

Tecplot, Inc.

Copyright © 1988-2025 Tecplot, Inc. All rights reserved worldwide. See the complete legal notice in the copyright section of this document.

Table of Contents

Introduction	6
Before You Start the Tutorials	6
Tutorials In This Guide	7
Part 1: General Application	8
External Flow	9
Loading and Manipulating Data	9
Step 1: Launch Tecplot 360 and Load the Data Set	9
Step 2: Rotate the Wing	12
Step 3: View Information About the Data Set	12
Step 4: Disable Bounding Box for Fluid Volume Zone	14
Step 5: View the Mesh Using the Context Toolbar	16
Step 6: Change the Mesh Color	16
Step 7: Set Up Contour Groups and Color Maps	19
Step 8: Change Contour Group on Plot	23
Step 9: Format Legend	24
Exploring a CFD Solution	27
Step 1: Add a Slice	27
Step 2: Set Slice Details	30
Step 3: Show Contours	32
Step 4: Prepare for Streamtraces	36
Step 5: Seed Streamtraces	39
Step 6: Seed Volume Ribbons	42
Step 7: Adjust Rotation and Lighting	46
Step 8: Surface Streamtraces	46
Step 9: Create Iso-Surfaces	49
Step 10: Probe Data	56
Comparing a CFD Simulation with Experimental Data	57
Step 1: Load Layout.	57
Step 2: Append Experimental Data	57
Step 3: Variable Load and Combine	59
Step 4: Normalize Y to Y/b	60
Step 5: Extract Slice from Simulation Data at Pressure Tap Location	63
Step 6: Normalize Slice's X to X/L	67
Step 7: Create XY Plot of Slice	69
Step 8: Plot Experimental Data	73
Step 9: Add Error Bars	77

Step 10: Final Polishing	
Next Steps	80
Understanding Volume Surfaces	81
Understanding Volume Surfaces	81
Step 1: Launch Tecplot 360 and Load the Data Set	81
Step 2: Surfaces to Plot and the Zone Style dialog	83
Step 3: Contours and Surface Data	
Step 4: Changed Surfaces to Plot in the Zone Style Dialog.	85
Next Steps	86
Transient Data.	87
Loading and Exploring Transient Data	87
Step 1: Launch Tecplot 360 and Load the Data Set	
Step 2: Get A Good Look	
Step 3: Dive into Data Set Info	
Step 4: Time Strands in the Zone Style Dialog	
Step 5: Visualizing a Contour Plot	
Step 6: Visualize the Flow With Streamtraces	
Step 7: Animate Your Plot	96
Extracting Data	
Step 1: Create a Vertical Line	
Step 2: Extract the Line Over Time	
Step 3: Create A Line Plot of Pressure	
Step 4: Link the Time Steps and Animate the Plot	
Step 5: Time Series Plot	
Frequency Analysis Using Fourier Transform	111
Step 1: Fourier Transform.	
Step 2: Re-Analyzing at a New Position	115
Calculations and Contour Cutoff	
Step 1: Set Field Variables	
Step 2: Calculate Vorticity Magnitude	
Step 3: Show Vorticity Magnitude on Contour Plot	
Step 4: Adjust Cutoff	123
Step 5: Polishing the Plot	
Step 6: Animate It	
Next Steps	
Finite Element Analysis	
Calculating Von Mises Stress Along a Surface	
Step 1: Load the Data Set	

Step 2: Slow Animation Down
Step 3: Calculate Von Mises Stress
Step 4: Contour by Von Mises Stress
Step 5: Isolate the Connecting Rod
Step 6: Identify the Critical Failure Threshold
Step 7: 3D Multi Frames
Data Alter with If Statements
Step 1: Specify an Equation
Step 2: Contour It
Step 3: Flood by Primary Cell Value
Plot Maximum Stress Over Time
Step 1: Load the Macro
Step 2: Macro Breakdown: Create a Zone
Step 3: Macro Breakdown: Alter Variables
Step 4: Macro Breakdown: Create XY Plot
Next Steps
Part 2: Internal Combustion Engines
Internal Combustion Engines
Loading and Manipulating Data
Step 1: Launch Tecplot 360 and Load the Data Set
Step 2: Disable Bounding Box for Fluid Volume Zone
Step 3: Hide Exhaust Ports
Exploring the Data Set
Step 1: Adding a Slice
Step 2: Set Slice Styles
Step 3: Showing Spray Parcels
Step 4: Changing Spray Parcel Size
Step 5: Changing Spray Parcel Color
Step 6: Creating an Iso-surface
Step 7: Using 3D Multi Frames
Step 8: Export to Movie File
Loading Cell Averaged Output Files (CONVERGE)
Step 1: Loading the data
Step 2: Add Multiple Y Axes
Step 3: Add Grid Lines
Step 4: Update Axis Labels
Next Steps
Part 3: Ocean Modeling

Ocean Modeling	180
Loading and Manipulating Data	180
Step 1: Launch Tecplot 360 and Load the Data Set	180
Step 2: Inspecting the Data	183
Making your first plot	184
Step 1: Turn on Contour to See the Domain	184
Step 2: Assign XYZ to Lon, Lat	185
Step 3: Adjusting Z-scaling	186
Step 4: Change View and Lighting	187
Step 5: Change the Contour Variable and Colormap	188
Step 6: Use Value Blanking to Hide Dry Land.	190
Step 7: Animating and Exporting a Movie File.	192
Specific Ocean Plots	194
Visualizing Surface Velocities	194
Step 1: Turn on the Vectors	194
Step 2: Value Blank Siglev	195
Step 3: Use Vector Tangents	196
Step 4: Calculate Velocity Magnitude	197
Step 5: Contour by Velocity Magnitude	200
Step 6: Resize Vectors	201
Insert a Georeferenced Image	203
Step 1: Set Up Coordinate System	203
Step 2: Insert Georeferenced Image	204
Step 3: Update Image location	204
Understanding Salinity Stratification.	206
Step 1: Turn off Zone Layers	206
Step 2: Show the Bathymetry	206
Step 3: Place a Slice.	209
Step 4: Contour the Slice by Salinity.	211
Advanced Topics	213
Vertical Transect (external video)	213
Time Average (external video)	213
Shapefile conversion (external video)	214
Next Steps	214
Copyright	215

Introduction

Tecplot 360 allows you to interactively explore, visualize, and analyze your CFD data, and then communicate your results. With Tecplot 360, you can produce high-quality plots for reports, papers, presentations, videos, or web sites.

The user documentation for Tecplot 360 includes these resources:

Getting Started Manual (this document)

Highlights how to work with key features through a tutorial revolving around data files similar to those you might use.

User's Manual

Complete documentation of all Tecplot 360 features.

Scripting Guide

Information on working with Tecplot 360 macros and a full syntax reference.

Quick Reference Guide

A handy reference for all the little details of using Tecplot 360, such as text placeholders, keyboard shortcuts, and much more.

Data Format Guide

Tecplot data formats and how to write them.

Installation Guide

How to install Tecplot 360 on your machine.

Release Notes

Information on the latest Tecplot 360 features along with platform-specific notes.

This manual includes four tutorials to help you get started with Tecplot 360. For in-depth information on any of the topics covered in the Getting Started Manual, please refer to the User's Manual which is included in your Tecplot 360 installation directory or on our website at: www.tecplot.com/ documentation.

Before You Start the Tutorials

Before beginning a tutorial from this guide, we suggest you read through Chapters 1 and 2 of the User's Manual to introduce yourself to the Tecplot 360 user interface and familiarize yourself with the product's basic concepts of operation. It may be helpful to have Tecplot 360 open while reading these chapters so you can experiment a little.

Chapter 1 - Introduction

Covers the Welcome Screen, supported input devices, the Tecplot 360 workspace, the menu bar and global toolbar, context menus and toolbars, and sidebars. Reading this chapter will orient you to the main user interface controls of Tecplot 360, making it easier to find what you're looking for when you are working on the tutorial.

Chapter 2 - Using the Workspace

Covers data hierarchy, coordinate systems, frames, workspace management, view modification, and the edit menu. You will get practice with many of these concepts in the pages that follow. You may want to review Chapter 2 again after completing the tutorial.

Tutorials In This Guide

This Getting Started Manual is divided into tutorials, each of which contains a series of exercises covering various aspects of the topic at hand. The tutorials are designed so you can do them in any order you like, but within a tutorial, it is best to go through the exercises in order if possible (though we do try to provide guidance on how to pick up in midstream).

The tutorials are:

External Flow

Using the Onera M6 wing model, covers loading the data, producing a basic plot, slicing, streamtraces, isosurfaces, probing, and comparing simulated and experimental data (including normalizing the data).

Understanding Volume Surfaces

Uses the DuctFlow data set as an example of how Tecplot 360 renders volume surfaces using Surfaces to Plot.

Transient Data

Uses a wind turbine data set with 127 time steps to understand how transient (time-based) data is structured and how to produce animated contour plots, extract data over time for analysis, and calculate and visualize additional variables using the Tecplot 360 analysis tools.

Finite Element Analysis

Uses a transient FEA dataset of a connecting rod created with LS-DYNA to explore multiple ways to visualize the maximum Von Mises Stress of the rotating rod.

All of the tutorials in this guide have datasets located in the Getting Started Bundle. A few datasets are located in the example folder in the installation directory, OneraM6Wing and DuctFlow.plt as well as the Getting Started Bundle.

Dive in and master the view!

Part 1: General Application

External Flow

The Onera M6 wing is a standard design for basic studies of 3D flows at high Reynolds numbers from low to transonic speeds (that is, local supersonic flow, shocks, and turbulent boundary layer separation). The wing was tested by NASA in a wind tunnel at four different Mach numbers and various angles of attack. This is now a classic CFD validation case for external flows because it has a simple geometry, complex flows, and because it includes experimental data against which a CFD solution can be validated. For more about the Onera M6 wing, see turbmodels.larc.nasa.gov/ onerawingnumerics_val.html.

This tutorial uses the Onera M6 data set as your entry into Tecplot 360, introducing basic operations at first (loading data and manipulating the view) and progressing to more intermediate and then advanced operations. We chose this data set because you may already be familiar with it (and if you're not, it's straightforward to understand).

The Onera M6 data set is in the OneraM6wing folder inside the Getting Started Bundle.

This tutorial is divided into three segments. We have provided a layout file for the end of each segment, so you can check your work. There is also a macro included to generate an enhanced version of one of the plots in the third segment, and a layout showing its results. The segments are:

Number and Level	Title and Description
1 - Beginner	Loading and Manipulating Data - Load the Onera M6 simulation data set into Tecplot 360 and create a contour plot, along the way discovering how to rotate a 3D plot, view and color a mesh, set up contour groups and color maps, and display a legend.
2 - Intermediate	Exploring a CFD Solution - Further explore and understand the Onera M6 simulation data set by adding slices, streamtraces, and iso-surfaces, and by probing data.
3 - Intermediate	Comparing a CFD Simulation with Experimental Data - Create a coefficient of pressure (Cp) plot comparing simulated data with experimental data, including normalizing the dimensions of the simulation data set.

A video version of this tutorial is available on the Web at www.tecplot.com/category/tecplot-360-videos/ external-flow. The videos may have minor differences from the printed version of the tutorial in this manual, but they end up in the same places.

Loading and Manipulating Data

Step 1: Launch Tecplot 360 and Load the Data Set

Start Tecplot 360 from the Start menu (Windows), by typing tec360 in a terminal window (Linux), or by double-clicking the application icon in the Applications folder (Mac). The Tecplot 360 Welcome Screen appears, as shown here. (We will show the Windows version of Tecplot 360 in this document, but the product looks substantially the same on other platforms.)



The Welcome Screen appears each time you launch Tecplot 360 and gives you easy access to layouts you have recently worked with, along with quick links to documentation and other resources to help you get the most out of the product.

To begin loading the Onera M6 data, click **Load Data** at the top of the Welcome Screen. (You may also choose **Load Data** from the **File** drop-down menu in the menu bar, or click the folder icon, second from the left, in the toolbar. These alternate methods are convenient when the Welcome Screen isn't visible.)

The Load Data dialog appears.

Ioad Data					×
\leftarrow \rightarrow \checkmark \uparrow \frown Desktop $>$ 30	60GettingStarted > OneraM6Wing >		~ C 5	Search OneraM6Wing	מ נ
Organize - New folder				≣ •	. 🗖 🔇
Ame Home	Name	Date modified	Туре	Size	
🔀 Gallery	📁 finallayouts	1/4/2024 3:41 PM	File folder		
	acros macros	8/12/2022 2:41 PM	File folder		
🔁 Documents 🔹 🖈	ArbitraryPoints.txt	8/12/2022 2:41 PM	Text Docume	nt	6 KB
	Onera_Experimental.dat	8/12/2022 2:41 PM	DAT		27 KB
	OneraM6_SU2_RANS.plt	8/12/2022 2:41 PM	Tecplot 360 E	X plot file 4,7	35 KB
Desktop					
> 📮 This PC					
> 🦆 Network					
> 🛕 Linux					
🗌 Data overri	de				
File name: OneraM6_	SU2_RANS.plt			All Supported Files (*.	bin;*.dat;*.

Navigate to your Tecplot 360 installation folder, then the examples folder, then the OneraM6wing folder. Then double-click the OneraM6_SU2_RANS.plt file to open it in Tecplot 360. (If you can't see this file, choose **All Files** in the menu at the bottom of the dialog.) The data file is opened and a 3D plot of the Onera M6 wing appears in the Tecplot 360 workspace, as shown here.



Step 2: Rotate the Wing

To rotate the view of the wing, hold down the Control key on the keyboard (Command on Mac), then hold down the right mouse button in the Tecplot 360 workspace and move the mouse to rotate the wing.

You'll notice that the range of rotation is not very great, making it hard to make significant changes to the view of the wing. This is because the rotation origin (the point around which rotation is performed) is not set anywhere near the wing. To change this, simply place the mouse pointer in the approximate center of the wing on the screen, then press the lowercase letter 0 (for *origin*) on the keyboard.

Then hold down the Control key (Command on Mac) and drag with the right mouse button again. You'll see it is now much easier to rotate the wing, since it rotates around its center.

Step 3: View Information About the Data Set

To see information about the Onera M6 data set, choose **Data Set Info** from the **Data** drop-down menu. The **Data Set Information** dialog, shown here, appears.

Data Set Information	×
Zone/Var Info Data Set	Aux Data
Zone(s)	Variable(s)
1: FluidVolume 2: WingSurface	1: x ^ 2: y 3: z 4: Density 5: Momentum U (Densi 6: Momentum V (Densi <
Zone Type: FE - Brick Pts: 46417 Elem: 43008	Var Type: Float Var Location: Nodal Var Status: Not Loaded Var Range Selected Zone: Min: -6.4472 Max: 5.6413 Var Range Active Zones: Min: -6.4472 Max: 5.6413 Load Variables
	Close Help

This dialog provides a wealth of information about the data set. The two lists at the top of the dialog show you the names of the zones and the variables in the data set.

The zones in this data set are:

FluidVolume

the air around the wing

WingSurface

the surface of the wing itself

You may wish to explore the other information in the dialog, which is divided into three pages. Click the **Help** button for more information about the information displayed on any page of the dialog. When you're finished, close the **Data Set Information** dialog.

Step 4: Disable Bounding Box for Fluid Volume Zone

As you rotated the wing in step 2, you may have noticed a dashed orange line swoosh by in the plot. We have caught a glimpse of it here.



This is the bounding box of the FluidVolume zone, which represents the air around the wing. This zone does not have any style (that is, visual appearance) so it would normally be invisible. Tecplot 360 adds the dashed orange line so that you know it's there and can see its dimension.



Choose **Fit Everything** from the **View** menu to see the full extent of the volume zone (see above). The wing is a tiny part of the data set! Choose **View**>**Last** to return to the previous view.

The bounding box does not add anything to the plot we're making, so let's turn it off. Choose **Show Bounding Boxes for Enabled Volume Zones with No Style** from the **Options** menu. The dashed orange line disappears.

Step 5: View the Mesh Using the Context Toolbar

To view the mesh for the WingSurface zone, right-click on the wing in the Tecplot 360 workspace. A context toolbar appears, as shown here.



From left to right on this toolbar are buttons that allow you to turn on and off the mesh, contours, vectors, shade, edge, and translucency. Click the first button to display the mesh for the WingSurface zone, as shown here.



Step 6: Change the Mesh Color



The top portion of the Plot sidebar lets you turn the layers of your plot on and off. The **Zone Style** button in this sidebar opens the **Zone Style** dialog, shown here.

	contour vector scatter	Shaue	Euge P	onits Surraces	volume	Effects		
Zone Number	Zone Name	Group Numb	o Show er Zone	Show Mesh Mesh Type	Mesh Color	Line Pattern	Pattern Length	Line Thickness
1	FluidVolume	1	\checkmark	Overlay		Solid	2.00%	0.10%
2	WingSurface	1	\checkmark	Overlay		Solid	2.00%	0.10%
Selection c	iteria:		Zones	Groups	Clear			

The Mesh Color column (right most green column) displays the color of the mesh for each zone, including WingSurface. Right-click the black color swatch to display the Color Chooser.



The Color Chooser lets you choose a single solid color, or, using the 1-8 buttons at the bottom of the dialog, you may choose to have the mesh colored using a contour, such as a gradient based on the value of some variable. (These numbers actually refer to Tecplot 360's eight contour groups, which associate variables and color maps. We will look at contours in more depth shortly.)

For now, let's choose a blue color for the mesh.



Close the Zone Style dialog, so that the wing surface is visible again.

Scenic Detour: Context Menu and Toolbar

Try right-clicking the wing. A context menu and toolbar appears, allowing you to make many of the same changes you can make in the **Zone Style** dialog—without needing to pull up the dialog. Here we are using the drop-down menu to the right of the mesh icon (the first icon on the toolbar) to choose the mesh color.



If you are changing the styles of many zones at once, it still makes sense to pull up the **Zone Style** dialog. For many other situations, the context menu and toolbar are faster.

We will use this context menu again in later steps.

Step 7: Set Up Contour Groups and Color Maps

The Onera M6 data contains a number of variables we might be interested in visualizing. Let's set up color maps so that we can see them as contours on the surface of the wing. First activate the Contour layer in the Plot sidebar, then click the *interestivate* button next to the Contour checkbox to open the **Contour & Multi-Coloring Details** dialog, seen here.

Plot	e, x
3D Cartesian	\sim
Snap to orientation view	
Show zone layers	
Mesh	
🗹 Contour	2
🗹 Shade	
Vector	- ⁴⁰
Edge	Ľ
Scatter	
Zone Style	
Solution times 0	-101
Soldton time. o	0.
	1
Show derived objects	
Iso-Surfaces	1
🗌 Slices 🛛 👰	(j ^{‡‡}
Streamtraces	Ç
Show effects	
🗹 Lighting 📜	.
Translucency	
Snap to	
None Paper Grid	ł
Redraw	
🗹 Auto redraw	

🗕 Contour & Multi-Coloring [Details >	<
4: Density	✓ 1 2 3 4 5 6 7 8	
Levels and Color Bands	Lines Labels Legend	
Contour levels		
1 : 0.464079	↑ Set Levels	
2:0.557233	Add Level 0	
4:0.743541		
5 : 0.836695	Remove Selected	
6:0.929849	Contour variable range	
8:1.11616	Min: 0.370926 Max: 1.86139	
	~	
Large R	Rainbow 🗸 👯	
Color map distribution method	Color cutoff	
O Banded O Continuous	S Cutoff above: 1.48877	
Contour values at endpoints	Cutoff below: 0.743541	
Min: 0.370926	Color map adjustments	
Max: 1.86139	Reset 👻 🗌 Reverse	
Approximate continuous	Repeat cycles: 1	
	Close Help	

At the top of the dialog is a drop-down menu for choosing a variable. Next to this are eight numbered buttons, which specify the *contour group* you are editing. Each contour group has its own settings for this dialog. The contour group provides a way to associate a variable with a color map and other settings. The color map specifies how the zone will be colored according to the value of the specified variable.

For our tutorial, we will set up two contour groups. The first will display density with the Large Rainbow color map. The second will display the pressure coefficient using the Magma color map.

First, move the dialog so you can see most of the plot and the dialog at the same time.

- 1. Make sure contour group 1 is set at the top of the **Contour & Multi-Coloring Details** dialog.
- 2. Make sure the Density variable is chosen in the drop-down menu at the top of the dialog.
- 3. From the drop-down menu under the color map preview, choose Large Rainbow.
- 4. Click **Continuous** for the Color Map Distribution Method.

The dialog should appear as shown above. You should have noticed the plot changing as you made each change in the dialog, since the wing surface is already using contour group 1 by default.

Next, we'll set up contour group 2 for the Pressure_Coefficient variable.

- 1. Click 2 at the top of the **Contour & Multi-Coloring Details** dialog to choose contour group 2.
- 2. Choose the Pressure_Coefficient variable in the drop-down menu at the top of the dialog.
- 3. From the drop-down menu under the color map preview, choose Magma.
- 4. Click **Continuous** for the Color Map Distribution Method.
- 5. We also need to change the levels for the color map. Click the **Set Levels** button, then, in the **Enter Contour Levels** dialog, change the Minimum level to -1 and the number of levels to 21, as shown here.

Enter Contour Levels	×	
Level Creation Mode		
Exact levels		
O Approximate levels for nice values		
Range Distribution		
Min, max, and number of levels		
O Min, max, and delta		
O Exponential distribution		
Minimum level: -1		
Maximum level: 1		
Number of levels: 21		
Delta: 0.2		
Reset Range	•	
OK Cancel Help		

6. Click **OK** to close the **Enter Contour Levels** dialog.

The **Contour & Multi-Coloring Details** dialog should now appear as shown here.

🗕 Contour & Multi-Coloring I	Details ×
12: Pressure_Coefficient Levels and Color Bands Contour levels	1 2 3 4 5 6 7 8 Lines Labels Legend
1: -1 2: -0.9 3: -0.8 4: -0.7 5: -0.6 6: -0.5 7: -0.4 8: -0.3 0: -0.2	 Set Levels Add Level Add Level Remove Selected Contour variable range Min: -1.30067 Max: 1.1383
Magma	
Color map distribution method Banded Continuous Contour values at endpoints Min: -1.30067 Max: 1.1383 Approximate continuous	Color cutoff Cutoff above: 0.528562 Cutoff below: -0.690923 Color map adjustments Reset Reset Repeat cycles: 1
	Close Help

Step 8: Change Contour Group on Plot

Now that we've set up our contour groups, close the **Contour & Multi-Coloring Details** dialog. Ensure the Contour layer is checked on the Plot sidebar.

Now, we can change the contour variable displayed on the wing by right-clicking the wing, clicking the drop-down menu next to the contour icon in the context menu, and choosing contour group 2 C2: Pressure Coefficient.



The plot so far is shown below.



Step 9: Format Legend

That contour legend is useful, but it overlaps our plot. Double-click the legend title to open the Legend page of the **Contour & Multi-Coloring Details** dialog, shown here.

Ontour & Multi-Coloring Details
12: Pressure_Coefficient ~ 1 2 3 4 5 6 7 8
Levels and Color Bands Lines Labels Legend
Show contour legend
Alignment: Horizontal V Anchor
X (%): 97.738 Y (%): 20.965 Legend Box
Separate color bands
Resize automatically Show color cutoff levels
Level skip: 1 🔹 Line spacing: 1.2
Label continuous colors position:
Text color: Black
Number font: Helvetica \checkmark B <i>I</i>
Size: 2.5 Frame% V Number Format
Show header
Header text: Use Variable Name 🗸
Header font: Helvetica \checkmark B <i>I</i>
Size: 2.5 Frame%
Close Help

This is the same dialog we just closed a moment ago, just a different page. You can also get to it by clicking the *button* next to Contour in the Plot sidebar, or by choosing **Plot>Contour/Multi-Coloring** from the Tecplot 360 menu bar.

In the Contour & Multi-Coloring Details dialog:

- 1. Change the alignment to horizontal
- 2. Turn on automatic resizing

The desired settings are shown above. Close the dialog after changing these settings.

