Tecplot
Enjoy the View™

TECPLOT 360™
2006

Getting Started Manual

Tecplot, Inc. Bellevue, WA 2006

Tecplot, Inc.
PO 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 documentation: documentation@tecplot.com
For more information, visit http://www.tecplot.com

06-360-02-1

Rev 03/2006

Table of Contents

Chapter 5 Transient Data Tutorial...........................117

Chapter 1 # Introduction

Tecplot allows you to interactively explore, visualize and analyze your data and communicate the results with high quality plots for reports, paper, presentations or web sites.

The user documentation for Tecplot 360 is divided into 7 books:

Getting Started Manual (This document)
User's Manual
Data Format Guide
Tecplot Reference Manual
Quick Reference Manual
Add-on Developer's Kit - User's Manual
Add-on Developer's Kit - Getting Results
Add-on Developer's Kit - Reference Manual
Installation Instructions
Release Notes

The *Getting Started Manual* is intended for beginning Tecplot 360 users. It provides a brief overview of the capabilities of Tecplot, as well as a series of tutorials. The User's Manual provides in-depth descriptions of working with Tecplot's features.The Add-on Developer's Kit manuals and Reference Manual are recommended for advanced Tecplot users.

Each tutorial takes approximately 20-25 minutes to complete.

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

- Internal Flow Tutorial - includes: value blanking and streamtrace animation

- Transient Data Tutorial - includes: animation, working with multiple data files

For in-depth information on the topics covered in the Getting Started Manual, please refer to the *Tecplot User's Manual* available on our website at: www.tecplot.com\support\360_documentation.asp.

Chapter 2 Overview

Tecplot 360 combines the following features to allow you to easily visualize complex physics and relationships:

- XY, Polar, 2D and 3D plotting and animation tools in one unified environment
- Unique multi-frame workspace allowing up to 128 drawing windows
- Customize the independent axes for specialty plots (e.g., Lift vs. Angle of Attack or Cp vs. Cord)
- Interactive slice, iso-surface and streamtrace tools
- Detect vortex cores, shock surfaces, and separation and reattachment lines
- Calculate critical flow functions and perform integrations
- Integrate particle trajectories with support for massed particles and drag effects
- Validate numerical models with test data in the same window
- Compare multiple models simultaneously including visualization of fluid-structures interaction
- Support for 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 directly into PowerPoint, Word and Microsoft Office applications
- Animating to a file for use in PowerPoint, Web and Framer (AVI, Flash, Raster Metafile)
- Using the Publish command to share results directly on the Web

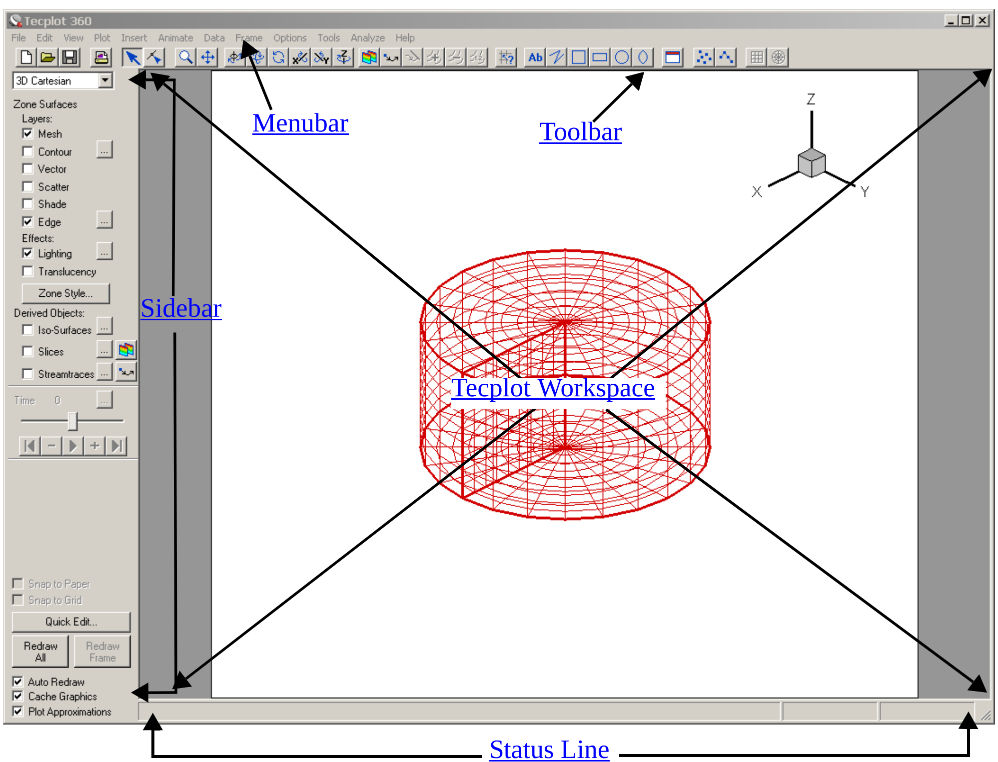
- Exporting presentation-quality vector and raster formats (JPEG, PNG, TIFF, BMP, WMF, PostScript, EPS)

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

- Create macros by recording or writing scripts
- Use the Quick Macro Panel for one-click macro access
- Batch process plotting and printing
- Extend Tecplot functionality with the Add-on Developer's Kit
- Customize the interface to your workflow

2 - 1 Interface

The Tecplot interface is divided into 5 major sections:



2 - 2 Menubar

The menu bar offers rapid access to most of Tecplot's features.

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

Tecplot's features are organized into the following menus:

- **File -** Use the **File** menu to control reading and writing of data files and plot layouts, printing and exporting of plots, recording and playing macros, setting and saving configuration preferences, and exiting Tecplot.

- **Edit -** Use the **Edit** menu for: undo functions, cutting, copying, pasting, and clearing objects, as well as pushing and popping (changing the draw order for selected items).

Cut, **Copy**, and **Paste** work only within Tecplot. To place a graphics image of your layout into another program, use **Copy Plot to Clipboard** (in Windows and Macintosh only)

- **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 sizing options for convenience. **Center** moves the plot image so that the data points are centered within the frame. (Only the data is centered; text, geometries, and the 3D axes are not considered.) **Fit to Full Size** fits the entire plot into the frame. **Nice Fit to Full Size** sets the axis range to begin and end on major axis increments (if axes are dependent Tecplot adjusts the vertical axis length to accommodate a major tick mark). **Make Current View Nice** modifies the range on a specified axis to fit the minimum and maximum of the variable assigned to that axis, then snaps the major tick marks to the ends of the axis. (If axis dependency is not independent this may affect the range on another axis.) **Data Fit** fits the data points to the frame.

- **Plot -** Use the **Plot** menu to control the style of your plots. The menu items available are dependent upon the active plot type (selected from the sidebar).

- **Insert -** Use the **Insert** menu to add text, geometries (polylines, circles, squares, ellipses, and rectangles), or image files.

- **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 include zone creation, interpolation, triangulation, as well as 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.

- **Tools -** Use the **Tools** menu to launch the **Quick Macro Panel** or an add-on

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

- **Help -** Use the **Help** menu to get quick help on features. By selecting **About Tecplot** 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 are dependent upon the plot type of the active frame. For 2D or 3D Cartesian plot types, you can add or subtract zone layers, zone effects 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:

- Select the desired Plot Types

- 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.

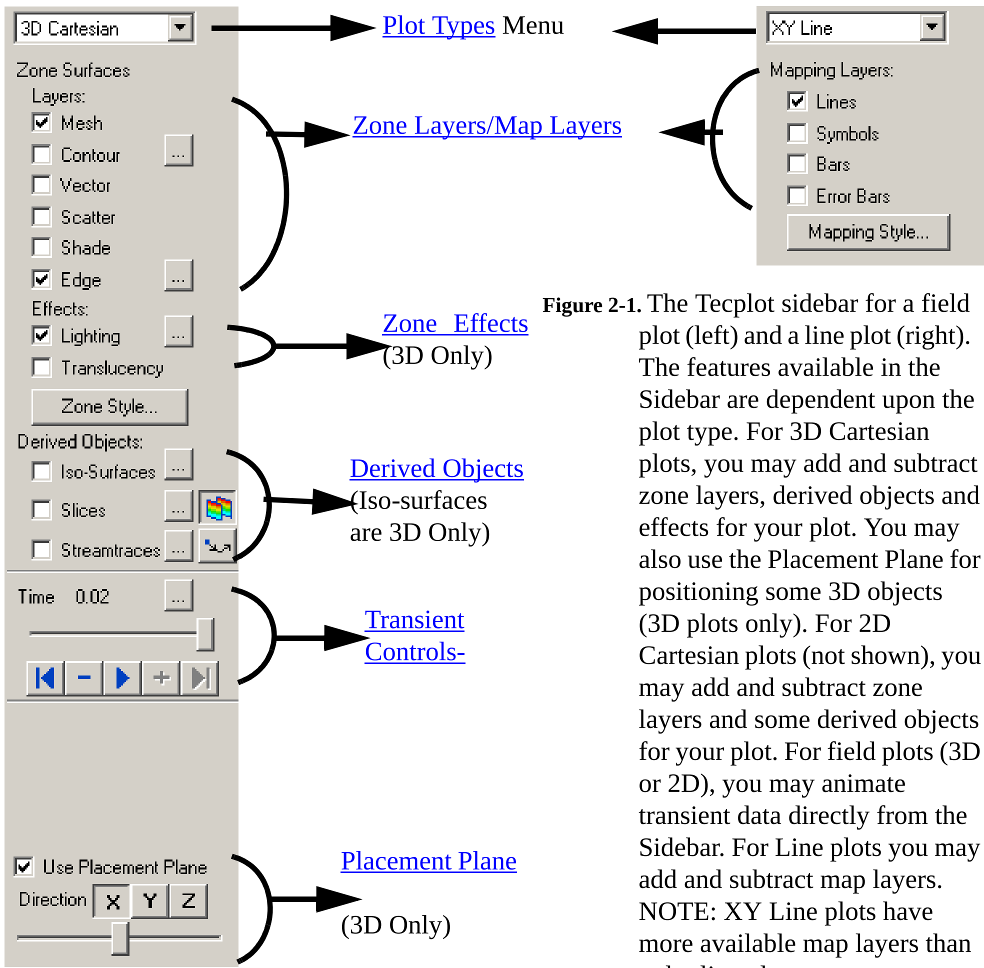


Figure 2-1. The Tecplot sidebar for a field plot (left) and a line plot (right). The features available in the Sidebar are dependent upon the plot type. For 3D Cartesian plots, you may add and subtract zone layers, derived objects and effects for your plot. You may also use the Placement Plane for positioning 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 more available map layers than polar line plots.

Plot Types

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

- **3D Cartesian -** 3D plots of surfaces and volumes.
- **2D Cartesian -** 2D plots of surfaces, where the vertical and horizontal axis are both dependent variables (i.e. x = f(A) and y = f(A), where A is another variable).
- **XY Line** - line plots of independent and dependent variables on a Cartesian grid. Typically the horizontal axis (x) is the independent variable and the y-axis a dependent variable, y = f(x).
- **Polar Line** - line plots of independent and dependent variables on a polar grid.
- **S (Sketch) -** Create plots without data, such as drawings, flow charts, and viewgraphs.

Zone Layers/Map Layers

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

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

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

- **Edge -** Zone edges and creases for ordered data and creases for finite-element data.

The four XY Line map layers are:

- **Lines -** Plots a pair of variables, X and Y, as a set of line segments or a fitted curve.
- **Symbols -** A pair of variables, e.g. X and Y, as individual data points represented by a symbol you specify.
- **Bars -** A pair of variables, X and Y, as a horizontal or vertical bar chart.
- **Error Bars -** Allows you to add error bars to your plot.

The two map layers for Polar Line are:

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

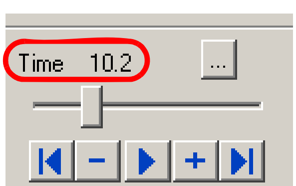
Zone Effects

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. Their corresponding **Details** dialogs can be accessed via the browse ... button.

Transient Controls-



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

The Animation Controls have the following function:

•  – Jumps to the *Starting Value*

• – Jumps toward the *Starting Value* by one step.

•  – Runs the animation as specified by the ‘Operation’ field of the **Time Details** dialog. The *Play* button becomes a *Stop* button while the animation is playing.

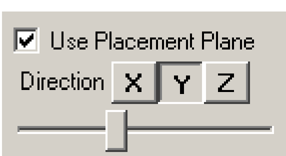
•  – Jumps toward the *Ending Value* by one step.

•  – Jumps to the *Ending Value*.

Use the button to launch the **Time Details** dialog.

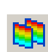
See the User’s Manual for more information on Time controls and the **Time Details** dialog. See the *Tecplot 360 User’s Manual* for more information on Time controls and the Time Details dialog.

Placement Plane



When you are using certain tools to add objects to your plot, toggle-on **Use Placement Plane** in the Sidebar to place them along a given plane (3D Plots only). Use the X,Y and Z buttons to select 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 )

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 to completely regenerate the workspace); ***Redraw (CTRL+R)*** redraws only the current frame.

Sidebar - Auto Redraw

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

NOTE: Even if Tecplot is in the middle of an auto-redraw, it can be interrupted with a mouse click or key press.

Sidebar - Cache Graphics

Tecplot uses OpenGL to render plots. OpenGL provides for the ability to cache graphic instructions for rendering and can re-render the cached graphics much faster than having Tecplot send the instructions again. This is particularly true for interactive manipulation of a plot. However this performance potential comes at the cost of using more memory. If the memory need is too high the overall performance could be less. Tecplot has three graphics cache modes: cache all graphics, cache only lightweight graphics objects, and do not cache graphics.

When ***Cache Graphics*** is selected in the Sidebar, Tecplot assumes there is enough memory to generate the graphics cache. Assuming this is true Tecplot's rendering performance will be optimal for interactive manipulation of plots.

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

See the *Tecplot 360 User’s Manual* for more information.

Sidebar - Plot Approximations

When ***Plot Approximation*** is selected, and if the number of data points is above the point threshold, (see below) Tecplot will render the approximate plot for style, data, and interactive view changes followed immediately by the full plot. This option provides for good interactive performance with the final plot always displayed in the full representation.

See the *Tecplot 360 User’s Manual* for more information.

2- 2.2 Toolbar

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

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

Selector Tool

 - Use the **Selector** tool to select objects in your workspace. The selected objects can be modified using the Quick Edit Dialog and (in some cases) the Selector tool itself.

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

- frames
- axis grid area
- text
- geometries

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

To select an object and open that object's attributes dialog either double-click on any object, or drag the cursor to select groups of objects (calls up **Group Select** dialog). Click OK, then **Object Details**.

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 data set 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** mode 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 click-and-drag to the desired location of the chosen data point. Other selected points will move as a unit with the chosen data point, maintaining their relative positions.

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

Group Select

The **Group Select** dialog is opened when you select a group of objects with the **Selector** or **Adjustor** tools. 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 (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 drop-down 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. You choose the appropriate color from the **Select Color** dialog.

Zoom Tool

 Zoom into or away from the plot.

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. Making the box larger than the frame zooms away from the plot.The region within the view box will be resized to fit into the frame.

If ***Snap to Grid*** is selected, 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)**.

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

- **2D Independent Axis Mode -** Allows the selected region to expand to exactly fit in the frame. The axes are rescaled independently to fit the zoom box.

- **2D Dependent Axis Mode** - In dependent mode, the axes are not fit perfectly to the zoom box. The longest dimension from the zoom box is applied to 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 (or CTRL+ right-click) to interactively zoom into or out of the plot.

Clicking anywhere in your plot while the zoom tool is active, centers the 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 interactively translate objects.

You can rescale your image by pressing “+” to magnify, “-” to shrink. If you are Shift-dragging to move the paper, the rescale buttons “+” and “-” will magnify or shrink the paper so long as you have the mouse button depressed

Three-Dimensional Rotation

Tecplot allows you to rotate your data in a variety of different ways. Choose one of the six 3D rotation mouse modes, then drag the pointer in the workspace to rotate your 3D image. The six rotation mouse modes can be entered by selecting the appropriate tools, as follows:

- Spherical  - Drag the mouse horizontally to rotate about the Z-axis; drag the mouse vertically to control the tilt of the Z-axis.

- Rollerball  - Drag the mouse in the direction to move with respect to the current orientation on the screen. In this mode, your mouse acts much like a rollerball.

- Twist  - Drag the mouse clockwise around the image to rotate the image clockwise. Drag the mouse counterclockwise around the image to rotate the image counterclockwise.

- X-axis  - Drag the mouse to rotate the image about the X-axis.

- Y-axis  - Drag the mouse to rotate the image about the Y-axis.

- Z-axis  - Drag the mouse to rotate the image about the Z-axis.

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

Drag	Rotate about the defined rotation origin with your current Rotate tool.
Alt-Drag	Rotate about the viewer position using your current Rotate tool.
Middle-Click/CTRL+right click	smooth zoom in and out of the data
right-click	translate the data
C	Move rotation origin to probed point, ignoring zones.
O	Move rotation origin to probed point of data.
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 **Slicing** 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 adds a slice. Primary Slices active [ONLY] - Turns on Start/End Slices and adds a slice Start/End Slices active [ONLY] - Turns on Start/End Slices and adds a slice
-	Primary Slices, Start End Slices Active - Removes start and end slices Primary Slices active [ONLY] - removes the primary slice Start/End Slices active [ONLY] - removes the Start and End Slices
Click/Drag	Updates the position of the primary slice (if active). If only start and end slices are visible, click updates the position of the 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, slices).
Shift-click	Place the start or end slice (whichever is closest to the initial click location). Show Start/End Slices is activated, if necessary.
Shift-drag	Move the start or end slice (whichever is closest to the initial click location). Show Start/End Slices is activated, if necessary.
I, J, K (ordered zones only)	Switch to slicing constant I-, J-, or K-planes respectively.
X, Y, Z	Switch to slicing constant X-, Y-, or Z-planes respectively.
1-4	Numbers one through four switch to the corresponding slice group.

Add Streamtrace

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

Refer to the *Tecplot 360 User's Manual* for more information.

D	Switch to streamrods.
R	Switch to streamribbons.
S	Switch to surface lines.
V	Switch to volume lines.
1-9	Change the number of streamtraces to be added when placing a rake of streamtraces.

Streamtrace Termination Line

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

To draw a **Streamtrace Termination Line**:

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

Add Contour Level

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

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

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

Delete Contour Level

 Select 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

 Select the **Add Contour Label** tool to switch to the **Contour Label** mode, enabling 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 at a slight offset from the lines they label.


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

Probe Tool


Select the **Probe At Tool** to probe for values of the data set's variables at a particular point.

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

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


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

Insert Text

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

Insert Geometries

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


Polylines

Squares

Rectangles

Circle

Ellipse

Create New Frame

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

To add a frame:

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

If you have data loaded into Tecplot and you create a new frame, you can attach the existing data set to the new frame by changing the plot type.

Extract Discrete Points

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

To select points:

- Click your left-hand mouse button at each location where you would like to extract a point.
- To end extraction, either double-click on the last point, or press `Esc`.

- 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

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

To select points:

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

Create Rectangular Zone

 Select the **Create Rectangular Zone** tool to add new 2D rectangular zones to the current Tecplot data set.

To create a rectangular zone:

- Click once in the current data region to anchor one corner of the zone.
- 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**).

 The current frame must have a data set attached to it, in order for this tool to be active.

Create Circular Zone

 Select the **Create Circular Zone** tool to add new 2D circular zones to the current Tecplot data set.

To create a circular zone:

- Click once in the current data region to specify the center of the zone.
- 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**).

The current frame must have a data set attached to it, in order for this tool to be active.

2- 2.3 Status Line

The status line, running along the bottom of the Tecplot window, gives “hover help.” When you move the pointer over a 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- 2.4 Tecplot 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 is set up to draw on, as well as a reference grid and rulers. The active frame, in which you are currently working, is on top. All modifications are made to the current frame.

2 - 3 Data Hierarchy

Tecplot structures data in two levels: data sets and zones. Data sets are contained within frames. Each data set 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 using subwindows called "frames". By default, one frame is open when you launch Tecplot. You can add frames to the workspace using the Frame menu. Data sets can be unique to the frame or shared between frames. Linking data between frames allows you generate unique plots of the same data. For more information on working with frames, please refer to the User's Manual.

2- 3.2 Data Sets

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

2- 3.3 Zones

Zones are a subset of data sets. A data set can be composed of a single zone or several zones. Zones are either defined in the data file or created directly in Tecplot. The number of zones in a concatenated data set 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 data set of an airplane many consist of a zone for each wing, each wheel, the nose, etc. Alternatively, zones may be defined based on the material. For example, a data set of a fluid tank may have a zone for the tank itself and additional zones for each fluid therein.

2 - 4 Data Structure

Tecplot 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 one-, two-, or three-dimensional array, where I, J, and K are the index values within the array. The number of data points is the product of all of the dimensions within the array.

- **One-dimensional array (I-ordered, J-ordered or K-ordered) -** A single 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 data set with I=5, J=K=1 has 5 data points.

- **Two-dimensional array (IJ-ordered, JK-ordered, IK-ordered) -** A two-dimensional array of data points where two of the three dimensions (I,J, K) are greater than one and the other dimension is equal to one. The number of data points in a two-dimensional ordered data set is the product of the all of the dimensions. For example, in an IJ-ordered data set, the number of data points is equal to I x J (where K=1).

- **IJK-ordered -** Three-dimensional array of data points where all three of the I-, J-, and K-dimensions are greater than one. The number of data points is the product of the I-, J-, and K-dimensions.

2- 4.2 Finite-Element Data

Finite-element data is arranged in two arrays, a variable array and a connectivity matrix. The variable array is a collection of points in 2D or 3D space that are connected into polygonal or polyhedral units called elements. The connections between the nodes are defined by the connectivity matrix.

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 data set, or the data points may be irregularly (randomly) positioned. For such difficult cases, you may be able to organize your data as a patchwork of elements. Each element can be independent of the other elements, so you can group your elements to fit complex boundaries and leave voids within sets of elements. Figure 2-2 shows how finite-element data can be used to model a complex boundary.

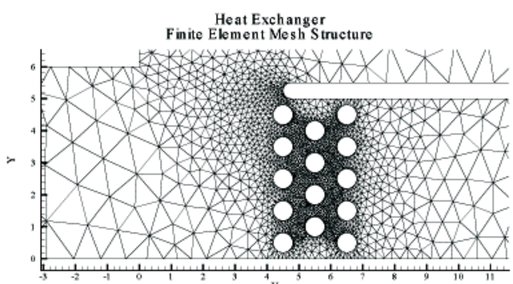


Figure 2-2. Finite-element data used to model a complex boundary.

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

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

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

You can simulate zones with mixed element types by repeating nodes as necessary. For example, a triangle element can be included in a quadrilateral zone by repeating one node in the element's connectivity list, and tetrahedral, pyramidal, and prismatic elements can be included in a brick zone by repeating nodes appropriately.

2 - 5 Creating Plots

The basic steps for creating a plot in Tecplot are:

1. Define your data set using one of the following methods:
   - **a.** This is typically accomplished by using the **Load Data File(s)** command from the **File** menu.
   - **b.** Use the **Open Layout** command from the **File** menu to load linked layout or layout package files.
   - **c.** Use any combination of the options in the **Create Zone** sub-menu of the **Data** menu or the **Insert** menu to create your data sets directly within Tecplot.
2. Select 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 (e.g. contour zone layer or symbols mapping layer). Use the details buttons (...) 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 buttons (...) to customize any derived objects.

You are not limited to working with only one plot at a time in Tecplot. You can create multiple files at one time 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 data set.

2 - 6 Output Formats

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

- **Printing** - accessed via **File>Print**
- **Exporting to an image file** - accessed via **File>Export** and selecting the desired image format in the **Export** dialog.
- **Exporting to an animation file** - accessed via any of the Animation dialogs (by selecting “To File” as the animation destination) 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 & Macintosh Only) - Use the **Copy Plot to Clipboard** option from the **Edit** menu to paste your plot into word processing software.

