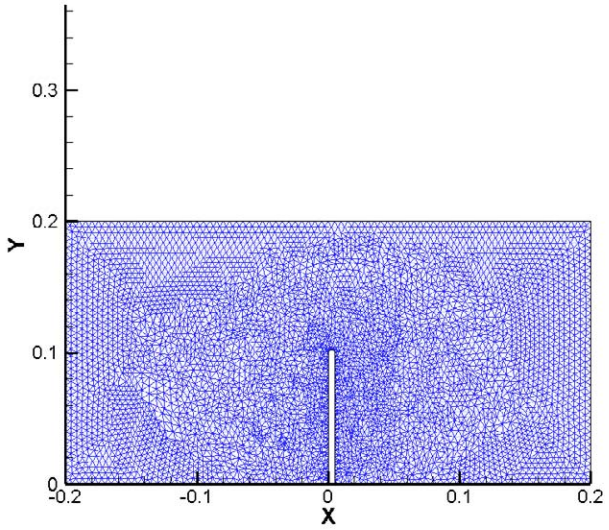
Step 4 Change the View Setting

Choose **View>Fit to Full Size** to fit the plot to the frame.



Tecplot 360 provides several methods of adjusting your current view. Experiment with the options under the **View** menu to see how you can change the plot view.

The plot will look as follows:

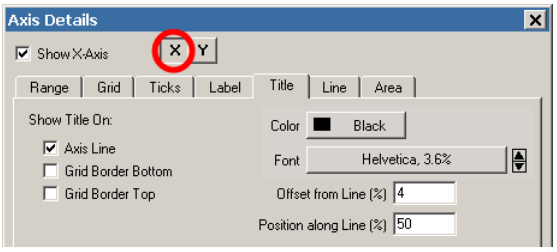


Step 5 Modify the Axis

We have modified the view of the plot to fit the screen, but to make the baffle more visible, we will zoom in on the X-axis.

A. Double-click on the X-axis to launch the **Axis Details** dialog[1].

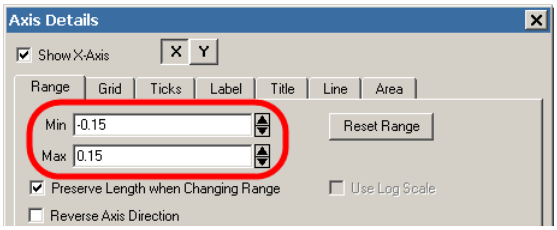
B. In the **Axis Details** dialog, click the X button to adjust the range for the X-axis.



C. On the Range page of the **Axis Details** dialog:

i. Enter "-0.15" for the Min.

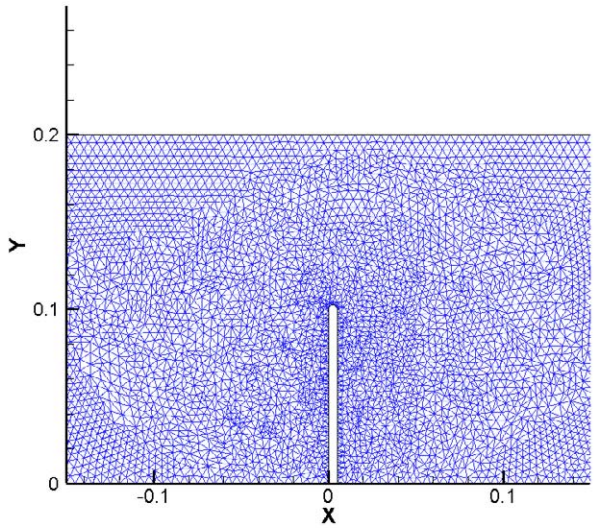
ii. Enter "0.15" for the Max.



iii. Click Close.

1. The **Axis Details** dialog can also be accessed via the **Plot** menu.

The modified plot will look as follows:



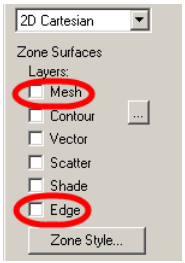
5 - 4 Working with Contour Groups

Now that the data is loaded, we would like to examine the effect of the pressure in the fluid body (FLUENT data) on the stress component of the baffle (Abaqus/Tecplot 360 data). To achieve this, we will create a contour plot and animate it. Displaying contours will give us a visual representation of the stress and pressure variables and how they interact with each other.

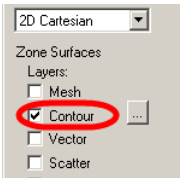
Step 6 Display Contours

We will now activate the contour layer.

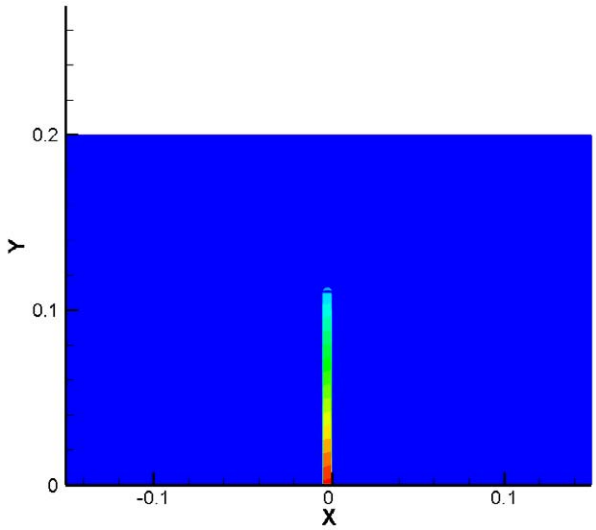
A. Toggle-off “Mesh” and “Edge” in the Sidebar. Turning off the Mesh and Edge layers will improve visibility of the Contour layer.



B. Toggle-on “Contour”.



Your plot will look as follows:



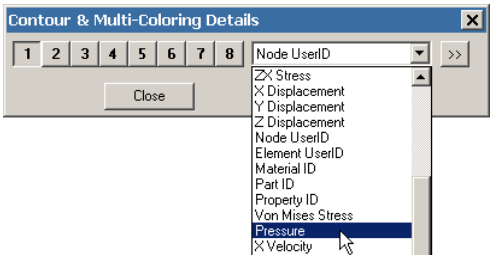
Step 7 Set the Contour Variable for Contour Group 1

Because we would like to create a contour plot of two separate variables simultaneously, we need to work with contour groups. A contour group is used to link a set of contour parameters (for example, the contour variable and the contour levels) to one another. Contour groups allow us to apply different levels, values, and colors to specific groups of data. We can also link a color map to a contour group and customize the color map attributes to make them easy to recognize.