Using the mouse, you may now drag the legend to the bottom of the plot. The final plot is shown here.



A Tecplot 360 layout (.lay) file containing a snapshot of the final result of this tutorial segment is in OneraM6wing/finallayouts/ExtenalFlowVideo1.lay in the examples folder in your Tecplot 360 installation folder. You can compare this with your result to see how you did.

You might even try saving a layout of your own work. Choose **File>Save Layout** from the Tecplot 360 menu bar, then navigate to the folder in which you want to save the layout, name it, and click **Save**.

🧕 Save Layout					×
← → • ↑ 📕 >	This PC > Local Disk (C:) > MyData	3	✓ Ŭ	Search MyData	م
Organize 🔹 New f	folder				∎== ▼ ?
📌 Quick access	Name	Date modif	fied Type	Size	
Creative Cloud Fil	les	No items match y	your search.		
lene One Drive					
This PC					
🕩 Network					
File name: E	ExternalFlowVideoTlay				~
Save as type:	Inked Data (*.lay)				~
	Use relative paths	Include preview image			
∧ Hide Folders			Help	Save	Cancel

Exploring a CFD Solution

This segment of the External Flow tutorial covers slicing, streamtraces, iso-surfaces, and probing. These are tools you'll frequently use to get a closer look at the volume data in your solution.

We'll continue from where we left off at the end of the previous segment, with the coefficient of pressure visible on the surface of the wing. If you have closed Tecplot 360 since completing that segment, you can load the provided layout file OneraM6wing/finallayouts/ExtenalFlowVideo1.lay in the examples folder in your Tecplot 360 installation folder, and continue from there.

Step 1: Add a Slice

Plot	₽×
3D Cartesian	\sim
Snap to orientation view	
Y Z Z Y ZY	
Show zone layers	
Mesh	
Contour	2
🗹 Shade	
Vector	2 ⁴⁹
Edge	Ľ
Scatter	
Zone Style	
	374
Solution time: 0	(1) ye
Show derived objects	
Iso-Surfaces	*
🗹 Slices 🛛 🙀) 1
Streamtraces	\$
Show effects	
🗹 Lighting	۶.
Translucency	
Snap to	
None O Paper O Gri	id
Redraw	
🗹 Auto redraw	

To add a slice, toggle on the Slices checkbox in the Plot sidebar. A slice appears through the volume zone.



As we determined in the first segment of this tutorial, the volume zone is much larger than the wing. The slice through that zone, therefore, dominates the plot. That's why the screen suddenly fills with a nearly solid color (yellow in our plot; it might be green or a different color in yours).

The Density legend appears because this slice is using contour group 1.

To move the slice with the mouse, first click the slice tool in the Plot sidebar, next to the Slices

checkbox.

Then click the surface of the wing. (You probably can't see the wing at first, with the slice in front of it, but you can still click it even though you can't see it.) The slice moves to pass through the point that you clicked. You can see the slice at various positions in the following images.



You can also *drag* the mouse on the surface of the wing to fine-tune the slice location. A blue-gray preview plane shows where the slice will be when you release the mouse button.



Step 2: Set Slice Details

Click the 🛹 button next to the Slices in the Plot sidebar to launch the Slice Details dialog.



Let's spend a moment on the Slice Details dialog.

Slice Details ×
Show group 1 1 2 3 4 5 6 7 8
Definition Contour Vector Other Animate
Slice location: Y-Planes \vee
Show primary slice
0
Show start/end slices
Start:
0.3821712494
End:
7.2612537386
Show intermediate slices Number of slices: 1
Slice through: Volume Zones \sim
Obey source zone blanking
Show solid plane while dragging (faster)
Extract Slices
Close Help

• There are eight buttons across the top, just as with the **Contour Details** dialog. This isn't a coincidence: Tecplot 360 supports eight slice groups, just as it supports eight contour groups. Each slice group uses the same normal and style, but can have its own positions. Various other settings are also unique to each slice group.

These numbered buttons tell Tecplot 360 what slice group you want to edit.

• You can manually change the slice orientation to a specific plane. Currently it is set to the X plane, which is the default. A Y plane slice is more useful here, since it shows the wake behind the wing. So let's change that.

The dialog also lets you choose an arbitrary slice orientation, though we won't use this capability in this tutorial.

• You can see that you can set up multiple slices by specifying a start slice, an end slice, and some number of intermediate slices. You can also choose what kind of data you want to slice through.

You can leave the **Slice Details** dialog open while you experiment with dragging the slice around. Notice that the slice position field in the dialog updates as you move the slice in the plot. (Most dialogs in Tecplot 360 can be left open while you work, updating whenever you change a plot by other means.)

Step 3: Show Contours

🔯 Slice Details				×
Show group 1 1 2	3 4 5 6 7 8			
Definition Contour	Vector Other Animate			
Show contours				
Contour type:	Flood	\sim		
Flood by:	C7: Pressure	\sim	2	
Contour lines by:	C1: Density	\sim	Ż	
Line color:	Black			
Line thickness (%):	0.1	\sim		
Use lighting effect:	Gouraud	\sim		
Use translucency:	10	*		
	Close		Help	

With the **Slice Details** dialog still open, let's take a quick detour to the Contour page of this dialog, which allows you to specify how the slice itself will be colored.

This ties back to the contour groups we discussed in the previous segment of this tutorial (see Step 7: Set Up Contour Groups and Color Maps). As a refresher, a contour group ties a color map to a variable, thereby establishing how an object in your plot will be colored based on the values of the selected variable.

On the **Contour** page of the **Slice Details** dialog, we can choose the contour group to be used for the slice, which in turn determines the color map and the variable used for the contouring. Flooding best represents continuous values, so we'll use the **Flood By** drop-down menu to choose the Pressure group.

If the variable we want to use isn't listed on the **Flood By** drop-down menu, we'll want to go to the **Contour and Multi-Coloring Details** dialog to set it up. Conveniently, that dialog can be accessed right

from the **Slice Details** dialog by clicking the gear icon next to the **Flood By** drop-down menu. Try it now if you like, just to see it's the same dialog, then close the **Contour and Multi-Coloring Details** dialog.





Scenic Detour: Creating Multiple Slices

Slices are great, so let's add more of them! We don't need to do this for the plot we're trying to make, which is why this section is labeled as a detour. You can skip it if you want.

- 1. On the Definition page of the **Slice Details** dialog, turn off **Show Primary Slice** and turn on **Show Start/End Slices**.
- 2. Rotate the plot so that you are looking nearly, but not quite, straight down on the wing from above. The slice, being perpendicular to the view, should be close to a line.

Reminder: To rotate the plot, hold down the Control key (Command on Mac) and hold down the right mouse button while moving the mouse. Hit o (the lowercase letter O) to set the origin of rotation to the mouse location.

Notice the *orientation axis* in the upper right corner of the plot, which makes it easy to tell when the plot's Z axis is pointing at you.



- 3. Using the Start slider, set the starting slice location to 0.
- 4. Set the ending slice location to the wing tip, around 1.16.
- 5. Turn on the Show Intermediate Slices checkbox and set the number of slices to 4. A total of six slices are shown.

Here's what the **Slice Details** dialog should look at this point, along with the plot.

Slice Details ×
Show group 1 1 2 3 4 5 6 7 8
Definition Contour Vector Other Animate
Slice location: Y-Planes \vee
Show primary slice
1.18
Show start/end slices
Start:
0
End:
1.16
Show intermediate slices Number of slices: 4
Slice through: Volume Zones \checkmark
Obey source zone blanking
Show solid plane while dragging (faster)
Extract Slices
Close Help


Step 4: Prepare for Streamtraces

Streamtraces are a useful tool for visualizing the flow around a surface. We'll try out three different types of streamtraces: volume lines, volume ribbons, and surface lines.

We only need a single slice for this, so if you followed the multiple-slice detour, go back to the **Slice Details** dialog, turn off Show Start/End Slices, and turn Show Primary Slice on.

We want our slice at the tip of the wing; that's where the pressure differential can cause vortex shedding. Enter 1.18 for the primary slice's location in the Slice Details dialog to place it close to the wing's tip.

The **Slice Details** dialog should now look as shown here. You can close the dialog at this point, as we won't need it for a while.

Slice Details ×
Show group 1 1 2 3 4 5 6 7 8
Definition Contour Vector Other Animate
Slice location: Y-Planes \vee
Show primary slice
1.18
Show start/end slices
Start:
0
End:
1.16
Show intermediate slices Number of slices: 4
Slice through: Volume Zones \checkmark
Obey source zone blanking
Show solid plane while dragging (faster)
Extract Slices
Extract Silces
Close Help

Now, rotate the wing so that the tip is pointing almost directly at us. (Hold Control, or Command on Mac, while dragging with the right mouse button.) The orientation axis in the upper right corner of the plot will help you get the wing pointed the right way. It should look roughly like the image below.

What are we looking at here? The green-blue background is our slice, and it's hiding most of the wing at this point, so we only see the wing tip.



Last but not least, we can specify the vector variables that describe the velocity of the flow around the wing. If we don't do it in advance, Tecplot 360 will prompt us to choose the variables as we begin adding our streamtraces. However, just so we know where the feature is, let's do it ourselves now.

Choose $Plot \rightarrow Vector \rightarrow Variables$ from the menu bar. The Select Variables dialog appears. We actually don't have velocity values in our data, but we do have momentum, and the vector field for momentum is the same as for velocity. (Momentum is just velocity times density. In the next segment of this tutorial, you'll learn a technique you could use to calculate true velocity values.)

Therefore, for our purposes, we can use the momentum variables. So choose Momentum U, Momentum V, and Momentum W for the U, V, and W vector variables, respectively.

Select Variables					
Cho	oose the vector variables.				
U	5: Momentum U (Density*U)	\sim			
v	6: Momentum V (Density*V) ~				
w	/ 7: Momentum W (Density*W) ~				
OK Cancel Help					

When the **Select Variables** dialog appears as shown here, click **OK**.

Step 5: Seed Streamtraces

Turn on the Streamtraces checkbox in the Plot sidebar, then click the Streamtrace tool button, as shown here.

You can now place a line with the mouse and seed streamtraces evenly along the line. To place the line, click and hold the mouse button at the starting point, drag to the end point, and release the mouse button. Try drawing a roughly vertical line along the leading edge of the wing, as shown here.



Tecplot 360 creates streamtraces showing the flow around the wing, as shown below. The streamtraces are generated by simulating the release of massless particles at the slice surface at equally-spaced

points along the line (called "seeding") and calculating their path of travel based on the vector variables that describe the flow.



Some of the streamtraces are behind the slice. To see the streamtraces better, let's change the slice to be 40% transparent. To do this, right-click the slice and choose 40 from the right-most drop down menu in the context toolbar.



Now, rotate the plot in three dimensions to better see the streamtraces and the wing. A possible result is shown below.



Step 6: Seed Volume Ribbons

By default, streamtraces are simple lines with arrowheads. Ribbons can allow us to see the flow better because they show local twist using a 2D surface. Here's how to switch to volume ribbons.

1. Open the Streamtrace Details dialog by clicking the 🚰 button next to Streamtraces in the Plot sidebar. Make sure the Placement page is active using the tabs at the top of the dialog.

Solution time: 0	0	
		🔯 Streamtrace Details 🛛 🛛 🗙
		_
	J.	Show streamtraces
		Placement Lines Rod/Ribbon Timing Animate Term Line Integration
Show derived objects		Create streamtraces with format: Volume Line \checkmark Direction: Both \checkmark
Iso-Surfaces		
		Seed streamtraces: On Surfaces of Active Objects
🗹 Slices 🛛 🐧	P 🕼	Number of seed points: 50 🜩
Streamtraces	¥ 👯	
	23	Info:
Show effects		• Dependently, distributes points proportional to the surface call density among all active report
🖂 Lighting	ž 🏄	 Randomiy distributes points proportional to the surface cell density among all active zones.
		Number of characteristics 10 County Characteristics (1) Delate tasts
		Number of streamtraces: 10 Create Streamtrace(s) Delete Last Delete All
		Close Help

- 2. Click **Delete All** in the **Streamtrace Details** dialog to remove the existing streamtraces.
- 3. Change the drop-down menu at the top of the dialog to Volume Ribbon.
- 4. Click the streamtrace tool button next to Streamtraces in the Plot sidebar. Leave the Streamtrace Details open, but move it out of the way.
- 5. As before, drag out a line along the leading edge of the wing to seed the streamtraces and create the ribbons.

The plot should look something like this with the ribbons.



Let's change them from white to a more attention-grabbing color. We can do this on the Rod/Ribbon page of the **Streamtrace Details** dialog.

🧕 Streamtrace Details							×
Show streamtraces							
Placement Lines Rod/	Ribbon	Timing	Animate	Te	rm Line	Integration	
Show paths							
Rod/ribbon width:	0.01						
Rod points:	3						
Show mesh							
Color:		📕 Bl	ack				
Line thickness (%):	0.1			\sim			
Show contour flood							
Flood by:	C1: Dens	sity		\sim	Dr.		
Show shade							
Color:		W	nite				
Use lighting effect:	Gouraud			\sim			
Use translucency:	50			*			
					Clos	se I	Help

Near the bottom of this dialog, under the Show Shade checkbox, click the button next to Color. Then, in the Color Chooser, click the red swatch.

🖲 Color Chooser					×
_					
				_	
■Multi 1	2 3 4	5	6 7	8	RGB
	Clos	e		Help)

The plot with red ribbons is shown below:



Step 7: Adjust Rotation and Lighting

You can adjust the direction of the lighting to make the ribbons stand out more. To do this, click the sun toolbar icon in the Plot sidebar.

Show effects	
🗹 Lighting	R
Translucency	Move light source tool

Click or drag around the plot to move the light. As you do so, you will find spots where the ribbons catch a highlight. You can also try rotating the plot to get the best view.

Step 8: Surface Streamtraces

So far, we've been looking at flow through the air around the wing: volume streamtraces and ribbons. We can also visualize the flow across the surface of the wing.

Just to be clear, these streamtraces will be calculated using the momentum data from the volume zone around the wing. The wing surface zone does not actually contain momentum or velocity data. The data file we provide for this tutorial has the zones set up to make this work correctly.

We used the slice to specify a plane on which we could draw a line to seed streamtraces in a volume. When creating surface streamtraces, we just seed directly on the surface. So click the Slices checkbox in the Plot sidebar to turn off the slice.

Show derived objects	
Iso-Surfaces	****
Slices	W
Streamtraces	~ ₹

Next, let's delete the ribbons we've created by clicking **Delete All** on the Placement page of the **Streamtrace Details** dialog. Then choose "Surface Line" at the top of the dialog.

Streamtrace Details	×
Show streamtraces	
Placement Lines Rod/Ribbon Timing Animate Term Line Integratio	n
Create streamtraces with format: Volume Line Virection: Both Volume Line Volume Line Volume Line Volume Line Volume Ribbon Surface Line Volume Ribbon Surface Line Volume Rod	
Info: • Randomly distributes points proportional to the surface cell density among all active zone	÷5.
Number of streamtraces: 0 Create Streamtrace(s) Delete Last Delete A	AII
Close He	lp

Let's rotate the wing again so that we can see across it. (Last reminder: hold Control, or Command on Mac, while dragging with the right mouse button.) We're going for an angle something like this.



As always, you can use the orientation axis in the upper right corner of the plot to help you stay oriented.

Now, just as we did before when seeding from the surface of our slice, we choose the streamtrace tool in the Plot sidebar, then drag a line across the wing surface, parallel to the leading edge. After Tecplot 360 creates the streamtraces, we end up with something like this.



We made our streamlines thicker so you can see them more easily. For extra credit, figure out how to make yours thicker, too. (The setting you need is on the Lines page of the Streamtrace Details dialog.)

We're done with streamtraces for now. Turn off the Streamtraces checkbox in the Plot sidebar when you're done to hide the streamtraces and show just the wing.

Step 9: Create Iso-Surfaces

Iso-surfaces are great way to visualize a constant value of a contour variable as a surface. In other words, you specify a value, and the iso-surface shows you where the specified variable has that value.

Let's use it to visualize the shockwave as the wing passes the speed of sound. We do this by creating an iso-surface at Mach 1.



- 1. Turn on the Iso-Surfaces checkbox in the Plot sidebar.
- 2. Click the 🛃 button next to Iso-Surfaces to open the Iso-Surface Details dialog.

道 Iso-Surface Details		×
Show group 1 1 2 3	4 5 6 7 8	
Definition Contour and S	hade Mesh and Vector	Animate
Define iso-surfaces using:	C1: Density	~ 🕸
Draw iso-surfaces at:	1 Specified Value	\sim
Value 1:	0.74354095	
Value 2:	1.1161563	
Value 3:	1.48877165	
Contour variable		
Min:	0.370926	
Max:	1.86139	
Surface generation method:	Auto	\sim
Obey source zone blankin	g	
	Close	Help

Now, we need to choose a contour group that associates the Mach variable with a color map. However, if you expand the Define Iso-Surfaces Using dropdown menu, you will see we don't have one. So we need to create one.

- 3. Click the gear icon next to this menu to open the Contour and Multi-Coloring Details dialog, which we've seen before.
- 4. Click the **3** button at the top of this dialog to define the third contour group.
- 5. Using the menu at the top of the dialog, choose the Mach variable.
- 6. Choose the Large Rainbow color map. Here's how it should look.

🗕 Contour & Multi-Coloring	Details	\times
13: Mach	✓ 1 2 3 4 5 6 7 8	
Levels and Color Bands	Lines Labels Legend	
Contour levels		
1:0.1	▲ Set Levels	
2:0.2	Add Level 0	
4:0.4		
5:0.5	Remove Selected	
6:0.6	Contour variable range	
7:0.7	Min: 0 Max: 1.57558	
0.00	¥	
Color map options		_
		<u> </u>
Large	Rainbow 🗸 🔅	
Color map distribution method	Color cutoff	
Banded Ocontinuous	s Cutoff above: 1.18169	
Contour values at endpoint	ts Cutoff below: 0.393895	
Min: 0	Color map adjustments	
Max: 1.57558	Reset 👻 🗌 Reverse	
Approximate continuou	IS Repeat cycles: 1	
	Close Help	

- 7. Close the **Contour and Multi-Coloring Details** dialog.
- 8. Back in the **Iso-Surface Details** dialog, choose contour group 3 from the *Define Iso-Surfaces Using* menu, then enter a value of 1 in the Value 1 field.

Iso-Surface Details		\times
Show group 1 1 2 3	4 5 6 7 8	
Definition Contour and S	Shade Mesh and Vector Animate	
Define iso-surfaces using:	C3: Mach 🗸 🔅	
Draw iso-surfaces at:	1 Specified Value $\qquad \qquad \lor$	
Value 1:	1	
Value 2:	1.1161563	
Value 3:	1.48877166183	
Contour variable		
Min:	0	
Max:	1.57558	
Surface generation method:	Auto \checkmark	
Obey source zone blanki	ng	
	Close Help	

The plot is shown here. We've rotated the wing to get a better view.



This iso-surface shows us where the air is moving at Mach 1 around the wing. Try adjusting the Value 1 field using the arrows next to the numeric entry field and watch the plot change in response.

It is often instructive to show more than one iso-surface at more than one value. Let's add a second isosurface at Mach 1.2. To do this, change the **Draw Iso-Surfaces** At menu to *2 Specified Values* and enter 1.2 in the Value 2 field.

The dialog options are shown here.

🗕 Iso-Surface Details	:	×
Show group 1 1 2 3	4 5 6 7 8	
Definition Contour and S	Shade Mesh and Vector Animate	
Define iso-surfaces using:	C3: Mach 🗸 🔅	
Draw iso-surfaces at:	2 Specified Values $\qquad \lor$	
Value 1:	1	
Value 2:	1.2	
Value 3:	1.48877166182	
Contour variable		
Min:	0	
Max:	1.57558	
Surface generation method:	Auto ~	
Obey source zone blankir	ng	
	Close Help	

Where's the second iso-surface? Well, it's closer to the wing than the first one, so it is hidden. Let's make the iso-surfaces partly transparent so we can see the second one inside the first (and also, dimly, the wing surface inside the second).

To do this, right-click the iso-surface in the plot, then turn on transparency by clicking the rightmost icon in the toolbar. Our iso-surfaces are both in the same group, so they both become transparent.

⊞ - Ø - ≩ - € - ঢ়ৢ -
Iso-Surface Group: 1
Hide Define iso-surface by Extract
lso-surface details

Now you can see the Mach 1.2 iso-surface inside the Mach 1 iso-surface. Rotate the plot and see! This is the shock surface where the transition between subsonic and supersonic is happening. Here's our final iso-surface plot.



Step 10: Probe Data

The last exploration technique we'll explore is probing. Probing shows the values of all variables at a location you click in the plot. Simply choose the Probe tool from the toolbar at the top of the screen.



Now, click points of interest in your plot. The Probe sidebar, which pops out along the right edge of the screen, displays the variable values at each clicked location.

You can also copy variable names, values, or both from the probe results by selecting the desired information and right-clicking, or by pressing Control-C (Command-C on Mac OS X).

robe		5	>
Probe At		(2
Variable	Value	٦	٧a
х	0.425983793755309592		'iable
у	0.338006084962933495		Valu
z	0.31262965120760311		ß
Density	1.00150962451886727		<u>6</u>
Momentum U (Density*U)	331.637199230642238		Cen
Momentum V (Density*V)	-9.69596885222857097		ter Va
Momentum W (Density*W)	14.8246646215857218		alues
Energy (Density*E)	N/A		
SA Turbulent Eddy Viscosity	N/A		Zone/
Pressure	N/A		Cell I
Temperature	N/A		đ
Pressure_Coefficient	N/A		교
Mach	1		ce V
Laminar_Viscosity	N/A	1	lighb
Skin_Friction_Coefficient	N/A		9
Heat_Flux	N/A		
Y_Plus	N/A		
Eddy_Viscosity	N/A		

As with any sidebar, you can move the Probe sidebar to another edge of the workspace, combine it with another sidebar, or even tear it off and move it outside the main Tecplot 360 window—to a different monitor, if you want.

As before, a Tecplot 360 layout (.lay) file containing a snapshot of the final result of this tutorial segment is in OneraM6wing/finallayouts/ExtenalFlowVideo2.lay in the examples folder in your Tecplot 360 installation folder.

Comparing a CFD Simulation with Experimental Data

In this segment, we'll be creating a classic Coefficient of Pressure (Cp) plot using both simulated and experimental data. To do this, we'll first append the experimental data, which was gathered via pressure taps along the wing in a wind tunnel. Then we'll extract a slice and create an XY line plot in a new frame.

The experimental data is normalized in both the span and chord directions: the Y direction is normalized by b (the span) and the X direction is normalized by c (the chord). The simulation data is not normalized, so part of our work will be to adapt the experimental data so that it can be compared to the simulation.

Step 1: Load Layout

We'll be continuing from where we left off with segment 1 of this tutorial, *not* from segment 2. Load the layout OneraM6wing/finallayouts/ExtenalFlowVideo1.lay in the examples folder in your Tecplot 360 installation folder and follow along.

Step 2: Append Experimental Data

Choose File \rightarrow Load Data from the Tecplot 360 menu, then open the experimental data file. This is a

Tecplot ASCII file named Onera_Experimental.dat and can be found OneraM6wing folder in your Tecplot 360 examples directory. (If you can't see this file, choose All Files in the menu at the bottom of the dialog.) The experimental data file we've included is a slightly modified version of data provided by NASA. For more information, see turbmodels.larc.nasa.gov/onerawingnumerics_val.html.

If you used All Files, the **Choose Loader** dialog appears after you select the file to ask what format the data is in.



Choose **Tecplot Data Loader** and click **OK**.

Next, Tecplot 360 asks whether you want the new data to replace the existing data, or to append the new data.



In the Load File(s) dialog, click **Append data to active frame**.