Chapter 3 External Flow Tutorial

3 - 1 Introduction

3- 1.1 Tutorial Summary

The tutorial should take approximately 30-40 minutes to complete and has the following steps:

Step 1: Load a Plot3D file (including a boundary file)
Step 2: Determine the Number of Zones and Variables.
Step 3: Isolate the Wing using the boundary file
Step 4: Adjust the View
Step 5: Calculate the Pressure Coefficient
Step 6: Display Contours
Step 7: Add a Contour Legend
Step 8: Modify the Contour Legend
Step 9: Add Contour Levels
Step 10: Modify the Contour coloring
Step 11: Extract a Slice
Step 12: Activate the Edge Layer for Zone 7 only
Step 13: Create a new frame
Step 14: Delete a group of mappings
Step 15: Activate the mapping
Step 16: Change the zone associated with the mapping
Step 17: Go to View>Data Fit
Step 18: Determine the maximum and minimum values of the x variable
Step 19: Create a new variable
Step 20: Change the x-axis variable
Step 21: Go to View>Data Fit
Step 22: Load the measured data file
Step 23: Change the variable name
Step 24: Create New Mappings
Step 25: Rename the Mappings
Step 26: Change the zone assignments for the experimental data mappings
Step 27: Activate the new mappings
Step 28: Reverse the Y-axes
Step 29: Adjust the Mapping Style
Step 30: Add a Line Legend
Step 31: Export the frames to a file

All supporting data files for this tutorial can be found in: $TEC360HOME/demo/tutorial/external_flow, where $TEC360HOME is the installation directory for Tecplot 360. For Windows users, this is typically C:/Program Files/Tecplot/Tec360.

3- 1.2 Background Information

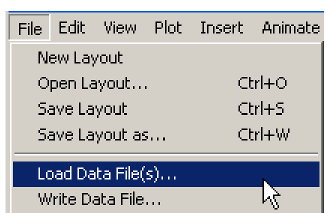
In 1972, the ONERA Aerodynamics Department designed a swept, semi-span wing, with no twist[1]. The wing was very well instrumented to be used as an experimental support for basic studies of three-dimensional flows at high Reynolds numbers from low to transonic speeds (i.e. local supersonic flow, shocks, and turbulent boundary layer separation). The following tutorial demonstrates: how to display pressure on an airplane wing and compare measured and simulated data on a two-dimensional projection plane.

The simulated data set is a PLOT3D file with freestream conditions M = .8395, Re = 11.72 x 106, Angle of Attack = 6 degrees and angle of side-slip = 0.0 degrees. The measured data set 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 12x 106. The measured data set is an ASCII file. The tutorial compares the data sets at M = .84 and y/b = .44 (where b is the wingspan).

3 - 2 Getting Started

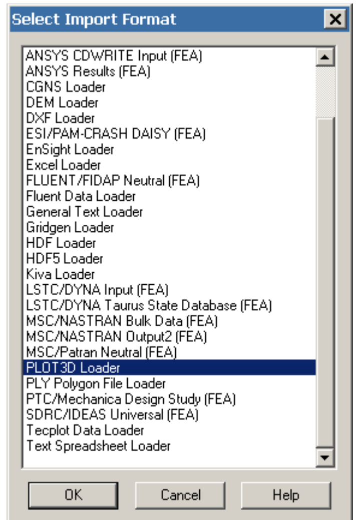
Step 1:Load a Plot3D file (including a boundary file)

A. Select **Load Data File(s)** from the **File** menu.

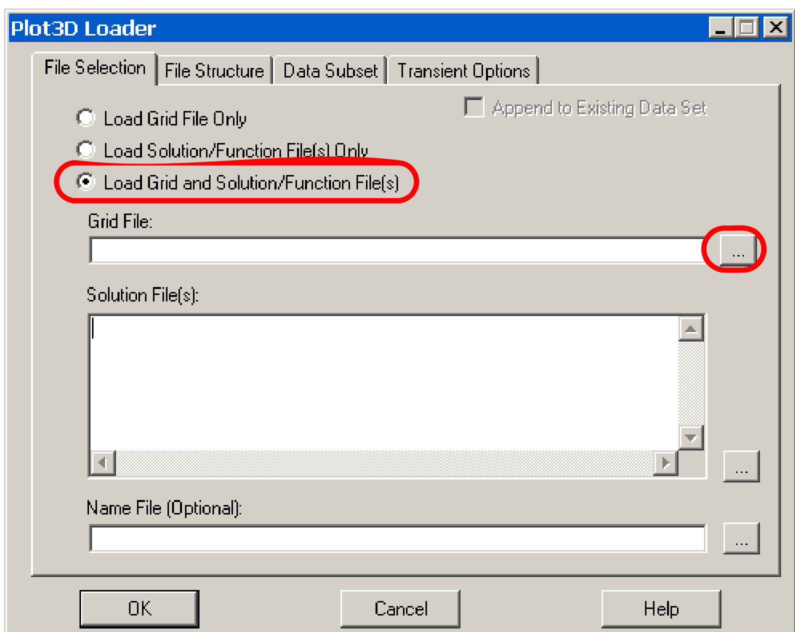


1. Schmitt, V. and F. Charpin, "Pressure Distributions on the ONERA-M6-Wing at Transonic Mach Numbers," Experimental Data Base for Computer Program Assessment. Report of the Fluid Dynamics Panel Working Group 04, AGARD AR 138, May 1979.

B. Select ***Plot3DLoader*** from the **Select Import Format** dialog.



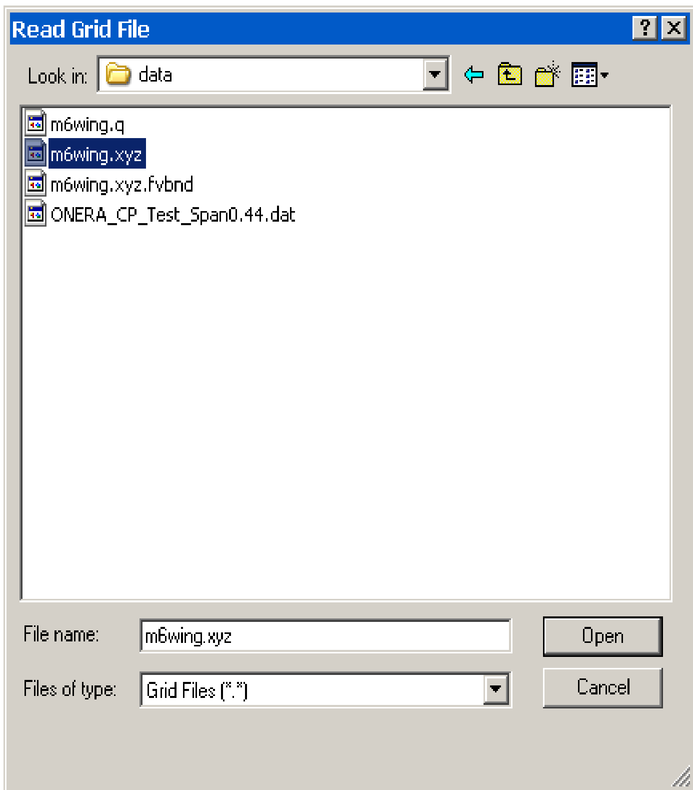
C. In the **Plot3D Loader** dialog:



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

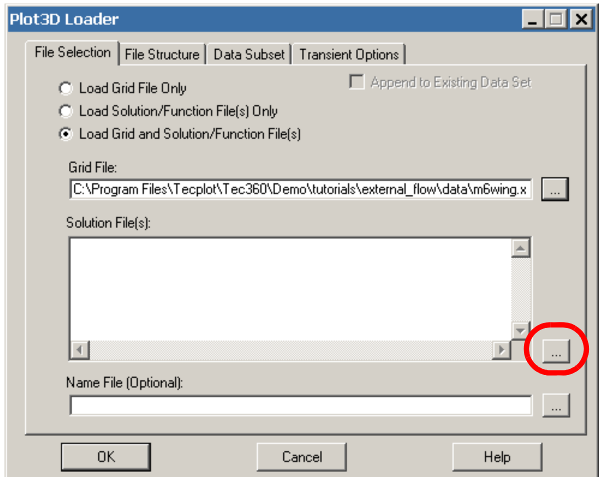
ii. Select the button to the right of “Grid File”.

D. In the **Read Grid file** dialog, navigate to: *$TEC360HOME/demo/tutorial/external_flow/data*[2] and select *m6wing.xyz*.



2. $TEC360HOME is the installation directory for Tecplot 360. For Windows users, this is typically *C:/Program Files/Tecplot/Tec360*.

E. In the **Plot3D Loader** dialog, select the button to the right of "Solution File(s)".

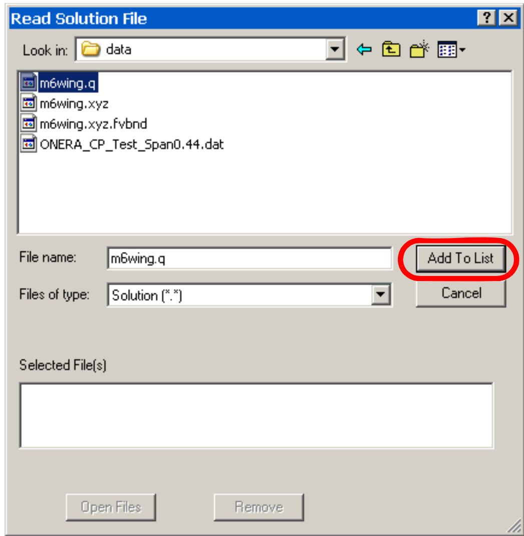


F. In the **Read Solution File** dialog:

i. Navigate to: *$TEC360HOME/demo/tutorial/external_flow/data*[3].

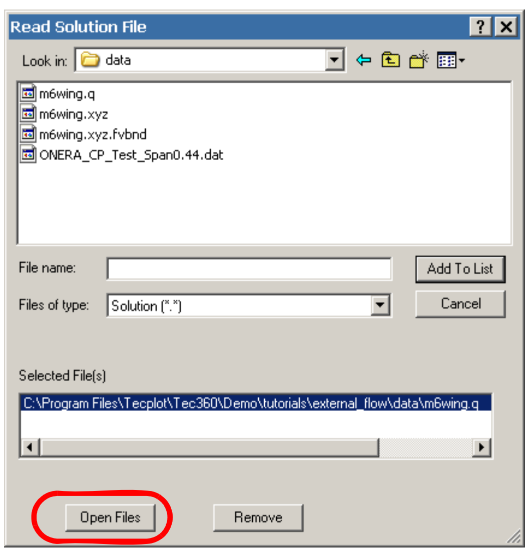
3. $TEC360HOME is the installation directory for Tecplot 360. For Windows users, this is typically C:/Program Files/Tecplot/Tec360.

ii. Select *m6wing.q*.

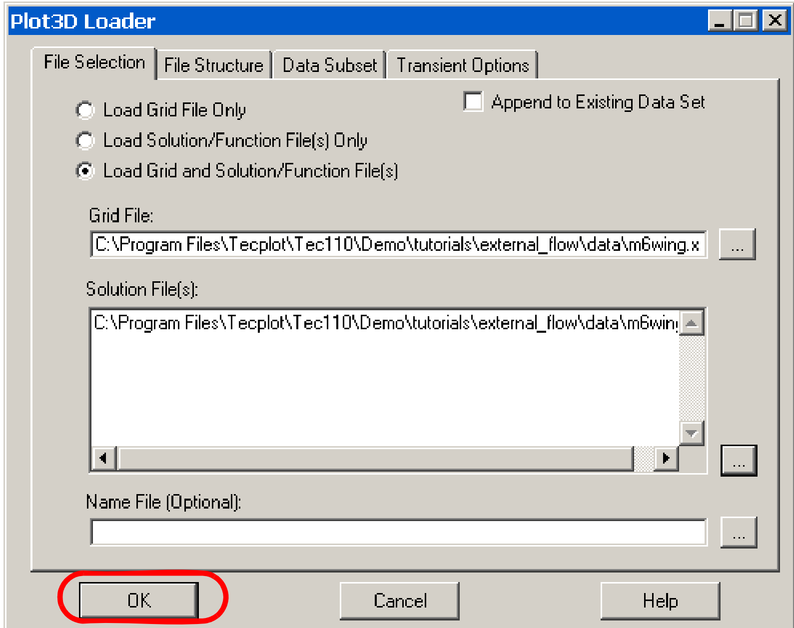


iii. Select the ***Add to List*** button.

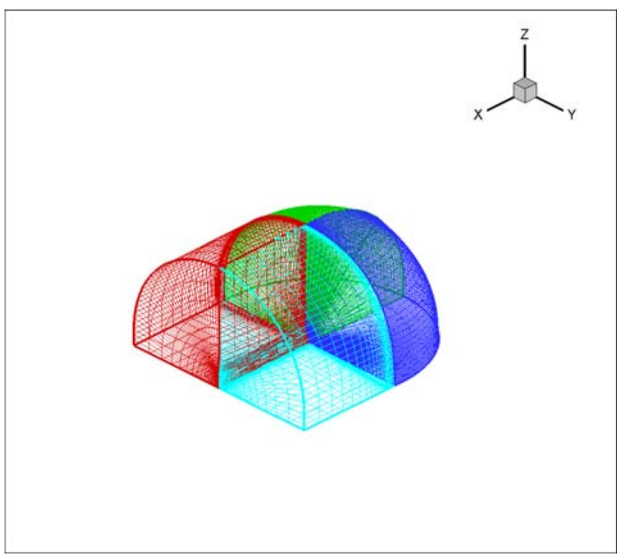
iv. Select the ***Open Files*** button.



G. Select "OK" after returning to the **PLOT3D Loader** dialog.



The initial plot will look as follows:



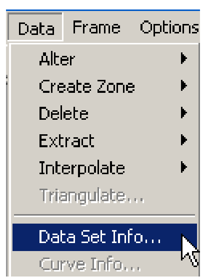
You have now successfully loaded a PLOT3D data file into the Tecplot workspace.

NOTE: The boundary file, *m6wing.xyz.fvbnd*, was automatically included in the data set because its filename prefix (*m6wing.xyz*) is identical to the grid file and the files are located in the same directory.

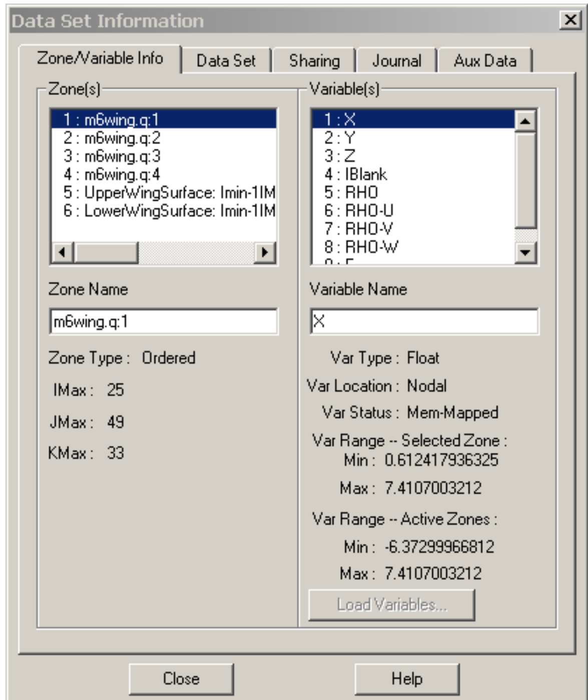
Step 2: Determine the Number of Zones and Variables.

In Tecplot, 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 pre-processor. See also 2- 3.3 “Zones” on page 25.

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



In this data set, there are six zones, each of which contain nine variables. The ***Aux Data*** tab details additional variables included in the data set.



B. Close the dialog.

Step 3: Isolate the Wing using the boundary file

This data set contains information for a very large volume and it is difficult to visualize the wing. In this particular case, there is a simple solution to isolating the wing boundaries.

A. Select the “Zone Style” button from 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 apply to the entire plot.

B. In the **Zone Style** dialog, select zones 1 - 4 by either:

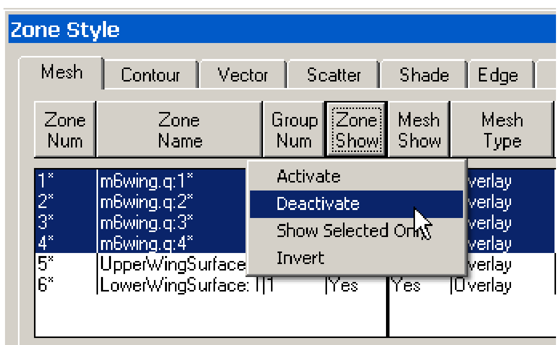
i. Selecting Zone 1, holding the SHIFT key and selecting Zone 4

ii. Selecting Zone 1 and clicking-and-dragging down to Zone 4

iii. Selecting “Select Range” from the *Zone Num* button and entering <u>1</u> in the ***Begin*** field and <u>4</u> in the ***End*** field of the **Enter Range** dialog.

iv. Selecting “Select by Name” from the *Zone Name* button and entering “m6*” in the **Enter Name** dialog.

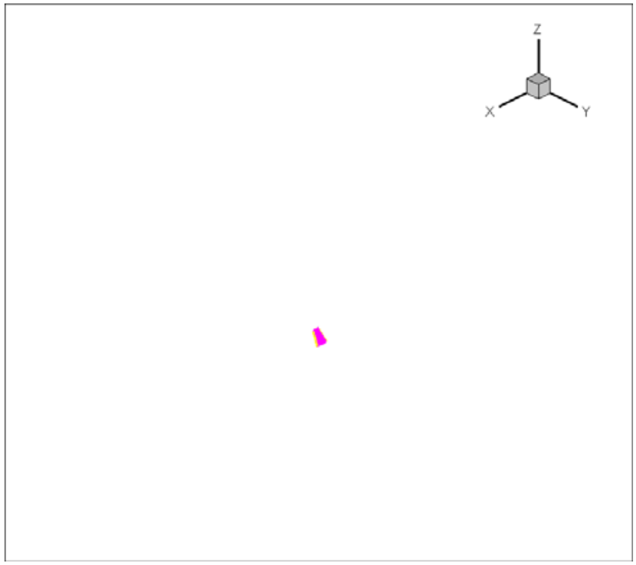
C. Select the *Zone Show* button and then select “Deactivate”



Alternatively, you can select Zones 5 - 6 and select “Show Selected Only” from the *Zone Show* button.

D. Close the dialog.

The plot should look as follows:

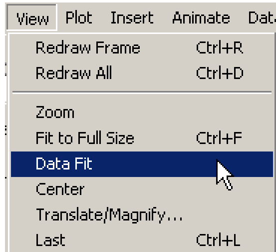


The specific zone(s) the wing is located in is a characteristic of the data file. If you have a data file that is not arranged in this manner, you may find the following methods helpful:

- For many viscous, CFD data sets 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're looking for by turning off the mesh, turning on the contour and turning on translucency.
- Experiment with turning zones on-and-off and surfaces to plot in the **Zone Style** dialog. In this particular example, if the boundary plot is not loaded, you can view the wing by selecting "J-Planes" as the surfaces to plot in the ***Surfaces*** page of the **Zone Style** dialog.

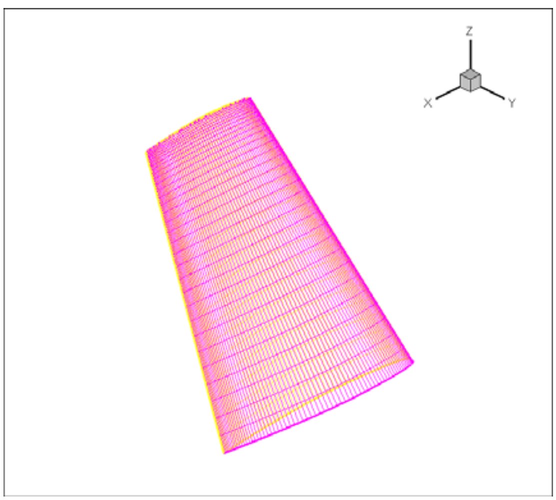
Step 4: Adjust the View

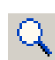
Select **Data Fit** from the **View** menu.



As its name implies, **Data Fit** resets the view so that the active zones fill the current frame.

The plot should look as follows:



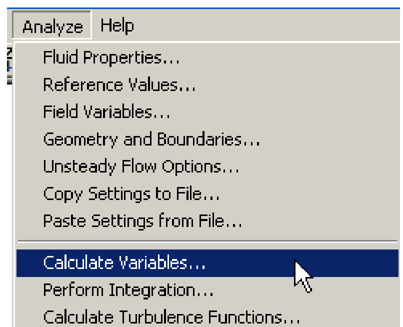
Alternatively you may use the  and  tools or the **Translate/Magnify** dialog (accessed via the **View** menu). You can also use the middle-mouse button to zoom your plot and the right-mouse button to translate.

3 - 3 Display Contour Map of the Pressure Coefficient

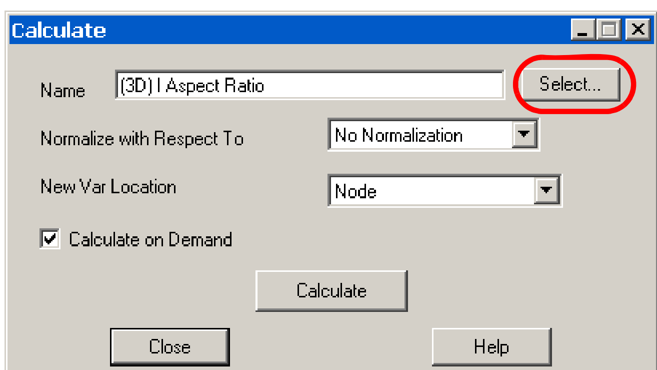
In order to have a more useful visualization of the wing, we would like to turn off the Mesh layer and turn on the Contour layer (using Pressure coefficient as the contour variable). In this case, the pressure data was not provided in the data file. Tecplot can calculate the pressure coefficient for us.

Step 5: Calculate the Pressure Coefficient

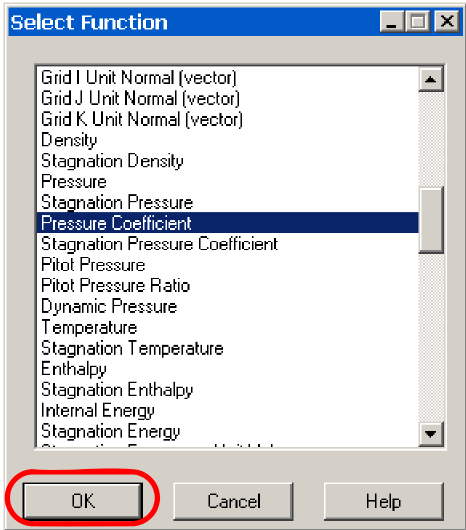
A. Select **Calculate Variables** from the **Analyze** menu.



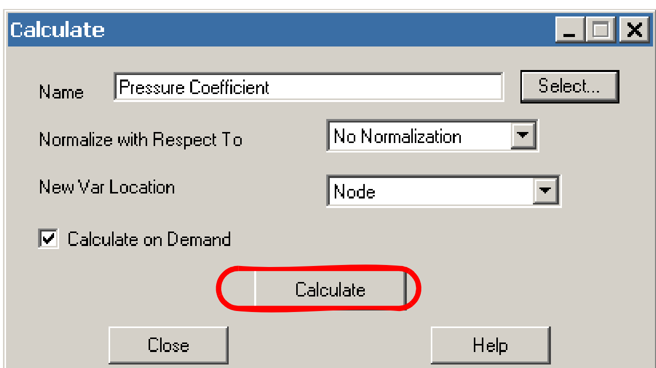
B. In the **Calculate** dialog, click on the ***Select*** button to change the variable to be calculated.



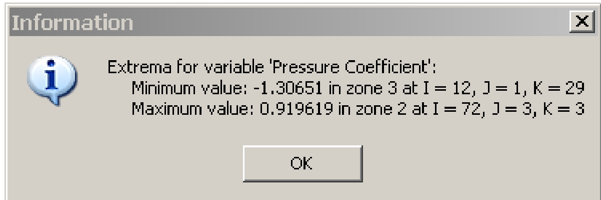
C. In the **Select Function** dialog, highlight “Pressure Coefficient” and select the OK button.



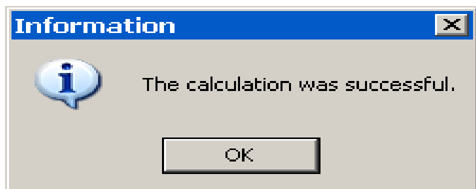
D. Select the ***Calculate*** button.