The first contour group we will create will be used to illustrate the pressure in the fluid body.

Open the **Contour Details** dialog by clicking the Details button to the right of "Contour" in the Sidebar.

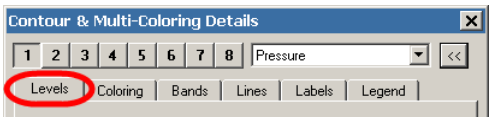
In the **Contour Details** dialog, click the 1 button to set the first contour group. Choose "Pressure" from the menu.



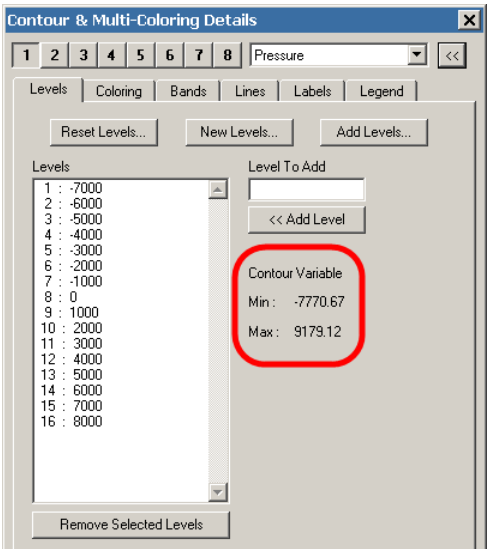
Step 8 Modify the Contour Levels for Contour Group 1

A. Expand the **Contour Details** dialog by clicking the >> button. This will fully open the **Contour Details** dialog, so we can work with the other options in the dialog.

B. To modify contour levels, switch to the Levels page of the dialog.



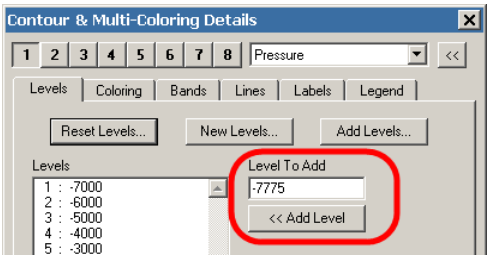
On the Levels page of the **Contour Details** dialog, a list of the current contour levels is displayed in the Levels region. In addition, the minimum and maximum values of the Contour Variable are displayed on the right-hand side of the page. We will use those values to add two more contour levels.



There are several methods for adding and deleting contour levels. In this tutorial, we will demonstrate using the << Add Level button. Alternatively, you may use the Reset Levels, New Levels, or Add Levels buttons located at the top of the page.

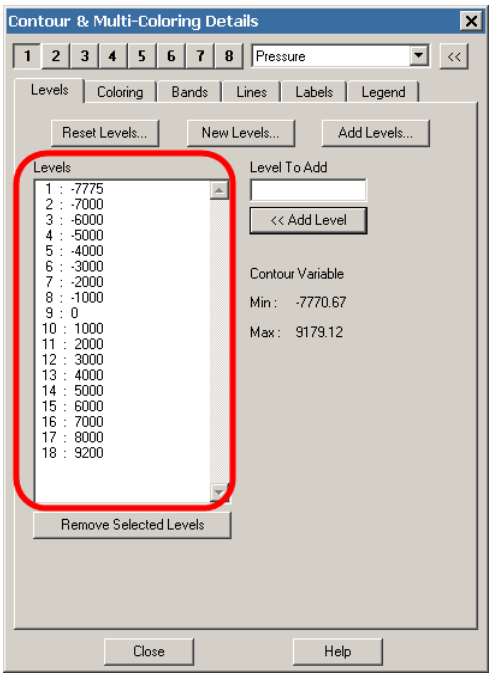
C. In the "Level To Add" field:

i. Enter: "-7775", and click the << Add Level button.



ii. Enter: "9200", and click the << Add Level button.

The new levels will be listed in the Levels region:

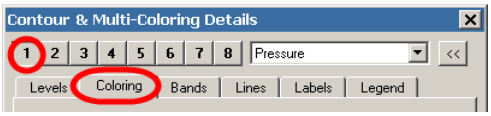


Step 9 Modify the Color Map for Contour Group 1

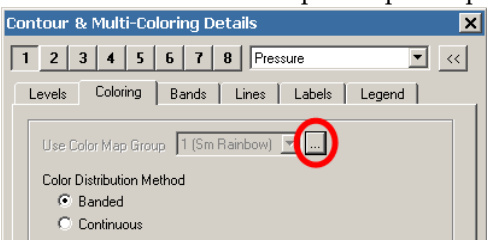
Because we are using two contour variables simultaneously, we would like to alter the coloring of each map to ensure the different contour groups are easily distinguishable. For this tutorial, we will use a GrayScale color map for Contour Group 1 and a Rainbow color map for Contour Group 2.

We will start with the pressure variable.

A. With the 1 button selected on the **Contour Details** dialog, switch to the Coloring page.



B. Click the Details button next to "Use Color Map Group" to open the **Color Map** dialog[1].

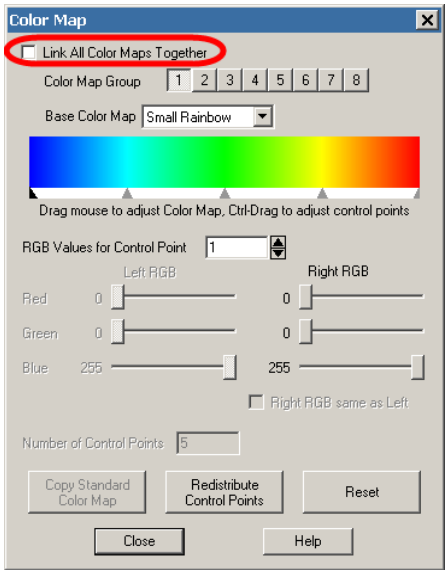


C. Tecplot 360 allows you to work with up to eight different color maps. By default, the maps are linked together (so that changing one color map changes the other color maps).

In the **Color Map** dialog:

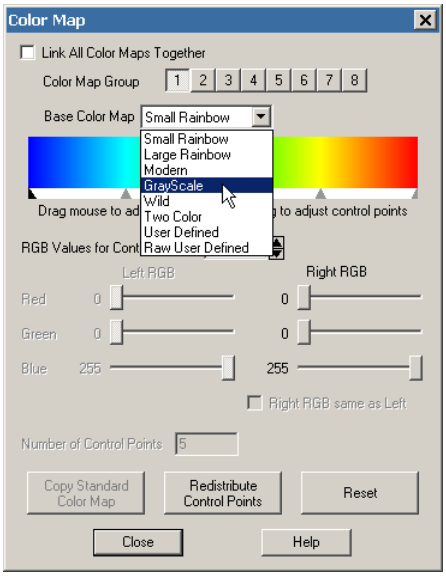
1. The **Color Map** dialog can also be launched via the **Options** menu.

i. Toggle-off "Link All Color Maps Together" to allow you to modify the color maps separately.