Step 3: Variable Load and Combine

The **Variable Load and Combine** dialog should appear after appending the data. This dialog gives the user options to combine certain variables as well as the option to not add certain variables. Variables that are automatically combined are colored blue. Manually combined variables are colored green. Anything else is colored black. Notice that Pressure_Coefficient has been automatically combined.

The files being opened do not contain the same variables. If two or more variables from different files have the same meaning, you may combine them by selecting an equal number of variables from both lists, then clicking Combine. Only variables that you move into the "Variables to load" list will be loaded. It is not necessary to combine (or load) all variables. Available variables Variables to load TapNumber X/L Y/b Z/L Cp Error Add Selected All	Variable Load and Combine		×
Only variables that you move into the "Variables to load" list will be loaded. It is not necessary to combine (or load) all variables. Available variables Variables to load TapNumber 1: x X/L 2: y Y/b 3: z Z/L 4: Density Cp Error 6: Momentum U (Density*U) 6: Momentum W (Density*V) 7: Momentum W (Density*V) 8: Energy (Density*E) 9: SA Turbulent Eddy Viscosity	The files being opened do not contain the same variables. If may combine them by selecting an equal number of variables	two or more variables from different files have th s from both lists, then clicking Combine.	e same meaning, you
Available variables Variables to load TapNumber 1: x Variable state: X/L 2: y 3: z Y/b 3: z 4: Density 4: Density Cp Error 5: Momentum U (Density*U) 6: Momentum V (Density*V) Not combined S: Add Selected S Add All S S A Turbulent Eddy Viscosity Variables.txt Alias file: C:\Program Files\Tecplot\Tecplot 360 EX Beta\variable_aliases.txt Apply OK Cancel	Only variables that you move into the $\\$ variables to load" list	will be loaded. It is not necessary to combine (or	load) all variables.
TapNumber 1: x Variable state: X/L 2: y 3: z Y/b 3: z 4: Density Z/L 4: Density 5: Momentum U (Density*U) G: Momentum V (Density*V) 6: Momentum V (Density*V) 7: Momentum W (Density*E) 9: SA Turbulent Eddy Viscosity 9: SA Turbulent Eddy Viscosity Add Selected State Add Selected Add All Alias file: C:\Program Files\Tecplot\Tecplot 360 EX Beta\variable_aliases.txt	Available variables	Variables to load	
Alias file: C:\Program Files\Tecplot\Tecplot 360 EX Beta\variable_aliases.txt Apply	TapNumber X/L Y/b Z/L Cp Error Add Selected	1: x 2: y 3: z 4: Density 5: Momentum U (Density*U) 6: Momentum V (Density*V) 7: Momentum W (Density*V) 8: Energy (Density*E) 9: SA Turbulent Eddy Viscosity	 Variable state: Combined Auto combined Not combined
	Alias file: C:\Program Files\Tecplot\Tecplot 360 EX Beta\var	riable_aliases.txt Apply	al Help

For this example, we will load all of the variables from the Experimental dataset. Select **Add All** and then select **OK** to close the **Variable Load and Combine** dialog.

To verify that the data was loaded correctly, open the **Data Set Information** dialog by choosing **Data** \rightarrow **Data Set Info** from the menu. Notice the seven new zones: Section 1 - 0.2, Section 2 -0.44, and so on. These zones contain the experimental data from the pressure taps placed along the wing.

🧕 Data Set Infor	mation			Х
Zone/Var Info	Data Set	Aux Data		
Zone(s)			Variable(s)	
1: FluidVolum 2: WingSurfa 3: Section 1 4: Section 2 5: Section 3 6: Section 4 7: Section 5 8: Section 6 9: Section 7	ne ce - 0.2 - 0.44 65 8 9 96 99		1: x A 2: y 3: z 4: Density 5: Momentum U (Density*U) 6: Momentum V (Density*V) 7: Momentum W (Density*W) 8: Energy (Density*E) 9: SA Turbulent Eddy Viscosity 10: Pressure 11: Temperature 12: Pressure_Coefficient 13: Mach	
Zone Type: Orde	ered		Var Type: Float	
IMax: 34			Var Location: Nodal	
JMax: 1			Var Status: Passive	
KMax: 1			Var Range Selected Zone: Min: N/A Max: N/A Var Range Active Zones: Min: -6.4472 Max: 5.6413 Load Variables.	
			Close Help	

You'll also notice several new variables, starting at variable #19: TapNumber, X/L, Y/b, Z/L, and Cp Error.

Step 4: Normalize Y to Y/b

We want to compare the experimental data with our simulation data at a pressure tap location. However, the experimental data is, as we mentioned, normalized: instead of using a Y that ranges from 0 to the span length, as in the simulation data, it uses a Y over b (Y/b) that ranges from 0 to 1.

If we open the **Data Spreadsheet** dialog (**Data** – **Spreadsheet**), choose our WingSurface zone, and

scroll over to see the "Y/b" variable, we can see that our simulation data does not contain this information. It is only in the experimental data set.

Zone:	2: Win	gSurface			Variat	ole: 1: x				
Plane:	К ~			Index: 1	- Forma	it: Best Floa	t ×			Precision: 3
N\V	efficie	Heat_Flux	Y_Plus	Eddy_Viscosity	TapNumber	X/L	Y/b	Z/L	Cp Error	^
1		Not Loa	Not Loa	Not Loaded	0	0	0	0	0	
2		Not Loa	Not Loa	Not Loaded	0	0	0	0	0	
3		Not Loa	Not Loa	Not Loaded	0	0	0	0	0	
4		Not Loa	Not Loa	Not Loaded	0	0	0	0	0	
5		Not Loa	Not Loa	Not Loaded	0	0	0	0	0	
6		Not Loa	Not Loa	Not Loaded	0	0	0	0	0	
7		Not Loa	Not Loa	Not Loaded	0	0	0	0	0	
8		Not Loa	Not Loa	Not Loaded	0	0	0	0	0	
9		Not Loa	Not Loa	Not Loaded	0	0	0	0	0	
10		Not Loa	Not Loa	Not Loaded	0	0	0	0	0	
11		Not Loa	Not Loa	Not Loaded	0	0	0	0	0	
12		Not Loa	Not Loa	Not Loaded	0	0	0	0	0	
13		Not Loa	Not Loa	Not Loaded	0	0	0	0	0	
14		Not Loa	Not Loa	Not Loaded	0	0	0	0	0	
15		Not Loa	Not Loa	Not Loaded	0	0	0	0	0	
16		Not Loa	Not Loa	Not Loaded	0	0	0	0	0	
17		Not Loa	Not Loa	Not Loaded	0	0	0	0	0	
18		Not Loa	Not Loa	Not Loaded	0	0	0	0	0	
19		Not Loa	Not Loa	Not Loaded	0	0	0	0	0	
<		Not Loo	NotLoo	Not Loaded	n	0	0	n	0	>
Alte	r in all sl	hared zones								Load Variables
									Close	Help

However, if you use the Zone menu to switch to one of the experimental data zones, such as Section 1 - 0.2, you will see that it has values for this variable.

🖲 Da	ta Spre	adsheet									×
Zone:	3: Se	ction 1 - 0.2			✓ Varia	ble: 1: x					\sim
Plane:	$K_{\rm e} \sim$			Index: 1	÷ Form	at: Best Floa	at 🖂			Precision: 3	•
I\V	efficie	Heat_Flux	Y_Plus	Eddy_Viscosity	TapNumber	X/L	Y/b	Z/L	Cp Error		^
1		0	0	0	12	0.9503	0.2	-0.00726	0.02		
2		0	0	0	11	0.81597	0.2	-0.02193	0.02		
3		0	0	0	10	0.66593	0.2	-0.03587	0.02		
4		0	0	0	9	0.56588	0.2	-0.04301	0.02		
5		0	0	0	8	0.46599	0.2	-0.04773	0.02		
6		0	0	0	7	0.3658	0.2	-0.04889	0.02		
7		0	0	0	6	0.26579	0.2	-0.04702	0.02		
8		0	0	0	5	0.16594	0.2	-0.04194	0.02		
9		0	0	0	4	0.04993	0.2	-0.02971	0.02		
10		0	0	0	3	0.02002	0.2	-0.02268	0.02		
11		0	0	0	2	0.00593	0.2	-0.01333	0.02		
12		0	0	0	1	0.00034	0.2	0.00292	0.02		
13		0	0	0	34	0.00216	0.2	0.00802	0.02		
14		0	0	0	33	0.00866	0.2	0.01592	0.02		
15		0	0	0	32	0.02037	0.2	0.02285	0.02		
16		0	0	0	31	0.03525	0.2	0.02701	0.02		
17		0	0	0	30	0.06036	0.2	0.0313	0.02		
18		0	0	0	29	0.09959	0.2	0.03617	0.02		
19		0	0	0	28	0.15037	0.2	0.0408	0.02		
20		0	0	0	77	0 20044	0.2	0.04412	0.02	>	~
	er in all	shared zones								Load Variable	
	or in all s	2010-2010-2010-201100-20110-201100-201100000000									
									Close	Help	

Therefore, we will need to calculate this variable for our simulation data. We can do this using an equation.

Close the **Data Spreadsheet** dialog, then open the **Specify Equations** dialog by choosing **Data** \rightarrow **Alter** \rightarrow **Specify Equations**. We know that the span of the wing, *b*, is 1.19 from publicly-available information about the Onera M6 model. So, in the Equation(s) field, enter:

 $\{Y/b\} = \{y\}/1.19$

We only want to calculate this variable in our simulation zones, since it already exists in our experimental zones, so make sure only FluidVolume and WingSurface are selected in the Zones to Alter list. (Hold down the Control key, or Command on Mac, while clicking zones to toggle them on and off.) The **Specify Equations** dialog should appear as shown below.

Specify Equations					×
Equation(s)					
$\{Y/b\} = \{y\}/1.19$					
Data Set Info V Ignore divide by zero errors FE derivat	ive <u>m</u> ethod:	Moving Least Squares 🗸	•	Save Equations	Load Equations
Default Equation Modifiers					
Zones to Alter	Index Ran	iges			
1: FluidVolume]	Start	End	Skip	
2: WingSurface	I-Index	1	Mx	1	
3: Section 1 - 0.2	<u>]</u> -Index	1	Mx	1	
4: Section 2 - 0.44	K-Index	1	Mx	1	
6: Section 48	New var	data type:	Auto		
7: Section 59			Auto		
All Active None	ive <u>w</u> var	location:	Auto		
			C <u>o</u> mput	te Close	Help

Click **Compute** to calculate the Y/b value for our simulation zones.

Finally, let's check our work by probing at the wing tip.

- 1. Click the Probe tool in the Tecplot 360 toolbar.
- 2. Click at the tip of the wing.
- 3. Observe the Y/b value in the Probe sidebar.

We should expect a value of 1 for *Y/b*. However, since the wing tip is beveled, the *Y/b* value will actually be slightly greater than 1. Everything is exactly as it should be. Mission accomplished!

Step 5: Extract Slice from Simulation Data at Pressure Tap Location

Let's go ahead and change the Y axis to Y/b. Choose **Plot** \rightarrow **Assign XYZ** from the Tecplot 360 menu bar. Then, in the Select Variables dialog, choose "Y/b" for the Y-axis variable, as shown here. Click **OK**. to save this change.

Select Variables	×
Choose the variables to assign to the axes.	
X 1: x	\sim
Y 21: Y/b	\sim
Z 3: z	\sim
OK Cancel Help	

The wing may rotate in the workspace when you change the Y variable. This is normal.

Our experimental data contains information only at the locations of the pressure taps on the wing. For this example, we'll compare the simulation data with the experimental data from the pressure tap at Y/b = 0.65. We will create a slice of the simulation at this location and extract that slice to a new zone.

All the simulated surface data along the wing at the location of the pressure tap, in other words, will be copied into a new two-dimensional zone. Here's how.

1. Turn on slices in the Plot sidebar.

Show derived objects	
Iso-Surfaces	*
Slices	W
Show slices es	~ \$ 🥸

- 2. Open the Slice Details dialog by clicking next to Slices in the Plot sidebar.
- 3. In the Slice Details dialog:
 - Change the Slice Location to Y-Planes
 - Make sure Show Primary Slice is enabled
 - $\,\circ\,$ Enter the value 0.65 for the location of the primary slice.
 - Set Slice Through to Surface Zones.

Slice Details		×
Show group 1 1 2 3	4 5 6 7	8
Definition Contour Vector	Other Anim	ate
Slice location: Y-Planes $$		
Show primary slice		*
	0.65	•
Show start/end slices		
Start:		
	0.321152305603	
End:		
	6.10189380646	
Show intermediate slices	Number of slices:	1
Slice through: Surface Zones	\sim	
Volume Zones		
Surfaces of Volum	e Z-kš na Surface Zones	1
Extract Slices		
	Close	Help
	0.000	nop

A band appears on the surface of the wing at the location of our slice, as shown below.



Choose the Extract Slices button to extract this slice to a new zone. (Or by selecting Data → Extract → Slices).

Extract Slice	\times
Group	
● Slice group: 1 ∨	
 All active slice groups 	
Time	
 Current solution time 	
All solution times	
Create a zone for each	
Since	
 Connected region 	
Resulting 1D zone type	
I-ordered if possible	
O FE line segment	
Extract Close Help	

- 5. In the **Extract Slice** dialog, leave all of the default values and click **Extract** to proceed with the slice extraction.
- 6. Close the **Extract Slice** dialog and the **Slice Details** dialog.

Our new slice zone is zone 10 and is named "Slice Y=0.65." You can verify this in Data Set Information, if you like.

Step 6: Normalize Slice's X to X/L

Now we can normalize the X dimension of our slice to the chord, in order to match the experimental data. We'll do this using the same **Specify Equations** dialog we used to normalize the Y dimension. In the experimental data, this is stored as the variable *X/L*. (If you're curious, you can verify this in the **Data Spreadsheet** dialog, as we did previously with *Y/b*.)

The equation we'll need is:

 $\{X/L\} = (\{x\} - MINX) / (MAXX - MINX)$

There's a twist that we need to address before we actually perform this calculation. The equation uses the MAXX and MINX intrinsic variables. These variables are provided by Tecplot 360 and refer to the maximum and minimum values of X in *all active zones*. We want the maximum and minimum X for just our slice, so as to calculate the chord, so we need to temporarily deactivate all other zones while we perform this calculation.

So open the **Zone Style** dialog by clicking the **Zone Style** button in the Plot sidebar.

Zone Number	Zone Name	Group Number	Show Zone	Use Value Blanking	Lighting Effect	Use Surface Translucency	Surface Translucency	~
1	FluidVolume	1	\checkmark	\checkmark	Gouraud	~	50	
2	WingSurface	1	\checkmark	~	Gouraud	~	50	
3	Section 1 - 0.2	1	\checkmark	~	Gouraud	~	50	
1	Section 2 - 0.44	1	\checkmark	\checkmark	Gouraud	~	50	
5	Section 365	1	\checkmark	\checkmark	Gouraud	~	50	
;	Section 4 9	1	1	. 1	Couroud	. 1	50	~
election cr	iteria:	Zo	nes	Groups	0	Clear		

Right-click the Show Zone checkbox for the extracted slice zone and choose Show Selected Only.

Zone Sty Mesh C	le Contour Vecto	or Scatt	ter S	Shade Ed	ge Point	s Surfaces	Volume Effec	ts
Zone Number	Zone Name	Group Number	Show Zone	Use Value Blanking	Lighting Effect	Use Surface Translucency	Surface Translucency	
5 6	Section 4	1			Gouraud		50	
7	Section 5	1	$\overline{\checkmark}$		Gouraud		50	
8	Section 6	1	\checkmark	~	Gouraud		50	
9	Section 7	1	\checkmark	\checkmark	Gouraud	Image: A start of the start	50	
10	Slice: Y=0.65	1			Gouraud	~	50	`
Selection criteria:				Activate Deactivate		Groups	Clear	
				Show selected only Invert				
			_			_		Close Help

The checkboxes next to all zones except the slice zone toggle off. Your plot also disappears; don't panic!

Now, choose $Data \rightarrow Alter \rightarrow Specify Equations$ and enter our equation in the Equation(s) field. For convenience, here it is again:

 $\{X/L\} = (\{x\} - MINX) / (MAXX - MINX)$

Since we need this calculation only for the slice, make sure our new slice zone, listed at the bottom of the Zones To Alter list as "10: Slice Y=0.65" is the only zone selected.

Specify Equations				×	
Equation(s)					
X/L = (x - MINX) / (MAXX - MINX)					
Data Set Info Ignore divide by zero errors FE derivat	ive <u>m</u> ethod: M	loving Least Squares 🗸	<u>S</u> ave E	quations	
Default Equation Modifiers					
Zones to Alter	Index Rang	jes			
5: Section 365	5	Start I	End	Skip	
6: Section 48	I-Index	1	Mx	1	
7: Section 59	<u>]</u> -Index	1	Mx	1	
9: Section 799	<u>K</u> -Index	1	Mx	1	
10: Slice: Y=0.65	New var d	lata <u>t</u> ype:	Auto		
	Ne <u>w</u> var le	location:	Auto	~	
			Compute	Close Help	
			Compute	ciose neip	

Now you're ready to click **Compute**.

After the calculation is complete, go back to the **Zone Style** dialog and turn the disabled zones back on by selecting all rows (Control-A, or Command-A on Mac, is a handy keyboard shortcut for this), then toggling on the Show checkboxes for any disabled zone. This toggles on all selected zones.

You can now close both the **Zone Style** and **Specify Equations** dialogs.

Step 7: Create XY Plot of Slice

We're ready to create an XY plot of the slice. We'll put it in its own frame: frames are the Tecplot 360 way of showing multiple plots on a page.

First, click the frame tool in the Tecplot 360 toolbar.



Next, use it to draw the new frame in the workspace. The placement and size don't matter, since we'll have Tecplot 360 set it up nicely for us in a moment. Click and hold the mouse button somewhere in your plot, then drag down and to the right to create the frame. Release the mouse button when a rectangle is visible in the plot.

The new frame might look something like this.



Next, choose $Frame \rightarrow Tile Frames$ in the Tecplot 360 menu bar. The Tile dialog appears, as shown here.



Click the bottom right button in the **Tile Frames** dialog to stack the frames on top of each other in the workspace. Then close the **Tile** dialog.

Your plot now looks something like this.



Now, using the drop-down menu at the top of the Plot sidebar, change from Sketch to XY Line.

Plot	5	×
Sketch		\sim
3D Cartesian 2D Cartesian		
XY Line	N	
Polar Line Sketch	XY Line	ſ

The **Create Mappings** dialog appears. In this dialog, we'll set the X-Axis to our *X/L* variable, our Y-Axis to **Pressure_Coefficient**, and the Zone to our extracted slice zone. The desired settings are shown here.
🧕 Create Ma	opings	×
X-Axis varia	ble versus Y-Axis variable for one zone	
🔘 X-Axis varia	ble versus all other variables for one zor	ne
🔘 Y-Axis varial	ole versus all other variables for one zor	ne
🔘 X-Axis varia	ble versus Y-Axis variable for all linear z	ones
Mapping name:	Map 0	
X-Axis:	20: X/L	\sim
Y-Axis:	12: Pressure_Coefficient	\sim
Zone:	10: Slice: Y=0.65	\sim
	OK Cancel Help	

Click **OK** to create the plot, which is shown here.



If you're familiar with Cp plots, you'll notice something odd about our current plot! The problem is that the Y axis is reversed from what it should be. Fortunately, this is easily corrected.

- 1. Choose **Plot** \rightarrow **Axis** from the menu bar to open the **Axis Details** dialog.
- 2. Click Y1 at the top to choose the Y axis.
- 3. Toggle on the Reverse Axis Direction checkbox.

🚳 Axis Details	×
Show Y1-Axis X1 Y1 X2 Y2 X3 Y3 X4 Y4 X5 Y5	
Range Grid Ticks Labels Title Line Area	
Min: -1.20455479622	
Max: 0.929452294815	
✓ Preserve axis length when changing range Use log scale	
Reverse axis direction	
Dependency	
O Dependent X to Y ratio: 1	
Independent	
Automatically adjust axis ranges to nice values	
	Close Help

4. When the settings are as shown here, close the **Axis Details** dialog.

Our plot now looks like this. Much better.



Step 8: Plot Experimental Data

We're in the home stretch now! Let's add the experimental data to our plot. First, open the **Mapping Style** dialog by clicking the **Mapping Style** button in the Plot sidebar.

🧕 Mappin <u>o</u>	g Style						×
Definitions	Lines	Curves	Symbols Error B	ars Bar Charts Ind	lices		
Map Number	Map Name	Show Map	X-Axis Variable	Y-Axis Variable	Zone	Sort	
1	Map 0	\checkmark	20: X/L	12: Pressure_Coeffic	10: Slice: Y=0.65	None	>
<							>
Selection cr	iteria:			Select Clear			
	c	Create Ma	р Сору Мар	Delete Map To Top	To Bottom	Close	Help

A *line map* is the Tecplot 360 way of associating (mapping) a variable with a visual style for each line in your plot. Tecplot 360 created our first line map for us when we switched to XY Line mode. The **Mapping Style** dialog is used to manage the line maps in our XY Line plot.

We want to show both the simulation and experimental data in our plot, so we'll need to create a second line map for the experimental data. The easiest way to do this is to copy the existing simulation data line map and modify it to display the experimental data.

- 1. Click the first row in the Mapping Style table to select the first line map.
- 2. Click the **Copy Map** button at the bottom of the dialog. A second line map appears in the table.
- 3. Right-click the Zone field in the second row of the table, then choose zone 5, Section 3 .65, as the zone for the second line map. This is the experimental data for the pressure tap at 0.65 along the wing, corresponding to the position of the slice we've taken of the simulation zone.
- 4. Toggle on the Show Map checkbox for the second map.
- 5. Double-click the name of each mapping and enter an appropriate name. Especially with complex line plots with many mappings, giving your mappings good names will make it much easier to make changes to the plot. We named them as follows:
 - Map 1: CFD Simulation: Y/b = 0.65
 - Map 2: Experimental: Y/b = 0.65

The Mapping Style dialog and the plot should now appear as shown here.



As you can see, there's a *slight* problem here: both the simulation and the experimental data have the same appearance, making it impossible to distinguish them. We can address this by using different colors for the two zones.

Additionally, the simulation data is continuous, while the experimental data, having been measured at specific points along the wing, is discrete. Therefore, we will display the experimental data using a symbol at each measurement location rather than as a line.

Finally, we have uncertainty information for the experimental data, which we can display using error bars.

All of these appearance changes can be managed using the **Mapping Style** dialog. Let's get to it.

- 1. Change to the Lines page of the **Mapping Style** dialog using the tabs at the top of the dialog, then toggle off the Show Lines checkbox for the Experimental mapping.
- 2. Also on the Lines page, right-click the line thickness for the CFD Simulation map and change it to 0.40% to make it thicker.

The Lines page should appear as follows.

Maria	Mar	Channe	Channe	Line	Line	Line	Detterm	
Map Number	Map Name	Map	Snow Lines	Color	Thickness	Pattern	Length	
	CFD Simulation: Y/b = 0.65	\checkmark	\checkmark		0.40%	Solid	2.00%	
2	Experimental: Y/b = 0.65	\checkmark			0.10%	Solid	2.00%	

3. Change to the Symbols page of the **Mapping Style** dialog. You'll note that all settings are grayed out here. To use the Symbols settings, we must enable the Symbols of our plot by toggling on the Symbols checkbox on the Plot sidebar.

XY Line	\sim
Show mapping layers	
✓ Lines	
Symbols	
Bars	
Error bars	
Mapping Style	

- 4. Now that the Symbols settings are available in the **Mapping Style** dialog, turn the symbols off for our CFD Simulation map.
- 5. Right-click the Outline Color for the Experimental map and choose a blue using the Color Chooser.
- 6. The symbols are way too big, so right-click the Symbol Size for this map and change it to 1.0. The Symbols page should look as follows.