E. Press “OK” in the following **Information** dialog detailing the extrema for the Pressure Coefficient variable.



F. Press “OK” in the following **Information** dialog.



G. Close the **Calculate** dialog.

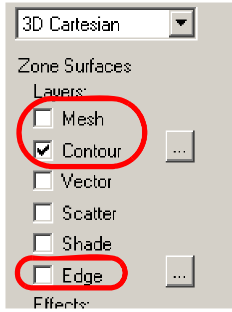
> NOTE: While calculating variables you may use any units. However, you must be consistent within your variable set!

Step 6: Display Contours

We would now like to visualize the pressure coefficient using contours.

A. Toggle-off Mesh and Edge layers in the Sidebar.

B. Toggle-on the Contour layer.



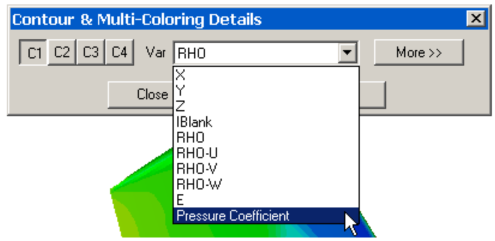
NOTE: Any combination of zone layers can be viewed at a time, we are simply turning the mesh and edge layers off to create a more pleasing image.

C. Open the **Contour Details** dialog by either:

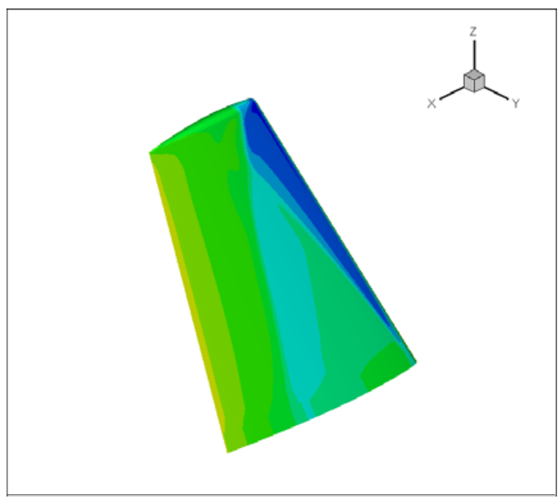
i. Selecting the button to right of Contour in the Sidebar.

ii. Selecting **Contour/Multi-Coloring** from the **Plot** menu.

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



The final image should look as follows:



3 - 4 Modify the Contour layer

Using the **Contour and Multicoloring Details** dialog box, you can customize: contour levels, coloring and more. The **Contour Details** dialog box can be accessed via **Plot> Contour/Multi-coloring** or by selecting the

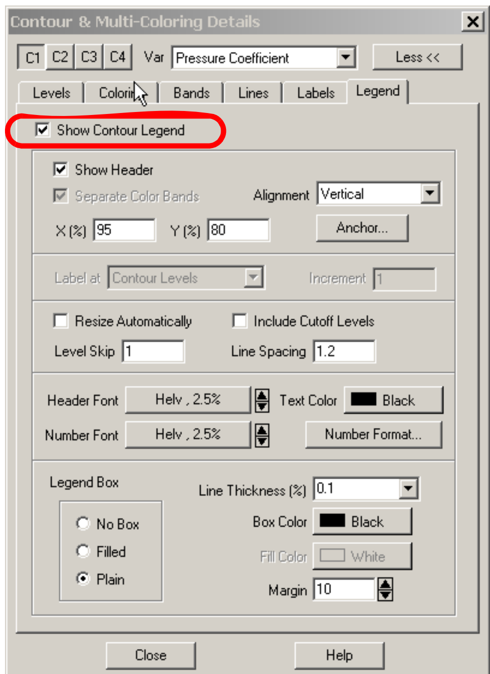
button to the right of Contour in the Sidebar.

Step 7: Add a Contour Legend

A. Expand the **Contour Details** dialog by selecting the More >> button.

B. Go to the ***Legend*** page of the dialog.

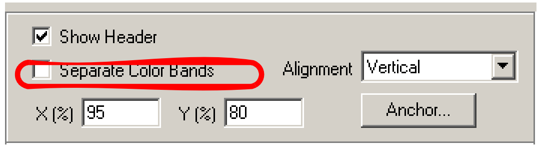
C. Toggle-on *Show Contour Legend*.



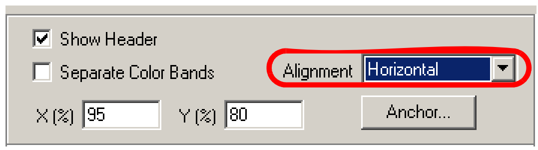
Step 8: Modify the Contour Legend

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

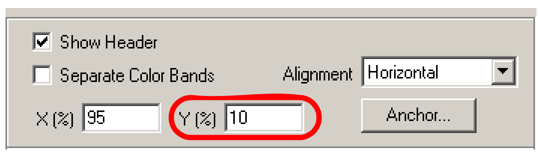
i. Toggle-off “Separate Color Bands” to remove the level lines from the legend.



ii. Change the Alignment to Horizontal.

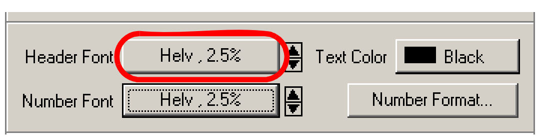


iii. Reposition the legend to 10% for Y.

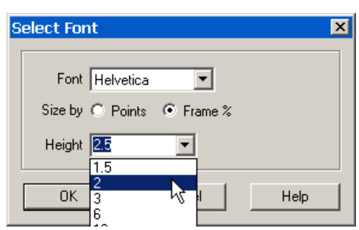


B. In the Font region of the page, perform the following steps:

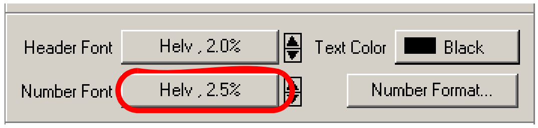
i. Select the Header Font button.



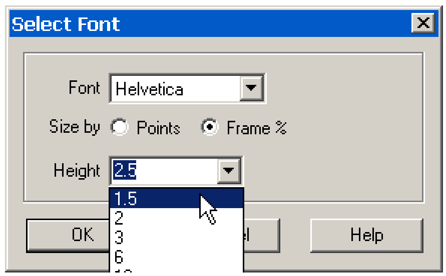
ii. In the **Select Font** dialog, change the header font size (height) to 2% and press ***OK*** to close the dialog.



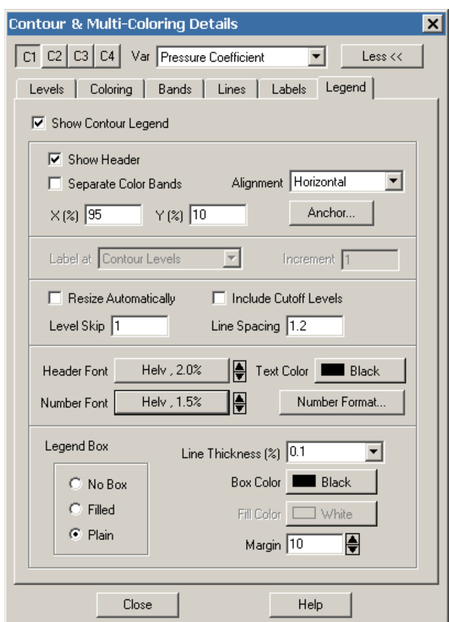
iii. Select the Number Font button.



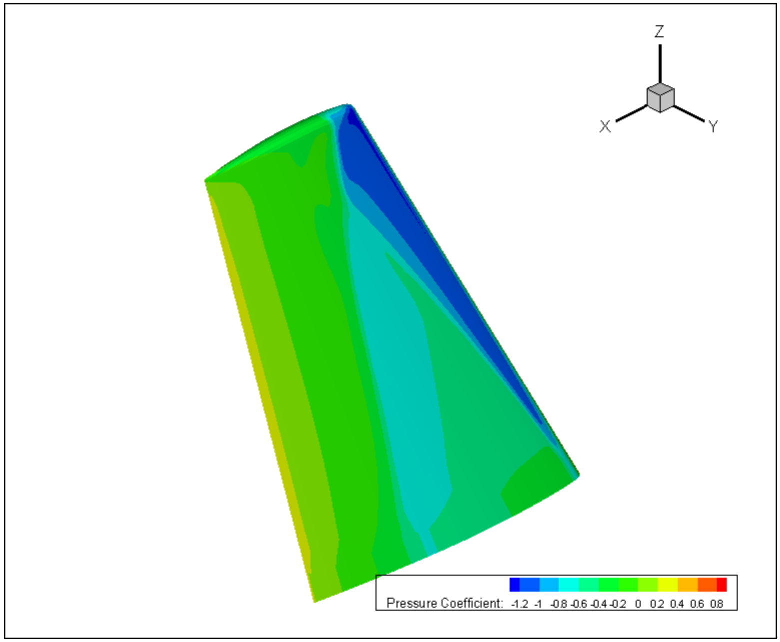
iv. Change the number font size to 1.5% and press ***OK*** to close the dialog.



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



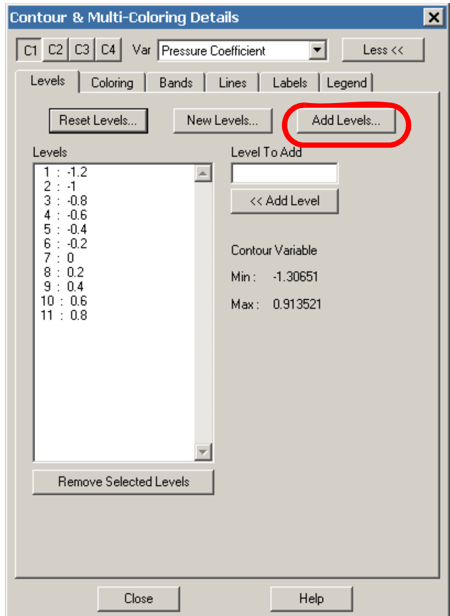
The plot will look as follows:



Step 9: Add Contour Levels

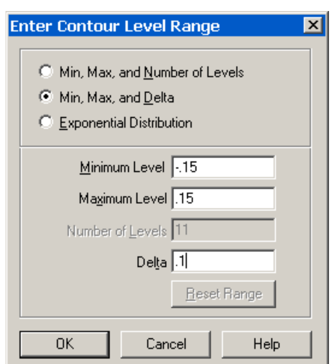
Judging from the legend, the bulk of the data values are between .2 and -.4. We would like to add more levels in that range to add definition to the plot.

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



B. Select the *Add Levels* button.

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



i. Select the ***Min, Max, and Delta*** radio button.

ii. Enter -.15 in the *Minimum Level* field.

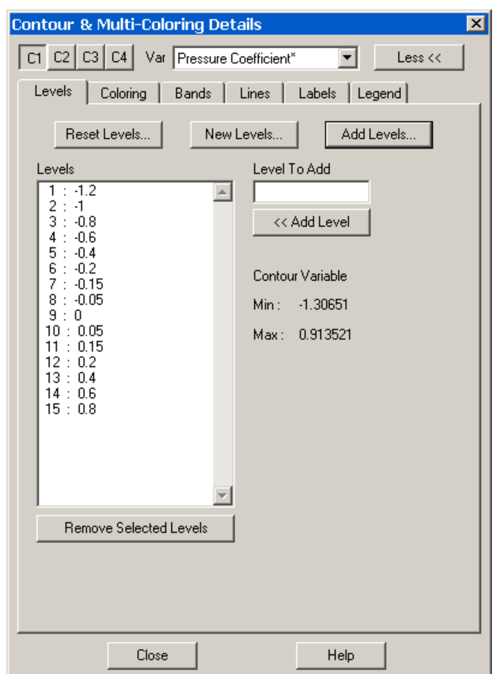
iii. Enter .15 in the *Maximum Level* field.

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

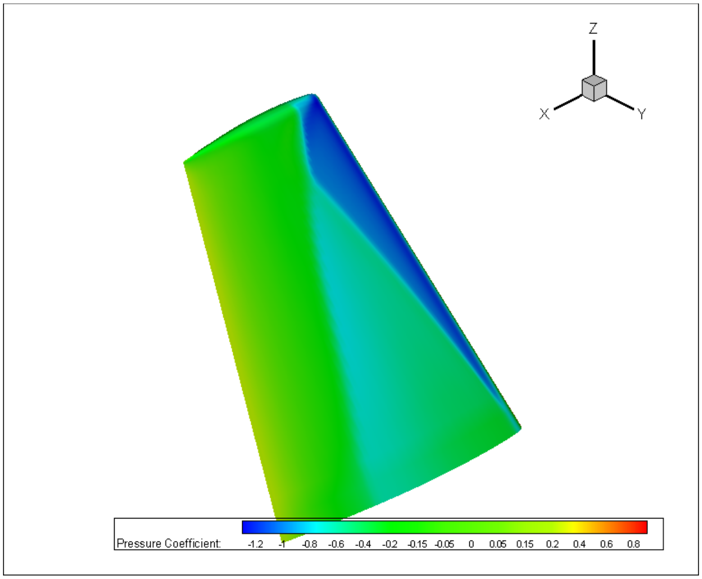
v. Click ***OK*** to apply the changes and close the dialog.

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

The ***Levels*** page will now look as follows:



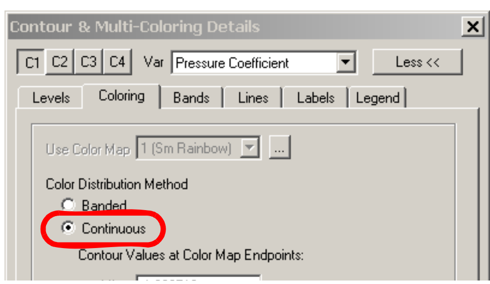
The plot containing a legend will look as follows:



Step 10: Modify the Contour coloring

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

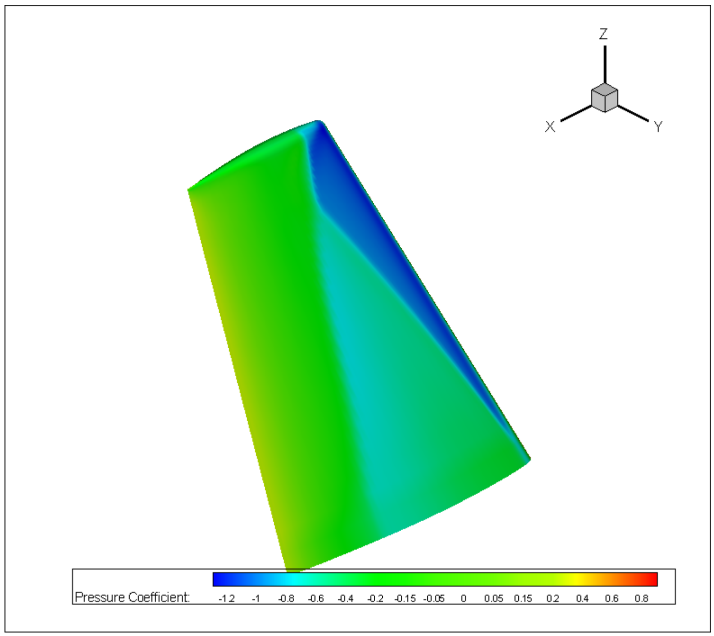
B. Select the “Continuous” radio button



C. Close the dialog.

You can adjust the color map and its control points by going to **Options>Color Map**.

The plot should look as follows:



3 - 5 Create a 2D Projection Plane

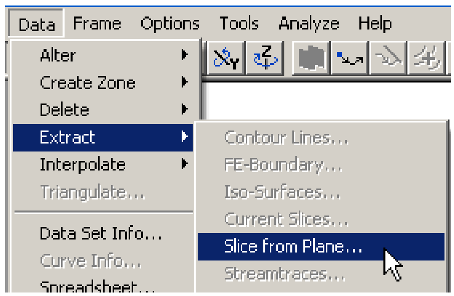
During the original ONERA wing study, the researchers compared simulation and measured data at y/b equal to .2, .44, .65, .8, .9, .95 and .99, where b is the wingspan.

The following steps will recreate the comparison plots for y/b equal to .44.

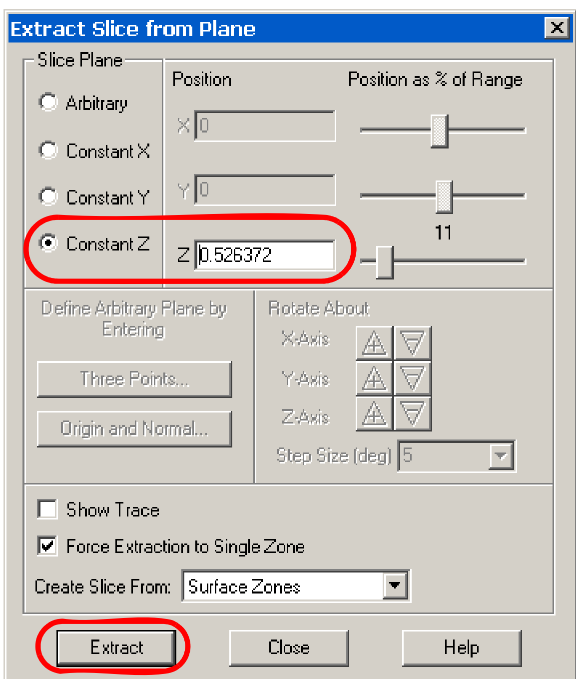
Step 11: Extract a Slice

The y-axis from the data is equivalent to the z-axis in Tecplot. From the ONERA study, we know that b (wingspan) is 1.1963 m. As such, we would like to extract a slice at z =.44 * b or z =.526372.

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



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

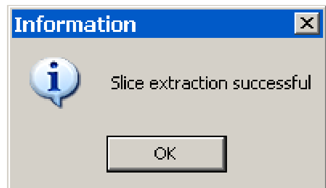


i. Select the “Constant Z” radio button.

ii. Set the “Z Position” equal to .526372.

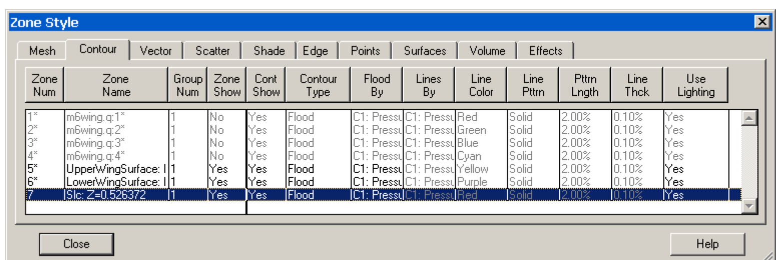
iii. Select the “Extract” button.

C. Select “OK” on the successful extraction dialog.



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

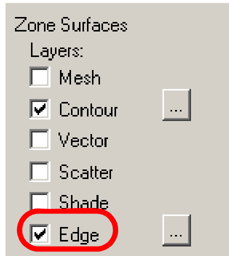
E. Go to the **Zone Style** dialog and verify that a new zone has been created. The new zone is named: SLC Z = .526372.



Step 12: 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 using the **Zone Style** dialog.

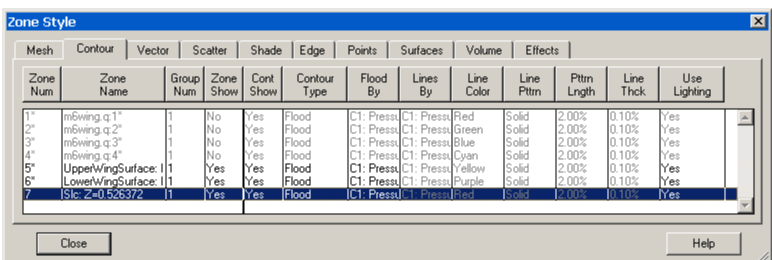
A. Toggle-on Edge in the Sidebar.



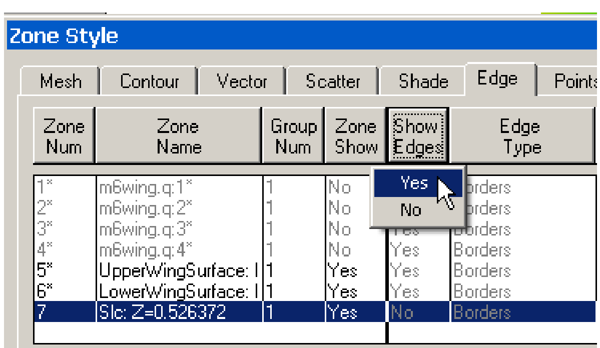
B. Open the **Zone Style** dialog.

C. On the ***Edge*** page of the **Zone Style** dialog perform the following steps:

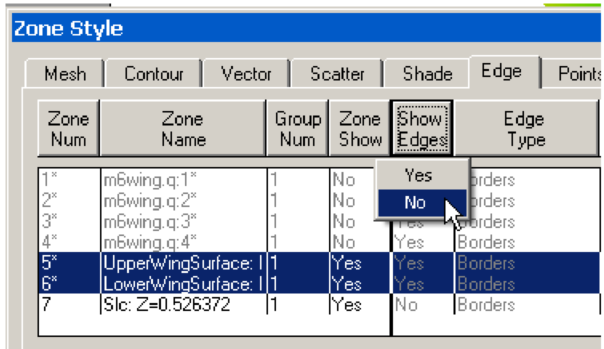
i. Select Zone 7 (the extracted slice).



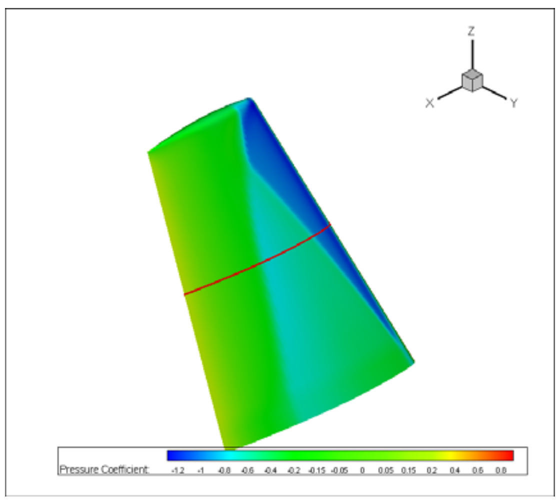
ii. Select “Yes” from the *Show Edge* menu.



iii. Select zones 5-6 and select: “No” from the *Show Edge* menu.



The image should look as follows:.



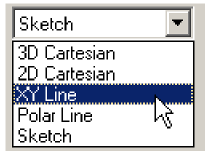
When wingspan is not recorded, you can approximate the wingspan using the probe  and a distance equation.

Step 13: Create a new frame

A. Select the “new frame” icon from the toolbar .

B. Click and drag in the workspace.

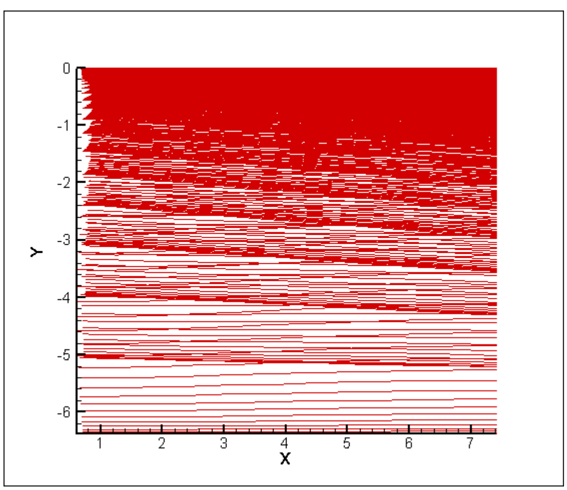
C. To associate the new frame with the existing data set, go to the Sidebar and switch the plot type to “XY Line”.



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

When working with multiple frames, use the **Order Frames** dialog (accessed via the **Frame** menu) to bring your frames backward and forward.

The plot will look as follows:



Step 14: Delete a group of mappings

The new plot currently displays an XY plot of the first zone in the original data set. 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. Selecting the Mapping Style button from the Sidebar.