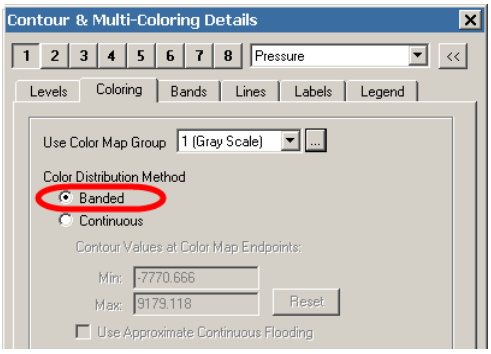
Unlinking color maps also results in separate color maps being assigned to each of the contour groups. Color Map 1 is assigned to Contour Group 1, Color Map 2 is assigned to Contour Group 2, and so on.

ii. Choose "GrayScale" from the **Base Color Map** menu.

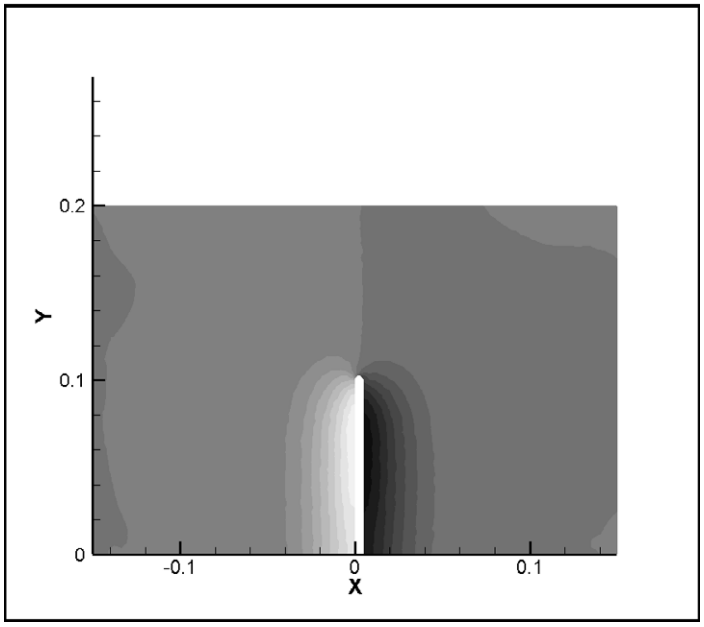


D. Click Close.

E. On the Coloring page of the **Contour Details** dialog, make sure that "Banded" is selected for the "Color Distribution Method". This will allow you to easily identify the different contour levels on the plot.



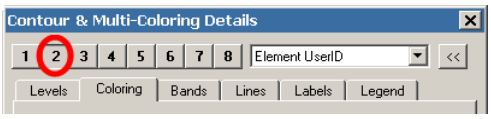
The plot will look as follows:



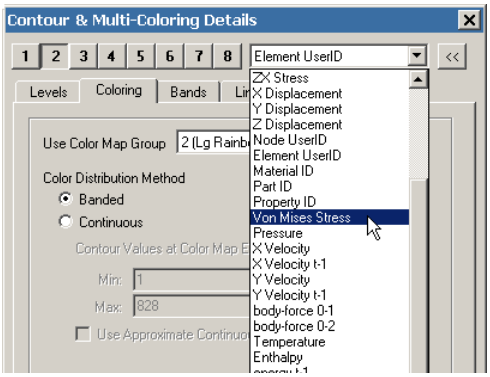
Step 10 Set the Contour Variable for Contour Group 2

We will now set up the second contour group (Contour Group 2) to illustrate the Von Mises Stress on the baffle.

A. In the **Contour Details** dialog, click the 2 button to switch to Contour Group 2.



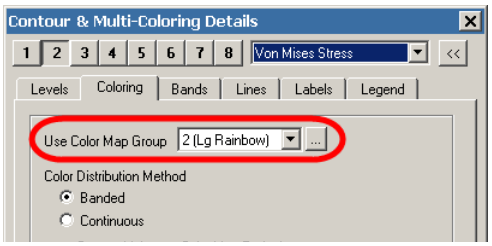
B. Choose “Von Mises Stress” from the menu.



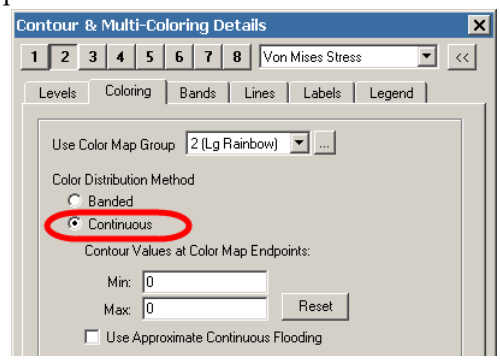
Step 11 Modify the Color Map for Contour Group 2

A. With the 2 button selected, switch to the Coloring page of **Contour Details** dialog.

B. On the Coloring page of the dialog, Color Map “Large Rainbow” (Lg Rainbow) will automatically be set to the second color map.



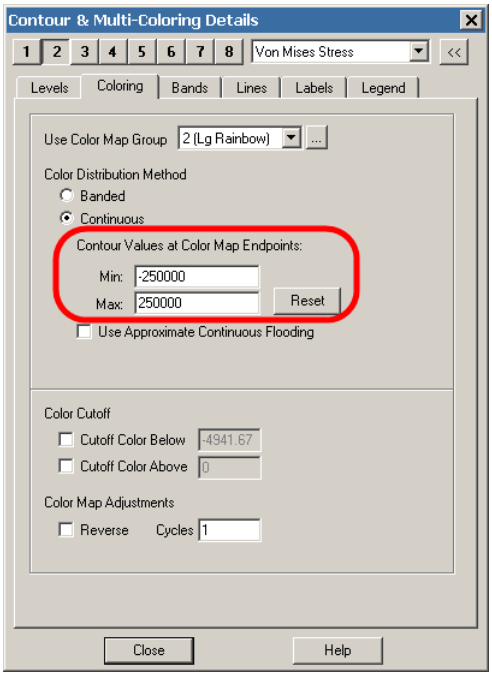
C. In the **Contour Details** dialog, choose the “Continuous” radio button. This will create a continuous color map between each contour level.



D. In the “Contour Values at Color Map Endpoints” region:

i. Enter “-250000” for Min.

ii. Enter “250000” for Max.