Mapping	Style								
Definitions	Lines Curves Symbols	Erro	or Bars E	Bar Charts In	dices				
Map Number	Map Name	Show Map	Show Symbols	Symbol Shape	Symbol Size	Symbol Spacing	Outline Color	Line Thickness	
1 2	CFD Simulation: Y/b = 0.65 Experimental: Y/b = 0.65	\checkmark		Square Square	2.50% 1.00%	Draw all Draw all		0.10% 0.10%	None None
<									>
Selection cri	iteria:		Select	Clear					
	Create Map Copy	у Мар	Delete M	1ар То То	р	o Bottom	Clos	se	Help

Scenic Detour: Line Map Context Menu and Toolbar

Earlier in this tutorial sequence, we suggested you right-click on your 3D plot to discover a quick way to make changes to your zone style—without a trip to the **Zone Style** dialog. Try right-clicking on a line in your plot to reveal the line map context menu and toolbar (shown at right). Then try to do the next step, adding error bars, using the context menu rather than the **Mapping Style** dialog.



Step 9: Add Error Bars

The experimental data includes an error variable calculated from data NASA provides about the Onera M6 wing, namely that the measurement error for the pressure taps was found to be ± 0.02 . Let's add error bars to the plot to visualize this information.

1. Change to the Error Bars page of the **Mapping Style** dialog. As with the Symbols page, all the Error Bars settings are initially grayed out. So we will need to toggle on Error Bars in the Plot sidebar to enable the Error Bars layer, which makes these settings available in Mapping Style.

- 2. With the Error Bar settings available in the **Mapping Style** dialog, toggle on Show Error Bars for the Experimental map.
- 3. Right-click the Error Bar Variable for this map and choose variable 23, "Cp Error."
- 4. Right-click the Error Bar Color for the map and choose the same blue you earlier chose for the map's symbols.

The Error Bars page should now look as follows.

Mapping	Style							
Definitions	Lines Curves Symbols	Erro	or Bars Ba	r Charts Indices				
Map Number	Map Name	Show Map	Show Error Bars	Error Bar Variable	Error Bar Type	Error Bar Spacing	Error Bar Color	Erro
1 2	CFD Simulation: Y/b = 0.65 Experimental: Y/b = 0.65	\checkmark		3: z 23: Cp Error	Vertical Vertical	Draw all Draw all		2.50 2.50
<								>
Selection cri	iteria:		Select	Clear				
	Create Map Cop	у Мар	Delete Ma	р То Тор	To Bottom	Close	He	elp

You can now close the Mapping Style dialog. The plot should look as follows.



Step 10: Final Polishing

As a finishing touch, we can add a legend to the plot. Choose $Plot \rightarrow Line Legend$ from the Tecplot 360 menu to open the Legend dialog, then toggle on the Show Line Legend checkbox.

Legend ×
✓ Show Line Legend✓ Show Mapping Names
Text
Font: Helvetica V B I
Size: 3 Frame% ~ Black
Position
X (%): 95 Anchor
Y (%): 80
Line spacing: 1.2
Legend box
🔿 No box 🔘 Outline 🔵 Fill
Line thickness (%):
Box color: Black
Fill color: White
Margin: 10
Close Help

Let's also choose $View \rightarrow Nice$ Fit to Full Size to make the plot look a little less crowded against the axes.

The final plot (including both frames) is shown below.



Naturally, a Tecplot 360 layout package (.lpk) file containing a snapshot of the final result of this tutorial segment is in OneraM6wing/finallayouts/ExtenalFlowVideo3.lpk in the examples folder in your Tecplot 360 installation folder.

Next Steps

This concludes the External Flow tutorial for Tecplot 360. At this point, you may wish to dig in to the User's Manual or Help (**Help** \rightarrow **Tecplot 360 Help**) for more details on the product in general or on specific features you've used in this tutorial.

We regularly create videos to introduce users to features of the product, not just introductory topics for new users, but also for advanced users and to highlight new features in the latest release. You can find these on our Web site at www.tecplot.com/category/tecplot-360-videos/?product=360 or on our YouTube channel at www.youtube.com/user/tecplot360.

Understanding Volume Surfaces

This tutorial shows an example of how Tecplot 360 renders volume surfaces using Surfaces to Plot. For this tutorial, we will be using the duct flow dataset which can be found in the <u>Getting Started Bundle</u>.

This tutorial contains only one segment. The level of complexity is shown below.

Number and Level	Title and Description
1 - Beginner	Understanding Volume Surfaces - Load the Duct Flow dataset and understand how Tecplot views volume surface data.

A video version of this tutorial is available on the Web at www.tecplot.com/2016/03/04/understanding-volume-surfaces. The videos may have minor differences from the printed version of the tutorial in this manual, but they end up in the same place.

Understanding Volume Surfaces

Step 1: Launch Tecplot 360 and Load the Data Set

Start Tecplot 360 from the Start menu (Windows), by typing tec360 in a terminal window (Linux), or by double-clicking the application icon in the Applications folder (Mac). We will show the Windows version of Tecplot 360 in this document, but the product looks substantially the same on other platforms.

To begin loading the surface plot data, click **Load Data** at the top of the Welcome Screen. (You may also choose **Load Data** from the **File** drop-down menu in the menu bar, or click the folder icon, second from the left, in the toolbar. These other methods are convenient when the Welcome Screen isn't visible.)

The Load Data dialog appears.

Load Data				×
\leftarrow \rightarrow \checkmark \uparrow ${\frown}$ \rightarrow Desktop \rightarrow 3	60GettingStarted > SimpleData		∨ C Sear	ch SimpleData
Organize • New folder				≣ • 🔲 💡
Home	Name	Date modified	Туре	Size
Gallery	🧕 3ElementWing.lpk	8/12/2022 2:41 PM	Tecplot360EX ligh	twe 1,770 KB
	🔯 CircularContour.plt	8/12/2022 2:41 PM	Tecplot 360 EX plo	ot file 13 KB
Documente 🔶	🔯 CustomLabels.lay	8/12/2022 2:41 PM	Tecplot 360 EX lay	yout 7 KB
	🧕 CustomLabels.plt	8/12/2022 2:41 PM	Tecplot 360 EX plo	ot file 2 KB
	🧕 DownDraft.plt	8/12/2022 2:41 PM	Tecplot 360 EX plo	ot file 38 KB
Downloads	🧕 DuctFlow.lay	8/12/2022 2:41 PM	Tecplot 360 EX lay	yout 25 KB
🛄 Desktop 🛛 🖈	0 DuctFlow.plt	8/12/2022 2:41 PM	Tecplot 360 EX plo	ot file 484 KB
	🧕 Eddy.plt	8/12/2022 2:41 PM	Tecplot 360 EX plo	ot file 1 KB
> 📮 This PC	🧕 F18.lay	8/12/2022 2:41 PM	Tecplot 360 EX lay	/out 55 KB
> 🧯 Network	🧕 F18.plt	8/12/2022 2:41 PM	Tecplot 360 EX plo	ot file 229 KB
> 👌 Linux	🧕 HeatExchanger.plt	8/12/2022 2:41 PM	Tecplot 360 EX plo	ot file 45 KB
	🧕 IndependentDependent.lpk	8/12/2022 2:41 PM	Tecplot360EX ligh	twe 4 KB
🗌 Data overri	de			
File name: DuctFlow.	bit		~ All S	upported Files (*.*case;*.*در ``
			Help C	Open 🔻 Cancel

Navigate to the examples/SimpleData directory located in the installation folder of Tecplot and select the DuctFlow.plt file. After opening this file, you'll see a 3D Cartesian plot like the one below.



Step 2: Surfaces to Plot and the Zone Style dialog

Currently we are looking at an empty box surrounded by orange dashed lines. These lines represent the boundary of the volume zones with no style applied. To understand what we are looking at, let's look at the Surface tab of the **Zone Style** dialog:

Plot	8	×	Г
3D Cartesian		\sim	
Snap to orientation view			
Show zone layers			
Mesh			
Contour	2	P.	
🗹 Shade			
Vector		P	
Edge	Ľ	Č,	
Scatter			
Zone Style			
کما Launch the zone style	: di	alog	3

- Open the **Zone Style** dialog from the Plot sidebar.
- Select the **Surfaces** tab located at the top and notice the Surfaces to Plot heading.

7	7	Group	Channel	Curfaces				Lindeto		
Zone Number	Zone Name	Group Number	Zone	to Plot	Range	Range	Range			
	Tetrahedral	1	\checkmark	None	1,1,1	1,1,1	1,1,1			
								1		

For performance reasons, the default option for **Surfaces to Plot** is set to "none" on volume zones. When you have a volume zone where you'd like to see the surface data then change this option. Note that if surface zones are in the dataset, they would be shown in the Zone Style dialog as "N/A".

Step 3: Contours and Surface Data

Now let's turn on Contour from the Plot sidebar. To show contours, however, Tecplot 360 must represent this on some sort of surface. Since this data only contains volume zones, a Question dialog appears asking to display the surfaces of volume zones. Alternatively, enabling the mesh, contour, shade, vector or scatter layer option on the plot sidebar will ask you if you want surfaces turned on for

active zones.



Turning on the surfaces of volume zones can be a resource-heavy operation as it requires loading the entire volume zone and calculating which cells represent the outer surface.

Clicking **Yes** on the Question dialog allows Tecplot 360 to change the Surfaces to Plot option in the Zone Style dialog. The volume surfaces will now be contoured by the first variable that is not an axis variable.

Step 4: Changed Surfaces to Plot in the Zone Style Dialog

If the Zone Style dialog was closed previously, open it again by clicking the Zone Style dialog in the Plot sidebar.

1esh (Contour Vecto	or Scatt	ter S	Shade Edge P	Points Su	irfaces	Volume	Effects	
Zone Number	Zone Name	Group Number	Show Zone	Surfaces to Plot	I-Plane Range	J-Plane Range	K-Plane Range		
	Tetrahedral	1	\checkmark	Boundary cell face	es 1.1.1	1.1.1	1.1.1		
				,		.,.,.			
						.,.,.			

Navigating to the Surfaces tab, you'll see that the Surfaces to Plot heading has changed to **Boundary cell faces** which allows us to see the contour of surfaces of exposed boundary cells.

The resulting plot should look similar to what is shown below. Since we did not change the Contour variable, the plot is displaying the U Vector variable.



Next Steps

This concludes the Volume Surfaces tutorial for Tecplot 360. At this point, you may wish to dig in to the User's Manual or Help (**Help>Tecplot 360 Help**) for more details on the product in general or on specific features you've used in this tutorial.

We regularly create videos to introduce users to features of the product, not just introductory topics for new users, but also for advanced users and to highlight new features in the latest release. You can find these on our Web site at www.tecplot.com/category/tecplot-360-videos/?product=360 or on our YouTube channel at www.youtube.com/user/tecplot360.

Transient Data

This tutorial uses a vertical-axis wind turbine data set to explore the transient (time-based) capabilities of Tecplot 360. This data set comprises 127 time steps and 254 separate files in Fluent format. Each time step has two files, one for the geometry and one for the data. These files are sizable (over a gigabyte compressed), so they are not included with your Tecplot 360 installation. Instead, download the Getting Started Bundle and unzip the compressed data to a convenient location.

This tutorial is divided into four segments. We have provided a layout file for the end of each segment, so you can check your work. The segments are:

Number and Level	Title and Description
1 - Beginner	Loading and Exploring Transient Data - Load the wind turbine simulation data set and see how it is organized inside Tecplot 360. Add streamtraces and create an on- screen animation of the data set.
2 - Intermediate	Extracting Data - Use Tecplot 360 tools and techniques to reduce the amount of data that needs to be analyzed, making it easier to understand.
3 - Expert	Frequency Analysis Using Fourier Transform - Perform a Fourier transform to extract frequency domain information from the simulation and analyze the sources of pressure variations.
4 - Expert	Calculations and Contour Cutoff - Calculate a new variable by using the Tecplot 360 CFDA Analyzer. Use the Contour Color Cutoff property to isolate a region of interest in the plot.

A video version of this tutorial is available on the Web at www.tecplot.com/2016/03/25/transient-seriesvideo-1-introduction-transient-data. The videos may have minor differences from the printed version of the tutorial in this manual, but they end up in the same places.

Loading and Exploring Transient Data

Step 1: Launch Tecplot 360 and Load the Data Set

Start Tecplot 360 from the Start menu (Windows), by typing tec360 in a terminal window (Linux), or by double-clicking the application icon in the Applications folder (Mac). We will show the Windows version of Tecplot 360 in this document, but the product looks substantially the same on other platforms.

To begin loading the wind turbine data, click **Load Data** at the top of the Welcome Screen. (You may also choose **Load Data** from the **File** drop-down menu in the menu bar, or click the folder icon, second from the left, in the toolbar. These other methods are convenient when the Welcome Screen isn't visible.) The Load Data dialog appears.

🖲 Load Data					×
\leftarrow \rightarrow \checkmark \uparrow \blacksquare \rightarrow Desktop	> 360GettingStarted > WindTurbineBlades		~ C	Search WindTurbineBlades	م
Organize • New folder				≣ · □	?
A Home	Name	Date modified	Туре	Size	
Callen/	vawtRun-3150.cas	8/12/2022 2:41 PM	CAS File	2,866 KB	- 1
Callery	vawtRun-3150.dat	8/12/2022 2:41 PM	DAT	11,564 KB	
	vawtRun-3155.cas	8/12/2022 2:41 PM	CAS File	2,866 KB	
Documents		8/12/2022 2:41 PM	DAT	11,588 KB	
Pictures	* 🗋 vawtRun-3160.cas	8/12/2022 2:41 PM	CAS File	2,866 KB	
Downloads	৵ vawtRun-3160.dat	8/12/2022 2:41 PM	DAT	11,561 KB	
🛅 Desktop	★ vawtRun-3165.cas	8/12/2022 2:41 PM	CAS File	2,866 KB	
	vawtRun-3165.dat	8/12/2022 2:41 PM	DAT	11,580 KB	
> 📮 This PC	vawtRun-3170.cas	8/12/2022 2:41 PM	CAS File	2,866 KB	
> 🎦 Network	vawtRun-3170.dat	8/12/2022 2:41 PM	DAT	11,599 KB	
A linux	vawtRun-3175.cas	8/12/2022 2:41 PM	CAS File	2,866 KB	
	vawtRun-3175.dat	8/12/2022 2:41 PM	DAT	11,571 КВ	
Data		0/40/0000 0 /4 854	CAC ET	2.000 //P	
Data	overnde				
File name: "vaw	rtRun-3165.cas" "vawtRun-3165.dat" "vawtRun-3170.	cas" "vawtRun-3170.dat" "vawtRun-317	5.cas" "vaw ~	All Supported Files (*.*case;*.	*cç ∽
			Help	Open 🔽 Cance	2

Navigate to the windturbineblades folder where you extracted the data files. Select all the files in this directory, for example by clicking the first one and then clicking the last one while holding the Shift key, then click **Open**. (If you can't see the files, choose **All Supported Files** in the menu at the bottom of the dialog.)

Opening all these data files will take a moment; when Tecplot 360 is finished, you'll see a 2D Cartesian plot like the one below.



Step 2: Get A Good Look

What exactly are we looking at here? This is the full 2D fluid domain, representing the wind turbine's blades and the air surrounding them. Let's get a better view:

Plot	8×
2D Cartesian	\sim
Show zone layers	
Mesh Show mesh layer	2
Shade	-0
Scatter	
Zone Style	

• Display the mesh by turning on the Mesh checkbox in the Plot sidebar. The mesh is a collection of

interlinked triangles; these are the individual cells that were used in the simulation.

• Click the Zoom tool on the toolbar, then use the mouse to draw a rectangle around the center of the plot. Repeat this two or three times until you see a plot like the one below.



i

You may also zoom by placing the mouse pointer in the frame to be zoomed, holding the mouse's middle button or scroll wheel, and moving the mouse up and down.

So we can see this is a cross-section of a three-bladed vertical wind turbine. The white areas are the blades, as you can intuit from their airfoil shape. You can even infer the direction of rotation: counterclockwise.

Step 3: Dive into Data Set Info



Let's take a closer look at the data we've loaded. Choose **Data Set Info** from the Tecplot 360 **Data** menu to open the Data Set Information dialog, shown here.

From here, we can see that there are more than a thousand zones in this data set. Each zone represents a region of the simulation at a particular point in time.

The number of zones may seem like a lot if you're not used to working with transient data, but if you do the math, this is only nine zones per time step. And indeed, if you look at the zone list here, you can see that the zone names repeat every nine entries: zone 1 and zone 10 are both named dom-1, zone 2 and zone 11 are both named dom-5, and so on.

So we have:

- 1143 total zones
- 127 time steps
- 9 zones per time step

If you click through the zones while keeping an eye on the Time Strand field in the panel below the zone list, you will discover that the zones that have the same names also have the same time strand. Time strands are how Tecplot 360 links the zones representing the same region throughout time.

You can also see the solution time of each zone above the time strand as you click on in the zone list. As you would expect, zones 1-9 have the same solution time, as do zone 10-18, and so forth. Thus, each set of nine zones represents the same point in time.

Step 4: Time Strands in the Zone Style Dialog

Let's close the Data Set Information dialog and open the Zone Style dialog (below) by clicking the Zone Style button in the Plot sidebar.

Zone Number	Zone Name	Group	Show Zone	Show Mesh	Mesh Type	Mesh Color	Line Pattern	Pattern	Line Thickness	
1*	dom-1	1			Overlay		Solid	2.00%	0.10%	
2*	dom-5	1	\checkmark	\checkmark	Overlay		Solid	2.00%	0.10%	
3*	blades-4	1	\checkmark	\checkmark	Overlay		Solid	2.00%	0.10%	
4*	blades-7	1	\checkmark	\checkmark	Overlay		Solid	2.00%	0.10%	
5*	blades-10	1	\checkmark	\checkmark	Overlay		Solid	2.00%	0.10%	
6*	outlet-17	1		~	Overlay		Solid	2.00%	0.10%	
7*	inlet-16	1		~	Overlay		Solid	2.00%	0.10%	
8*	int_interface-13	1	\checkmark	\checkmark	Overlay		Solid	2.00%	0.10%	
9*	ext_interface-18	1	\checkmark	\checkmark	Overlay		Solid	2.00%	0.10%	

Here you'll see nine entries: one for each time strand. (The asterisk next to each zone number indicates that it's not a single zone, but a group of zones having the same time strand.) In this way, the Zone Style dialog lets you make changes to the "same zone" throughout time all at once, as you'll usually want to do, instead of having to make those changes separately at each time step. So although there are technically hundreds of zones in the data, Tecplot 360 lets you style them as though there were only nine.

We're done with the Zone Style dialog, so close it.

Step 5: Visualizing a Contour Plot

Right-click the plot to display the context toolbar and menu, as shown here. These allow you to make changes to the selected zone.

From left to right, the icons are Mesh, Contour, Vector, Shade, and Edge. Click the first icon to turn off the Mesh layer, then the second icon to turn on the Contour layer. (You can do this in the Plot sidebar if you prefer.



Now your screen should be showing you something like the plot below.



Step 6: Visualize the Flow With Streamtraces

Tecplot 360 can help visualize the flow through the wind turbine by adding streamtraces. In the Plot sidebar, turn on the Streamtraces checkbox.



The Select Variables dialog appears to choose the variables that represent velocity through the fluid; the defaults that Tecplot 360 has chosen are correct, so just click OK.

🧕 Select Variables	×
Choose the vector variables.	
U 4: X Velocity	\sim
V 7: Y Velocity	\sim
OK Cancel Help	

After you've accepted the vector variables, choose the streamtrace tool in the main toolbar, or in the Plot sidebar next to the Streamtraces checkbox.

You are now ready to add streamtraces to your plot, a process called *seeding*. It is convenient to seed a number of equally-spaced points along a line, which is referred to as a *rake*.

To do this, click in the plot near the top blade of the turbine, then hold the mouse button while dragging down between the two lower blades. Tecplot 360 seeds ten streamtraces at equally-spaced points along the line you placed Tecplot 360 calculates the path of a massless particle forward and backward from each seeded point and draws the path on the plot.



You may seed additional streamtraces by placing additional rakes in the same way, or seed individual points by clicking. A Tecplot 360 layout (.lay) file containing a snapshot of the final result of this tutorial segment is in WindTurbineBlades/FinalLayouts/transient_1.lay in the Getting Started bundle.

Step 7: Animate Your Plot

Click the **Play** button in the Plot sidebar to see how pressure and fluid flow change as the wind turbine blades rotate. (Just as we deduced earlier from the blade shape, the turbine turns counterclockwise.)



Scenic Detour: Animation Speed

🧕 Time Animatio	n Details		×
Starting time:	3.15	🖨 Mi	n: 3.15
Ending time:	3.78	🖨 Ma	ax: 3.78
Time step skip:	1	÷ Nu	imber of steps: 127
Animation direction:	Forward	\sim	
Current step:	1		Go to Step
Current value:	3.15		
Limit animation s	peed to 12		frames/second
For strands with no	zones at the curren	t solutior	n time:
Show zones a	at or before solution	n time	
Show zones of the second se	only at solution time		
		Close	Help

The animation speed can be limited in the Time Animation Details dialog, which can be reached by clicking the *intervention* button next to the animation controls in the Plot sidebar.

The maximum animation speed is limited by the speed at which Tecplot 360 can draw an individual frame, which depends a lot on how fast your computer is and how much of the data has already been loaded into memory. However, you can keep animations from going too fast by clicking the Limit Animation Speed checkbox and entering the desired number of frames per second.

Extracting Data

In this exercise, we will cover extraction techniques that helps you visualize details about your data at a particular location, allowing you to focus on exactly what you want to learn. First, we'll extract data at a single time step to a line plot. Then we'll extract data at a single point over time into a time series plot.

But before we get started, turn off the streamlines you added at the end of the last exercise by turning off the Streamtraces checkbox in the Plot sidebar and return to the first time step of the data. Your plot should look something like this again.



Save a layout at this point to make it easy to return this exact place for the next exercise.

Step 1: Create a Vertical Line

To draw the line along which we will extract the pressure data, first click the polyline geometry tool in the Tecplot 360 toolbar. A geometry is a shape that appears on your plot. This particular geometry tool can be used to create both lines and polygons, hence the name *polyline*.



The line we want to draw is in roughly the same place as the rake we placed for creating streamtraces in the previous lesson. But the polyline tool, since it can be used to create both simple lines and more complex shapes, works differently from that tool.

To place the line:

• Click and release the mouse button with the pointer positioned just under the topmost blade of the

turbine. With the polyline tool, you don't drag the mouse with the button held down.

- Press the V key on the keyboard to constrain the line to vertical. Now you don't need to worry about making the line perfectly vertical; Tecplot 360 does that for you.
- Move the mouse down to the bottom of the green area. You will notice that a vertical black line follows the pointer. No matter how you move the mouse horizontally, the line stays put.
- *Double-click* the mouse to end the creation of the line. (If you had single-clicked, you would then be able to move the mouse and create an additional line segment attached to the first.)

The line should look roughly like this on the plot. We have made our line thicker so you can see it better. You don't have to do the same, but if you have a moment, see if you can figure out how.



Step 2: Extract the Line Over Time

Let's see how pressure changes along our line *over time* by creating an animated plot of the Y coordinate vs. pressure. To do that, choose **Extract Polyline Over Time** from the **Extract** submenu of the **Data** menu.



If we were interested only in a single time-step, we could right-click the line and choose **Extract Points** from the context menu.

You will be asked for the number of points to be extracted along that line. The default value, which should be 100, is fine.

Prompt for text strir	ng X						
Enter a number between 2 and 1000000 for the number of points to extract.							
ОК	Cancel						

Tecplot 360 will take just a moment to go through all the time steps and extract the data along this line. You can open the Data Set Information dialog and see a new zone for each time step, named "Extracted Points," that were created beginning with zone number 1144 and ending with zone number 1270.



Each new zone is assigned the correct solution time and given a time strand to connect it to the others, just as we saw when we inspected the data in our file earlier.

That's 127 new zones in all, which matches the number of time steps in the file. All of these new zones are assigned to time strand 10, which makes sense because, previously, there were nine variables per time step and thus nine time strands. You can also see that each of the new zones has the same variable names as the previously-existing zones.