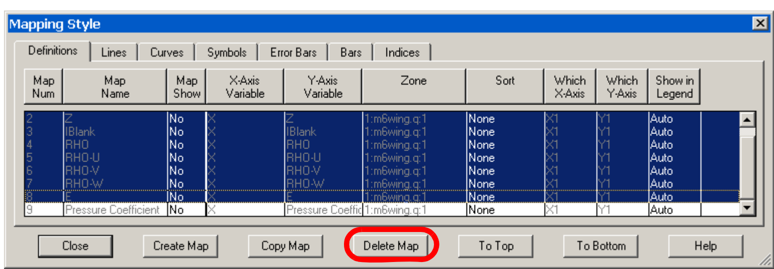
ii. Selecting **Plot>Mapping Style**.

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

i. Selecting Map 1, holding down the SHIFT key and Selecting Map 8.

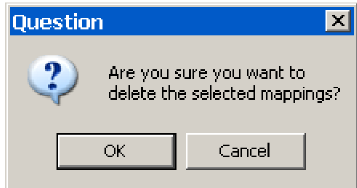
ii. Selecting Map 1 and clicking-and-dragging to Map 8.

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



C. With Maps 1 - 8 selected, select the *Delete Map* button.

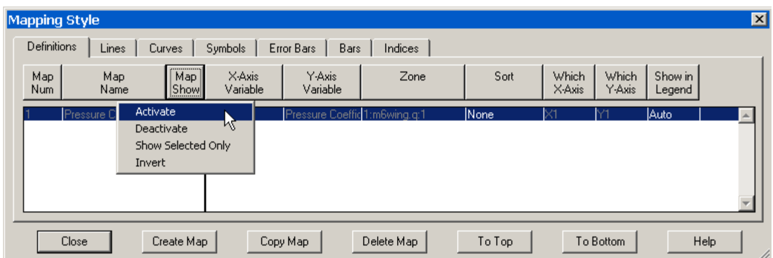
D. Select OK in the confirmation dialog.



Step 15: Activate the mapping

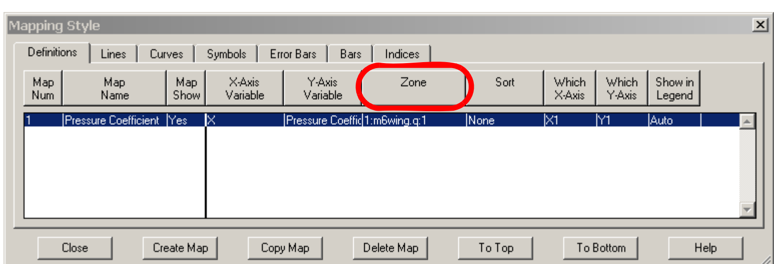
A. With the **Mapping Style** dialog still open, highlight the remaining mapping.

B. Select “Activate” from the *Map Show* button.



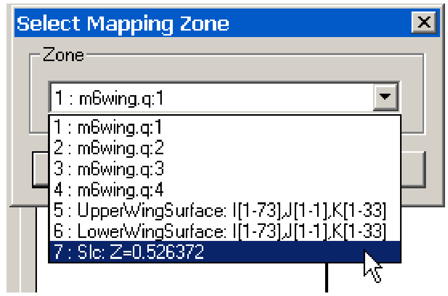
Step 16: Change the zone associated with the mapping

A. With the **Mapping Style** dialog still open, highlight the remaining mapping.



B. Select the *Zone* button.

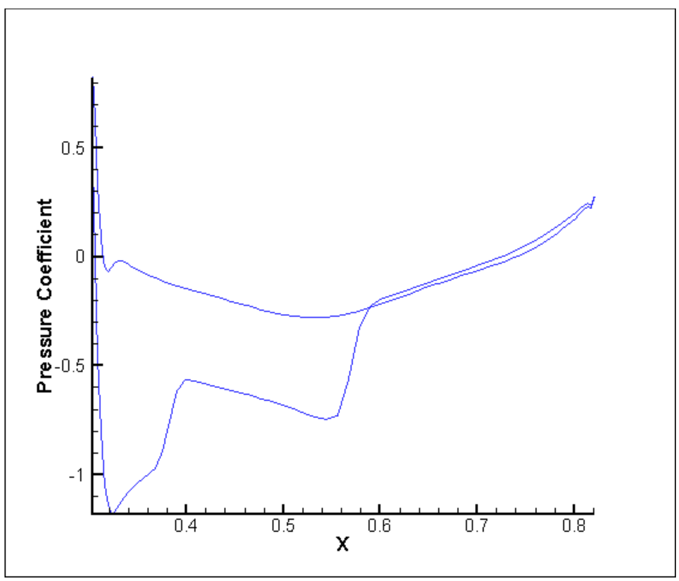
C. Select *Zone 7: SLC z = .526372* from the **Select Mapping Zones** dialog.



D. Close the dialog.

Step 17: Go to View>Data Fit

The plot will look as follows:



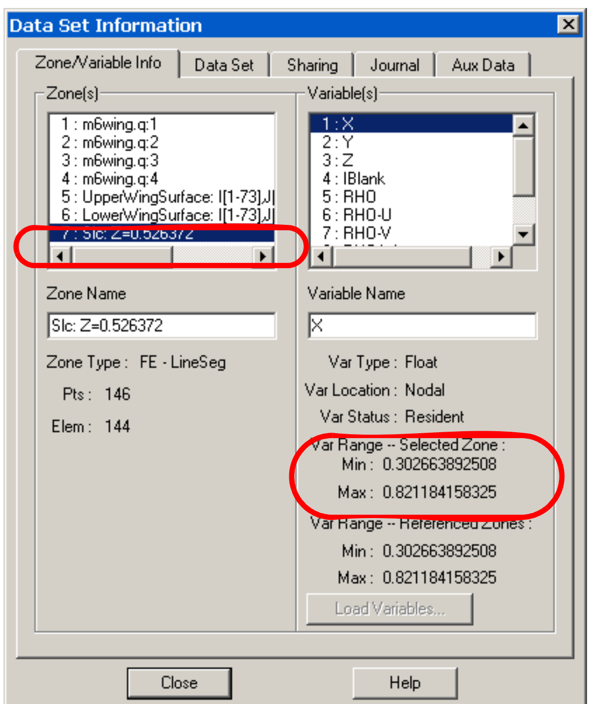
3 - 6 Data Manipulation

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 data sets, we will create a new variable {x/c} that shifts the current x variable to 0 and scales the value by c.

Step 18: Determine the maximum and minimum 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 Info** 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

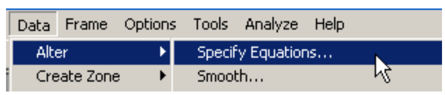
ii. Max = 0.8212

iii. Range = .5185 (not listed)

D. Close the dialog.

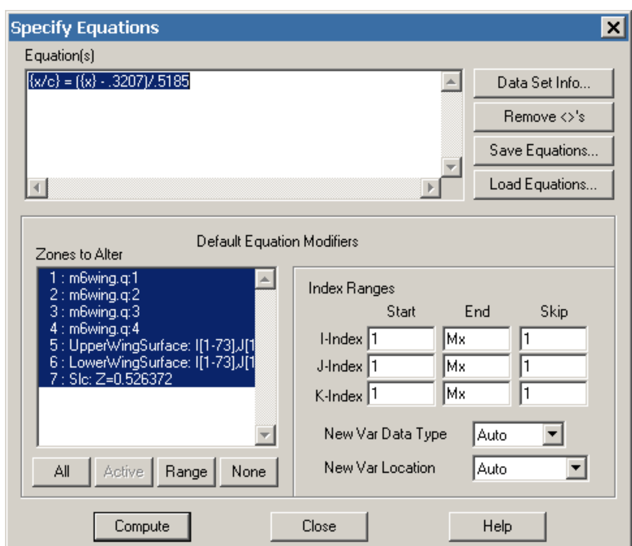
Step 19: Create a new variable

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



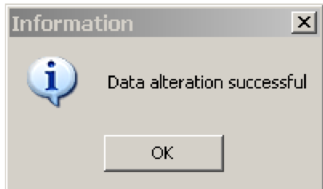
B. Enter the following equation in the **Specify Equations** dialog:

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



C. Select the ***Compute*** button.

D. Select ***OK*** in the **Information** dialog.



Data set 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.

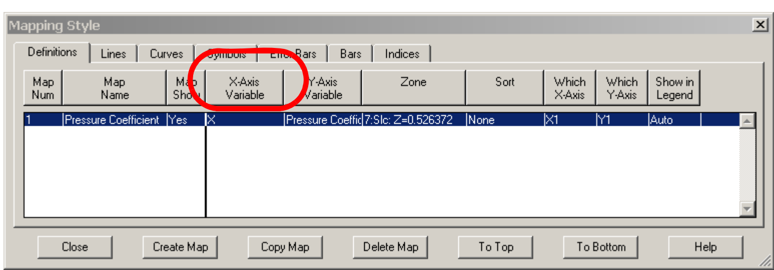
In the above dialog, a new variable {x/c} was created using shifted and scaled values of the existing {x} variable.

E. Close the dialog.

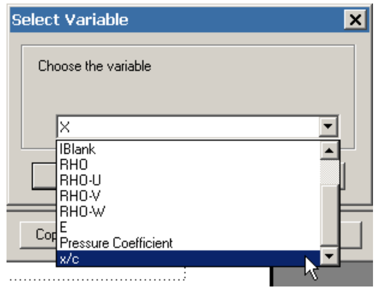
Step 20: Change the x-axis variable

A. With the **Mapping Style** dialog still open, highlight the remaining mapping.

B. Select the *X-axis Variable* button.

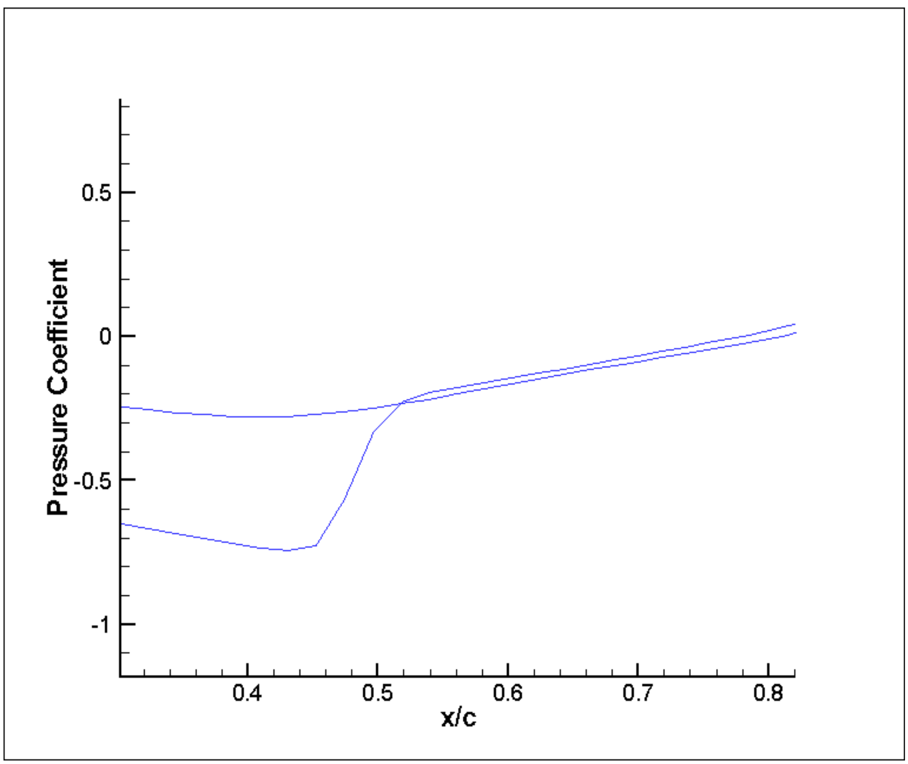


C. In the **Select Variable** dialog, select “x/c”.

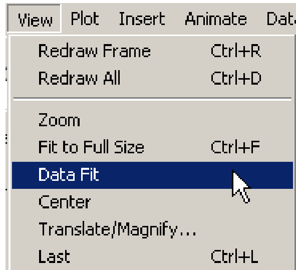


D. Press “OK” to assign the variable and close the dialog.

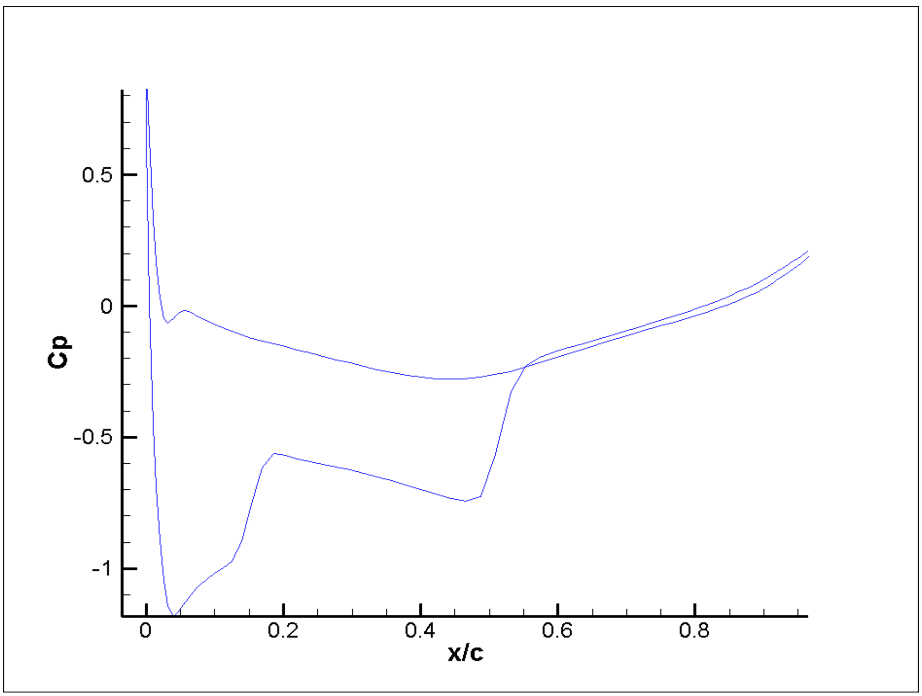
The plot will look as follows:



Step 21: Go to View>Data Fit



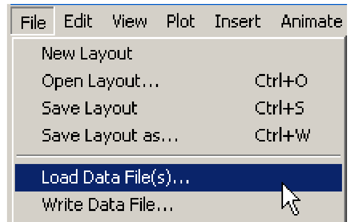
The plot will look as follows:



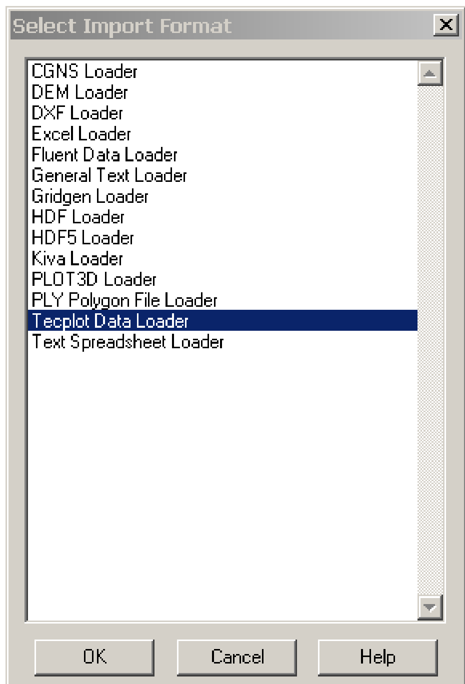
3 - 7 Compare Simulation Data with Measured Data

Step 22: Load the measured data file

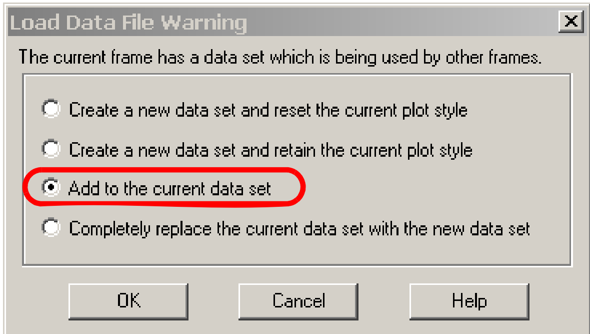
A. Go to **File>Load Data File**



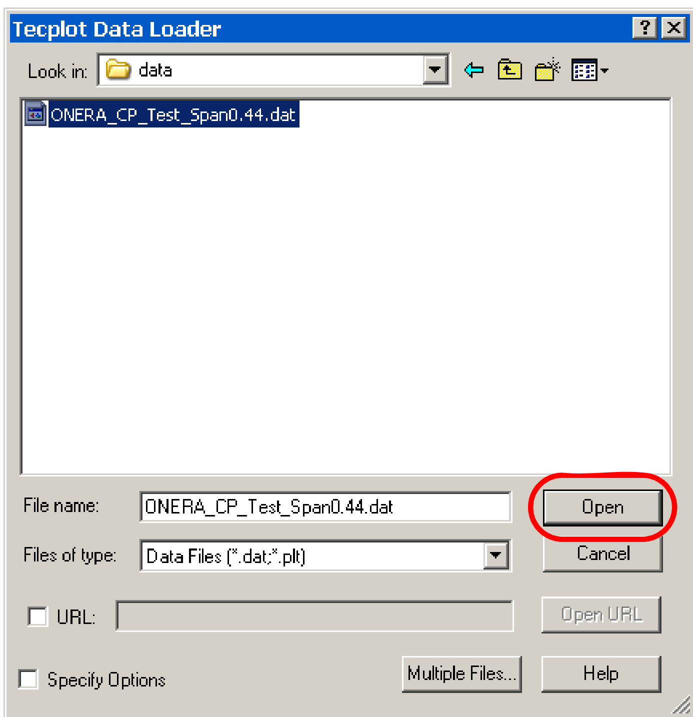
B. In the **Select Import Format** dialog, select the "Tecplot Data Loader".



C. In the **Load Data File Warning** dialog, select the “Add to the current data set” radio button and select the ***OK*** button.



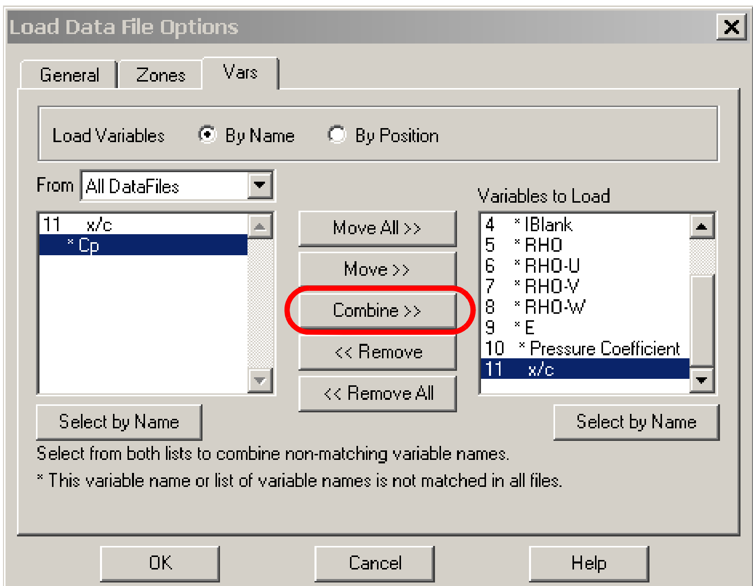
D. In the **Tecplot Data Loader** dialog, navigate to: *$TEC360HOME/demo/tutorials/external_flow/data* and select “*Onera_CP_Test_Span0.44.dat*”



E. Select the ***Open*** button to load the file.

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

G. Go to the ***Vars*** page of the **Load Data File 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 data set. 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 data sets, we need to combine the two pressure variables.

i. Select the Cp variable in the left-hand column.

ii. Select the Pressure Coefficient variable in the right-hand-column.

iii. Select “Combine” from the center column.

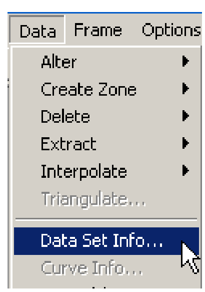
iv. Select ***OK*** to apply your changes and close the dialog.

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

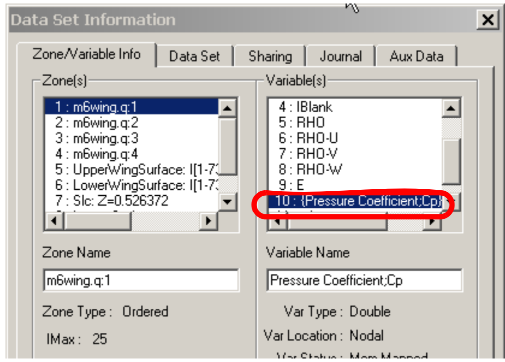
Step 23: Change the variable name

To simplify our axes labels, we are going to change the combined variable name.

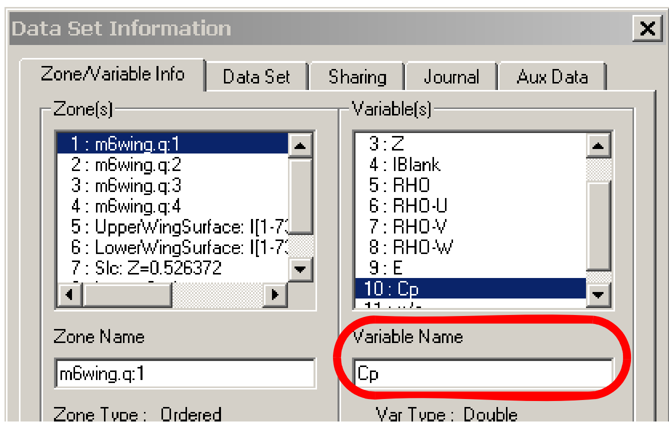
A. Go to **Data>Data Set Info**



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



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



The name change will be applied to all zones.

D. Close the dialog.

Step 24: Create New Mappings

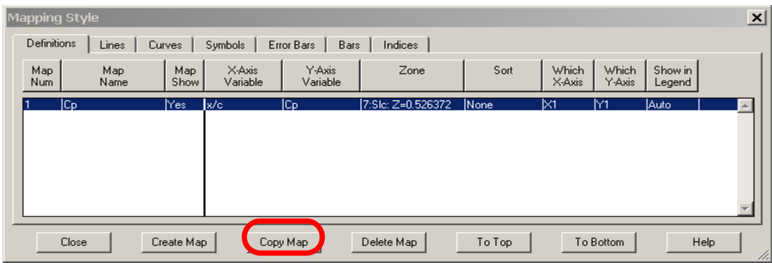
We now need to add mappings of the measured data set to our plot.

A. Open the **Mapping Style** dialog, by either:

i. Selecting the Mapping Style button from the Sidebar.

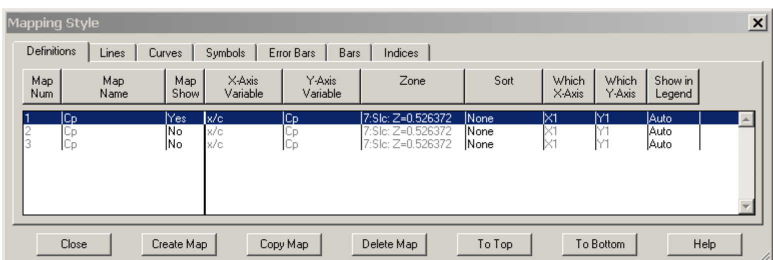
ii. Selecting **Plot>Mapping Style**.

B. Select the existing mapping.



C. Select the *Copy Map* button twice.

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



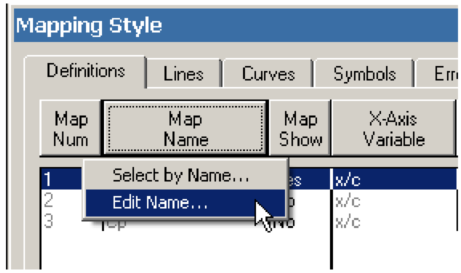
Step 25: Rename the Mappings

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

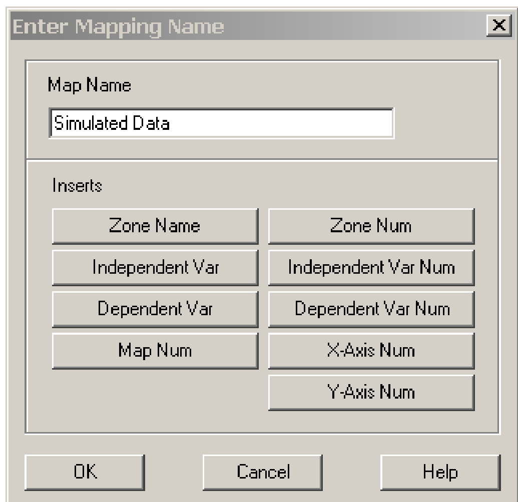
A. Rename Map 1.

i. Select Map 1.

ii. Select “Edit Name” from the *Map Name* menu.