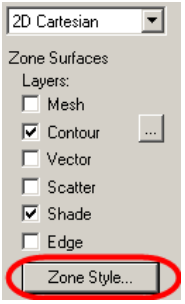
You may see Min and Max values at zero. This will occur if the Von Mises Stress is not defined at the current time step. The Contour Values will automatically adjust with the time step.

E. Click Close.

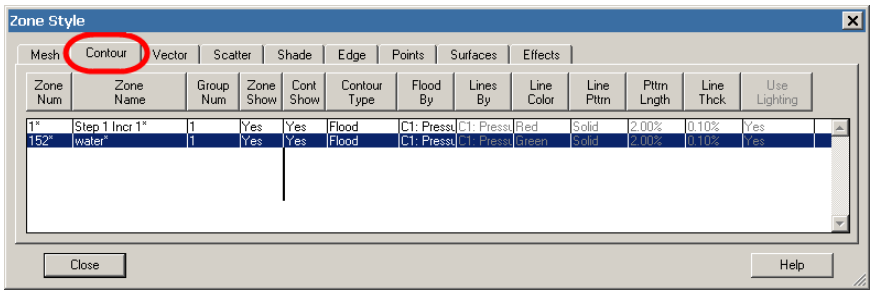
Step 12 Change the Contour Group Assigned to a Zone

By default, all zones are using Contour Group 1. This means we do not currently see Contour Group 2 on our plot. We will use the **Zone Style** dialog to assign Contour Group 2 to the baffle (Abaqus zone).

A. Launch the **Zone Style** dialog by clicking the Zone Style button in the Sidebar or by double-clicking on your plot.



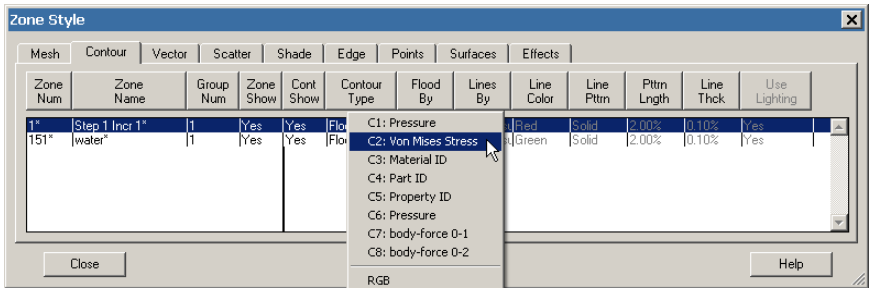
B. Switch to the Contour page of the dialog.



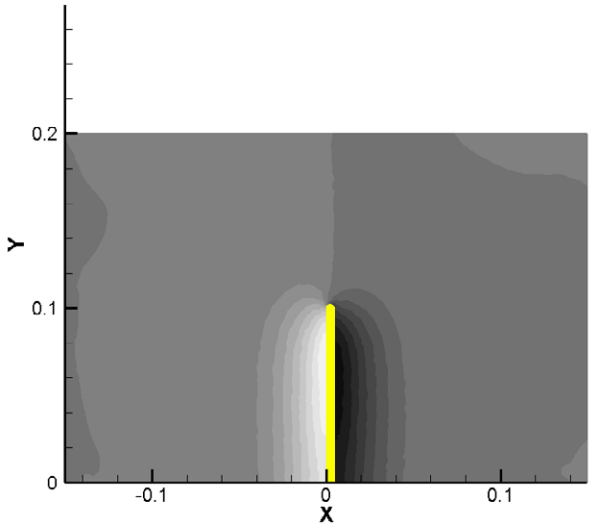
On the Contour page, you can see a list of all of the active zones for this time step. The first zone, Step 1 Incr 1, is the baffle (Abaqus data). The remaining zone is the fluid body (FLUENT data).

i. Choose the baffle (Step 1 Incr 1).

ii. Select "C2: Von Mises Stress" from the Flood By button.



C. Click Close. The plot will look as follows:



5 - 5 Customizing Contour Legends

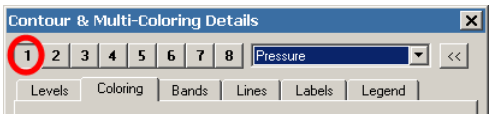
Contour legends are used as a key to define the numerical value of the variable tied to each color in the plot. The attributes of the legend of each contour group can be specified within the **Contour Details** dialog.

Step 13 Add Contour Legends

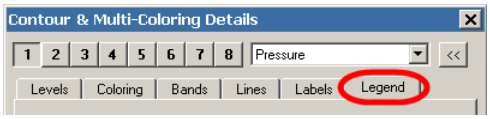
Now that we have defined our contour groups, we will add contour legends to the plot. One legend will be for the pressure variable and the other will be for Von Mises Stress.

A. Launch the **Contour Details** dialog by clicking the Details button next to "Contour" in the Sidebar.

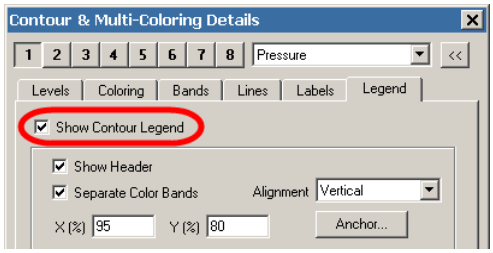
B. Click the 1 button in the **Contour Details** dialog to begin making changes for the first contour group.



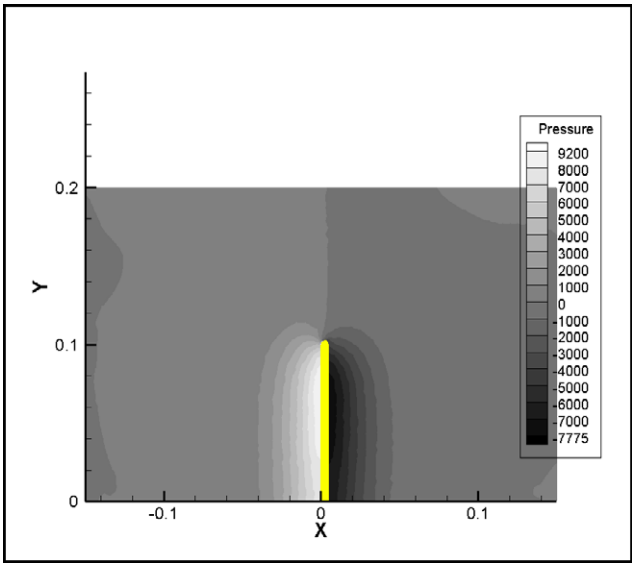
C. Switch to the Legend page of the dialog.