So, as we expect, we have extracted the data along the line over time.

Step 3: Create A Line Plot of Pressure

Let's see what the pressure looks like along our extraction line by creating a line plot. We can display this line plot side-by-side with the contour plot and even animate them together.

First, choose the New Frame tool in the Tecplot 360 toolbar. Then hold the mouse button down and drag out a rectangle on the plot. It doesn't matter where; we will resize and reposition it in the next step.

Ab	\checkmark		\bigcirc	\bigcirc	6	
						Create a new frame

Your new frame might look something like this.



Let's get that new frame into position. Choose **Tile Frames** from the **Frame** menu, then click the bottom-right button.



This will stretch both frames to the width of the workspace and position them one above the other, as seen here.



Let's go ahead and zoom out that top frame a bit and reposition it so that we can see the whole wind turbine. To do this, we will use the Zoom and Translate tools in the toolbar.



- Choose the Translate tool and drag the plot in the top frame so that it is centered
- Choose the Zoom tool, then hold down the Control key while clicking the plot to zoom out one step.

The top frame should look more like this now.



Now we'll turn our attention to the bottom frame, where we'll create a line plot of pressure along the line we placed on the plot earlier.

- Choose the selector (arrow) tool from the Tecplot 360 toolbar and click the bottom frame to select it.
- Change the plot mode to 2D Cartesian using the menu at the top of the Plot sidebar.

P	ot		₽×
	2D Cartesian		-
	3D Cartesian		
	2D Cartesian		
	XY Line Polar Line Sketch	2D Cartesian	
	🗹 Shade		
	Vector		<u>.</u>
	🗌 Edge		
	Scatter		
		Zone Style	

You might think to use an XY Line plot for this kind of data, and normally you could, but XY Line plots don't have the transient capabilities of 2D Cartesian plots.

In the bottom frame, you'll see a gray circle very similar to the plot that appeared when we first

opened this data set.

• Use the Zone Style dialog to turn off all zones except for the Extracted Points zone. Click the **Zone Style** button in the Plot sidebar, then right-click in the Show Zone column of the last zone listed and choose **Show Selected Only**.

2* dom- 3* blade 4* blade	-5 es-4	1				Pattern	Length	Thickness	
3* blade 4* blade	es-4		\checkmark		Overlay	Solid	2.00%	0.10%	
4* blade		1			Overlay	Solid	2.00%	0.10%	
	es-7	1	\checkmark		Overlay	Solid	2.00%	0.10%	
* blade	es-10	1			Overlay	Solid	2.00%	0.10%	
i* outlet	et-17	1			Overlay	Solid	2.00%	0.10%	
/* inlet-1	-16	1		~	Overlay	Solid	2.00%	0.10%	
8* int_int	nterface-13	1	\checkmark	~	Overlay	Solid	2.00%	0.10%	
* ext_in	nterface-18	1	\checkmark	~	Overlay	Solid	2.00%	0.10%	
144* Extrac	cted Points	1	2	\sim	Overlay	Solid	2.00%	0.10%	

Note that this zone has an asterisk by its number, indicating that it has a time strand, as we have previously seen in the Data Set Information dialog. Thus, when we make only this zone visible, it is the only zone visible at all time steps.

Close the Zone Style dialog after doing this.

• To assign variables to the axes, choose **Assign XY** from the **Plot** menu, then selecting the Y and Pressure variables for the X and Y axes as shown here.

۱	Select Variables	×
Ch	oose the variables to assign to the axes.	
x	2: Y	\sim
Y	3: Pressure	\sim
	OK Cancel Help	

Your line plot still looks very blank at the moment.

• Activate the Mesh checkbox in the Plot sidebar to see the line.

It's all scrunched up along the left side of the plot because the axes aren't set up properly. We'll take care of that next.

• Choose **Axis** from the **Plot** menu to open the Axis Details dialog. Change the Dependency radio button for the X axis to Independent, as shown here.

Axis Details	×
Show X-Axis X Y	
Range Grid Ticks Labels Title Line Area	
Min: -0.973917727947 Reset Range	
Max: 88.3510997315	
✓ Preserve axis length when changing range Use log scale	
Reverse axis direction	
Dependency	
O Dependent X to Y ratio: 1	
● Independent	
Automatically adjust axis ranges to nice values	
Close Help	

We'll let Tecplot 360 choose the range for the axis. Leaving the Axis Details dialog open, choose **Data Fit** from the **View** menu in the main Tecplot 360 window. (You can also just press Control-F.)

• The Data Fit operation also set a range for the Y axis, but it did so only considering the Y-axis range of the first time step. The range of Pressure at the first time step is not representative of all time steps, so we will manually specify an appropriate Y-axis range.

Change to the Y axis using the button at the top of the Axis Details dialog, then enter -100 and -30 as the minimum and maximum.

🔯 Axis Details	×
Show Y-Axis X Y	
Range Grid Ticks Labels Title Line Area	
Min: -100 Reset Range	
Preserve axis length when changing range Use log scale Reverse axis direction	
Dependency	
O Dependent X to Y ratio: 1	
Independent	
Automatically adjust axis ranges to nice values	
	Close Help

• Close the Axis Details dialog.

The plot in the bottom frame should now look a lot like this.



Step 4: Link the Time Steps and Animate the Plot

If you try animating the plot by pressing the **Play** button on the Plot sidebar, you will notice that only the active frame animates. To get both frames animating together, we will link them so they always display the same time step.
Solution time: 3.15	Ĩ

To do this, choose **Frame Linking** from the **Frame** menu to open the Set Links for Active Frame dialog.

Set Links for Active Frame	×	
Between Frames Within Frame		
Frame size and position	3D plot view	
Solution time	Slice positions	
X axis range	Iso-surface locations	
Y axis range	Contour levels	
XY axis position	Value blanking constraints	
Polar plot view		
Frame is member of group: 1 \sim]	
Apply Settings to All Frames of this group Apply Settings to Selected Frames of this group		
	Close Help	

On the Between Frames page, mark the Solution Time checkbox, then click the **Apply Settings to All Frames of this Group** button. Close the dialog and try animating again!

Scenic Detour: Add Solution Time Caption

It would be nice to be able to see what the solution time is right on the plot, without having to look at the Plot sidebar. This might be of particular utility if you were exporting a video. To do this:

• Click the Add Text tool in the Tecplot 360 toolbar.



• Click on the plot where you want the text to appear.

This displays the Text Details dialog, where you can enter text and choose its formatting.

• In the Text Details dialog, enter Solution Time: &(SOLUTIONTIME)

The text starting with an ampersand is a dynamic text placeholder. Tecplot 360 replaces it with the current solution time when displaying it in the plot.

I Size: 14	■ Black
Frame \vee Angle (deg): 0	~ Anchor
	Text Box
	Frame V Angle (deg): 0

Solution Time: 3.15

• Click Accept to close the Text Details dialog.

Animate again and watch the solution time change!

Step 5: Time Series Plot

We have seen how to create a line plot showing the pressure along a line at each time step. Using animation, we can show how that changes over time. Now we'll reduce the dimensionality even further by creating a single line plot showing how pressure varies at a single point.

To create this time series plot, choose **Probe to Create Time Series Plot** from the **Tools** menu, then click as close to 0, 0 as you can get on the contour plot (top frame). Hint: the X and Y coordinates of the mouse cursor are displayed in the bottom right corner of the Tecplot 360 workspace.

After you click, Tecplot 360 spends a moment gathering the pressure value for that point for all time steps, then displays the time series plot in a new frame overlapping the contour plot.



When you animate, a moving vertical line appears in this plot to indicate the current time step.



All three plots (the contour plot, the pressure line plot, and the time series plot) animate at the same time.

Try using the Tile Frames tool we used earlier to find a good arrangement for your three frames. You'll probably need to re-position and zoom out on the contour plot again.



Frequency Analysis Using Fourier Transform

As we can see from the time series plot we just created, pressure varies over time and appears to do so as a wave. In this exercise, we'll analyze the fundamental frequency of the pressure waves in the data using a Fourier transform, and see how this varies when changing the probe location.

Before starting, let's delete the middle frame (which shows the pressure along the line). First, choose the selector tool in the toolbar, if it isn't already selected.



Then click the edge of the middle frame to select it, then press the Delete key on your keyboard.

Finally, re-tile the frames by choosing **Tile Frames** from the **Frame** menu. You may want to zoom in on the contour plot and/or re-center it. The plot should look something like this.

Solution Time: 3.15



Step 1: Fourier Transform

Select the bottom frame (containing the time series plot) with the selector (arrow) tool, then choose **Fourier Transform** from the **Data** menu. The Discrete Fourier Transform dialog, shown here, appears.

Discrete Fourier Transform	×
Independent variable: 1: Solution Time \checkmark	
Dependent variables	
2: X	^
3: Y	
4: Pressure	
5: X Velocity	×
Source zones 1: Time Series Plot Zone	
Window function: Rectangular $ \lor $ Plot placement: In corner of source frame	e 🖂
Include conjugates Replace zones	
Obey source zone blanking Replace variables	
Close Transform Help	

In this dialog, choose Pressure as the dependent variable and make sure the Time Series Plot Zone is selected as the source zone. Using the **Plot Placement** drop-down menu, choose to tile the new frame with existing frames. Leave everything else the same. Click **Transform** to perform the Fourier transform.

A new frame appears, showing Frequency (Solution Time)⁻¹ on the X axis and Amplitude (Pressure) on the Y axis.



Looking at this plot, we can easily see that there is a low-frequency spike at around 5 Hz and a higher-frequency spike at about 77 Hz. This matches what we see in the time series plot: a lower-frequency (high-period) wave overlaid by a higher-frequency (low-period) wave. The combination makes the time series plot look a little jagged.

From watching the animation, we can attribute the lower-frequency pressure changes to the movement of the blades. However, we don't have an obvious explanation for the higher-frequency changes.

Before going further, let's have a quick look at what the Fourier Transform actually did to our data set by popping into Data Set Information (choose **Data Set Info** from the **Data** menu). Make sure that the Fourier plot frame is selected.

Data Set Information	×
Zone/Var Info Data Set Aux Data	
Zone(s)	Variable(s)
1: Time Series Plot Zone 2: Fourier Transform (Time Series Plot	31: Wall shear-2 32: Wall shear-3 33: Boundary heat flux 34: Boundary rad heat flux 35: Wall yplus 36: Wall yplus utau 37: Frequency (Solution Time) <sup< td=""> 38: Amplitude (Pressure) 39: Phase (radians)</sup<>
< >	< >
Zone Type: Ordered	Var Type: Double
IMax: 64	Var Location: Nodal
JMax: 1	Var Status: Heap
KMax: 1	Var Range Selected Zone:
	Min: 0
	Max: 100
	Var Range Referenced Zones:
	Min: 0
	Max: 100
	Close Help

As you can see, Tecplot 360 has created a new Fourier Transform zone based on the Time Series Plot and added three new variables starting with variable 37. (The other zone and the first 36 variables were created when we generated the time series plot.)

The new variables are the frequency, amplitude, and phase output from the Fourier transform. We only selected Pressure as our dependent variable in the transform, but if we had selected more, there would be three new variables for each one we selected.

Step 2: Re-Analyzing at a New Position

Let's see if we can find out what's causing the higher-frequency changes. To do this, we'll re-probe well outside the turbine, near the left inlet. Then we'll create a new time series plot and re-do the Fourier transform.

Zoom out the contour plot so that you see most or all of the fluid domain surrounding the turbine. The blades will be very small in the center of the plot. At right you'll see we've zoomed all the way out. The mouse pointer is placed roughly where we'll be probing.



Solution Time: 3.15

Now choose **Probe to Create Time Series Plot** from the **Tools** menu again, and click close to the indicated position on the plot. A new time series plot is created.



Right away, you can see that this plot has less obvious order in it.

Since our Fourier transform plot was calculated from the data in the old time series plot, it disappears. But we can re-create it easily. Simply right-click the line in the time series plot to display the context menu and toolbar. Then choose **Fourier Transform** from the menu.



A new Fourier transform plot is created using the options previously set in the Discrete Fourier Transform dialog. It looks something like the plot below.



Here, the main frequency is the one we previously saw at around 77 Hz. The lower frequency around 5 Hz is no longer there, indicating that we are probing far enough away to avoid the effects of the turbine blades. We can conclude that the 77 Hz frequency is an input boundary condition.

A Tecplot 360 layout (.lay) file containing a snapshot of the final result of this tutorial segment is in WindTurbineBlades/FinalLayouts/transient_2.lay in the Getting Started bundle.

Your Turn

Try analyzing other variables, such as Turbulent Viscosity. By probing near where vortices are shed, you can see how changes in turbulent viscosity are solely due to the motion of the turbine blades. We will delve into this in more detail in the next exercise.

Calculations and Contour Cutoff

Tecplot 360's CFD analysis tools have many tricks up their sleeve. One of the most important is the ability to calculate commonly-needed variables that are not actually present in the data set. Why have your solver calculate, say, vorticity magnitude (and expend disk space storing these values) when it can be easily derived from values already in the data set?

Furthermore, Tecplot 360 can calculate these values on demand, so no time is spent on calculating them unless and until we choose to visualize them. This means that until we use vorticity magnitude in a plot, it does not need to be calculated at all. And even when the variable is being used in a plot, it will not be calculated for a given time step until we actually view that time step. (Once calculated, however, the value remains in memory, so returning to a time step usually won't require it to be calculated a second time.) So, if we will be looking at only ten or fifteen time steps in our data set, using calculate-on-demand will save us around 90% of the time we might have spent calculating vorticity magnitude.

In this exercise, we'll calculate Vorticity Magnitude using the Tecplot 360 CFD analysis tools. Then we'll adjust the contour color cutoff to isolate interesting regions of the plot.

We'll only need the contour plot for this exercise. You can delete the frames containing the time series

and Fourier plots you created in the previous exercise and re-tile so that the contour plot frame fills the workspace. Re-zoom and re-center as needed.

Step 1: Set Field Variables

Before we can calculate the new variable, we must make sure that the baseline variables are assigned in the Field Variables dialog. To open this dialog, choose **Field Variables** from the **Analyze** menu.

🧕 Field	Variables				×
Convec	tive Variables				
(Velocity		U:	X Velocity	Select
(O Momentum		V:	Y Velocity	
			w:		
State V	/ariables				
For	Pressure	\sim	Use	Pressure	Select
For	Density	\sim	Use	Density	
				OK	Help

Tecplot 360 has chosen defaults that match what it thinks you want to do:

- X Velocity and Y Velocity have been chosen as the convective variables U and V.
- Pressure and Density have been chosen as the corresponding state variables.

For this data set, these defaults are correct, so you can just click **OK**. For other data sets, you may need to choose different variables.

Step 2: Calculate Vorticity Magnitude

To calculate vorticity magnitude, just choose **Calculate Variables** from the **Analyze** menu. This displays the Calculate dialog.

🥘 Calcu	late		\times
Name	(3D) I Aspect Ratio		Select
Normaliz	e with Respect To	No Normalization	~
New Var	Location	Node	~
🗸 Calc	ulate on Demand		
		Calculate	
		Close	Help

Click the **Select** button to choose the variable to be calculated. This displays the Select Function dialog.

Select Function	\times
X Momentum	^
Y Momentum	
Z Momentum	
Entropy	
Entropy Measure S1	
X Vorticity	
Y Vorticity	
Z Vorticity	
Vorticity Magnitude	
Q Criterion	
Swirl	
Velocity Cross Vorticity Magnitude	
Helicity	~
OK Cancel Help	

As you can see, there are literally dozens of variables that Tecplot 360 can calculate for you. Choose Vorticity Magnitude in the list then click **OK** to return to the Calculate dialog. The default options in the Calculate dialog are acceptable (note that the Calculate on Demand checkbox is checked!) so go ahead and click **Calculate**.

Tecplot 360 sets up calculate-on-demand for Vorticity Magnitude and adds the variable to the data set, and tells us that's what it has done via an alert.

We can now close the Calculate dialog.

Step 3: Show Vorticity Magnitude on Contour Plot

To actually show our newly-calculated variable on our plot, open the Contour Details dialog by clicking the 📝 button next to the Contours checkbox in the Plot sidebar.

Plot	8 ×
2D Cartesian	\sim
Show zone layers	
Mesh	
🗹 Contour	2
🗹 Shade	45
Vector	3 ⁴⁰
Edge	Ē
Scatter	
Zone Style	

The Contour Details dialog appears.

🙋 Contour & Multi-Coloring Details 🛛 🕹 🗙				
36: Vorticity Magnitude	1 2 3 4 5 6 7 8			
Levels and Color Bands	Lines Labels Legend			
Contour levels				
1:0	▲ Set Levels			
2:2.5	Add Level 0			
4 : 7.5	Remove Selected			
5:10				
7:15	Contour variable range			
8:17.5	Min: 0 Max: 3.09289€+07			
Color map options				
$\diamond \diamond \diamond \diamond$	$\diamond \diamond \diamond \diamond \diamond \diamond$			
Sequ	ential - Viridis 🗸 🙀			
Color map distribution method	Color cutoff			
Banded O Continuou	S Cutoff above: 2.31967e+07			
Contour values at endpoir	ts Cutoff below: '.73222e+06			
Min: 0	Color map adjustments			
Max: 0.09289e+07	Reset 🔻 🗌 Reverse			
Approximate continuo	us Repeat cycles: 1			
	Close Help			

In the Contour Details dialog:

- Choose **Vorticity Magnitude** from the menu at the top of the dialog.
- Click the Set Levels button to display the Enter Contour Levels dialog, shown at right.

In the Enter Contour Levels dialog:

- Choose "Exact levels" for the level creation mode.
- Choose "Min, max and number of levels" for the range distribution.
- Enter 0, 50, and 21 as the minimum, maximum, and number of levels respectively.

Enter Contour Levels				
Exact levels Approximate levels for nice values				
Range Distribution Min, max, and number of levels Min, max, and delta Exponential distribution				
Minimum level:	0			
Maximum level:	50			
Number of levels:	21			
Delta:	2e+06			
		Reset Range 👻		
ок	Cancel	Help		

When the dialog is as shown, click **OK**.

You may now move the Contour Details dialog to the side to get a good look at the plot. However, leave it open; we will have need for it again soon.

The contour plot should now look something like this.



Step 4: Adjust Cutoff

This plot is good, but we can make it better by only coloring areas with vorticity above a critical value, improving the impact of locations of higher vorticity.

Back in the Contour Details dialog, set the contours to cut off below a value of 7. This field is in the lower right portion of the dialog.

Color map options	
$\diamond \diamond \diamond \diamond \diamond \diamond$	$\diamond \diamond \diamond \diamond \checkmark$
Sequential - Viridis	~ 🙀
Color map distribution method	Color cutoff
Banded O Continuous	Cutoff above: 2.31967e+07
Contour values at endpoints	✓ Cutoff below:
Min: 0	Color map adjustments
Max: 3.09289e+07 Reset 👻	Reverse
Approximate continuous	Repeat cycles: 1
	Close Help

This makes our vortices stand out by eliminating all contouring below 7. You might experiment with slightly lower and higher values to see more or less. Now we must remove the legend as it is overlapping a region of interest on our plot.

Step 5: Polishing the Plot

🔯 Contour & Multi-Coloring Details	×
36: Vorticity Magnitude v 1 2 3 4 5 6 7 8	
Levels and Color Bands Lines Labels Legend	
Show contour legend	
Alignment: Horizontal \checkmark Anchor	
X (%): 72.124 Y (%): 96.999 Legend Box	
Separate color bands	
Resize automatically Show color cutoff levels	
Level skip: 1 🔹 Line spacing: 1.2	
Label continuous colors position: Contour levels \checkmark Increment: 3.093e+06]
Text color: Black	
Number font: Helvetica \checkmark B <i>I</i>	
Size: 2.5 Frame% V Number Format	
Show header	
Header text: Use Variable Name 🗸	
Header font: Helvetica V B I	
Size: 2.5 Frame% ~	
Close Help	

One last trip to the Contour Details dialog. Switch to the dialog's Legend page, change the alignment to horizontal, and activate the Resize Automatically checkbox.

You can close the Contour Details dialog. Switch to the Selector arrow tool, select the legend by clicking on the box around it, and drag the legend to the top center of the plot.



Using the translation tool in the toolbar, move the wind turbine blades a little to the left, so we can see more of the vortices being shed.



You can also translate a plot by holding down the right mouse button and dragging. This works when any tool is active.

You can see our final vorticity plot below.



A Tecplot 360 layout (.lay) file containing a snapshot of the final result of this tutorial segment is in WindTurbineBlades/FinalLayouts/transient_3.lay in the Getting Started bundle.

Step 6: Animate It



Click the **Play** button in the Plot sidebar to begin animating the plot.

You'll notice that the first time through, the animation is a little slow. This is because Vorticity Magnitude is a calculate-on-demand variable, as we discussed earlier. It takes a moment to calculate this variable the first time each time step is plotted.

After the first time through, the animation will repeat. Since the Vorticity Magnitude variable has already been calculated for each time step, it does not need to be calculated again, and the animation

moves much more quickly and smoothly the second and subsequent times through.

Next Steps

This concludes the Transient tutorial for Tecplot 360. At this point, you may wish to dig in to the User's Manual or Help (**Help>Tecplot 360 Help**) for more details on the product in general or on specific features you've used in this tutorial.

We regularly create videos to introduce users to features of the product, not just introductory topics for new users, but also for advanced users and to highlight new features in the latest release. You can find these on our Web site at www.tecplot.com/category/tecplot-360-videos/?product=360 or on our YouTube channel at www.youtube.com/user/tecplot360.

Finite Element Analysis

This tutorial uses a transient FEA dataset of a connecting rod created with LS-DYNA. The connecting rod data contains 32 time steps with 32 different variables from the LS-DYNA program of a rod rotating about a point. The data may be downloaded from the Getting Started Bundle.

The Finite Element Analysis tutorial contains three different segments. The segments will explore multiple ways to visualize the maximum Von Mises Stress of the rotating rod. We have provided a layout file for the end of each segment, so you can check your work. The overall difficulty, description, and features used in each segment are shown below:

Number and Level	Title and Description	Features Used
1 - Intermediate	Calculating Von Mises Stress Along a Surface - Calculate the Von Mises stress of the connecting rod data set and contour to find areas of high stress.	 Animation Color Cutoff Contour FEA Post-Processing Value Blanking 3D Multi Frames
2 - Intermediate	Data Alter with If Statements - Use an if statement to calculate where a stress threshold is being crossed.	Data AlterZone Style
3 - Intermediate	Plot Maximum Stress Over Time - Use a macro to plot Maximum Stress over time and link solution times between frames.	Frame LinkingMacro CommandsXY Plotting

A video version of this tutorial is available on the web at: www.tecplot.com/category/tecplot-360videos/finite-element-analysis. The videos may have minor differences from the printed version of the tutorial in this manual, but they cover the same material. The third segment of this tutorial is not in the video series as it contains few visual steps and is better explained in writing.

Calculating Von Mises Stress Along a Surface

This section will calculate the Von Mises stress of the connecting rod data set and contour to find areas of high stress.

Step 1: Load the Data Set

To begin loading the connecting rod data, select **Load Data** at the top of the Welcome Screen. (You may also choose **Load Data** from the **File** drop-down menu in the menu bar, or click the folder icon, second from the left, in the toolbar. These other methods are convenient when the Welcome Screen isn't

visible.) The Load Data dialog appears.

🚺 Load Data					×
\leftarrow \rightarrow \checkmark \uparrow \blacksquare \rightarrow Desktop	> 360GettingStarted > ConnectingRod		~ C	Search ConnectingRod	Q
Organize - New folder				≣ - □	?
A Home	Name	Date modified	Туре	Size	
🔁 Gallery	FinalLayouts	4/16/2024 9:05 AM	File folder		
	D3PLOT	8/12/2022 2:41 PM	File	490 KB	
Documents	*				
🔀 Pictures	<i>\$</i> *				
Downloads	*				
🔚 Desktop	*				
> 📮 This PC					
> 🦆 Network					
> 👌 Linux					
Data	override				
File name: D3P	.OT		~	All Supported Files (*.*case;*.*	cc ~
			Help	Open 🔽 Cancel	

Navigate to the downloaded and extracted ConnectingRod folder, open the menu on the bottom right of the dialog, and select All Supported Files. Load the D3PLOT file.