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

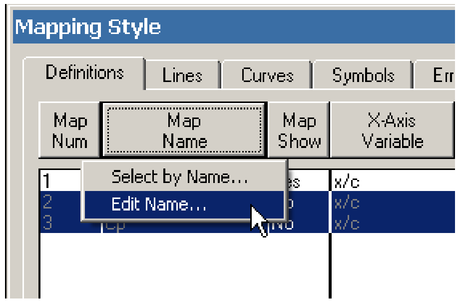


iv. Select the ***OK*** button to apply the changes and close the dialog.

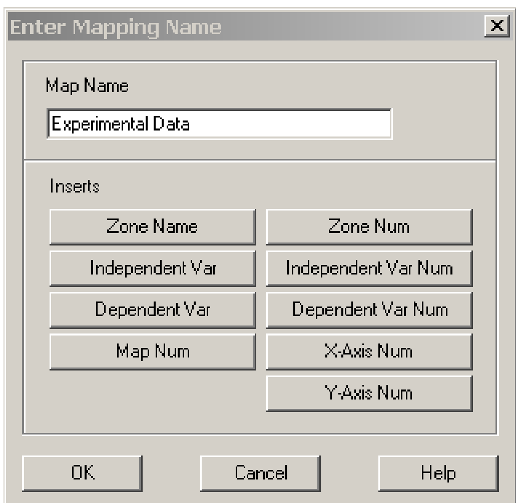
B. Rename Maps 2 & 3.

i. Select Maps 2 & 3.

ii. Select “Edit Name” from the *Map Name* menu.



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



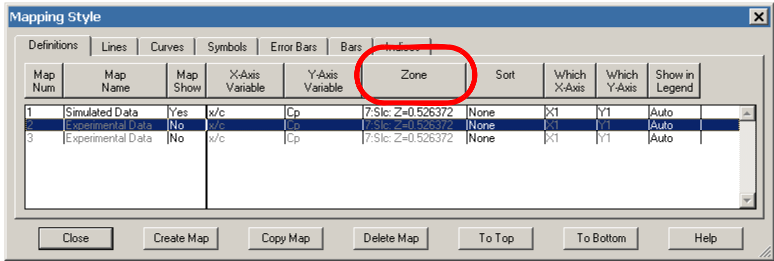
iv. Select the ***OK*** button to apply the changes and close the dialog.

Step 26: Change the zone assignments for the experimental data mappings

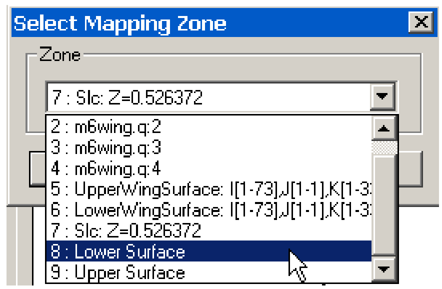
A. Map Num 2:

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

ii. Select the *Zone* button.



iii. Select “Zone 8” from the **Select Mapping Zone** dialog.

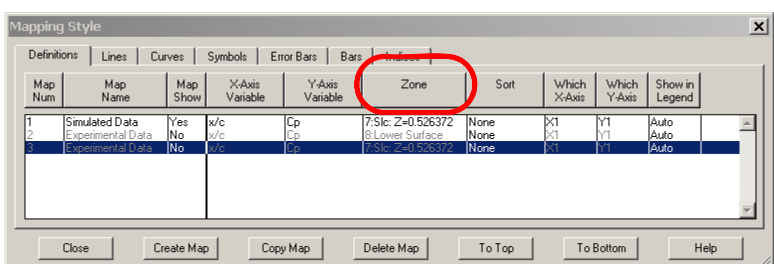


iv. Click ***OK*** to apply the changes and close the dialog.

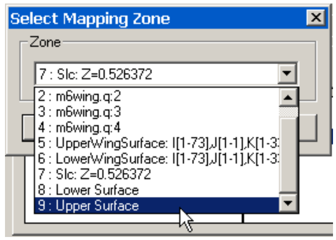
B. Map Num 3:

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

ii. Select the *Zone* button.



iii. Select “Zone 9” from the **Select Mapping Zone** dialog.



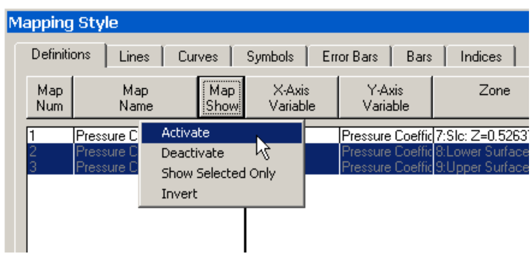
iv. Click ***OK*** to apply the changes and close the dialog.

Step 27: Activate the new mappings

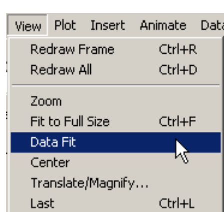
The new mappings have not appeared in the plot yet because they have not been activated.

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

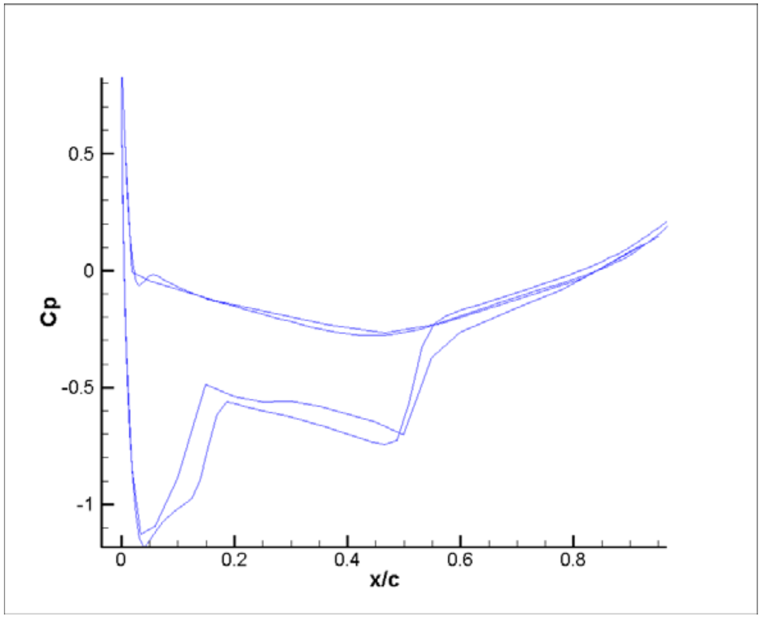
B. Select “Activate” from the *Map Show* menu.



C. Go to **View>Data Fit.**



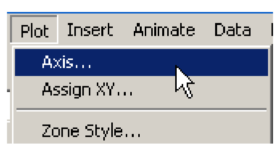
The plot should look as follows:



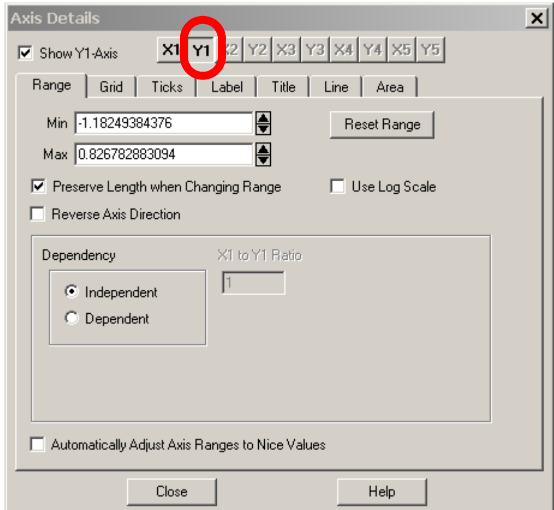
Step 28: Reverse the Y-axes

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

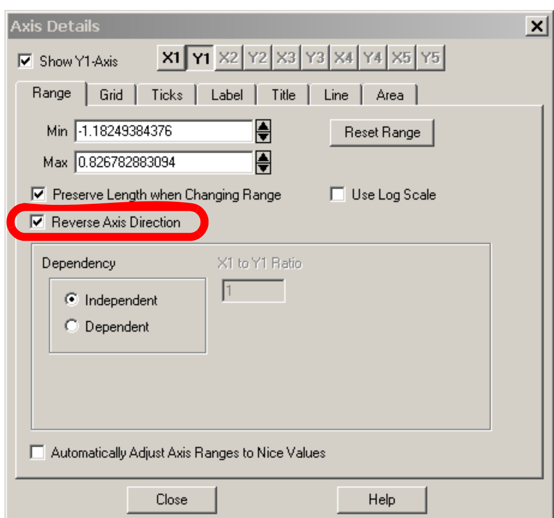
A. Go to **Plot>Axis**



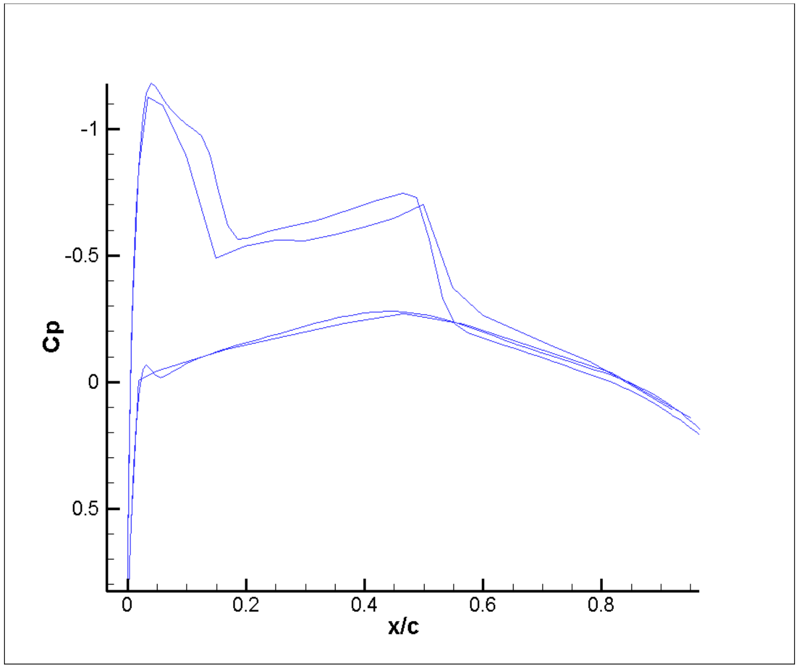
B. In the **Axis Details** dialog, select the Y1 button.



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



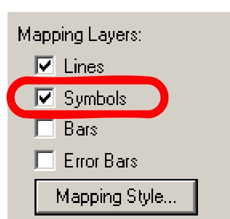
The plot will look as follows:



Step 29: 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.

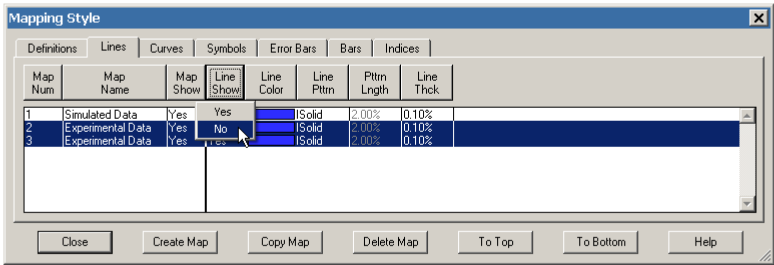
A. Toggle-on the Symbol layer in the Sidebar.



B. On the ***Lines*** page of the **Mapping Style** dialog:

i. Select Maps 2 & 3.

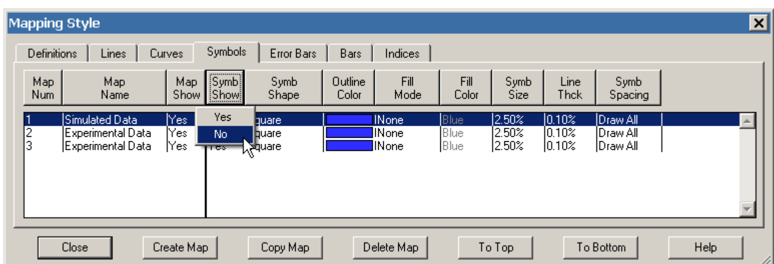
ii. Select the “No” from the *Line Show* button.



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

i. Select Map 1.

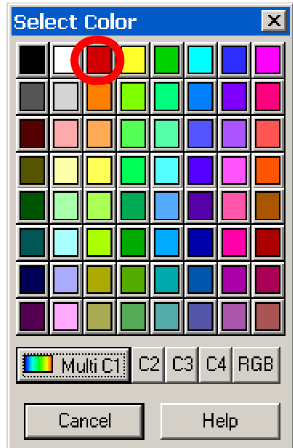
ii. Select “No” from the *Symb Show* button.



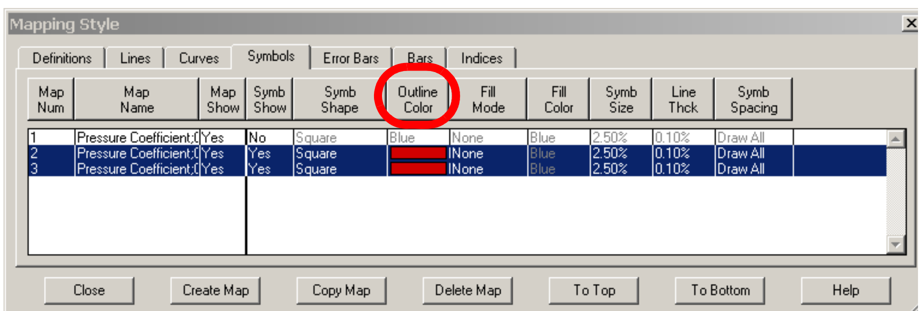
iii. Select Maps 2 & 3.

iv. Select the *Outline Color* button.

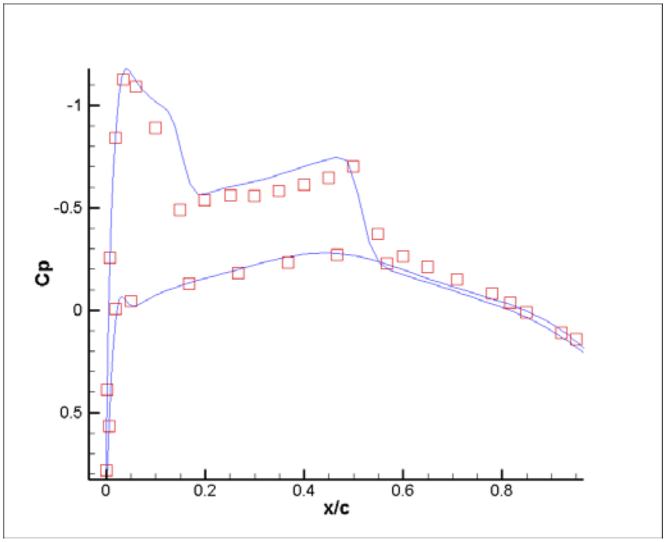
v. Select “Red” from the **Select Color** dialog.



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



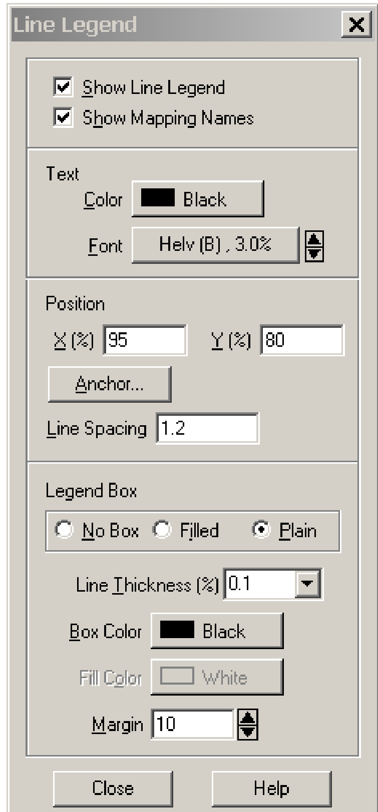
The plot will look as follows:



Step 30: Add a Line Legend

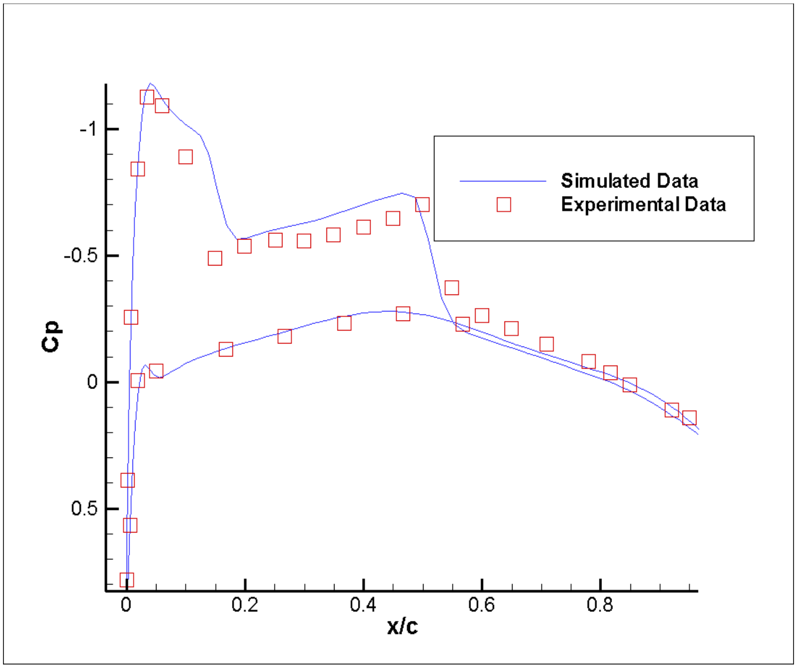
A. Go to **Plot>Line Legend.**

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



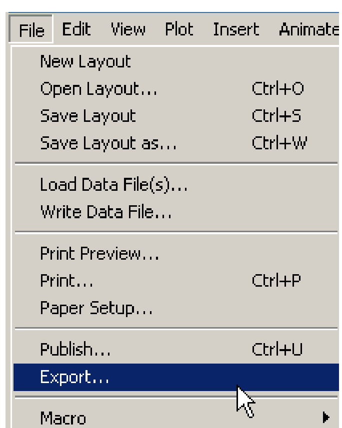
C. Accept the remaining default settings by closing the dialog.

The final plot will look as follows:

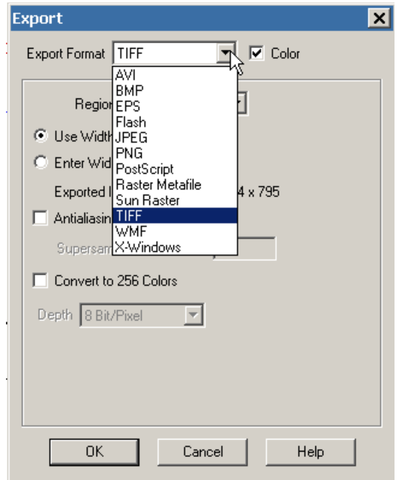


Step 31: Export the frames to a file

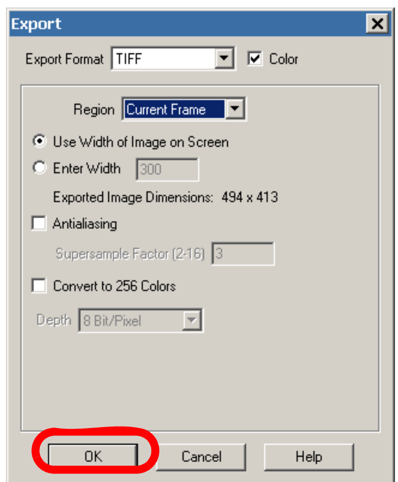
A. Go to **File>Export.**



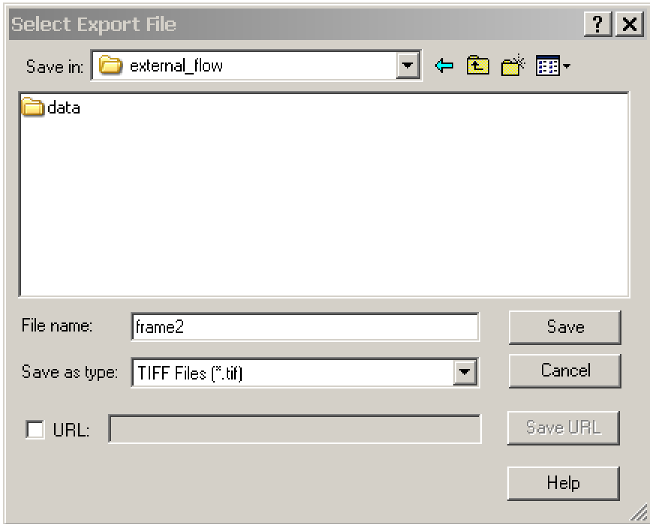
B. In the **Export** dialog, select TIF from the ***Export Format*** menu.



C. Accept the remaining default settings by pressing the OK button.



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



E. Press the ***Save*** button to export the image.

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

Chapter 4 Internal Flow Tutorial

4 - 1 Introduction

4- 1.1 Tutorial Summary

The following tutorial demonstrates how to visualize the internal flow during the mixing process. The tutorial should take approximately 20 minutes to complete and includes the following topics:

Step 1: Load a Fluent File
Step 2: Use the Sidebar to activate and deactivate Plot Layers
Step 3: Rotate the plot
Step 4: Use Depth Blanking
Step 5: Turn off Shading for selected zones
Step 6: Accentuate the Blade
Step 7: Lighting Effect
Step 8: Change the Frame Background Color
Step 9: Adjust the Lighting
Step 10: Activate Streamtraces
Step 11: Position streamtraces using the Add Streamtraces tool
Step 12: [OPTIONAL] Position streamtraces using the Streamtrace Details dialog.

All supporting data files for this tutorial can be found in: *$TEC360HOME/demo/tutorial/internal_flow*, where $TEC360HOME is the installation directory for Tecplot 360. For Windows users, this is typically C:/Program Files/Tecplot/Tec360.

4- 1.2 Background information

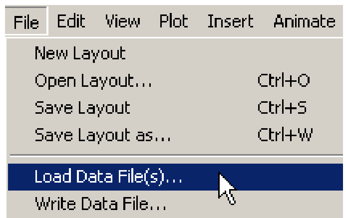
There are four primary steps for manufacturing soap and detergent: mixing, heat transfer, drying and separation. In both liquid and powder detergent manufacturing, dry and liquid ingredients are blended in a uniform mixture using static mixers, called crutchers. Location of the dry and liquid inlets,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

4 - 2 Getting Started

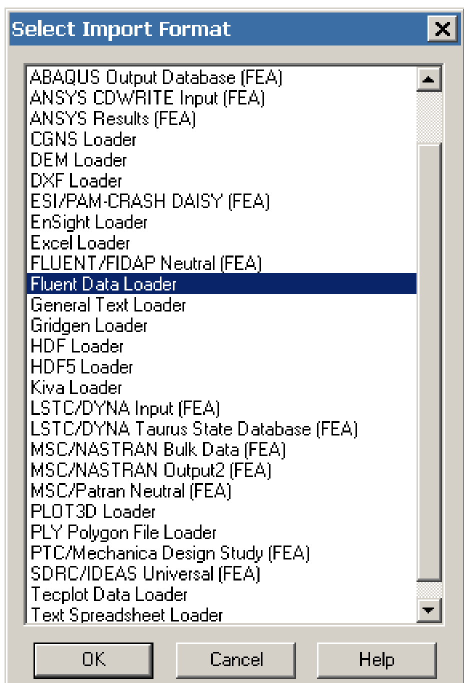
Step 1:Load a Fluent File

The **Fluent Data Loader** allows you to load case and/or data files. For this tutorial, we will load one set of case and data files.