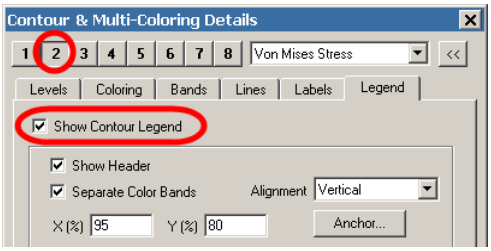
D. On the Legend page, toggle-on "Show Contour Legend".



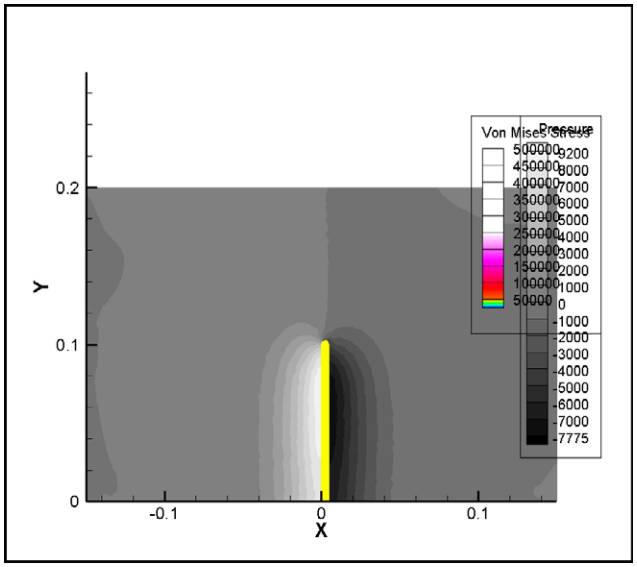
A contour legend will appear on the plot:



E. Choose the 2 contour group to add a legend for the second contour group, Von Mises Stress. On the Legend page, toggle-on "Show Contour Legend".



The plot will look as follows:

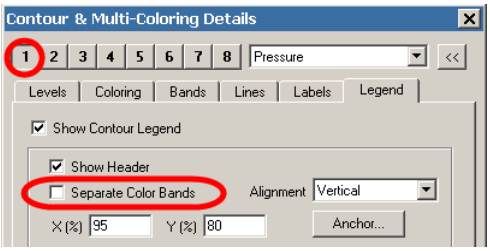


Step 14 Modify the Contour Legends

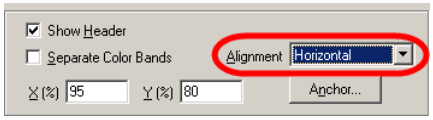
As you can see, the contour legends overlap one another. We will modify their size, placement, and appearance so that they are easier to read. As you work your way through the following steps, note how the legends change.

A. Choose 1 in the **Contour Details** dialog, and set the following:

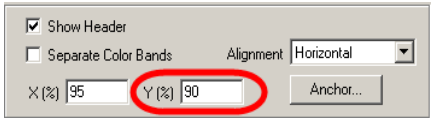
i. Toggle-off "Separate Color Bands" to remove the level lines from the legend.



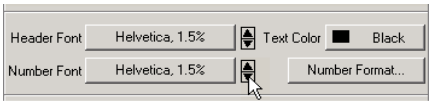
ii. Change the Alignment to "Horizontal".



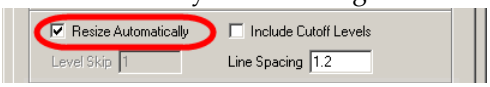
iii. Change the Y% to "90". This places the top left-hand corner of the contour legend up 90% of the frame's height (in relation to the bottom left-hand corner of the frame).



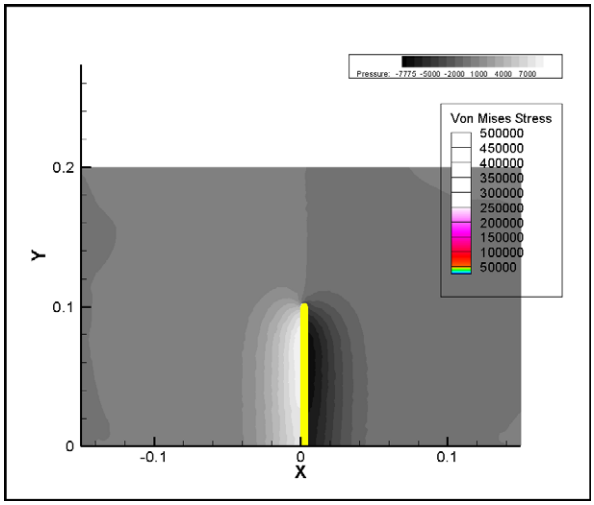
iv. Using the arrows in the Font region, change both the Header Font and the Number Font to "1.5%".



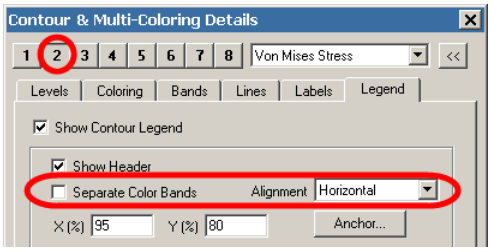
v. Toggle-on "Resize Automatically" to fit the legend to the plot.



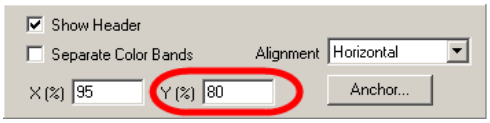
The plot will look as follows:



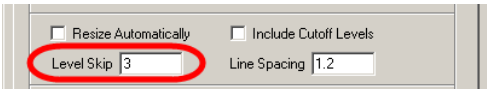
B. Repeat Step i and Step ii in Step A from above for 2 (that is, toggle-off "Separate Colors Band" and set the Alignment to "Horizontal").



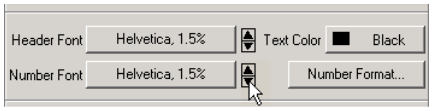
C. We will leave Y% at its default of "80" so that the legend for Contour Group 2 will sit below Contour Group 1.