Your loaded data only shows an orange dotted box, select **Contour** on the Plot sidebar, select **Yes** on the question dialog that appears, and your plot should look similar to the above picture. Refer to Understanding Volume Surfaces to learn about the Surfaces to Plot option.

Step 2: Slow Animation Down

Press the **play button**, **()**, on the plot sidebar and notice the machine parts in motion. If the animation is too fast, press the *(Press)* button above the animation slider and select the **checkbox** next to **Limit animation speed to**, limiting the animation to 12 frames per second.

🧕 Time Animatio	n Details	×
Starting time:	-210.96414	➡ Min: -210.964
Ending time:	114.13696	🖨 Max: 114.137
Time step skip:	1	Number of steps: 14
Animation direction:	Forward	\sim
Current step:	1	Go to Step
Current value:	-210.964	
Limit animation sp	peed to 12	frames/second
For strands with no a	zones at the current at or before solution	solution time: time
Show zones of the second se	only at solution time	
		Close Help

The resulting animation is now slower the next time the **play button** is pressed. Close the **Time Animation Details** window.

Step 3: Calculate Von Mises Stress

Open the **Data Set Information** window by selecting **Data>Data Set Info...** and notice how the data is configured. Each Zone pertains to a different time step and has the same variables. In the variables box, notice that the stress components for each direction are included in the dataset (variables 10-15) thus allowing the ability to calculate the Von Mises stress.

🧕 Data Set Infor	mation			\times
Zone/Var Info	Data Set	Aux Dat	a	
Zone(s)			Variable(s)	
1: Step 1 Incr 2: Step 2 Incr 3: Step 3 Incr 4: Step 4 Incr 5: Step 5 Incr 6: Step 6 Incr 7: Step 7 Incr 8: Step 8 Incr 9: Step 9 Incr 10: Step 10 Inc 11: Step 11 Inc 12: Step 12 Inc	0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0		6: W 7: X Acceleration 8: Y Acceleration 9: Z Acceleration 10: XX Stress 11: YY Stress 12: ZZ Stress 13: XY Stress 14: YZ Stress 15: ZX Stress 16: Equivalent, Plastic Strain 17: Nx	
Zone Type: F Pts: 4 Elem: 3 Solution Time: 0 Time Strand: 1	E - Brick 843 300		Var Type: Double Var Location: Cell Centered Var Status: Not Loaded Var Range Selected Zone: Min: N/A Max: N/A Var Range Active Zones: Min: N/A Max: N/A Load Variabl	es
			Close He	lp

Next, select **Tools>FEA Post-Processing** to open the **FEA Post-Processing Window**. Under the **Derive Variables** section derive **"Von Mises Stress/Strain"** from **"Stress"** (the Stress variables shown below). Click **Add to Data Set**. Now on the Data Set Information page, Von Mises Stress appears as variable 33. Close the FEA Post-Processing dialog and close the **Data Set Information** dialog.

FEA Post-Processing		\times
Deform Plots by Factor:	1	
Animate Deformation	On Screen V	
Number of Steps:	10	
Number of Cycles:	1	
	Animate	
Derive Variables Derive: Von Mises Stress/Strai Principal Stresses/Strai Vector Magnitude	n ^ ins v	
From: Stress		
	Add to Data Set	
	Close Help	

Step 4: Contour by Von Mises Stress

To begin contouring, select the we button next to the Contour checkbox on the plot sidebar to open the Contour & Multi-Coloring Details dialog. Currently, X Acceleration is the contour variable of the first contour group. Change the contour group variable to Von Mises Stress by selecting the drop-down box at the top of the dialog.

Tecplot 360 allows up to eight contour groups, each with their own contour variable and levels. This allows multiple contour surfaces to be plotted at the same time. In this tutorial, only one contour surface needs to be plotted.

🙋 Contour & Multi-Coloring Details	×
33: Von Mises Stress V 1	2 3 4 5 6 7 8 Labels Legend
Contour levels 1:0 2:47.3684 3:94.7368 4:142.105 5:189.474 6:236.842 7:284.211 8:331.579 2:270.047 Cont Min Color map options Color map options	Set Levels Add Level 0 Remove Selected cour variable range n: 0 Max: 866.29
Sequential - Virid	is v 🚳
Color map distribution method Banded Continuous Contour values at endpoints Min: 0 Max: 406.533 Reset	Color cutoff Cutoff above: 1003.27 Cutoff below: 800 Color map adjustments Repeat cycles: 1
	Close Help

Next, select the **Set Levels...** button and change the minimum level to be 0, maximum level to be 900, and the number of levels to be 21. The resulting **Enter Contour Levels** dialog should look like the picture below. Select **OK** and Close the **Contour & Multi-Coloring Details** dialog for now.

Enter Contour Levels ×				
Level Creation M	Level Creation Mode			
Exact levels				
Approximate	e levels for nice	values		
Range Distributio	Range Distribution			
Min, max, ar	nd delta			
C Exponential	Exponential distribution			
Minimum level:	0			
Maximum level:	900			
Number of levels: 21				
Delta:	47.3684			
		Reset Range 👻		
ОК	Cancel	Help		

Step 5: Isolate the Connecting Rod

Currently, the crankshaft and cylinder are in the way of viewing the connecting rod. Since the data set consists of one zone and many time steps, Value Blanking will allow the connecting rod to be seen more clearly. Select **Plot>Blanking>Value Blanking** from the toolbar to open the Value Blanking window. Under the Blank when section change the blanking constraints to be when **Von Mises Stress is equal to 0**. Select both the **Include Value Blanking checkbox** as well as the **Active checkbox** to enable the current blanking constraint. Now only the connecting rod is shown. Close the **Value Blanking** dialog.

🙋 Value Blanking 📃 💌
Include value blanking
Blank entire cells when:
any corner is blanked 🔻
Constraint 12345678
C Active
Blank when
[1: X ▼
is less than or equal to 🛛 👻
🔘 Variable 💿 Constant
0
Close Help

The resulting plot isolates the connecting rod because the Von Mises stress contour values are equal to zero on the crankshaft and cylinder. Though the parts still exist in the data set, they do not give helpful information. In the Value Blanking dialog, multiple constraints can be enabled at once to confine the plot view to very specific areas. For this situation, only one constraint was needed.

Step 6: Identify the Critical Failure Threshold

When clicking the play button again, the Von Mises Stress ranges between 0 and 900 at different steps in the cycle. The values of interest are when the Von Mises Stress exceeds 800. This is the critical failure threshold. To highlight the critical failure threshold, turn on **Color Cutoff** located in the Contour & Multi-Coloring Details dialog and cutoff below 800.

🗕 Contour & Multi-Coloring E	Details ×
33: Von Mises Stress	✓ 1 2 3 4 5 6 7 8
Levels and Color Bands	Lines Labels Legend
Contour levels	
1:0 2:45 3:90 4:135 5:180 6:225 7:270 8:315 0:260 Color map options	 Set Levels Add Level Add Level Remove Selected Contour variable range Min: 0 Max: 866.29
Sequent	tial - Viridis 🗸 🙀
Color map distribution method	Color cutoff
Banded O Continuous	Cutoff above: 1003.27
Contour values at endpoints Min: 0 Max: 406.533 F Approximate continuous	Cutoff below: 800 Color map adjustments Reset Repeat cycles: 1
	Close Help

Close the **Contour & Multi-Coloring Details** dialog.

Now when playing the animation, there are two steps that are above the critical failure threshold. Turning on **translucency** by the checkbox on the plot sidebar can give a better view of the part.



Step 7: 3D Multi Frames

Visualization can be further improved by selecting the 3D Multi Frames option. Select **Frame>3D Multi Frames** then select the **top left option** and three extra frames will appear. These frames are extra orthographic projections of the connecting rod and are created automatically. Notice when the **play** button is pressed again, the frames are automatically linked to each other and all frames animate together.



A Tecplot 360 layout (.lay) file containing a snapshot of the final result of this tutorial segment is in ConnectingRod/FinalLayouts/connecting_rod_1.lay in the Getting Started bundle.

Data Alter with If Statements

This section will use an if statement to calculate where a stress threshold is being crossed.

If you have continued from the previous section, be sure to return your screen to one frame. This can be done quickly by selecting Frame>3D Multi Frame and selecting the icon with the red x (on the bottom right). The screen should now only display one frame.

If you have not completed the previous section, calculate the Von Mises stress (see Step 3: Calculate Von Mises Stress), and turn on Value Blanking for Von Mises Stress equal to zero (see Step 5: Isolate the Connecting Rod).

Step 1: Specify an Equation

Select **Data**>**Alter**>**Specify Equations** from the menu bar to launch the Specify Equations dialog. Type the following equation into the text field: {maxStress} = IF ({Von Mises Stress}>800, 0, 1).

Specify Equations					×
Equation(s)					
{maxStress} = IF ({Von Mises Stress}>800, 0, 1)					
Data Set Info 🛛 Ignore divide by zero errors 🛛 FE derivat	tive <u>m</u> ethod: M	loving Least Squares 🗸	Sav	e Equations	Load Equations
Default Equation Modifiers					
Zones to Alter	Index Rang	es			
21: Step 21 Incr 0	S	Start I	End	Skip	
22: Step 22 Incr 0	I-Index	1	Mx	1	
23: Step 23 Incr 0	1-Index	1	My	1	
24: Step 24 Incr 0	2 macx		PIX		
25: Step 25 Incr 0	K-Index	1	Mx	1	
26: Step 26 Incr 0	New var d	lata <u>t</u> ype:	Auto		~
	Ne <u>w</u> var lo	ocation:	Auto		~
<u>Au</u> Acu <u>v</u> e <u>N</u> one					
			Compute	Close	Help
					· · · ·

Press the **Compute** button at the bottom and an Information popup will appear, "Data alteration successful." Click **OK** and close the **Specify Equations** dialog.

Step 2: Contour It

Open the **Contour & Multi-Coloring Details** dialog again by selecting the *intersection* button next to Contour on the plot sidebar. Change the contour variable to be the variable just created, maxStress. Check that the Contour settings looks similar to the image below. Remember to **remove the color cutoff** if it was set in the previous section.

🧕 Contour & Multi-Coloring Details	×
34: maxStress V 1 Levels and Color Bands Lines Li	2 3 4 5 6 7 8 abels Legend
1:0.05 2:0.1 3:0.15 4:0.2 5:0.25 6:0.3 7:0.35 8:0.4 0:0.4F Color map options	Set Levels Add Level 0 Remove Selected ur variable range 0 Max: 1
Sequential - Viridis	
Color map distribution method Banded Continuous Contour values at endpoints Min: 0 Max: 1 Reset Approximate continuous	Color cutoff Cutoff above: 0.75 Cutoff below: 0.25 Color map adjustments Reverse Repeat cycles: 1
	Close Help

If your data does not contain the amount of contour levels as the above picture, select the **Set Levels** button. This will bring up the Enter Contour Levels dialog. Set the **Level Creation** mode to "**Exact Levels**" and set the **minimum, maximum, and delta to 0.05, 0.95, and 0.05, respectively**. See Step 4: Contour by Von Mises Stress for more information.

Now the regions where the threshold of 800, specified in the if statement, are revealed. However, Tecplot 360 is not only plotting 0 and 1, as expected. By default, Tecplot 360 will interpolate values between cell values. Close the **Contour & Multi-Coloring Details** dialog.



Step 3: Flood by Primary Cell Value

To minimize interpolation by contouring, open the **Zone Style** dialog located on the plot sidebar. Select the **Contour** tab and set the **Contour Type** to **"Primary value flood"**.

		Shade Euge	Points Surface	es Volume E	Effects	
Zone Number	Zone Name	Group SI Number Z	how Show one Contour	Contour Type	Flood By	Lines By
*	Step 9 Incr 0	1 🗸	Prin	nary value flood	C1: maxStress	C1: maxStress
C						
election cr	iteria:	Zones	s Groups	Clear		

Close the **Zone Style** dialog and notice that the resulting plot contains only yellow and blue. The blue cells occur when the Von Mises stress threshold is met or being exceeded.



Plot Maximum Stress Over Time

This section will use a macro to plot Maximum Stress over time and link solution times between frames.

Step 1: Load the Macro

If you are continuing from the previous section, return the Contour variable back to Von Mises Stress (see Step 4: Contour by Von Mises Stress for more information). Then, click Scripting>Play Macro/Script... and the Open Macro dialog will appear. Navigate to the connecting rod folder again and select PlotMaxContourOverTime.mcr.

After selecting Open, the macro will automatically run, creating an XY plot of the maximum value of Von Mises stress at each time step. When the play button is pressed, a marker line follows over the XY plot to track the solution time.


A Tecplot 360 layout (.lay) file containing a snapshot of the final result of this tutorial segment is in ConnectingRod/FinalLayouts/connecting_rod_2.lay in the Getting Started bundle.

Step 2: Macro Breakdown: Create a Zone

This first portion of the macro file takes the first contour variable and finds its name. Then, the macro finds the number of time steps in the transient data and creates a zone.

```
# Get the variable name of the contour variable (limited to Contour Group #1)
$!EXTENDEDCOMMAND
 COMMANDPROCESSORID='extendmcr'
 COMMAND='QUERY.VARNUMBYASSIGNMENT "C" ContourVarNum'
$!EXTENDEDCOMMAND
 COMMANDPROCESSORID='extendmcr'
 COMMAND='QUERY.VARNAMEBYNUM |ContourVarNum| ContourVarName'
#
# Create a new zone that represents the MAXC value
# over time. MAXC returns the maximum value of the
# variable which is assigned to Contour Group #1. See
# the scripting guide for more detail on MAXC.
#
$!EXTENDEDCOMMAND
 COMMANDPROCESSORID='Extend Time MCR'
 COMMAND='QUERY.NUMTIMESTEPS NUMTIMESTEPS'
$!CREATERECTANGULARZONE
 IMAX = |NUMTIMESTEPS|
 JMAX = 1
 KMAX = 1
 X1 = 0
 Y1 = 0
 Z1 = 0
 X_{2} = 1
```

```
Y2 = 0
Z2 = 0
$!VARSET |TimeZone| = |NUMZONES|
$!RENAMEDATASETZONE
ZONE = |TimeZone|
Name = "Max |ContourVarName| over Time"
```



Places where vertical bars are used indicate the use of a macro variable. These can be defined by an intrinsic variable (see Scripting Guide), the \$!VARSETCOMMAND, or specific \$!EXTENDEDCOMMAND calls.

To begin, the first two extended commands in conjunction get the variable name by extracting the variable number by the variable defined in Contour Group 1, then using that to extract the name. The script will then use this as part of the new zones name.

The script then determines the number of time steps using the QUERY.NUMTIMESTEPS which is then used to create an I-ordered zone (rectangular zone where J = K = 1) dimensioned by the number of time steps.

Using the **\$!VARSET** command, a new internal macro variable is created based on the total number of zones. Since the new zone is the last zone created, its index is equal to the maximum number of zones. Then the zone is renamed using the Contour variable name found earlier.

Step 3: Macro Breakdown: Alter Variables

The next step is to populate the zone created in step 2 with the maximum value of Von Mises Stress. The key to this process is using the |MAXC| intrinsic variable which returns the maximum value of the currently active zones.

```
$!LOOP |NUMTIMESTEPS|
 $!EXTENDEDCOMMAND
   COMMANDPROCESSORID='Extend Time MCR'
   COMMAND='SET.CURTIMESTEP |LOOP|'
 $!EXTENDEDCOMMAND
    COMMANDPROCESSORID='Extend Time MCR'
   COMMAND='QUERY.TIMEATSTEP |LOOP| SolutionTime'
 # Instead of creating new variables, we just reuse variables
 # #1 and #2. This keeps the dataset a little cleaner, but if we
 # really wanted to create new variables we could do so using
 # the $!ALTERDATA command
 # Variable #1 represents Solution Time
 $!SETFIELDVALUE
   ZONE = |TimeZone|
   VAR = 1
   INDEX = |LOOP|
   FIELDVALUE = |SolutionTime|
```

```
# Variable #2 represent the Max Contour Value
$!SETFIELDVALUE
ZONE = |TimeZone|
VAR = 2
INDEX = |LOOP|
FIELDVALUE = |MAXC|
$!ENDLOOP
```



[LOOP], when inside a loop, is an intrinsic variable to return the value of the current loop count.

This portion of the macro loops through all time steps activating the current time step though SET.CURTIMESTEP which ensure that only the zone in current time are active and used in the |MAXC| calculation.

Also in the loop, the **\$!SETFIELDVALUE** command is used to set the point value for the specified variable and zone. In this case we are setting the solution time extracted by the QUERY.TIMEATSTEP and the |MAXC| value to the zone created in step 2.

In preparation for creating a new XY plot the macro performs some simple actions to clean up the plot.

We deactivate the zone we just created because we don't want it # to display in the current plot. We'll show it in a new frame instead. \$!ACTIVEFIELDZONES -= [|TimeZone|] # Turn on Time linking because we'll be turning on the # Solution Time axis marker on the following XY frame and # we want that marker to update as we animate over time. \$!LINKING BETWEENFRAMES {LINKSOLUTIONTIME = YES} # Make sure the active frame is at the top of the frame stack. This # ensures that the new frame we create below will inherit this dataset \$!FRAMECONTROL MOVETOTOPACTIVE

First, the newly created zone's field map is removed from showing the original plot, since it may affect how it is displayed. Then Frame linking is turned on and the frame is brought to the top.

Step 4: Macro Breakdown: Create XY Plot

The next step is to create a new XY plot for the newly created zone with a marker which indicates the current time step of the 3D plot.

Now plot the new zone in an XY plot
\$!CREATENEWFRAME
XYPOS

```
{
    X = 1.3947
    Y = 4.6447
 }
 WIDTH = 8.1217
 HEIGHT = 3.2862
$!PLOTTYPE = XYLINE
$!DELETELINEMAPS
$!CREATEL TNEMAP
$!LINEMAP [1] NAME = 'Max |ContourVarName| Over Time'
$!LINEMAP [1] ASSIGN{ZONE = |TimeZone|}
$!ACTIVELINEMAPS += [1]
$!VIEW FIT
$!XYLINEAXIS XDETAIL 1 {TITLE{TITLEMODE = USETEXT}}
$!XYLINEAXIS XDETAIL 1 {TITLE{TEXT = 'Solution Time'}}
$!XYLINEAXIS YDETAIL 1 {TITLE{TITLEMODE = USETEXT}}
$!XYLINEAXIS YDETAIL 1 {TITLE{TEXT = 'Max |ContourVarName|'}}
# Show the solution time axis marker in the XY frame. We turn
# on solution time frame linking to ensure the line updates when
# we animate in the other frame.
$!LINKING BETWEENFRAMES {LINKSOLUTIONTIME = YES}
$!XYLINEAXIS XDETAIL 1 {MARKERGRIDLINE{SHOW = YES}}
```

The XY position and dimensions are provided to create a new frame. This new frame automatically becomes the new active frame. For this reason, changing the plot type and other commands called after the \$!CREATENEWFRAME command only affect the newest frame and not the original.

By default, when the plot type is changed to **XY Line**, line maps are automatically created. The \$!DELETELINEMAPS command must be called in order to create custom line maps. In this case, only one line map is needed. The name and zones are assigned to the newly created line map and it is then activated. By default, line maps will use the first two variables of each zone, hence why only the first two variables were edited in step 3.

To clean up the plot for presentation, the title is set using the XYLINEAXIS commands. The final two lines ensure that the new frame is linking solution time with the previous one and create a marker line to track the current solution time between both frames.

Next Steps

This concludes the Finite Element Analysis tutorial for Tecplot 360. At this point, you may wish to dig in to the User's Manual or Help (**Help>Tecplot 360 Help**) for more details on the product in general or on specific features you've used in this tutorial.

We regularly create videos to introduce users to features of the product, not just introductory topics for new users, but also for advanced users and to highlight new features in the latest release. You can find these on our Web site at www.tecplot.com/category/tecplot-360-videos/?product=360 or on our

YouTube channel at www.youtube.com/user/tecplot360.

Part 2: Internal Combustion Engines

Internal Combustion Engines

This tutorial uses a transient CONVERGE dataset of a single cylinder engine. The data may be downloaded from the Getting Started Bundle.

The Internal Combustion Engine tutorial contains three different sections. The first section is general Tecplot knowledge of loading the data and changing plot styling. The second section is more objectivebased and each segment will be individual. We have provided a layout file alongside the data for the end of each segment, so you can check your work. The overall difficulty, description, and features used in each segment are shown below:

Number and Level	Title and Description	Features Used
1 - Beginner	Loading and Manipulating Data - Load the CONVERGE single cylinder data set into Tecplot 360.	 Data Loading Zone Style
2 - Intermediate	Exploring the Data Set - Adding plot styling and derived objects to create an animation plot and exporting to a movie file.	SlicesZone StyleIso-surfacesScatter
3 - Beginner	Loading Cell Averaged Output Files (CONVERGE) - Load a CONVERGE-specific data file and set up an XY Plot.	Data LoadingXY Plot

Loading and Manipulating Data

Step 1: Launch Tecplot 360 and Load the Data Set

Start Tecplot 360 from the Start menu (Windows), by typing tec360 in a terminal window (Linux), or by double-clicking the application icon in the Applications folder (Mac). The Tecplot 360 Welcome Screen appears, as shown here. (We will show the Windows version of Tecplot 360 in this document, but the product looks substantially the same on other platforms.).



The Welcome Screen appears each time you launch Tecplot 360 and gives you easy access to layouts you have recently worked with, along with quick links to documentation and other resources to help you get the most out of the product.

To begin loading the data, click Load Data at the top of the Welcome Screen. (You may also choose Load Data from the File drop-down menu in the menu bar, or click the folder icon, second from the left, in the toolbar. These alternate methods are convenient when the Welcome Screen isn't visible.).

🚺 Load Data								×
$\leftarrow \rightarrow \checkmark \uparrow$	📒 > De	esktop > 36	50GettingStarted > ICE		~ C	Search ICE		م
Organize - Ne	ew folder						≣ • 🗖	?
A Home			Name	Date modified	Туре		Size	1.1
🔀 Gallery			ICE0000352.10964e+02.plt	8/12/2022 2:41 PM	Tecplot 360	EX plot file	121,739 KB	- 1
			🖲 ICE0000401.85969e+02.plt	8/12/2022 2:41 PM	Tecplot 360	EX plot file	188,242 KB	- 1
			🗵 ICE0000451.60894e+02.plt	8/12/2022 2:41 PM	Tecplot 360	EX plot file	124,073 KB	- 1
			ICE0000501.35999e+02.plt	8/12/2022 2:41 PM	Tecplot 360	EX plot file	94,770 KB	- 1
Pictures		7	ICE0000551.10927e+02.plt	8/12/2022 2:41 PM	Tecplot 360	EX plot file	78,580 KB	- 1
Downloads		*	ICE0000608.59035e+01.plt	8/12/2022 2:41 PM	Tecplot 360	EX plot file	66,419 KB	- 1
🛄 Desktop		*	🖲 ICE0000656.08400e+01.plt	8/12/2022 2:41 PM	Tecplot 360	EX plot file	55,099 KB	- 1
			ICE0000703.57969e+01.plt	8/12/2022 2:41 PM	Tecplot 360	EX plot file	45,873 KB	- 1
> 📮 This PC			🖲 ICE0000751.09957e+01.plt	8/12/2022 2:41 PM	Tecplot 360	EX plot file	52,160 KB	- 1
> 🛬 Network			ICE000080_+1.40034e+01.plt	8/12/2022 2:41 PM	Tecplot 360	EX plot file	53,840 KB	- 1
> 👌 Linux			ICE000085_+3.90251e+01.plt	8/12/2022 2:41 PM	Tecplot 360	EX plot file	53,680 KB	- 1
			ICE000090_+6.40715e+01.plt	8/12/2022 2:41 PM	Tecplot 360	EX plot file	55,841 KB	
		Data overri	de					
	File name:	"ICE000060)8.59035e+01.plt" "ICE0000656.08400e+01.	.plt" "ICE0000703.57969e+01.plt" "ICE0	000751. ~	All Support	ed Files (*.*case;*.	*cç ∽
					Help	Open	Cance	el

Navigate to your Tecplot 360 Getting Started Bundle folder then the Internal Combustion Engine (ICE) folder. Select all of the .plt files and select Open. The data files are opened and a 3D plot of the Combustion Engine appears in the Tecplot 360 workspace, as shown here:



Step 2: Disable Bounding Box for Fluid Volume Zone

You may have noticed a dashed orange line when you first loaded your example file. This is the bounding box of the grid volume zone, which represents the air within the engine. This zone does not have any style (that is, visual appearance) so it would normally be invisible. Tecplot 360 adds the dashed orange line so that you know it is there and can see its dimension. You can choose to hide this line. From the Options menu, select "Show Bounding Boxes for Enabled Volume Zones with No Style". The dashed orange line disappears.