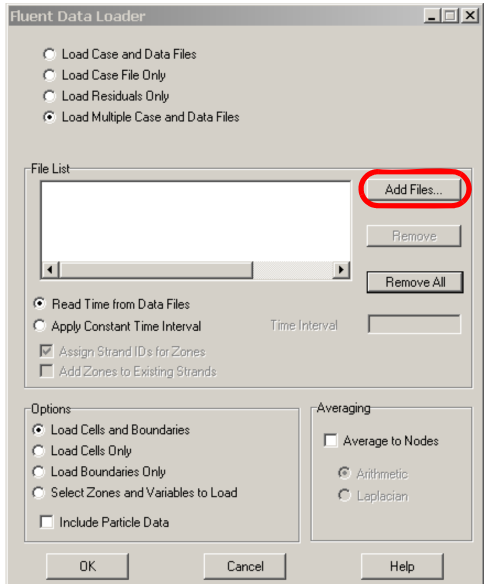
A. Go to **File>Load Data File(s)**.



A. Select **Fluent Data Loader** from the **Select Import Format** dialog.

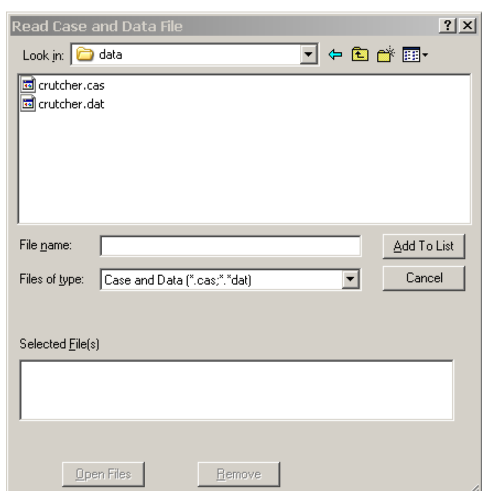


B. In the **Fluent Data Loader** dialog, select the ***Add Files...*** button.

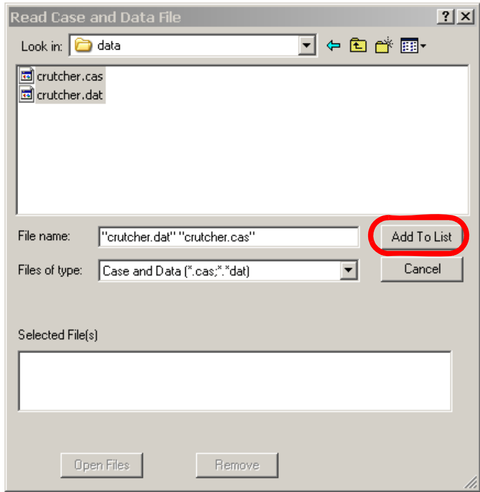


C. In the **Read Case and Data File** dialog:

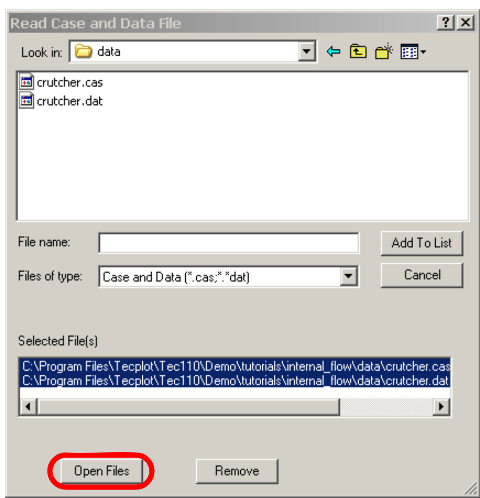
i. Navigate to: *$TEC360HOME/demo/360_tutorials/internal_flow/ data*[4]



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

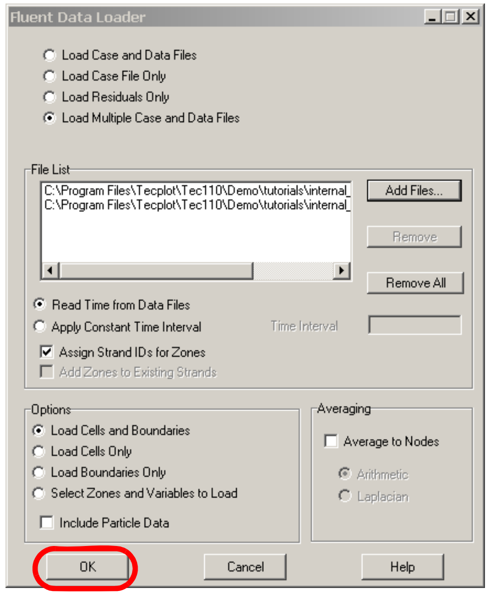


iii. Select the ***Open Files*** button

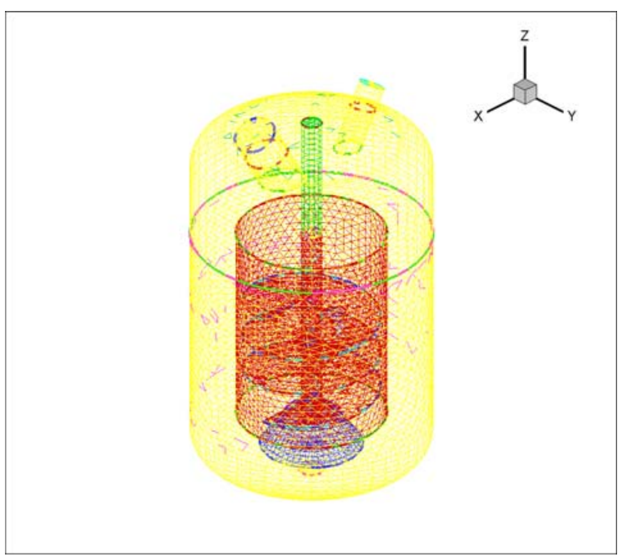


4. $TEC360HOME is the installation directory for Tecplot 360. For Windows users, this is typically C:/Program Files/Tecplot/Tec360.

D. In the **Fluent Data Loader** dialog, accept the remaining default settings and select “OK” to close the dialog.

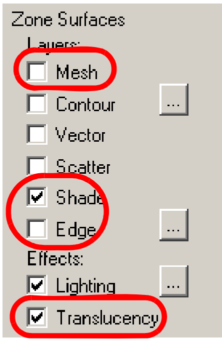


The initial view of the plot will look as follows:



Step 2: Use the Sidebar to activate and deactivate Plot Layers

By default, the Mesh and Edge layers are toggled-on when a data file is first loaded into Tecplot. In this case, the number of zones in the data (indicated by the different mesh colors) make it difficult to discern the objects in the plot.



A. Using the Sidebar, toggle-off the Mesh layer.

B. Toggle-on the Shade layer.

C. Toggle-off the Edge layer.

D. Toggle-on the Translucency zone effect.

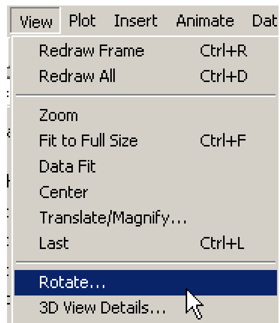
With a few clicks of the mouse, we have transformed the appearance of our data set.

4 - 3 Adjust the Plot View

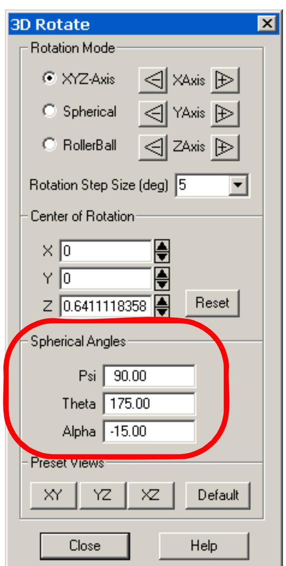
Step 3: Rotate the plot

We would like to have a better view of the mixer's interior, without eliminating any zones. Depth Blanking will allow us to remove a percentage of the physical data in the plot, independently of the zones. The percentage is calculated based upon the current position of the plot, so we must rotate the image to avoid removing the spouts from the plot.

A. Select **Rotate** from the **View** menu.



B. In the **3D Rotate** dialog,



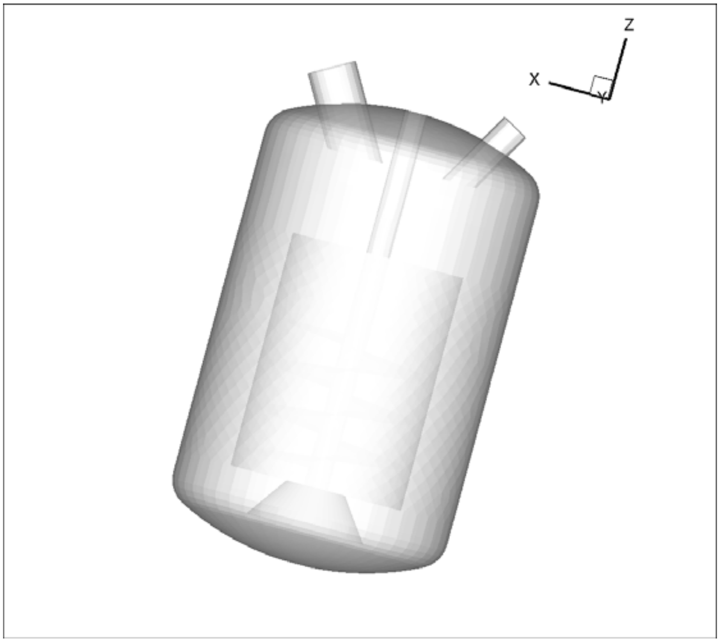
i. Set Psi = 90.

ii. Set Theta = 175.

iii. Set Alpha = -15.

iv. Close the dialog.

Your plot should look as follows:



You can also rotate your plot using the rotation tools from the sidebar.

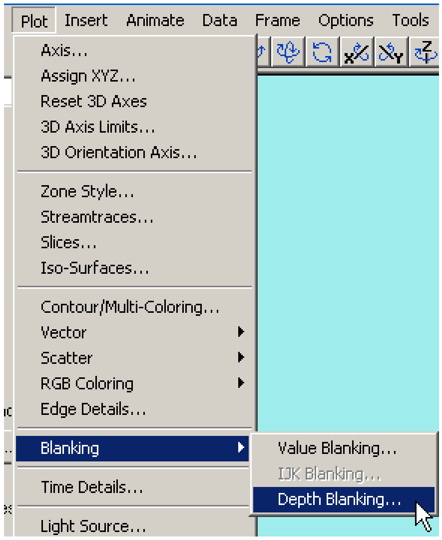
- Spherical  - Drag the mouse horizontally to rotate about the Z-axis; drag the mouse vertically to control the tilt of the Z-axis.
- Rollerball  - Drag the mouse in the direction to move with respect to the current orientation on the screen. In this mode, your mouse acts much like a rollerball.
- Twist  - Drag the mouse clockwise around the image to rotate the image clockwise. Drag the mouse counterclockwise around the image to rotate the image counterclockwise.
- X-axis  - Drag the mouse to rotate the image about the X-axis.
- Y-axis  - Drag the mouse to rotate the image about the Y-axis.
- Z-axis  - Drag the mouse to rotate the image about the Z-axis.

You can change the center of rotation, in any rotation mode, by selecting a point with your mouse and hitting “o” on the keyboard.

Step 4: Use Depth Blanking

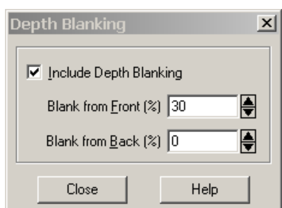
The rotation settings used in Step 3: A. were selected in anticipation of depth blanking.

A. Go to the **Plot** menu and select **Blanking>Depth Blanking**.



B. In the **Depth Blanking** dialog, toggle-on ***Include Depth Blanking***.

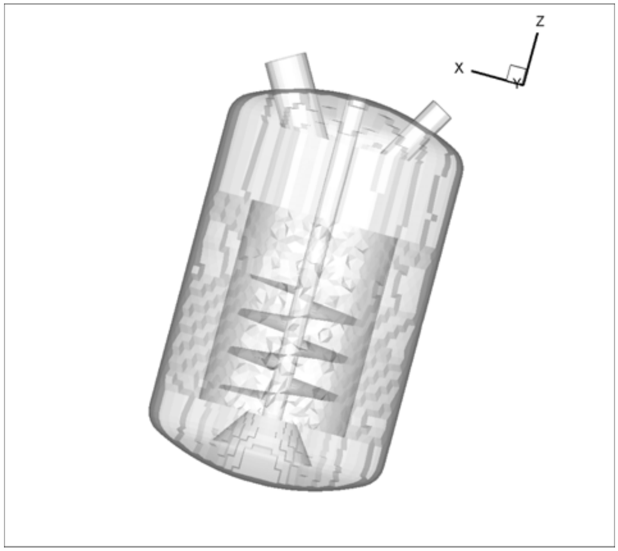
C. Enter “30” in ***Blank from Front (%)*** field.



NOTE: Depth Blanking blanks whole cells in your plot.

D. Close the dialog.

The plot should look as follows.



4 - 4 Use the Zone Style dialog to customize settings

In the previous steps of this tutorial, we were changing settings globally (to all zones). Now we will begin to fine-tune the settings for individual zones in the data set. In the Zone Surfaces block of the Sidebar, select the “Zone Style” button.

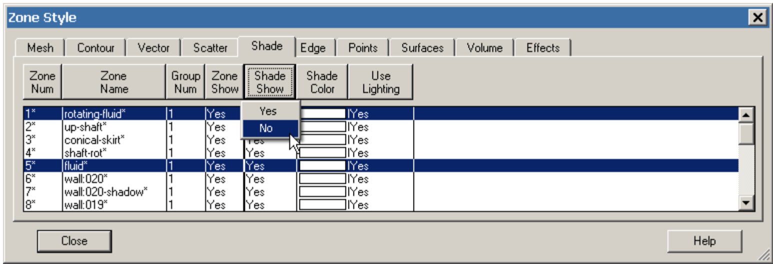
Step 5: Turn off Shading for selected zones

There is stair-stepping in our current image due to whole-cell blanking of the rotating fluid.

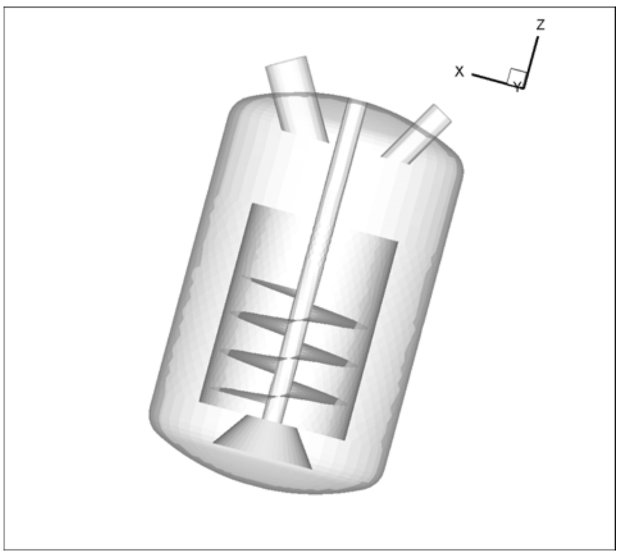
A. On the ***Shade*** page of the **Zone Style** dialog, select Zone 1 (rotating-fluid) and Zone 5 (fluid) using the CTRL key.

Alternatively, you can select the zones by name by choosing “Select by Name” from the *Zone Name* drop-down and entering “*fluid*” in the **Enter Text String** dialog.

B. Select the *Shade Show* column label and Select “No”.



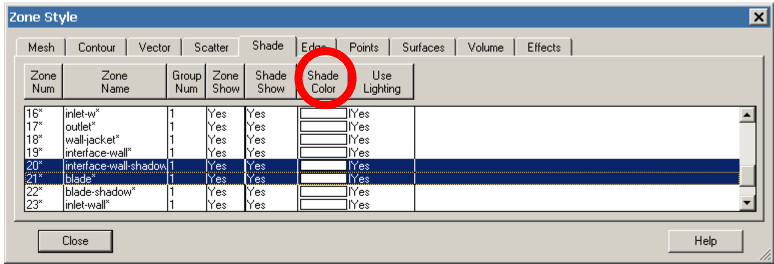
The plot will look as follows.



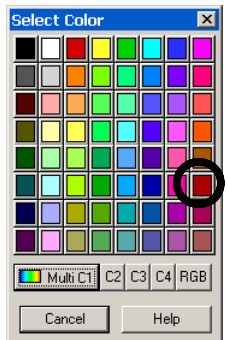
Step 6: Accentuate the Blade

A. While still on the ***Shade*** page of the **Zone Style** dialog, highlight zones 21 & 22 (using the SHIFT key) or by clicking-and-dragging with your mouse.

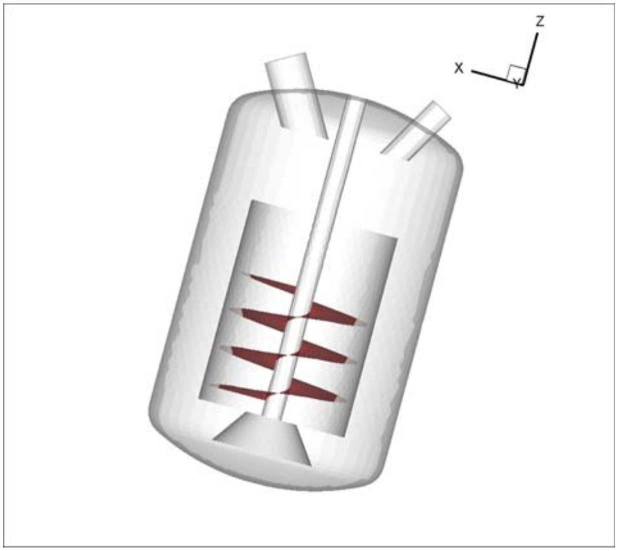
B. Select the *Shade Color* button.



C. Select a red color from the **Select Color** dialog,



Your plot should look as follows:

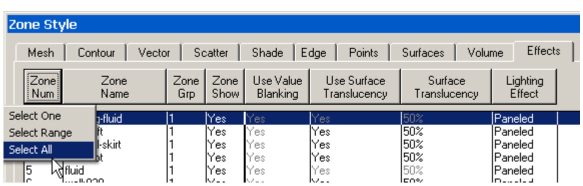


Step 7: Lighting Effect

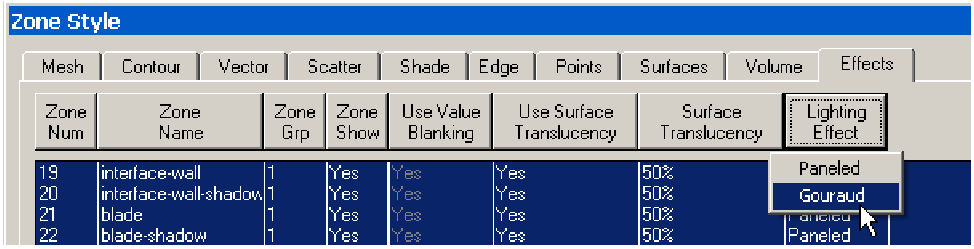
By default, Tecplot sets the shading to be paneled (a single shading value is used for each cell). Tecplot also offers a continuous shading (Gouraud), which provides a cleaner image.

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

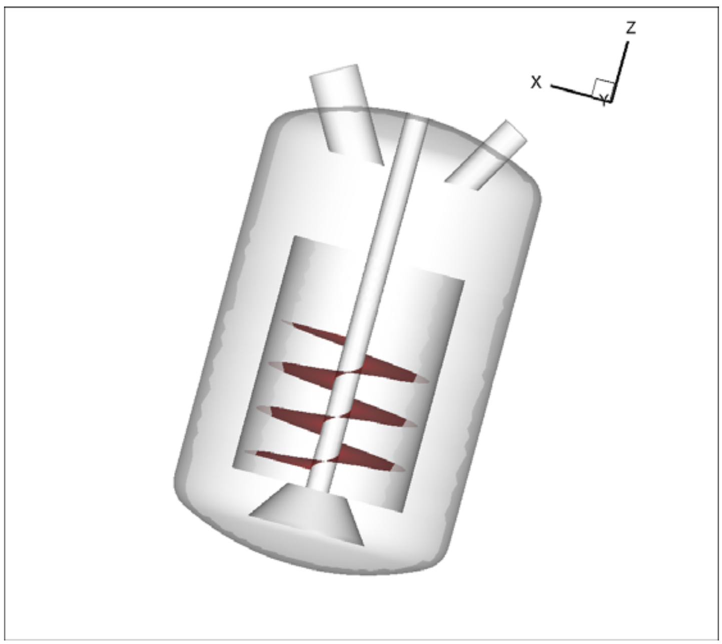
B. Click on the *Zone Num* column and select “Select All”.



C. Go to the *Lighting Effect* column and select “Gouraud”.



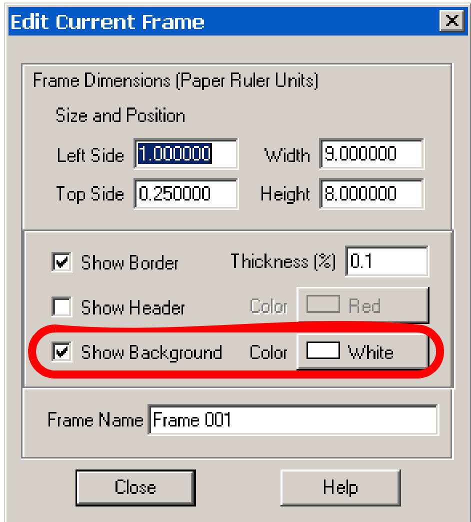
The result should look as follows:



Step 8: Change the Frame Background Color

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

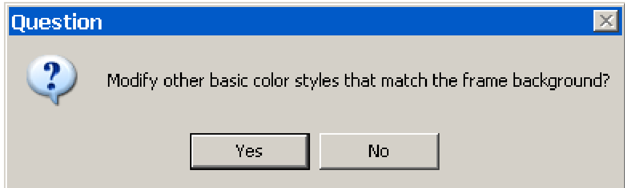
A. Go to **Frame>Edit Current Frame**



B. Toggle-on ***Show Background***

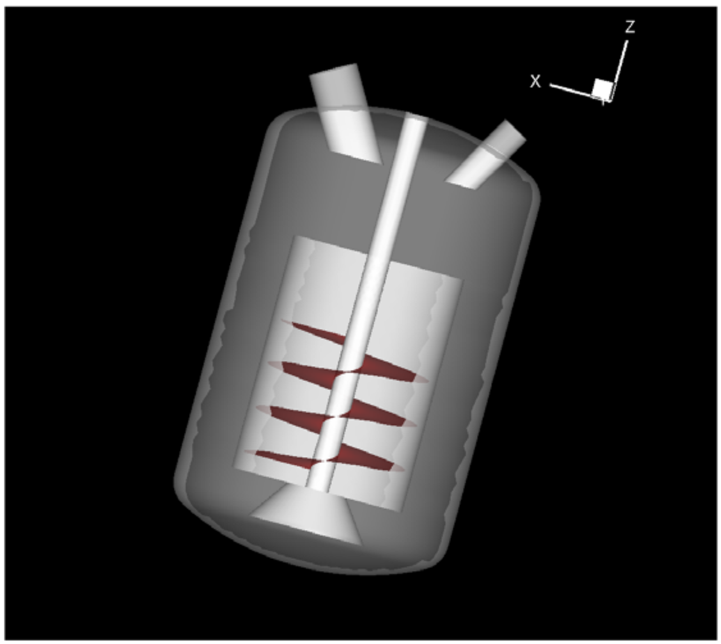
C. Select the color box and select the black box in the top-left corner of the **Select Color** dialog.