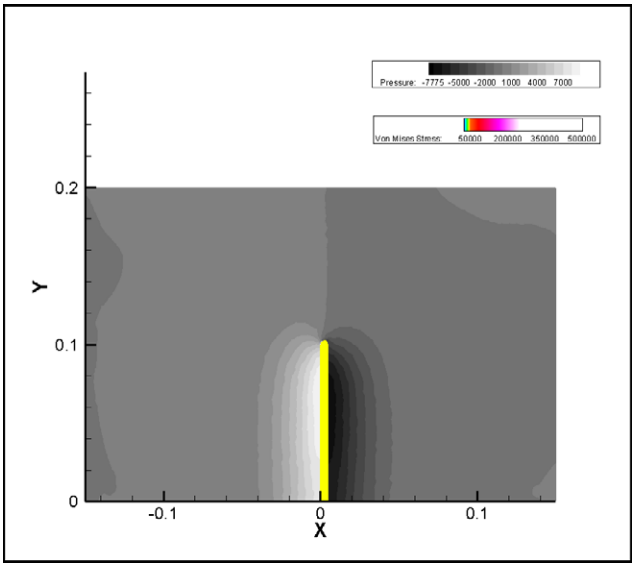
D. Toggle-off "Resize Automatically" and enter "3" into Level Skip to display only every third contour level in the legend, compressing it.



E. Using the arrows in the Font region, change both the Header Font and the Number Font to "1.5%".



F. Click Close. The plot will look as follows:



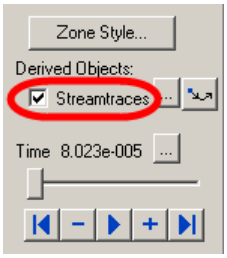
5 - 6 Using Streamtraces

Streamtraces allow you to better visualize how the fluid pressure and the baffle affect one another. There are two main types of streamtraces: surface line streamtraces and volume streamtraces. Volume streamtraces are created in 3D volume zones, while surface line streamtraces are used on 3D or 2D surfaces (such as we have for our plot). For more information about streamtraces, see Chapter 15: "Streamtraces" in the User's Manual.

Step 15 Display Streamtraces

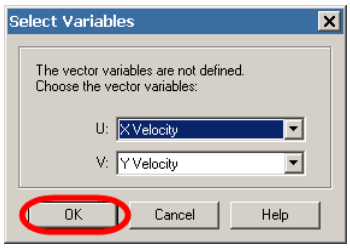
We will now add streamtraces to our plot.

A. Toggle-on "Streamtraces" in the Sidebar. This will allow you to include streamtraces in your plot.

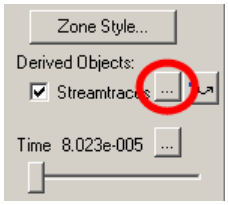


You can turn off "Streamtraces" to hide streamtraces from your plot. Turn on the Streamtraces control in the sidebar to add your previously created streamtraces back into your plot.

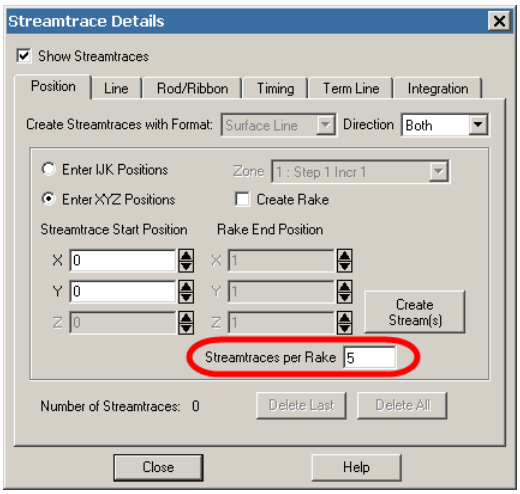
B. The **Select Variables** dialog will be launched automatically. The **Select Variables** dialog allows you to specify the vector variables for your streamtraces. Accept the defaults by clicking OK.



C. Click the Details button next to "Streamtraces" in the Sidebar.

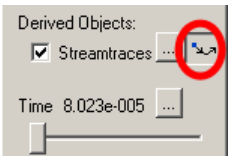


D. On the Position page of the **Streamtrace Details** dialog, change Streamtrace per Rake to "5". This will allow you to draw individual streamtraces on the plot by creating a rake.



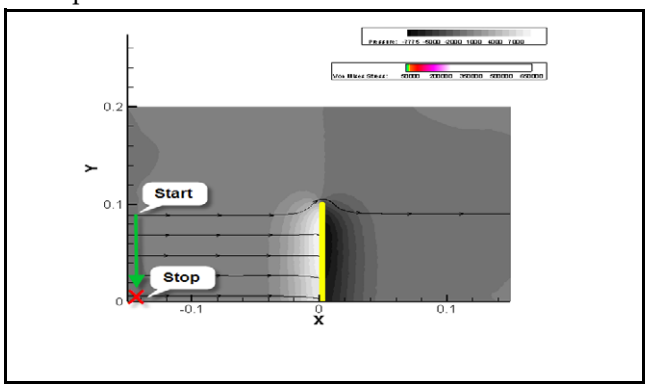
E. Click Close.

F. Click the Add Streamtrace button to the right of "Streamtraces" in the Sidebar[1].

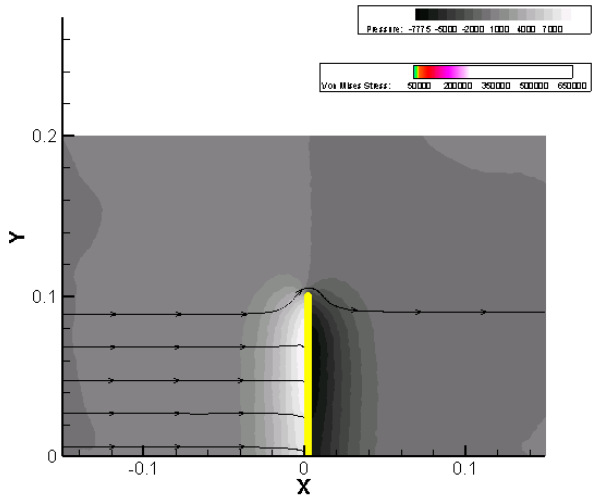


1. Double-clicking this tool will also bring up the Streamtrace Details dialog.

G. On the plot, draw a rake of streamtraces parallel with the X-axis by dragging the mouse vertically down the plot.



Your plot will look similar to the following:



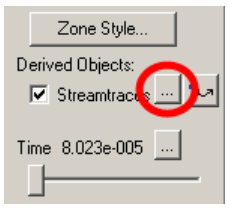
H. You may animate the plot by clicking the Play button in the Sidebar.

The first pass through the animation will take a few minutes because the streamtraces are calculated for each time step. Subsequent passes will be faster because the information has already been calculated.