Step 3: Hide Exhaust Ports

Since most of the data of interest lies within the cylinder, hide the exhaust zones by right-clicking the visible zones and selecting "Hide".



Try to hide as many of the exhaust port zones as possible. After hiding, the plot should look something like this. Do not worry if there are a few zones still active.



Exploring the Data Set

For this segment, we will be continuing from where we left off. If you have closed Tecplot 360 since the first segment, you can load the provided layout file in ICE/finallayouts/ICE_6S1.lay

Step 1: Adding a Slice

To add a slice, toggle on the Slices checkbox in the Plot sidebar.

Show derived objects	
Iso-Surfaces	1
Slices	R 🖗
Streamtraces	1
Show effects	
🗹 Lighting	<mark>0</mark>
🗹 Translucency	

A slice appears through the volume zone but may be hidden by surfaces. To be able to see the slice through the middle of our engine we must turn on Translucency in the Plot Sidebar as well.

From there the slice can be moved by selecting the slice interactor tool to place the slice interactively. The interactor tool can move the slice by clicking or dragging to a desired location. This can also be achieved by selecting the interactor to the Slices checkbox. For now, leave the slice in its default location. The Slice details dialog appears with the current Slice location and orientation.



The Slice Details dialog displays the exact slice position.

🧕 Slice Details		×
Show group 1 1 2 3	4 5 6 7 8	
Definition Contour Vector	Other Animate	
Slice location: X-Planes $$		
Show primary slice		‡
	0	▲ ▼

Step 2: Set Slice Styles

Setting the Contour

If you have the Slice Details dialog still open, navigate to the Contour tab. You will see our slice is being contoured by "mass" as this is the variable currently assigned to Contour Group 1. To change the variable in Slice Group 1, click the *state* button next to "Contour" on the Plot sidebar. This will open the Contour Details dialog. This dialog can also be opened by selecting the gear icon next to the Flood By field on the Slice Details dialog.

	🙋 Contour & Multi-Coloring Details 🛛 🕹 🗙
Slice Details ×	4: mass ~ 1 2 3 4 5 6 7 8
Since Details ×	4: mass 1 2 3 4 5 7 8 Levels and Color Bands Lines Labels Legend Contour levels 1: 1e-08 2: 2e-08 3: 3e-08 4: 4e-08 5: 5e-08 6: 6e-08 7: 7e-08 8: 8e-08 Color map options Color map options Color map distribution method Color cutoff Image: Sequential - Viridis Image: Sequential - Viridis
Close Help	Close Help

To change the variable of Contour Group 1, select the "1" button at the top of the Contour Details dialog

and select the dropdown list that currently displays *mass*. Select *temp* for now. Close the Contour Details dialog.

Slice Details			\times
_			
Show group 1 1	2 3 4 5 6	7 8	
Definition Contour	Vector Other	Animate	
Show contours			
Contour type:	Flood	\sim	
Flood by:	C1: temp	× 🏟	
Contour lines by:	C1: temp	× 🔅	
Line color:	Black		
Line thickness (%):	0.1	\sim	
Use lighting effect:	Gouraud	\sim	
Use translucency:	10		
		Close Help	
	L	nop	

Back in the Slice Details dialog, you should see the variable under the Contour tab change to temp as well. The contour group changed successfully!

Turning on the Mesh

We also want to see the mesh of our slice so we will turn that on as well. Right-click the slice on your screen to open the context menu and select the Mesh option. Shown below:



The slice mesh can also be turned on via the Other tab of the Slice Details dialog.Select the magnifying glass in the toolbar to zoom in for a better look.

Using the Forward, Back, and Play buttons on the Plot Sidebar, see how the plot changes over time.

Step 3: Showing Spray Parcels

The slice gives a good indication of the temperatures in the cylinder through time. However, the slice is not the only tool at our disposal. This data set also contains spray information that may be beneficial for our visualization of the engine. To follow along, be sure your Solution Time in the Plot sidebar is set to the first time step.

Open the Zone Style dialog from the plot sidebar and scroll to the bottom of the Zone list. There you will see a zone called "Spray". We want to style this zone to our specifications. To do this, check the Scatter checkbox from the Plot sidebar.

The plot will update with scatter symbols for every zone. This is not what we want though. We only want to show the scatter symbols for the "Spray" zone. In the Zone Style dialog, select the Scatter tab and highlight every zone except the "Spray" zone, right-click the "Show Scatter" column and select No.

🧕 Zone Style

Mesh	Contour	Vector	Scatter	Shade	Edge	Points	Surfaces	Volume	Effects				
Zone Numbe	er	Zone Name		Group Number	Show Zone	Show Scatter	Symbol Shape	Scatter Size	Outline Color	Line Thickness	Fill Mode	Fill Color	^
13*	T=-210	.964 boun	d=12	1	\checkmark			2.50%		0.10%			
14*	T=-210	.964 boun	d=13	1	\checkmark		Square	2.50%		0.10%	None		
15*	T=-210	.964 boun	d=14	1	\checkmark	\sim	Square	2.50%		0.10%	None		
16*	T=-210	.964 boun	d=15	1	\checkmark	\checkmark	Square	2.50%		0.10%	None		
17*	T=-210	.964 boun	d=16	1	\checkmark	\checkmark	Square	2.50%		0.10%	None		
18*	T=-210	.964 boun	d=17	1	\checkmark	\checkmark	Square	2.50%		0.10%	None		
19*	T=-210	.964 boun	d=18	1	\checkmark	\checkmark	Square	2.50%		0.10%	None		
20*	T=-210	.964 boun	d=20	1	\checkmark	2	Source	2.50%		0.10%	None		
21*	T=-210	.964 spray		1	\checkmark	Ye	s are	2.50%		0.10%	None		
83*	T135	999 houn	d–19	1	~	C No	0 170	2 50%		0.10%	None		*
Selection	criteria:				Zones	-6	Groups	Clear					
												Close	Help

Step 4: Changing Spray Parcel Size

Now the "Spray" zone is the only one showing a scatter plot but displaying as same sized squares. In the Scatter tab, right click the symbol shape and change it to Sphere.

Then we also want to change the symbol size, right click the column for Symbol Size and select the "Select Variable" option.

esn C	Contour Vector S	Scatter	Shade	Edge	Points	Surfaces	Volume	Effects				
Zone umber	Zone Name		Group Number	Show Zone	Show Scatter	Symbol Shape	Scatt Size	er e	Outline Color	Line Thickness	Fill Mode	Fill Colo
*	T=-210.964 bound=1	3 '	1	\checkmark		Square	2.50%			0.10%	None	
*	T=-210.964 bound=1	4 '	1	\checkmark		Square	2.50%			0.10%	None	
j*	T=-210.964 bound=1	5	1	\checkmark		Square	2.50%			0.10%	None	
7*	T=-210.964 bound=1	6	1	\checkmark		Square	2.50%			0.10%	None	
*	T=-210.964 bound=1	7	1	\checkmark		Square	2.50%			0.10%	None	
)*	T=-210.964 bound=1	8 '	1	\checkmark		Square	2.50%			0.10%	None	
)*	T=-210.964 bound=2	20 '	1	\checkmark		Square	2.50%			0.10%	None	
*	T=-210.964 spray		1	\checkmark	\checkmark	Square	2.50%	0.50			None	
*	T125 000 hound-1	a ·	1			Sausre	2 50%	1.00			None	>
ection cr	iteria:			Zones		Groups	Clear	2.50				
								4.00 8.00				
								Enter			Close	Help
								By: 4	mass		199	1215123
								Solor	t variable	0		

This will open the Scatter Size/Font menu.

🚳 Scatter Size/Font	×
ASCII symbol font: Helvetica	→ B I
Scatter-size variable:	14: DP_radius 🗸 📗
Size multiplier	10: Vel_mag
Grid units/magnitude:	12: DP_film_flag
Cm/magnitude:	14: DP_radius
Recalculate Size	15: DP_temp 16: DP_Vel_U 17: DP_Vel_V
	18: DP_Vel_W 19: DP_Vel_mag

We want to have the Scatter symbols sized by their radius, from the scatter symbol variable dropdown menu, select the variable DP_radius. However the plot still hasn't updated yet. Finally right-click the symbol size dropdown menu again and select the DP_radius variable. Your Zone Style should look like this and the plot should show scatter spheres sized by radius.

Zone Sty	ſle									
Mesh C Zone Number	Contour Vector Scatter S Zone Name	Shade Edg Group Number	ge P Show Zone	Show Scatter	Surfaces Vo Symbol Shape	lume Effects Scatter Size	Outline Color	Line Thickness	Fill Mode	^
19* 20*	T=-210.964 bound=18 T=-210.964 bound=20	1 1			Sphere Sphere	0.50% 0.50%		0.50% 0.50%	None None	_
83*	T=-210.964 spray T=-135.999 bound=19	1	\checkmark		Sphere	0.50%		0.50%	None	> ~
Selection cr	Selection criteria: Zones Groups Clear									
Selection cr		20	nes	Grou	ups Ci	ear				

Step 5: Changing Spray Parcel Color

The spray has differing size but it still does not give us much information. Contouring the scatter symbols by temperature would be helpful. Open the Zone Style dialog again to the Scatter tab. Scroll down to the "Spray" zone and then scroll to the right to the Outline Color. Right-click the Outline Color and the Color Chooser menu will appear.



Select Multi 2. This corresponds to the Contour & Multi-Coloring group 2. To change the variable of group 2, we will go back to the Contour Details dialog accessed by the *processed* by the solution on the sidebar next to the Contour toggle. For the parcels, we want the color to be contoured by DP_temp. Now you can see how the parcels change temperature and size through time.

🧕 Contour & Mult	ti-Coloring Details	×	(
15: DP_temp	V 1	2 3 4 5 6 7 8	
Contour levels 1:330 2:340 3:350 4:360 5:370 6:380 7:390 8:400 0:410	Con Mi	Set Levels Add Level Remove Selected tour variable range n: 319.707 Max: 335	
>	Sequential - Virio	is V 🗞	
Color map distributi	on method	Color cutoff	
Banded C) Continuous	Cutoff above: 331.177	
Contour value	s at endpoints	Cutoff below: 323.531	
Min: 319.	707	Color map adjustments	
Max: 335	Reset	Reverse	
Approxima	ate continuous	Repeat cycles: 1	
		Close Help]

You'll notice after changing the Parcel color that two contour legends appear on screen. This is because the parcels are colored by DP_temp while the slice is contoured by temp. Since both contours are in reference to temperature, we can update the contour levels of both contour groups.

First remove the legend on contour group 2 by right-clicking the legend and selecting Hide. Then in the Levels and Color tab and select the Set Levels button. Enter the level information like below.

🧕 Enter Contou	ır Levels	×						
Level Creation M	Level Creation Mode							
Exact levels								
O Approximate	e levels for nice	values						
Range Distribution Min, max, and number of levels								
Min, max, a	na aeita dictribution							
	distribution							
Minimum level:	400							
Maximum level:	1800							
Number of levels:	12							
Delta:	200							
		Reset Range 👻						
ОК	Cancel	Help						

Now repeat with group 1. This will update temp and DP_temp to the same scale.



Spheres of Parcel spray can be graphically intensive for some systems. If that is the case, use Point shapes or circles. Points cannot be resized but can use colors much like Spheres.

Scenic Detour: Setting Default Scatter style in the Configuration File

If typical workflows include scatter on a single zone changing the default style can save many clicks.

These commands will change the default values for Scatter to "off" and the shape to "Point". This is especially helpful in situations like our example where we only want to turn on one zone's Scatter points. Note that color contours cannot be set in the configuration file.

The tecplot.cfg file, located in the installation folder, is able to set a default scatter shape and state for simplicity. Open the file in a text editor of your choosing and add the following lines to your configuration file:

```
$!FieldMap Scatter{
   Show = No
   SymbolShape{GeomShape = Point}
}
```

Step 6: Creating an Iso-surface

For this next step we will use iso-surfaces to visualize a flame front during the ignition phase of our engine. Be sure the slice created in steps 1 and 2 is turned off by unchecking Slices on the sidebar but keep the spray turned on, optionally, with the Point scatter shape. Your data should look something similar to this:



Next, we will turn on the Iso-surface option from the Plot sidebar. Select the checkbox next to "Iso-Surfaces" and then the 🛃 button (similar to how we opened the Slice details dialog). This will open the Iso-surface Details dialog.

🗕 Iso-Surface Details	×
Show group 1 1 2 3	4 5 6 7 8
Definition Contour and S	Shade Mesh and Vector Animate
Define iso-surfaces using:	C1: temp 🗸 🔅
Draw iso-surfaces at:	1 Specified Value $$
Value 1:	459.466003418
Value 2:	568.465881348
Value 3:	677.465759277
Contour variable	
Min:	350.466
Max:	786.466
Surface generation method:	Auto ~
✓ Obey source zone blankir	ig
	Close Help

Since we want to visualize a flame front, use the variable temp set to Contour Group 1. Also select the Contour tab on the Iso-surface Details dialog and verify the Iso-surface is being contoured by temp as well.

🙋 Iso-Surface Details							
Show group 1 1 2 3 4 5 6 7 8							
Definition Contour an	nd Shade Mesh and Vector Animate						
Show contour							
Contour type:	Flood \sim						
Flood by:	C1: temp $~~$						
Contour lines by:	C1: temp 🗸 🙀						
Line color:	Black						
Line thickness (%):	0.1 ~						
Use lighting effect							
Show shade							
Color:	White						
✓ Use lighting effect							
Effects							
Lighting effect	Gouraud 🗸 🗸						
Use translucency:	50						
	Close Help						

Back on the details page, set the value of the Iso-surface to be 2000, which represents the flame front.

Iso-Surface Details						
Show group 1 1 2 3	4 5 6 7 8					
Definition Contour and S	hade Mesh and Vector Animate					
Define iso-surfaces using:	C1: temp 🗸 🙀					
Draw iso-surfaces at:	1 Specified Value $$					
Value 1:	2000					
Value 2:	568.465881348					
Value 3:	677.465759277					
Contour variable						
Min:	350.466					
Max:	786.466					
Surface generation method:	Auto \checkmark					
✓ Obey source zone blanking	ig					
	Close Help					

Pressing the play button on the Plot sidebar to animate through the flame front propagation.

Step 7: Using 3D Multi Frames

Our visualized data is starting to come together. However, it can be convenient to view data from multiple angles. This will give a better understanding of the data from all sides. This can be easily done by selecting **Frame** \rightarrow **3D Multi Frames**. For this example, we will use the top-left option.



This will give us orthogonal views of our data set and we can derive a better understanding of our data throughout time.



Step 8: Export to Movie File

Now that we have our plot set up as we want it, let's export our final product to a movie file. Select the button next to the Solution Time to open the Time Animation Details dialog.

Solution ti	ime: -:	210.96	4		¢,
	\mathbb{N}				

This dialog displays more advanced animation options than the Plot sidebar.

🗕 Time Animation	n Details	×					
Starting time:	-210.96414	Min: -210.964					
Ending time:	114.13696	Max: 114.137					
Time step skip:	1	Number of steps: 14					
Animation direction:	Forward ~	•					
Current step:	1	Go to Step					
Current value:	-210.964						
Limit animation sp	Limit animation speed to 12 frames/second						
For strands with no a	zones at the current so	ution time:					
Show zones a	Show zones at or before solution time						
Show zones of the second se	only at solution time						
	Clo	se Help					

From the Time Animation details dialog, select the "Export to File" button next to the VCR buttons. This will open up the Animation Export dialog. The default animation export is MPEG-4. Because of the 3D Multi Frames in our workspace, ensure the region is "All Frames".

Export ×	
Export format: MPEG-4 \checkmark	
✓ Color	
Region: All frames V	
 Use width of image on screen 	
Enter width: 4096	
Export dimensions: 4096 x 3088	
Antialiasing	
Supersample factor (2-16): 3	
Animation speed (frames/sec): 10	
OK Cancel Help	

You can also choose to export a larger image width than your current view. This is especially helpful if the movie is going to be displayed on a high-resolution screen. For this example, we want our final product to be viewed on a 4K resolution display so we will enter a width of 4096. Select OK and save the final file.

Loading Cell Averaged Output Files (CONVERGE)

This section will use data time history exported from CONVERGE. Although somewhat specific to CONVERGE users, it will show genearal time history capabilities. If you are continuing from the previous segment, select **File > New Layout** to clear the previous plot.

Step 1: Loading the data

Load area_avg_flow.out from the ICE directory of the Getting Started Guide Bundle. Select **File > Load Data**. Once prompted, select the CONVERGE Output Loader.

🗕 Choose Loader (*.out)	\times
Multiple loaders support this kind of file. Please choose which loader you would like	to use.
Suggested Loaders	^
CONVERGE Output File Loader	
MSC/Patran Neutral (FEA) い	
Other Available Loaders	
3D Systems STL (FEA)	
ABAOUS .fil Data (FFA)	~
<	>
OK Cancel He	elp

Selecting **OK** will load the plot in **XY** Line type.

Step 2: Add Multiple Y Axes

The plot by default is in XY Line mode which contains pre-loaded XY Line maps. Select the Mapping Style button on the Plot Sidebar to see which Mappings are available. You can see that this file was loaded with many variables premade vs. the Crank Angle. For this example, cange the plot to show Avg_Temp and Mass_Flow_Rate by Crank Angle. Turn off the default map for Crank vs Cycle Number. Turn on Map numbers 2 and 5.

efinitions	Lines Curv	/es S	Symbols Error Bars	Bar Charts Indices			
Map Number	Map Name	Show Map	X-Axis Variable	Y-Axis Variable	Zone	Sort	Which W X-Axis Y-
	Cycle_Number		1: Crank (DEG)	2: Cycle_Number	1: area_avg_flow bound_id_7	None	X1 Y1
	Mass_Flow_R	\checkmark	1: Crank (DEG)	3: Mass_Flow_Rate (1: area_avg_flow bound_id_7	None	X1 Y1
	Total_Pres (Pa)		1: Crank (DEG)	4: Total_Pres (Pa)	1: area_avg_flow bound_id_7	None	X1 Y1
	Static_Pres (Pa)		1: Crank (DEG)	5: Static_Pres (Pa)	1: area_avg_flow bound_id_7	None	X1 Y1
	Avg_Temp (K)	\checkmark	1: Crank (DEG)	6: Avg_Temp (K)	1: area_avg_flow bound_id_7	None	X1 Y1
	Avg_Velocity	L.S	1: Crank (DEG)	7: Avg_Velocity (m/s)	1: area_avg_flow bound_id_7	None	X1 Y1
	Avg_Density		1: Crank (DEG)	8: Avg_Density (kg/	1: area_avg_flow bound_id_7	None	X1 Y1
	Avg_Mach		1: Crank (DEG)	9: Avg_Mach	1: area_avg_flow bound_id_7	None	X1 Y1
:	Tot Darcel In		1. Crank (DEG)	10: Tot Darcel In	1: area ave flow bound id 7	None	¥1 V1
election cr	iteria:		Sele	ct Clear			

Notice how the plot automatically merges both lines to one Y Axis. The min/max values of Mass_Flow_Rate and Avg_Temp are very different, so putting one of the mappings in a second Y axis will help.



Select the Mapping Style button again on the Plot Sidebar. Scroll to the right to the "Which Y-Axis" column and right click the Avg_Temp row. Then select Y2 as the Y-Axis. The plot will automatically update.

Mapping Style

∕lap ame	Show Map	X-Axis Variable	Y-Axis Variable	Zone	Sort	Which X-Axis	Which Y-Axis	Show in Legend	î
Number		1: Crank (DEG)	2: Cycle_Number	1: area_avg_flow bound_id_7	None	X1	Y1	Auto	
Flow_R	\checkmark	1: Crank (DEG)	3: Mass_Flow_Rate (1: area_avg_flow bound_id_7	None	X1	Y1	Auto	
Pres (Pa)		1: Crank (DEG)	4: Total_Pres (Pa)	1: area_avg_flow bound_id_7	None	X1	Y1	Auto	
Pres (Pa)		1: Crank (DEG)	5: Static_Pres (Pa)	1: area_avg_flow bound_id_7	None	X1	Y1	Auto	
emp (K)	\checkmark	1: Crank (DEG)	6: Avg_Temp (K)	1: area_avg_flow bound_id_7	None	X1	Y1	Auto	
elocity		1: Crank (DEG)	7: Avg_Velocity (m/s)	1: area_avg_flow bound_id_7	None	X1	Y1	Y1	
ensity		1: Crank (DEG)	8: Avg_Density (kg/	1: area_avg_flow bound_id_7	None	X1	Y1	Y2	
1ach		1: Crank (DEG)	9: Avg_Mach	1: area_avg_flow bound_id_7	None	X1	Y1	×Y3	-
rcel In		1. Crank (DEG)	10: Tot Darcel In	1: area ava flow bound id 7	None	¥1	V1	VA .	~
<								14 >	,
election crit	eria.		Select	Clear				Y5	
needon en			boloci	oldi					

 \times



Finally close the Mapping Style dialog to see the final plot. If your plot doesn't automatically update to the new axis bounds, select Ctrl+F (or use **View** > **Fit** to Full Size).

To increase the line thickness, right-click the line to bring up the right-click context menu and update the the thickness as desired. For black and white printing, consider changing one of the lines to dashed.

Step 3: Add Grid Lines

Select **Plot** > **Axis** to open the **Axis Details** menu (or double click on the axis). From the Axis Details menu, you can see the different tabs to update the plot axes. Notice the X1, Y1, X2 … buttons, these control the axis being acted on. For now, we want to update the X1 Grid Lines. Select the Grid tab to update the grid lines for the X1 axis. Select the Y1 button as well to update the Grid options for Y1. In our case, Y1 is the Mass_Flow_Rate.

🧕 Axis Details			×
Show Y1-Axis X1 Y1	X2 Y2 X3 Y3 X4 Y4	4 X5 Y5	
Range Grid Ticks	Labels Title Line	Area	
	Gridlines	Minor gridlines	
	Show	Show	
Line pattern:	Solid \sim	Dotted \sim	
Pattern length (%):	2 ~	2 📉	
Thickness (%):	0.1 ~	0.1 📉	
Gridline color:	Black	Black	
Gridline draw order:	First \vee	First ~	
Show precise dot grid			
Dot size (cm): 0.0045	Dot color:	Black	
			Close Help

Select the Show checkboxes under Gridlines and Minor Grid Lines. Notice how the Major Tick labels have the Solid gridlines and the Minor Tick labels have the Dotted Gridlines.

Next select the Y2 button to edit the Grid Lines for Avg_Temp. Like before, show the Gridlines but this time leave the Minor Gridlines off. Also notice that there are no distinguishing differences between the major gridlines. For that reason, we want to update the Gridlines color to "Blue".