D. Select “Yes” in the confirmation box.



E. Close the **Edit Current Frame** dialog.

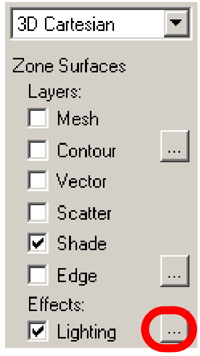
The result should look as follows:



Step 9: Adjust the Lighting

A. Open the **Light Source** dialog by either:

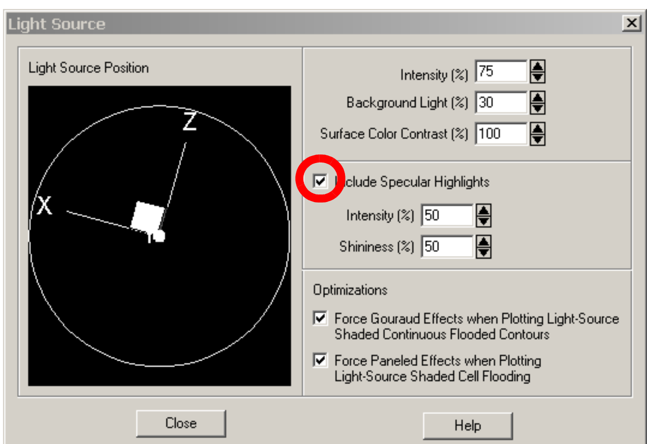
i. Selecting ... button to the right of Lighting in the Sidebar.



ii. Going to **Plot>Lighting Source**.

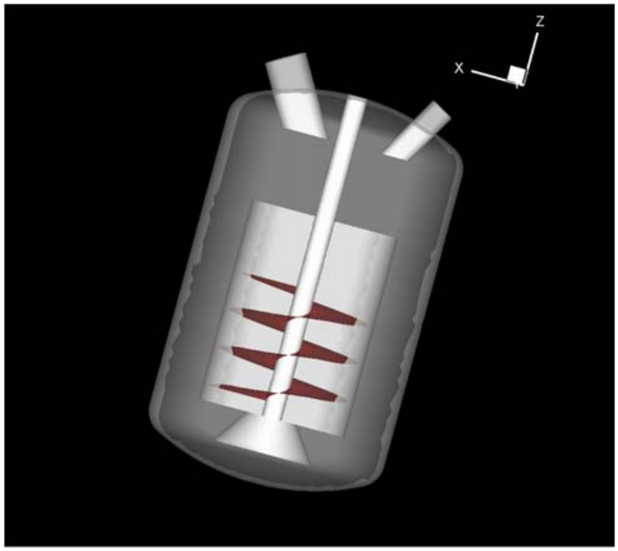
B. In the **Light Source** dialog, toggle-on ***Include Specular Highlighting***.

C. Accept the remaining default settings and close the dialog.



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

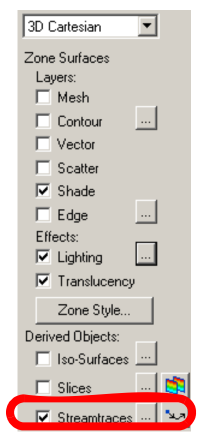
The result should look as follows:



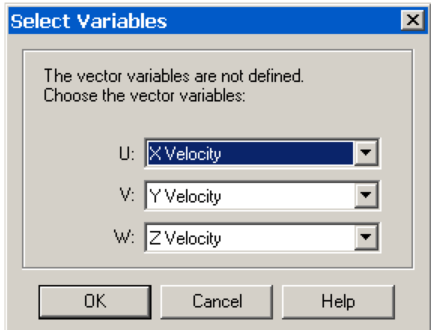
Step 10: Activate Streamtraces

We would like to visualize the internal flow of the mixer using streamtraces. Streamtrace controls are located in the Sidebar and in the **Streamtrace Details** dialog (accessed via **Plot>Streamtraces**).

A. Toggle-on the Streamtraces button in the Sidebar.



B. Accept the default vector variables in the **Select Variables** dialog by selecting “OK”.



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

Step 11: Position streamtraces using the Add Streamtraces tool

Streamtraces can be added to the plot interactively using either the  tool or the ***Position*** page of the **Streamtrace Details** dialog (accessed via **Plot>Streamtraces** or the button in the Sidebar).

A. Select the  tool from the Toolbar.

NOTE: The cursor has changed from an arrow to a “+”.

B. Click-and-drag within your plot. Streamtraces will automatically be added to your plot.

Step 12: [OPTIONAL] Position streamtraces using the Streamtrace Details dialog.

To precisely match our plot, you can also place streamtraces using the **Streamtrace Details** dialog.

A. Open the **Streamtrace Details** dialog by either selecting

Plot>Streamtraces or the ... button to the right of Streamtraces in the Sidebar.

B. Specify the following settings:

i. Toggle-on *Create Rake*

A rake is a group of Streamtraces. The number of streamtraces in the rake is specified at the bottom of the dialog.

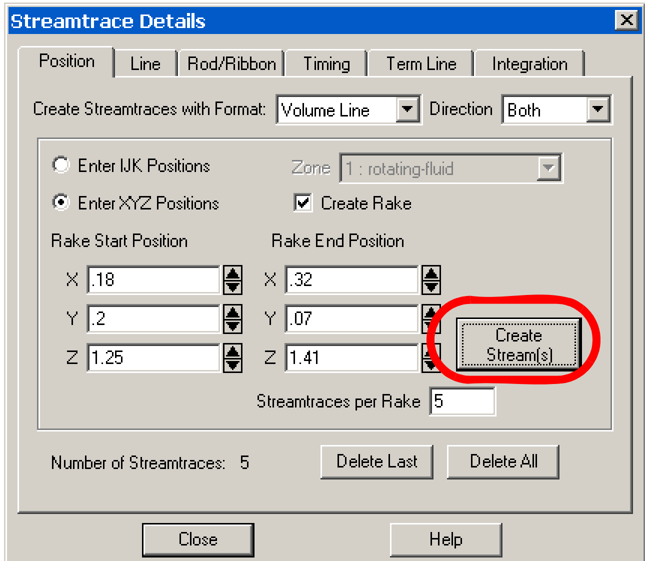
ii. *Rake Start Position*:

X = .18
Y = .2
Z = 1.25

iii. Rake End Position:

X = .32
Y = .07
Z = 1.41

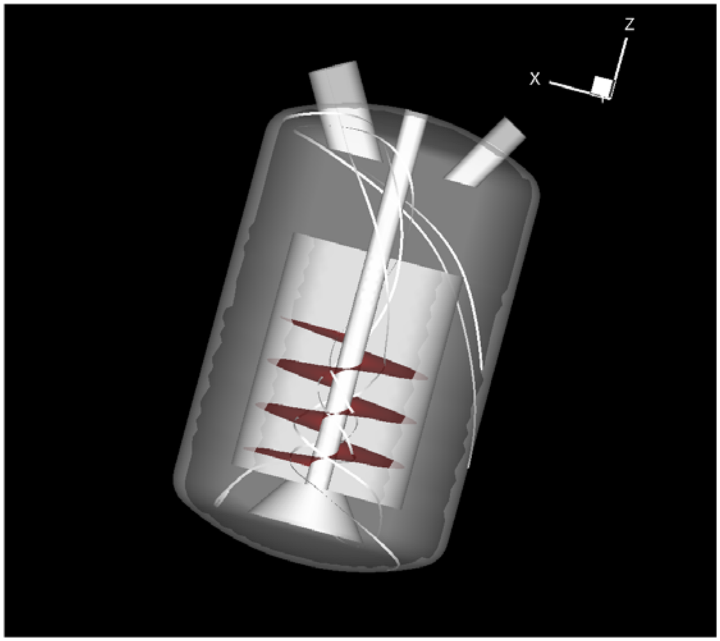
iv. *Streamtraces per Rake* = 5



C. Select the *Create Stream(s)* button.

D. Close the dialog.

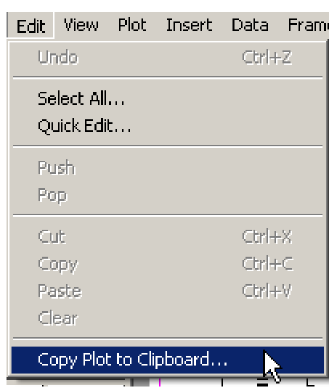
Your final plot should look as follows:



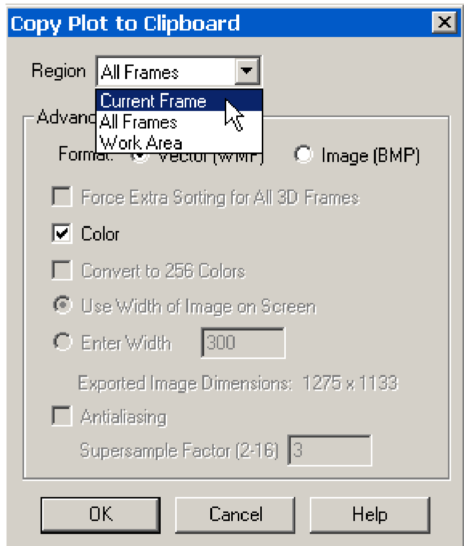
Step 13: Copy Plot to Clipboard

Now that your plot is complete, you can copy it to the Tecplot clipboard to be pasted into other software (such as Power Point).

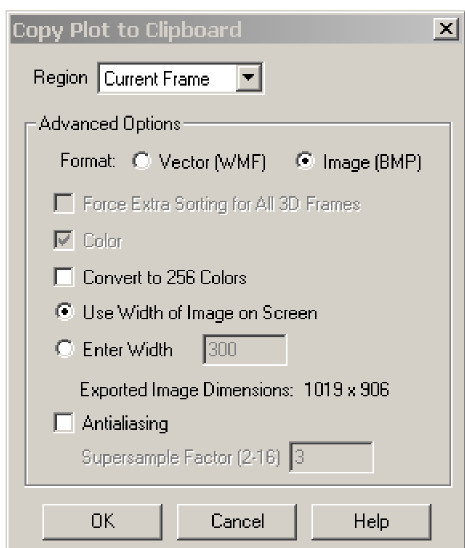
A. Go to **Edit> Copy Plot to Clipboard**



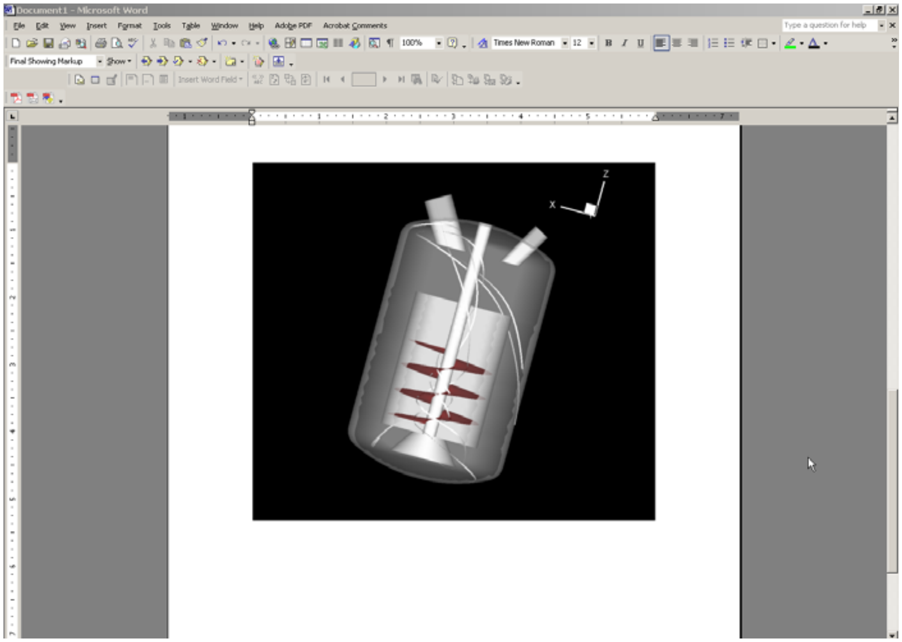
B. In the **Copy Plot to Clipboard** dialog, select *Current Frame* from the ***Region*** drop-down menu.



C. Select the Image (BMP) radio button.



D. Select the ***OK*** button. The plot can now be pasted into other software, such as Microsoft Word.



Chapter 5 Transient Data Tutorial

5 - 1 Introduction

5- 1.1 Tutorial Summary

Tecplot is an extremely useful tool for comparing simulated and measured data sets. In this tutorial, we present how to load a Fluent data set, generate iso-surfaces and animate the time steps in a data file.

The tutorial takes approximately 15-20 minutes to complete and includes the following steps:

Step 1: Load a Fluent Data File
Step 2: Focus on the Region of Interest (ROI)
Step 3: Rotate the Plot
Step 4: Translate the Data
Step 5: Change the active plot layers
Step 6: Turn-off select zones
Step 7: Change the Shade color for select zones.
Step 8: Toggle-on Iso-surfaces in the Sidebar.
Step 9: Define the Iso-surface positions
Step 10: Add translucency to the iso-surfaces.
Step 11: Animate the store separation on your screen
Step 12: Save your animation to a file

All supporting data files for this tutorial can be found in: *$TEC360HOME/demo/tutorial/transient/data*, where $TEC360HOME is the installation directory for Tecplot 360. For Windows users, this is typically C:/Program Files/Tecplot/Tec360. Additional time steps can be downloaded from our website at ftp://ftp.tecplot.com/pub/tecplot/training/transient_data.zip. The steps in the tutorial can be completed without the additional time steps, but the screenshots are taken with all time steps included.

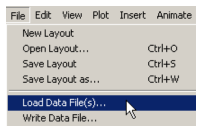
5- 1.2 Background Information

The data set provided in this tutorial, which presents simulations of the release of a store from an aircraft, was presented by Snyder *et al*[5] at the 33rd Annual AIAA Fluid Dynamics Conference and Exhibit. The wing is a 45-degree clipped delta with 25 foot root chord length and 21.67 foot semi-span. The store consists of a tangent-octave forebody, clipped tangent-octave afterbody, and cylindrical centerbody of 1.67 foot diameter. The store includes 4 fins, each with a 45-degree sweep clipped delta wing.

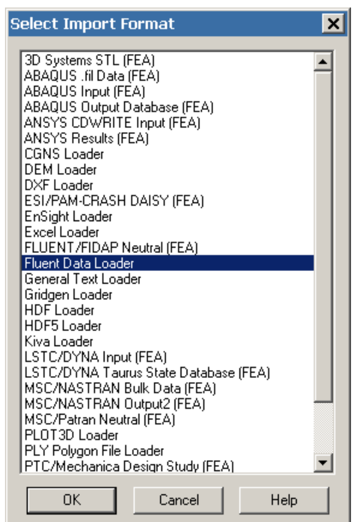
5 - 2 Getting Started

Step 1:Load a Fluent Data File

A. Select **Load Data Files** from the **File** menu



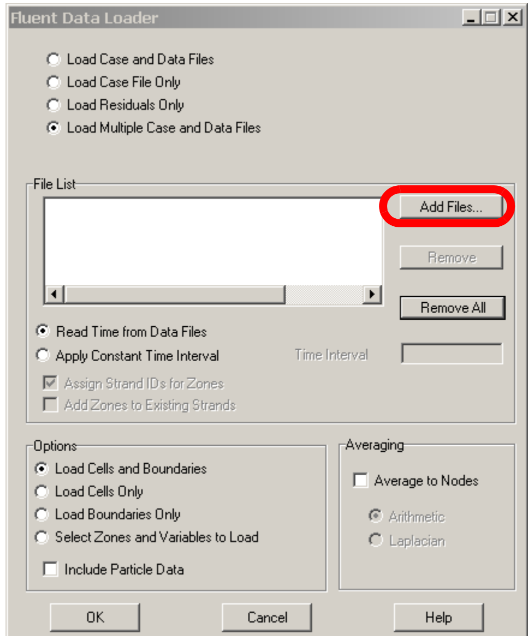
B. Select the “Fluent Data Loader” from the **Select Import Format** dialog



Be sure to select “Fluent Data Loader” and not “FLUENT/FIDAP Neutral - FEA”.

5. D. Snyder, E. Koutsavdis, and J. Antonnen, “Transonic Store Separation Using Unstructured CFD with Dynamic Meshing,” 33rd AIAA Fluid Dynamics Conference and Exhibit, Orlando, FL, June 23–26, 2003 (AIAA-2003-3919).

C. In the **Fluent Data Loader** dialog, select the ***Add Files*** button and navigate to: *$TEC360HOME/demo/tutorial/transient/data*[6]



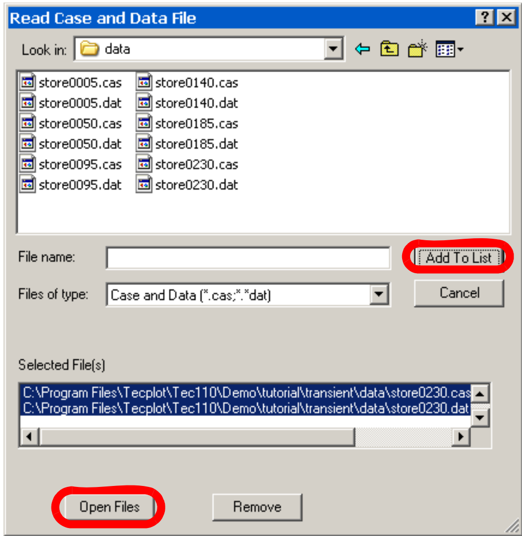
D. In the **Read Case and Data File** dialog:

i. Select all available files (using the SHIFT key).

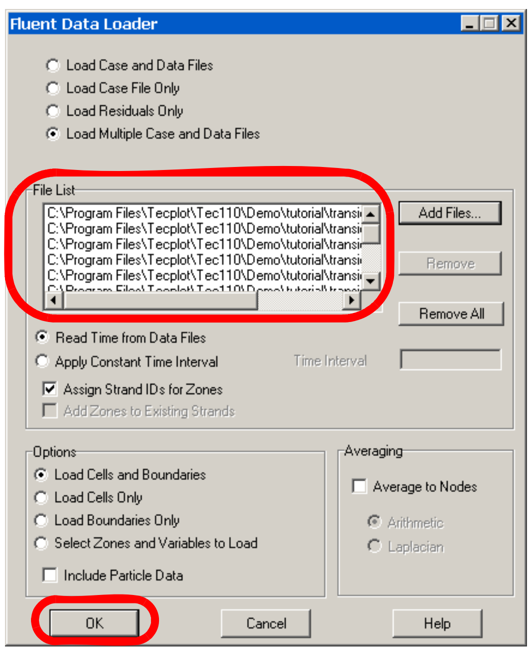
ii. Select the ***Add Files.../Add*** button (Windows/Unix).

6. $TEC360HOME is the installation directory for Tecplot 360. For Windows users, this is typically C:/Program Files/Tecplot/Tec360.

iii. Select the ***Open Files/OK*** button (Windows/Unix)



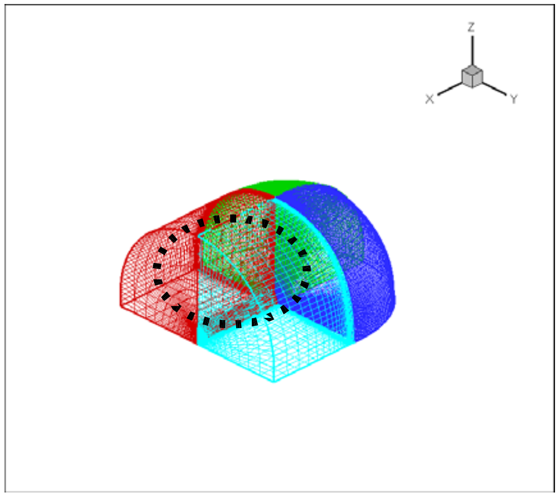
E. Select the ***OK*** button to finish the loading process.

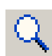


The imported data set is shown in the following figure.

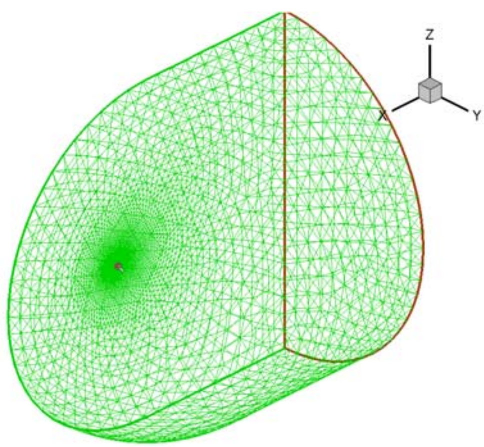
Step 2: Focus on the Region of Interest (ROI)

The region of interest is indicated with a dotted-black line in the following figure.

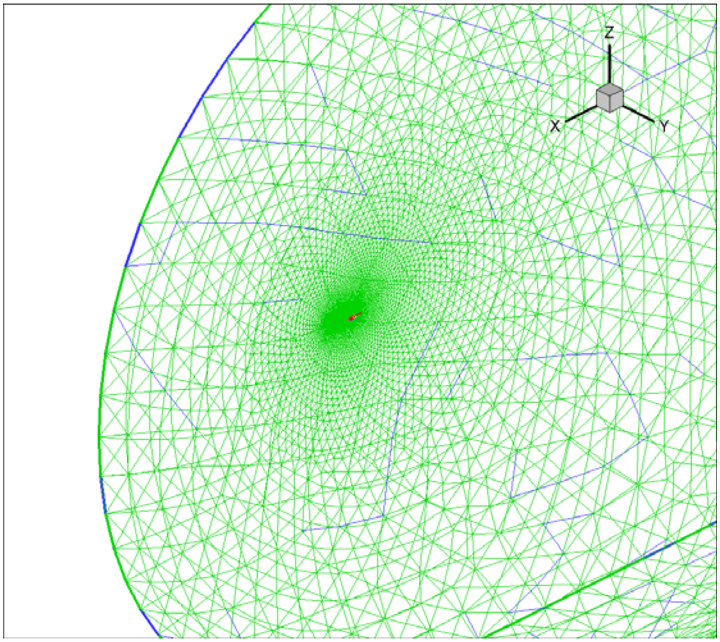


A. Select the  tool from the toolbar.

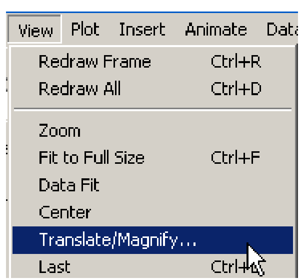
B. Center the cursor over the ROI and click once, as shown below.



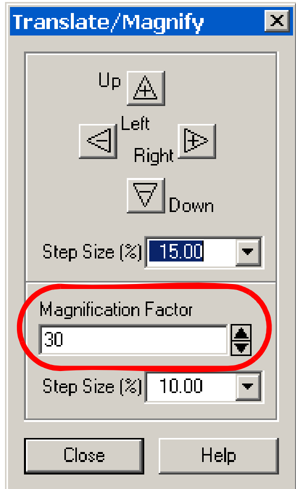
The result will look approximately like the following image:



C. Select **Translate/Magnify** from the **View** menu.



i. In the **Translate/Magnify** dialog, enter 30 in the *Magnification Factor* field.



ii. Close the dialog.

You can continue zooming by any of the following methods:

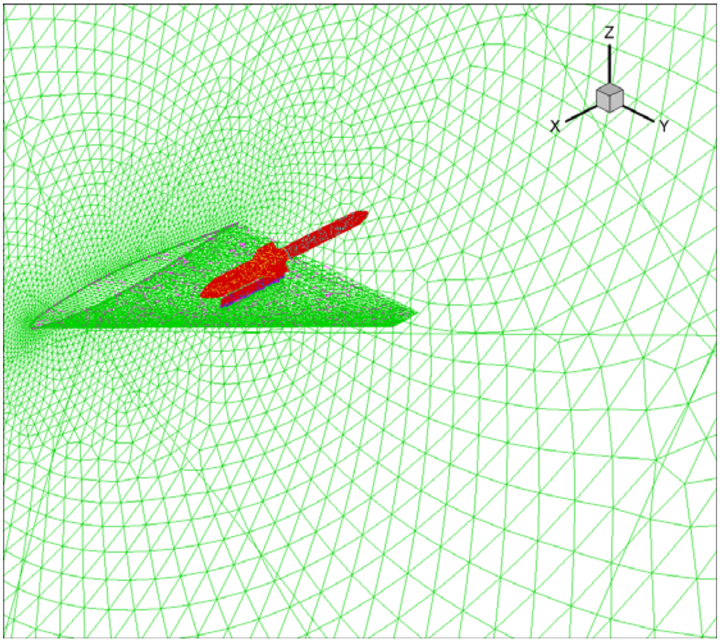
i. With the Zoom tool still active, continue clicking on the plot.

ii. With the Zoom tool still active, clicking-and-dragging the cursor to create a zoom window. Upon release, the frame is zoomed to the rubber band box.

iii. Using the **Translate/Magnify** dialog. We will use this dialog, so that you are able to perfectly replicate our plots.

iv. Using the middle-mouse button and clicking and dragging.