Step 16 Modify Streamtraces

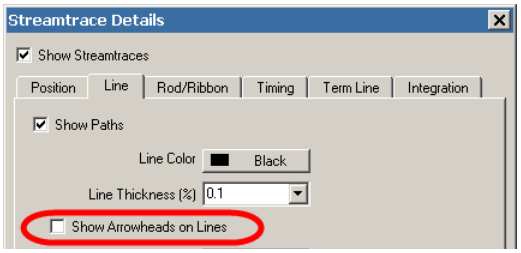
We will now modify the appearance of the streamtraces using the **Streamtrace Details** dialog.

A. Open the **Streamtrace Details** dialog by clicking the Details button to the right of "Streamtraces" in the Sidebar.

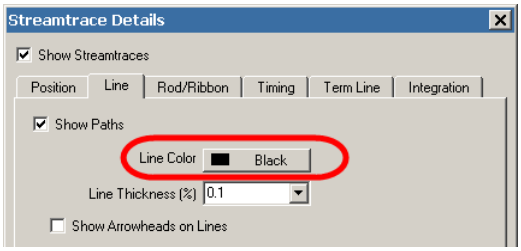


B. Go to the Line page in the **Streamtrace Details** dialog.

C. Toggle-off "Show Arrowheads on Lines". The arrows will disappear, leaving only lines on your plot.

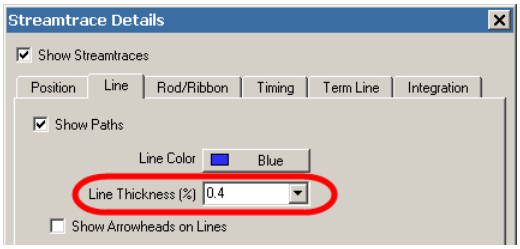


D. Click the button next to Line Color.



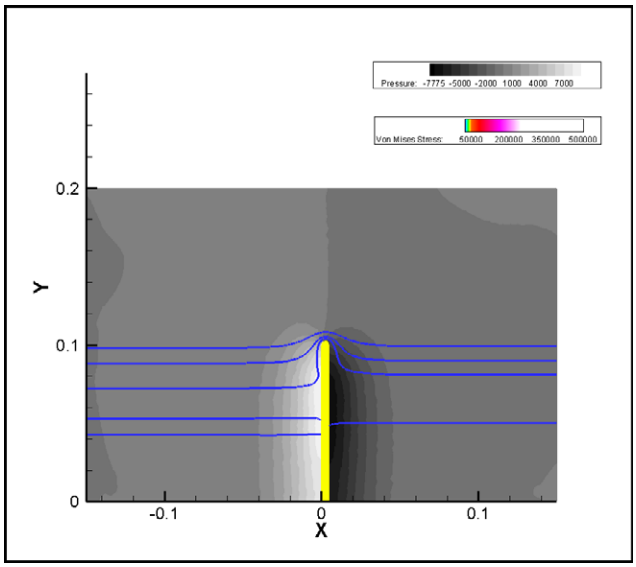
E. In the **Select Color** dialog, choose Blue.

F. In the **Streamtrace Details** dialog, change the Line Thickness (%) to "0.4".



G. Click Close.

The plot will look as follows (when reset):



5 - 7 Animating Your Plot, Displaying Result in PowerPoint

Now that we have set up our plot, we are ready to animate it. Tecplot 360 offers several forms of animation, including: time steps, zones, and streamtraces. In this example, we will animate the data over time. The animation will illustrate the deformation of the baffle due to the changes in the surrounding fluid pressure.

The full animation could take up to five minutes, depending on your machine and RAM. To pause the animation, click the Pause button. (The Play button becomes the Pause button once the animation has started.)

Step 17 Animate on the Screen

You can animate the plot to run through all the time steps or one at a time by:

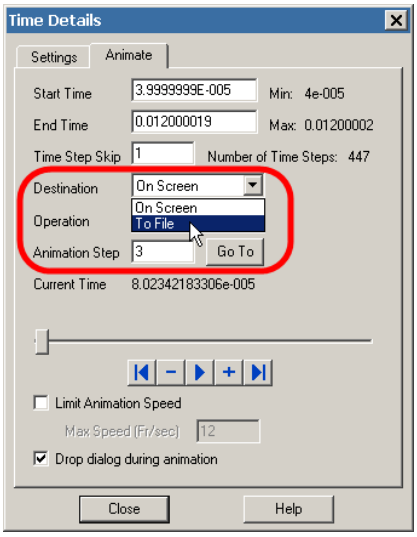
- Clicking the Play button in the Time Region of the Sidebar to animate through all the time steps.
- Clicking the + or - buttons to move through the time steps individually.

Step 18 Export Your Animation to a File

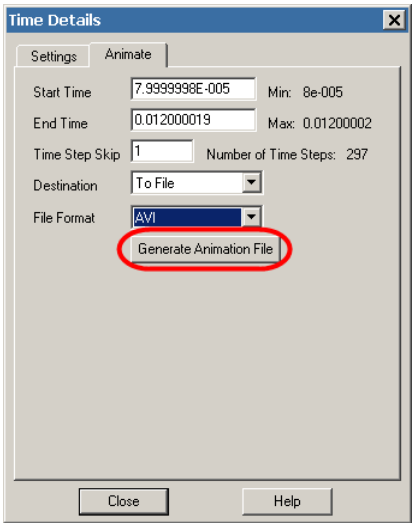
You can export an animation to a file for use in Microsoft PowerPoint® presentations or websites.

A. Launch the **Time Details** dialog by selecting the Details button to the right of Time in the Sidebar.

B. On the Animate page of the **Time Details** dialog, choose "To File" from the **Destination** menu.



C. Choose AVI from the File Format menu, and click the Generate Animation File button.

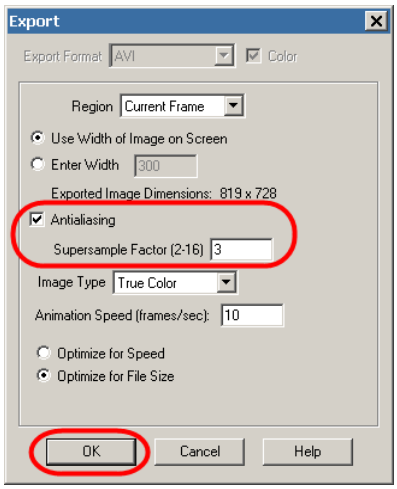


D. The **Export** dialog will launch automatically:

i. Toggle-on "Antialiasing".

Choosing "Antialiasing" will improve the appearance of text in the final output. It is also important to note that when using the "Use width of image on screen" option, the size of the frame window at the time of export will prescribe the video dimensions.