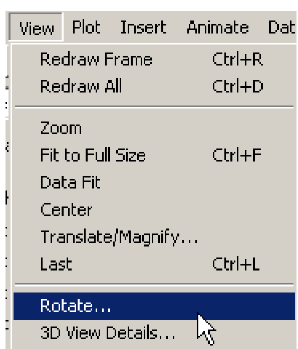
The result should look as follows:



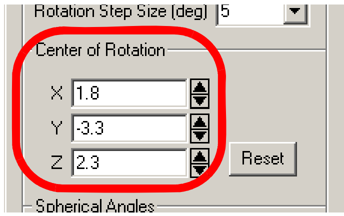
Step 3: Rotate the Plot

In order to get a better view of the wing and store, we will rotate the plot.

A. Select **Rotate** from the **View** menu.



B. In the **3D Rotate** dialog, enter the following coordinates for the ***Center of Rotation***:

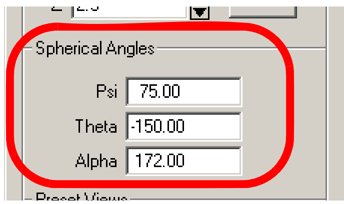


i. X = 1.8

ii. Y = -3.3

iii. Z = 2.3

C. Enter the following spherical angles:



i. Psi = 75

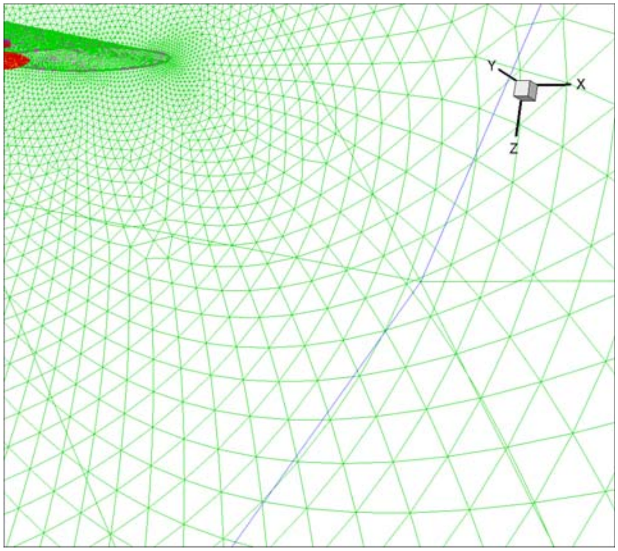
ii. Theta = -150

iii. Alpha = 172

Freehand spherical rotation can also be accomplished using the Spherical Rotation tool .

D. Close the dialog.

The rotation results should look as follows:



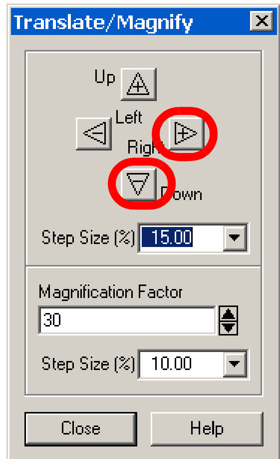
Step 4: Translate the Data

The rotation performed above moved part of the ROI outside of the frame. This can be easily remedied using any of the following options:

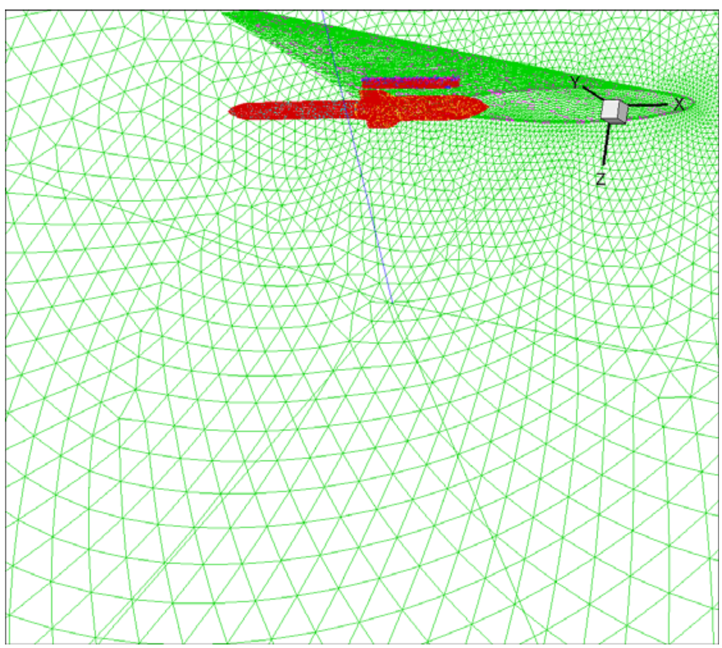
A. Selecting the Translate tool  .

B. Right-clicking-and-dragging the data.

C. Opening the **Translate/Magnify** dialog (accessed via the **View** menu). To closely match our plot, change the ***Step Size (%)*** to 15 and hit the right arrow five times and the down arrow once.



The result should look as follows:



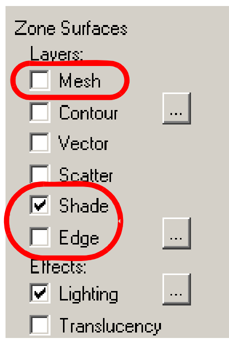
5 - 3 Customize the Plot Layers

The ultimate goal of this tutorial is to view an animation of the store separation and two pressure iso-surfaces. As such, the mesh and edge layers are more information than we need.

Step 5: Change the active plot layers

A. In the Sidebar, toggle-off Mesh and Edge layers.

B. Toggle-on the Shade layer.



The plot looks empty because the outer zone in the plot is no longer transparent.

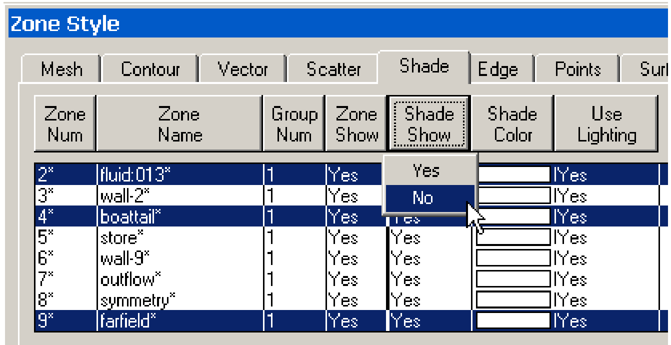
Step 6: Turn-off select zones

A. Open the **Zone Style** dialog (accessed via the Sidebar or the Plot menu).

B. Go to the ***Shade*** page of the dialog:

C. Select Zones 1, 2, 4 & 9 (using the CTRL key).

D. Select “No” from the *Shade Show* menu.



If your zones are not numbered 1 - 9, press the  button in the Sidebar to go back to the first time step. See 5 - 6 “Working with Transient Data in Tecplot” on page 137.

Step 7: Change the Shade color for select zones.

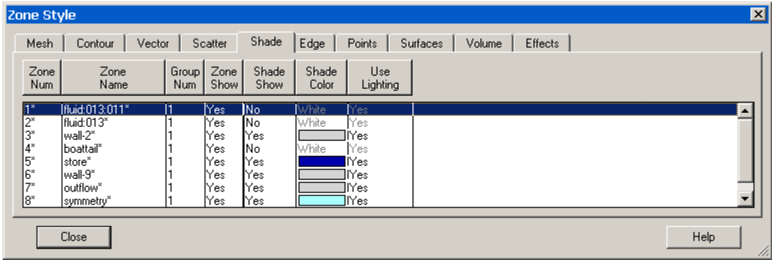
A. On the ***Shade*** page of the **Zone Style** dialog, select Zone 8 - Symmetry, set the *Shade Color* to “light blue”.

B. Select Zones 3,6 and 7 (using the CTRL key) and set the *Shade Color* to light gray.

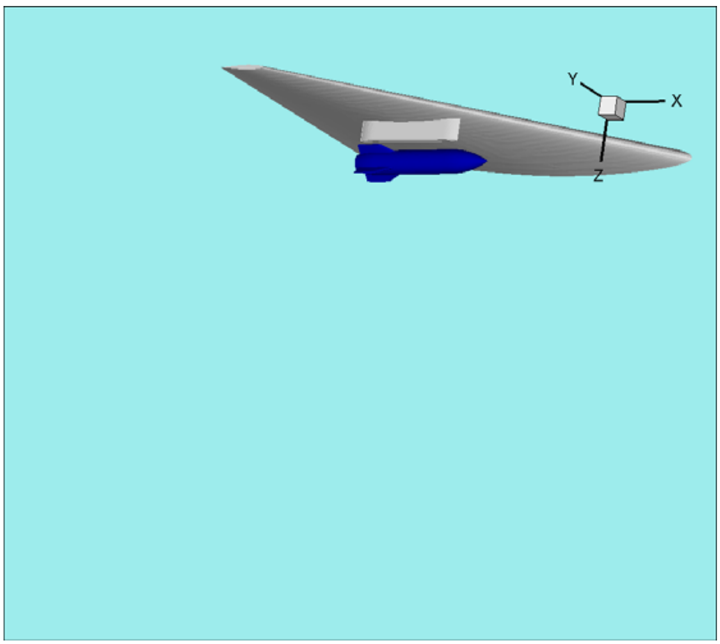
C. Select Zone 4 and set the *Shade Color* to dark gray.

D. Select Zone 5 and set the *Shade Color* to dark blue.

The ***Shade*** page of the **Zone Style** dialog should look as follows:



And the plot should look as follows:



You can remove the 3D Orientation Axis by simply selecting it and then pressing the delete button. Alternatively, you select **Plot>3D Orientation Axis** and toggle-off ***Show 3D Orientation Axis*** in the resulting dialog.

5 - 4 Add Iso-surfaces

Although, we could have animated the data at any time, the plot is now ready to be animated. Before we proceed to animation, however, we will enhance the plot by adding iso-surfaces.

Step 8: Toggle-on Iso-surfaces in the Sidebar.

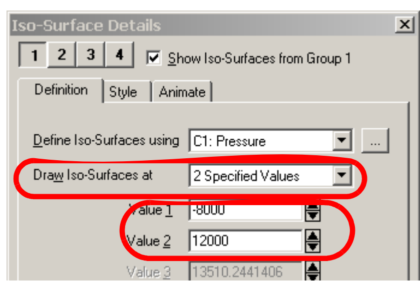
Step 9: Define the Iso-surface positions

A. Open the **Iso-surface Details** dialog by selecting the button next to Iso-surfaces in the Sidebar or by selecting **Iso-surface Details** from the **Plot** menu.

B. On the ***Definition*** page of the dialog, select *Draw Iso-surfaces at: 2 Specified Values*.

C. Set Value 1 = -8000.

D. Set Value 2 = 12000.



The Iso-surfaces are defined using Contour Variable (C1) (as indicated at the top of the dialog). Contour Variable C1 is set to the Pressure variable from the data set via the Contour Details dialog.

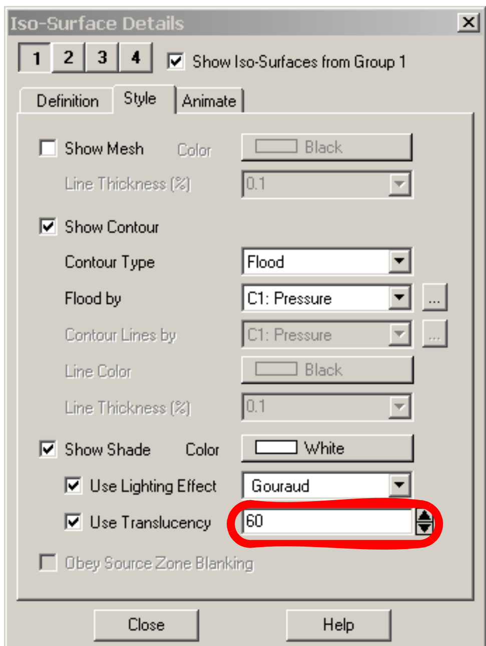
Step 10: Add translucency to the iso-surfaces.

A. Go to the ***Style*** page of the **Iso-surface Details** dialog.

A. Toggle-on *Show Shade.*

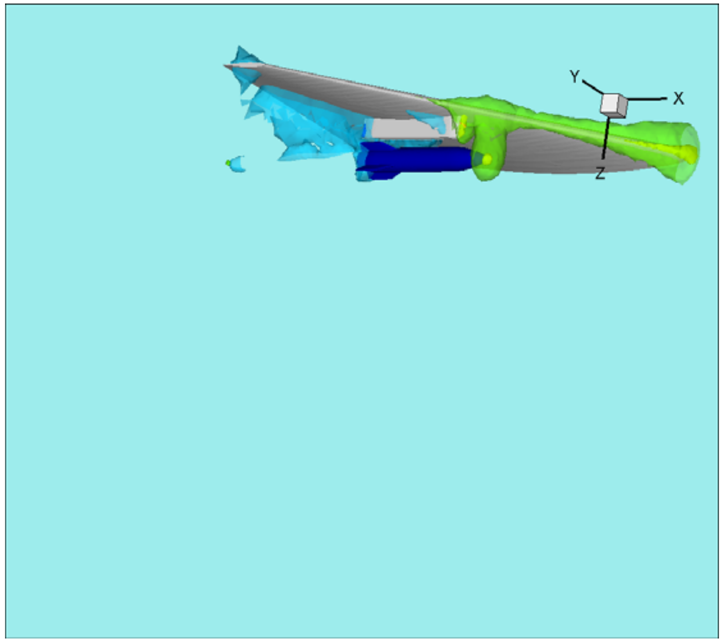
B. Toggle-on *Use Translucency.*

C. Increase the translucency from 50 to 60.



D. Close the dialog.

The plot should look as follows (where the blue and yellow cloud regions are the two iso-surfaces):



5 - 5 Animate Time Steps

Step 11: Animate the store separation on your screen

Animate your plot, by selecting the  button in the Sidebar.

You can also animate the plot over time by selecting Time from the Animate menu.

Step 12: Save your animation to a file

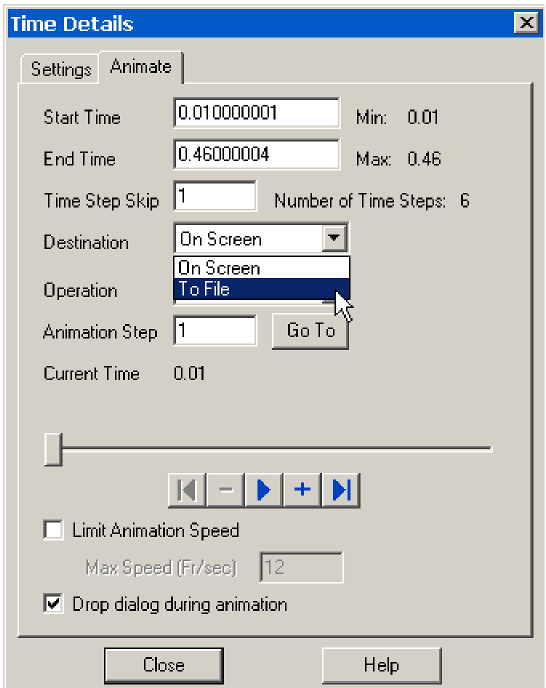
A. Open the ***Animate*** page of the **Time Details** dialog, by either

i. Selecting the button to the right of Time in the Sidebar

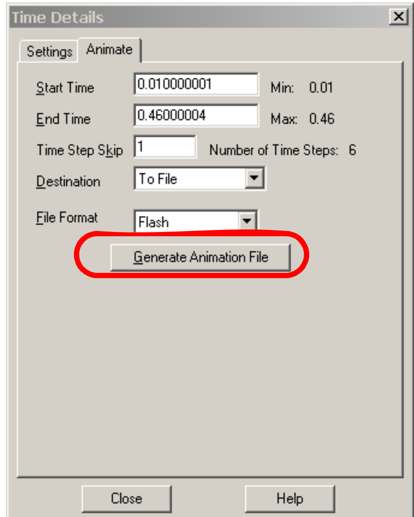
ii. Selecting **Time Details** from the **Plot** menu and selecting the ***Animate*** page.

iii. Selecting **Time** from the **Animate** menu.

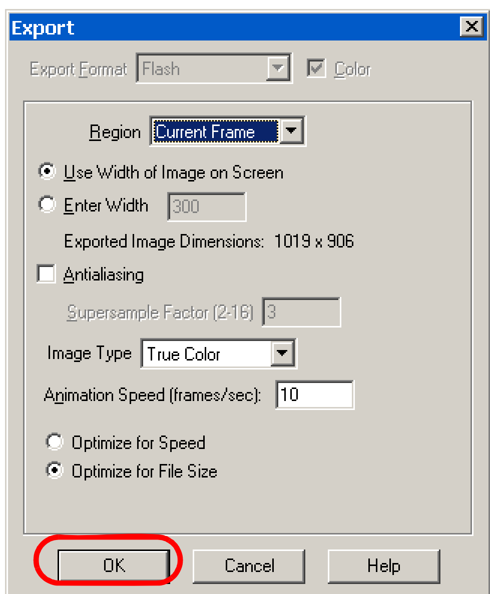
B. On the ***Animate*** page of the **Time Details** dialog, select “To File” from the *Destination* drop-down menu.



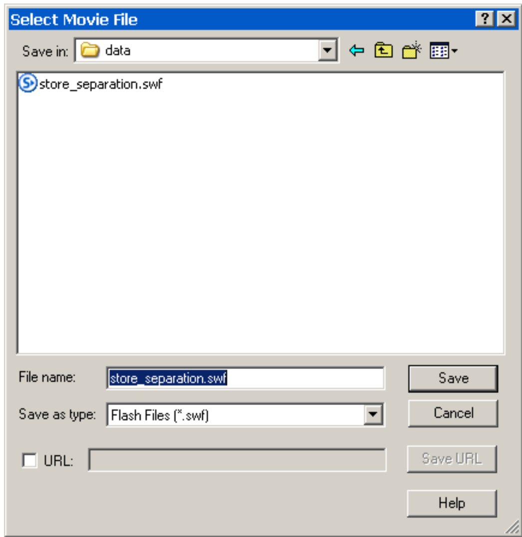
C. Accept the default format (Flash) and select the *Generate Animation File* button.



D. In the **Export** dialog, accept the default settings by pressing ***OK***.



E. In the **Select Movie File** dialog, navigate to the desired directory and press ***OK***.



A copy of our exported movie file is included with your Tecplot 360 installation at: /demo/tutorials/transient/data.

The movie file can now be imported into a Power Point presentation or placed on a website.

The easiest way to insert and play SWF files into Microsoft Power Point presentations is to download the Swiff Point Player — a free Microsoft Power Point Add-In.

5 - 6 Working with Transient Data in Tecplot

Field plots containing transient data are displayed slightly differently in the Zone Style dialog. When transient data is loaded, each **transient zone** is assigned to a given time **strand** and only the **relevant zone** for the **current solution time** is displayed (see the definitions below).

- **Transient zones** - Zones associated with time. The transient zone(s) displayed in the current frame are dependent upon the current solution time. All of the zones in this tutorial are transient zones.

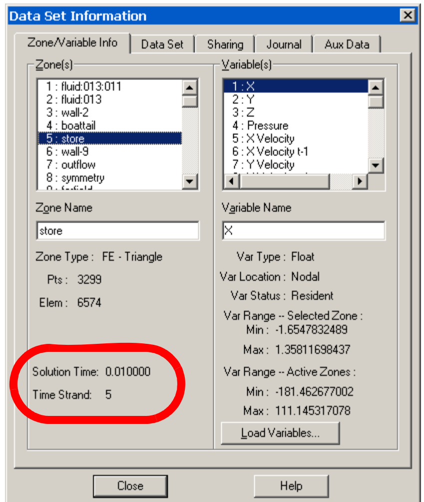
- **Static zones** - Zones not associated with time. They are displayed regardless of the current solution time. None of the zones in this tutorial are static zones.

- **Current Solution Time** - The value which determines which transient zones are displayed in the current frame. The value of Current Solution Time is specified on the ***Settings*** page of the **Time Details** dialog and the Sidebar.

- **Strand** - A series of transient zones of that represent the same part of a data set at different times, such as the Store. This data set contains 9 strands, 1 for each of physical regions of the data.

- **StrandID** - An integer value defined for each transient zone. The StrandID of a given zone is determined by the data loader. Zones 5, 14, 23, 32, 41 and 50 in are all have the same StrandID (5) and therefore the are part of the same strand.

In this tutorial, there are 6 time steps, 9 strands and 54 zones. The solution time and strand ID for a given zone can be viewed in the **Data Set Info** dialog (accessed via the **Data** menu).



Each strand contains the information for one physical region of the plot over time. For example, strand3 contains information for the boattail over time, strand4 contains information for part of the wing over time and strand5 contains the information for the store zone over time.

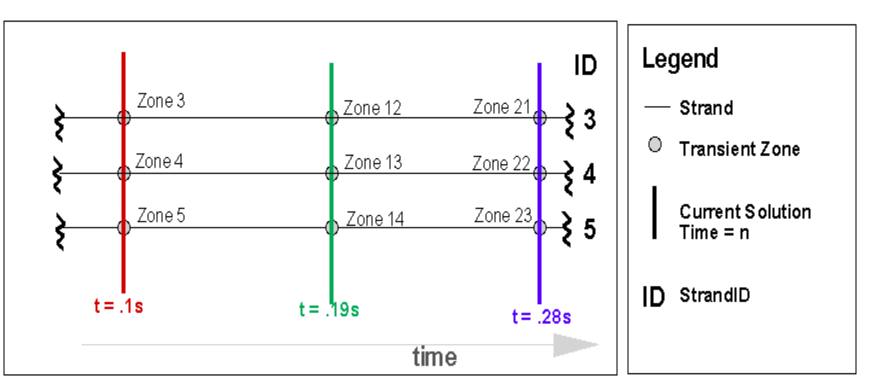


Figure 5-1. A truncated depiction of the data set. Time steps 2 -4 for strandIDs 2-5 are displayed. At Current Solution Time (t) = .1 s, Zones 3, 4 and 5 are displayed in Tecplot. At Current Solution Time (t) = .19 s, Zone 12, 13 and 14 are displayed in Tecplot.

The Data Set Info dialog allows you to view all of the zones in the data set regardless of the Current Solution Time. The Zone Style dialog displays only the relevant zones for the Current Solution Time. This is denoted by an “*” after the *Zone Num and Zone Name* columns. For example, at Current Solution Time equal to.01, Zones 1 - 9 are displayed in the Zone Style dialog (Figure 5-2) and at Current Solution Time equal to.1, Zones 10 - 18 are displayed in the Zone Style dialog (Figure 5-3).

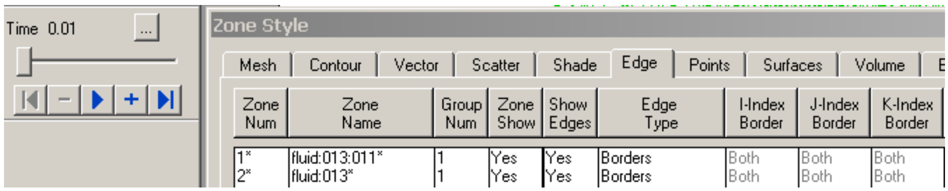


Figure 5-2. The **Zone Style** dialog at Current Solution Time = .01.

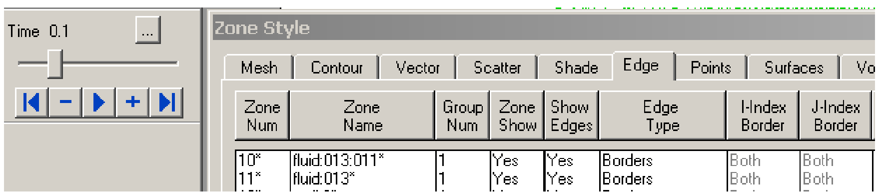


Figure 5-3. The **Zone Style** dialog at Current Solution Time = .1.

Changes made in the **Zone Style** dialog to any zone in a with a given StrandID are propagated to all zones with that StrandID.