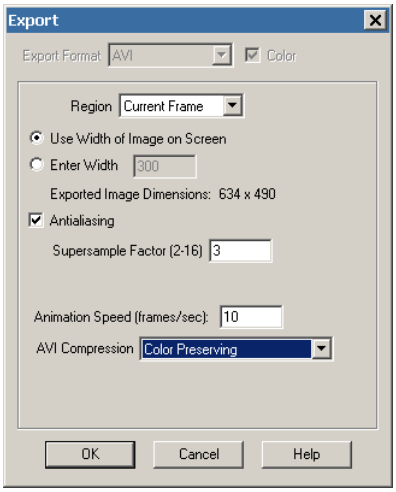
ii. Set “Supersample Factor” to “3”.



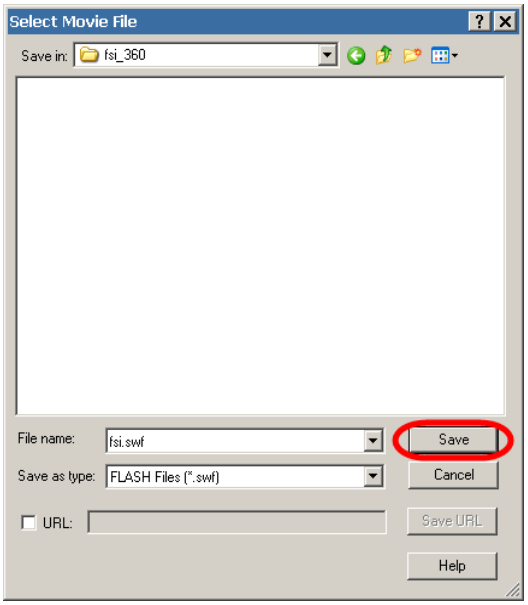
iii. Choose one of the following AVI Compression options:

- **Color Preserving** - This compression option optimizes the color quality of your output.
- **Line Preserving** - This compression option optimizes the quality of lines in your output.
- **Lossless Uncompressed** - This compression option gives the highest picture quality and preserves complete 24-bit pixel information. Note that this option also creates the largest file size.

iv. Accept the other default settings by clicking OK.



E. In the **Select Movie File** dialog, navigate to the desired directory and click Save to save the file as *fsi.avi*.



Tecplot 360 exports the data images to the file so that it may be viewed via another program. This process may take a few moments.

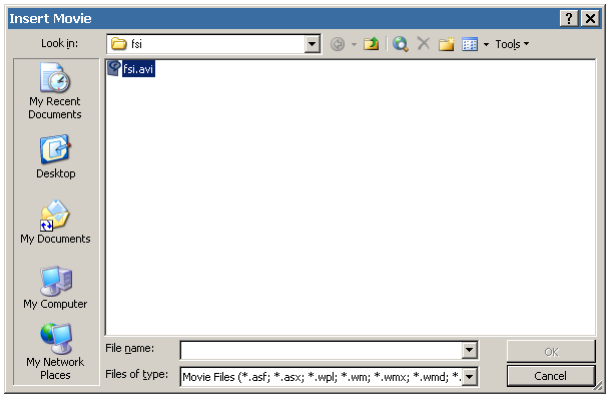
Step 19 Insert a Tecplot 360 AVI Animation into PowerPoint

AVI movies can be inserted directly into PowerPoint. We will now go through the steps involved in inserting an exported Tecplot 360 AVI animation file into PowerPoint.

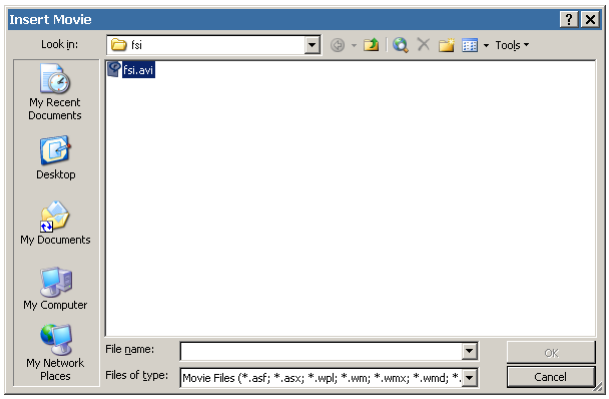
A. Launch PowerPoint, and choose a slide layout of your choosing.

B. **PowerPoint Version 2007** - Go to **Insert>Movie>Movie From File** to launch the Insert Movie dialog.

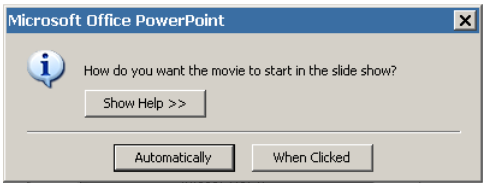
PowerPoint Version 2003 or Older - Go to **Insert>Movies and Sounds>Movie from File** to launch the **Insert Movie** dialog.



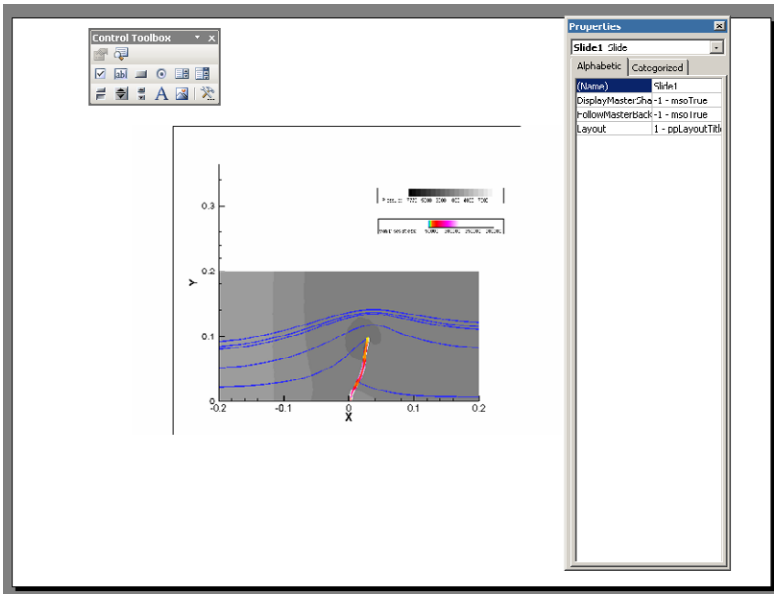
C. Choose your AVI file in the **Insert Movie** dialog.



D. Choose whether you would like your movie to start automatically in your slide show, or by a mouse click.



Once the slide show has played, the animation will show on the slide workspace.



5 - 8 Conclusion

This concludes the FSI tutorial. Having completed this tutorial, you should now be familiar with contour groups, color maps, streamtraces, and animation. Refer to the User's Manual for details regarding any of the features discussed in the tutorial.