🧕 Axis Details			×
Show Y2-Axis X1 Y1	X2 Y2 X3 Y3 X4 Y4	4 X5 Y5	
Range Grid Ticks	Labels Title Line	Area	
	Gridlines	Minor gridlines	
	Show	Show	
Line pattern:	Solid \vee	Dotted \sim	
Pattern length (%):	2 ~	2 ~	
Thickness (%):	0.1 ~	0.1 ~	
Gridline color:	Blue	Black	
Gridline draw order:	First \vee	First \vee	
Show precise dot grid			
Dot size (cm): 0.0045	Dot color:	Black	
			Close Help

Step 4: Update Axis Labels

Next, we want to update the Axis Labels to be spaced more cleanly. For Avg_Temp, the axis labels are spaced well with 100 points between each step so Y2 will not need to be updated. However, we will update Y1 to a better distribution. From the Axis Details dialog select the Labels tab.

Deselect the Auto Spacing option at the bottom to give us control of the Label Spacing. Then in the Spacing field, update the number to 0.025. Close the Axis Details dialog.

🧕 Axis Details								\times
Show Y1-Axis X1 Y	1 X2 Y2 X3 Y3	X4 Y4 X5 Y5						
Range Grid Ticks	Labels Title	Line Area						
Show labels on:	Color:	Black						
Grid border left	Font:	Helvetica	\sim	3		▲ ▼ B	Ι	
Grid border right	Format:	Number Format						
	Offset from line (%):	1						
	Orient labels:	At angle	\sim					
	Angle (deg):	0	\sim					
		Show label at axis intersection						
Label skip: 1]	✓ Erase behind labels						
Tick Mark and Label Spacing	ng: 0.025 j	Anchor: 0						_
					Close		Help	

The final plot is shown below:



Next Steps

This concludes the Internal Combustion Engine tutorial for Tecplot 360. At this point, you may wish to dig in to the User's Manual or Help (**Help>Tecplot 360 Help**) for more details on the product in general or on specific features you've used in this tutorial.

We regularly create videos to introduce users to features of the product, not just introductory topics for new users, but also for advanced users and to highlight new features in the latest release. You can find these on our Web site at www.tecplot.com/category/tecplot-360-videos/?product=360 or on our YouTube channel at www.youtube.com/user/tecplot360.

Part 3: Ocean Modeling
Ocean Modeling

This tutorial uses a transient FVCOM NetCDF data set of the Boston harbor. The data may be downloaded from the Getting Started Bundle.

This data set was provided by UMASS Chen's lab. The original data is located here.

The FVCOM NetCDF data file must have nv (element to node connectivity), siglev (sigma level), h (depth), zeta (surface elevation), (lon,lat) or (x, y) coordinates for Tecplot to be able to load. Tecplot 360 is not able to deal with FVCOM NetCDF data without the above variables.

Number and Level	Title and Description	Features Used
1 - Beginner	Loading and Manipulating Data - Load the Boston example dataset and inspect the data.	Data LoadingData Set Information
2 - Intermediate	Making your first plot - Adding plot styling and value blanking to create an animation plot and exporting to a movie file.	 Contours Zone Style Axis Scaling Value Blanking
3 - Intermediate	Specific Ocean Plots - Visualize surface vector fields, insert a georeferenced image and show iso-surfaces.	 Visualizing Surface Velocities Insert a Georeferenced Image Understanding Salinity Stratification
4 - Advanced	Advanced Topics - Links to advanced topics available on the Tecplot website.	 PyTecplot Averages Shapefiles

The overall difficulty, description, and features used in each segment are shown below:

Loading and Manipulating Data

Step 1: Launch Tecplot 360 and Load the Data Set

Start Tecplot 360 from the Start menu (Windows), by typing tec360 in a terminal window (Linux), or by double-clicking the application icon in the Applications folder (Mac). The Tecplot 360 Welcome Screen appears, as shown here. (We will show the Windows version of Tecplot 360 in this document, but the product looks substantially the same on other platforms).



The Welcome Screen appears each time you launch Tecplot 360 and gives you easy access to layouts you have recently worked with, along with quick links to documentation and other resources to help you get the most out of the product.

To begin loading the data, click Load Data at the top of the Welcome Screen. (You may also choose Load Data from the File drop-down menu in the menu bar, or click the folder icon, second from the left, in the toolbar. These alternate methods are convenient when the Welcome Screen isn't visible).

🖲 Load Data						×
\leftrightarrow \rightarrow \checkmark \uparrow \square \rightarrow De	esktop → 360	DGettingStarted > Ocean >		~ C	Search Ocean	Q
Organize - New folder					≣ • □	?
A Home		Name	Date modified	Туре	Size	
Gallery		finallayouts	8/12/2022 2:41 PM	File folder		
		NECOFS_FVCOM_OCEAN_BOSTON_FORECAST	8/12/2022 2:41 PM	NC File	748,971 KB	
Documents	*					
Pictures	*					
Downloads	*					
🛄 Desktop	*					
> 📮 This PC						
> 🖆 Network						
> 👌 Linux						
	Data overrid	e				
File name:	NECOFS_FV	COM_OCEAN_BOSTON_FORECAST_2048-2071.nc		~	All Supported Files (*.*case;*.*	יכ <u>נ</u> ~
				Help	Open 🔽 Cance	1

Navigate to your Tecplot 360 Getting Started Bundle folder and then the Ocean folder. Highlight the .nc file and select Open to open it in Tecplot 360. (If you can't see this file, choose All Files in the menu at the bottom of the dialog.) The data file is opened and a 3D plot of the Boston harbor loads in the Tecplot 360 workspace, as shown here, an orange box shows the bounds of the volume data. See Understanding Volume Surfaces.



Step 2: Inspecting the Data

The orange bounding box represents a volume dataset that is loaded with no style. For more information on Surfaces to Plot see Understanding Volume Surfaces. To see what data has been loaded, open the Data Set Info dialog. From the top of the screen select Data \rightarrow Data Set Info. This will bring up a dialog with lists of loaded zones and variables.

🙋 Data Set Information	×
Zone/Var Info Data Set Aux Data	
Zone(s)	Variable(s)
1: NECOFS_FVCOM_OCEAN_BOSTON_FORE(2: NECOFS_FVCOM_OCEAN_BOSTON_FORE(3: NECOFS_FVCOM_OCEAN_BOSTON_FORE(4: NECOFS_FVCOM_OCEAN_BOSTON_FORE(5: NECOFS_FVCOM_OCEAN_BOSTON_FORE(6: NECOFS_FVCOM_OCEAN_BOSTON_FORE(7: NECOFS_FVCOM_OCEAN_BOSTON_FORE(8: NECOFS_FVCOM_OCEAN_BOSTON_FORE(9: NECOFS_FVCOM_OCEAN_BOSTON_FORE(10: NECOFS_FVCOM_OCEAN_BOSTON_FORE(11: NECOFS_FVCOM_OCEAN_BOSTON_FORE(11: NECOFS_FVCOM_OCEAN_BOSTON_FORE(12: NECOFS_FVCOM_OCEAN_BOSTON_FORE(12: NECOFS_FVCOM_OCEAN_BOSTON_FORE(13: NECOFS_FVCOM_OCEAN_BOSTON_FORE(14: NECOFS_FVCOM_OCEAN_BOSTON_FORE(15: NECOFS_FVCOM_OCE	1: x ^ 2: y 3: lon 4: lat 5: z 6: art1 7: art2 8: hs 9: kh 10: km 11: kq 12: L 12: L
<pre> V: NECOES EVCOM OCEAN BOSTON FORE V V V V V V V V V V V V V V V V V V V</pre>	12:1 13: ntsn
Zone Type: FE - Brick Pts: 436020 Elem: 725705 Solution Time: 58078.3 Time Strand: 1	Var Type: Float Var Location: Nodal Var Status: Heap Var Range Selected Zone: Min: 809618 Max: 850800 Var Range Active Zones: Min: 809618 Max: 850800 Load Variables
	Close Help

In the Zones field, we can see that the NetCDF ocean model has loaded 24 zones. In this case, each zone represents a new timestep of the data so we have 24 timesteps in total. We can also see in the other column the variables that have been loaded from the dataset. Close the Data Set Info dialog for now.

Making your first plot

Step 1: Turn on Contour to See the Domain

For the first representation of the data, turn on the Contour layer. Select the checkbox next to Contour on the Plot Sidebar.

Contour	2
Shade	

A pop-up will appear asking about Surfaces to Plot. For now, select "Yes". A contour plot will appear with the default contour variable selected. In this case it is "lon". This is because "lon" is the first, non-XYZ variable in the dataset.

Step 2: Assign XYZ to Lon, Lat

For this plot, change the X and Y coordinates to Lon and Lat. This will make it easier for things like georeferenced images to be placed in the plot (see Insert a Georeferenced Image). To do this, select $Plot \rightarrow Assign XYZ$ and change the X variable to "lon" and the Y variable to "lat".

۲	Select Variables	×		
Choose the variables to assign to the axes.				
x	3: lon	\sim		
Y	4: lat	\sim		
z	5: z	\sim		
	Of Cancel Help			

The plot will automatically update on pressing OK.

Step 3: Adjusting Z-scaling

You may have noticed pop-ups that say, "Aspect ratio exceeds ratio limit. The ratio has been adjusted." This is because Tecplot attempts to maintain a visually interesting axis ratio to prevent any one direction from being too small. However, adjust this ratio because the current plot is too tall in the Z direction. Select **Plot** \rightarrow **Axis** to open the Axis Details dialog.

Towards the bottom of the dialog is the Dependency field. These options can determine XYZ dependency based on your information. The Z-axis needs to be smaller so set the plot "XY Dependent". Select the button next to "XY Dependent" and adjust the Z-axis size factor to be something like "0.000667".

log Axis Details	×
Show X-Axis X Y Z	
Range Grid Ticks Labels Title Line Area	
Min: -71.2888702393 Reset Range	
Max: -70.7392059326	
Preserve axis length when changing range Use log scale	
Reverse axis direction	
Dependency Size factors	
◯ Independent X to Y ratio: 1 X: 1	
XY dependent Y: 1	
O XYZ dependent X to Z ratio: 235.606 Z: 0.000667 Image: Colored	
Reset Size/Dependency	
Class	
Close Help	

Note that the Z-axis is still exaggerated even at this ratio, but it gives a better understanding of the overall scale.

Step 4: Change View and Lighting

If you want the map to change to a top-down view, select the rollerball rotate (shown below), click, and drag the screen around to get a top-down perspective.

Another option is selecting "Snap to Orientation view" in the Plot Sidebar and selecting the XY perspective.

From this angle, lighting effects wash out the contour color. For now we can turn off lighting by

unchecking the "Lighting" checkbox on the Plot sidebar.

Show effects	
	e ^{pa}
Show lighting	

Now the plot should have no lighting effect and the contour data can be seen more clearly.

Step 5: Change the Contour Variable and Colormap

The contour variable has remained at the default of "lon" up until now, update to a more data relevant variable like "Salinity". To do this, select the button next to Contour on the Plot sidebar. The Contour & Multi-Coloring details menu appears.

Notice the dropdown list at the top of the dialog is set to "lon". Select the dropdown list and all the variables in your dataset appear. Now select "salinity" as your new contour variable. The plot will automatically update.

2: lon	
3: ntsn 4: ntve 5: omega	Labels Legend
6: q2 7: q2l 8: qb 9: salinity 0: siglay 1: siglev 2: temp 5: -71.05 6: -71 7: -70.95 8: -70.9 0: 70.95	 Set Levels Add Level Add Level Remove Selected Contour variable range Min: -71.2639 Max: -70.7642
Color map options	
Color map options	tial - Viridis
Color map options	tial - Viridis Viridis
Color map options	tial - Viridis Color cutoff Cutoff above: -70.8891
Color map options	Itial - Viridis Color cutoff Color cutoff Cutoff above: -70.8891 Cutoff below:
Color map options	 Itial - Viridis Color cutoff Cutoff above: -70.8891 Cutoff below: -71.139 Color map adjustments
Color map options Color map distribution method Banded Continuous Contour values at endpoints Min: -71.2639 Max: -70.7642 R	Itial - Viridis Color cutoff Color cutoff Cutoff above: Cutoff below: Color map adjustments

Also select a new colormap based on the cmocean colormaps, in this case, the "cmocean - haline" colormap which is designed to be used with salinity plots. To change the colormap select the colormap dropdown from the middle of the Contour & Multi-Coloring Details dialog, and select "cmocean - haline" option.

Step 6: Use Value Blanking to Hide Dry Land

After updating the plot to "salinity" notice the shoreline of Boston harbor within the plot. However, the contours extend out from beyond the shoreline. This dataset has a variable called "wet_nodes" to determine what is wet (denoted by the variable = 1) and what is dry (denoted by the variable = 0).

To hide the dry nodes on the plot where "wet_nodes" is equal to 0. Go to $Plot \rightarrow Blanking \rightarrow Value$ Blanking to pull up the Value Blanking dialog.

🧕 Value Blanking 🛛 🗙 🗙
✓ Include value blanking
Blank entire cells when: any corner is blanked \sim
Constraint 1 2 3 4 5 6 7 8
Blank when
27: wet_nodes V
is equal to \sim
🔿 Variable 💿 Constant
0
Close Help

Then turn on the "Include value blanking" checkbox. This will tell the plot that we want to blank something. Change the Blank when fields to be when "wet_nodes", "is equal to", "Constant" and "0". Then select the "Active" checkbox. The plot will update accordingly. The exact value blanking options are in the image above and final plot should look like this:

Step 7: Animating and Exporting a Movie File

Now that the plot is looking good, let's animate through time. Select the Play button on the Plot sidebar to play the animation. The plot will show the ebbing and flowing of salinity in the Boston harbor.

Note that the first animation takes longer because it needs to load data. The second play through will be faster as the data has already been loaded.

Now we want to save this animation to a movie file. On the Plot sidebar select the button next to Solution time and the Time Animation details dialog will appear. This dialog can give more control over your plot animation but since the default settings are already sufficient, select the "Export to File" button next to the VCR buttons on the Time Animation details dialog.

🧕 Time Animation	n Details	×
Starting time:	58078.332	Min: 58078.3
Ending time:	58079.293	Max: 58079.3
Time step skip:	1	Number of steps: 24
Animation direction:	Forward	~
Current step:	1	Go to Step
Current value:	58078.3	
Limit animation sp	peed to 12	frames/second
For strands with no a	zones at the current s	olution time:
Show zones a	t or before solution ti	ne
Show zones of the second se	only at solution time	
	C	lose Help

From the Export dialog, we can select things such as export format, image width, antialiasing, and animation speed. For this instruction, select the MPEG-4 format. This will create a file with an .mp4 extension that is usable by most video viewers. Also, turn on antialiasing as well to get the definition on the shoreline and have smoother lines. Keep everything else the default and click OK.

Export	×
Export format: MPEG-4 \checkmark	
✓ Color	
Region: All frames ~	
Use width of image on screen	
O Enter width: 512	
Export dimensions: 1920 x 1706	
Antialiasing	
Supersample factor (2-16): 3	
Animation speed (frames/sec): 10	
Cancel Help	

Next, save the file to a convenient location as "boston_salinity.mp4". Congratulations you've created your first Tecplot ocean animation!

Specific Ocean Plots

The next few sections focus on specific quick plots that can be achieved with this dataset. The goal being to teach how to create interesting plots quickly.

Visualizing Surface Velocities

This section is a continuation from the previous section. If you are joining partway through, open the Ocean1.lay from the finallayouts directory.

Step 1: Turn on the Vectors

First turn on the Vector layer from the Plot Sidebar to visualize surface velocities. Select the checkbox next to Vector much like how we turned on the contour layer (see Step 1: Turn on Contour to See the Domain). The Select Variables menu should appear allowing the selection of the U, V, and W variables.

Select Variables				
Choose the vector variables.				
U	24: u	\sim		
v	25: v	\sim		
w	30: ww	\sim		
	OK Cancel Help			

For the next section, select the u, v, and "ww" variables from the dropdown.

Step 2: Value Blank Siglev

At this point all surfaces are showing vectors including the sea floor. To only show the sea surface, use value blanking to isolate the ocean surface by blanking when "siglev" is less than 0. Using the second value blanking constraint, select Plot \rightarrow Blanking \rightarrow Value Blanking and update the options to match the image below. Be sure to change the "Blank entire cells when" option to "all corners are blanked".

🧕 Value Blanking	×
✓ Include value blanking	
Blank entire cells when: all corners are blanked V	2
Active	,
Blank when	
21: siglev ~	e
is less than \sim	•
🔿 Variable 🔘 Constant	
0	
Clos	

Step 3: Use Vector Tangents

Because the view is still 3D model, some vectors will look like dots as the vector is pointing vertically toward the screen. To prevent this turn on the Vector Tangents from the Zone Style dialog. Select the Zone Style button from the Plot Sidebar and the Zone Style dialog will appear.

Zone Sty	yle									
Mesh	Contour	Vector	Scatter	Shade	Edge F	oints	Surface	s Volume	Effects	
Zone Number		: 1	Zone Name		Group Number	Show Zone	Show Vector	Vector Type	Ve Tan	ector Head gents Style
1*	NECOFS	S_FVCOM_	OCEAN_BC	STON_F	1	\checkmark	\checkmark	Tail at point	4	Plain
									45	
<										>
election c	riteria:				Zones	Gr	oups	Clear		
								г	Class	Uala
									I INCO	HOIN

Select the Vector tab and then select the checkbox under "Vector Tangents". This will ensure that the vectors will be displayed tangential to the surface. Close the Zone Style dialog.

Step 4: Calculate Velocity Magnitude

Since the Velocity Magnitude isn't located in the current dataset, we will need to calculate it. Typically new equations can be calculated using the Data Alter menu (Data \rightarrow Alter \rightarrow Specify Equations). However, the CFD Analyze menu can also be used to calculate Velocity magnitude which will prevent the need to type the equations in manually.

Firstly, we will need to set up the CFD Analyze menu by letting the Analyzer know the data's field variables. Select Analyze \rightarrow Field Variables to bring up the Field Variables dialog. From this dialog we can set the necessary variables for analysis. For this calculation, setting the convective variables is sufficient. Be sure the Velocity button is checked and the vectors variables are the same as what is shown on the plot (u, v, and ww).

🧕 Field	Variables			\times
Convec	tive Variables			
(Velocity	U:	u	Select
(Momentum	V:	v	
		w:	ww	
State \	/ariables			
For	Not Used \sim	Use		Select
For	Not Used $~~$	Use		
			ov. Durad	
			OK Cancel	неір

Next select Analyze \rightarrow Calculate Variables. This will bring up the Calculate Variables page. Select Velocity magnitude as the variable calculation and uncheck the Calculate on Demand button. This will make sure the points are calculated now instead of when the variable is plotted.

🧕 Calcul	ate			\times
Name	Velocity Magnitude		Select	
Normaliz	e with Respect To	No Normalization	\sim	
New Var	Location	Node	\sim	
Calcu	llate on Demand			
	Cal	culate		
		Close	Help	

Then select Calculate and the Velocity Magnitude variable will begin the calculation process for all zones. Information will appear upon completion that the calculation was successful. Now in the Data Set Info dialog (Data \rightarrow Data Set Info) you can see the Velocity Magnitude variable at the end of the variable list.

🗕 Data Set Information

Х

Zone/Var Info	Data Set	Aux Data					
Zone(s)				Variable(s)			
1: NECOFS_F	VCOM_OCEA	N_BOSTON	FORECAST ^	8: hs			~
2: NECOFS_F		AN_BOSTON	FORECAST	9: kh			
3: NECOFS_F	VCOM_OCE	AN_BOSTON	I_FORECAST	10: km			
4: NECOFS_F	VCOM_OCE	N_BOSTON	_FORECAST	11: kq			
5: NECOFS_F	VCOM_OCE	N_BOSTON	_FORECAST	12:1			
6: NECOFS_F	VCOM_OCE	N_BOSTON	_FORECAST	13: ntsn			
7: NECOFS_F	VCOM_OCE	AN_BOSTON	I_FORECAST	14: ntve			
8: NECOFS_F	VCOM_OCE	AN_BOSTON	I_FORECAST	15: omega			
9: NECOFS_F	VCOM_OCEA	AN_BOSTON	I_FORECAST	16: q2			
10: NECOFS_F	VCOM_OCE	AN_BOSTON	N_FORECAS	17: q2l			
11: NECOFS_F	VCOM_OCE	AN_BOSTON	N_FORECAS	18: qb			
12: NECOFS_F	VCOM_OCE	AN_BOSTON	N_FORECAS	19: salinity			
13: NECOFS_F	VCOM_OCE	AN_BOSTON	N_FORECAS	20: siglay			
14: NECOFS_F	VCOM_OCE	AN_BOSTO	N_FORECAS	21: siglev			
15: NECOFS_F	VCOM_OCE	AN_BOSTON	N_FORECAS	22: temp			
16: NECOFS_F	VCOM_OCE	AN_BOSTO	N_FORECAS	23: tpeak			
17: NECOFS_F	VCOM_OCE	AN_BOSTO	N_FORECAS	24: u			
18: NECOFS_F	VCOM_OCE	AN_BOSTON	N_FORECAS	25: v			
19: NECOFS_F	VCOM_OCE	AN_BOSTO	N_FORECAS	26: wdir			
20: NECOFS_F	VCOM_OCE	AN_BOSTO	N_FORECAS	27: wet_nodes			
21: NECOFS_F	VCOM_OCE	AN_BOSTON	N_FORECAS	28: wet_nodes_prev_int			
22: NECOFS_F	VCOM_OCE	AN_BOSTON	N_FORECAS	29: wlen			
23: NECOFS_F	VCOM_OCE	AN_BOSTON	N_FORECAS	30: ww			
<			>	31: Velocity Magnitude	A.C.		~
Zone Type: F	E - Brick			Var Type: Float	•		
Pts: 4	36020			Var Location: Nodal			
Elem: 7	25705			Var Status: Heap			
Solution Time: 5	8078.3			Var Range Selected Zone:			
Time Strand: 1				Min: 0			
				Max: 1.17643			
				Var Range Active Zones:			
				Min: 0			
				Max: 1.17643			
						Load Var	iables
					Close		Help
					-		

Step 5: Contour by Velocity Magnitude

To contour by velocity magnitude, see the previous section Step 5: Change the Contour Variable and Colormap to update the Contour variable. Then switch the colormap to "cmocean-speed", designed to be used with velocity. We also want to change the contour levels to only include data relevant to our study.

To do this, select Set Levels from the Contour & Multi-Coloring details dialog. When the dialog appears, change the option of Range Distribution to Min, Max, and Delta. With the minimum level at 0 and maximum level at 0.6 with a delta of 0.1.

🗕 Enter Contou	ur Levels	×
Level Creation M	lode	
Exact levels		
O Approximate	e levels for nice	values
Range Distributio	on nd number of le	evels
🔘 Min, max, a	nd delta	
Exponential	distribution	
Minimum level:	0.1	
Maximum level:	0.6	
Number of levels:	6	
Delta:	0.1	
		Reset Range 👻
ОК	Cancel	Help

Click OK and close the Contour & Multi-Coloring Details dialog. Here is what your plot should look like currently.

Step 6: Resize Vectors

Since the vectors on screen are a little bit too large in the more dense areas of the plot, we should change the length. Select $Plot \rightarrow Vector \rightarrow Details$ to adjust the vector length from the Vector Details dialog.

🧕 Vector Details		×
Definition Arrowhead Reference	e Vector	
Vector variables		
U: 24: u		~
V: 25: v		\sim
W: 30: ww		\sim
Vector length		
Relative (grid units/magnitude):	0.005	•
Relative (cm/magnitude):	0.14247873	*
O Uniform (%):	2	* *
		Reset Length
Use even vector spacing		
Approximate spacing between vector	rs (grid units)	
X: 0 🗘 Y: 0) 🗘 Z: 0	*
		Reset Spacing
	Close	Help

Set the Length to Relative (grid units/magnitude) and set the number to 0.005. This will update the vectors to be smaller and more manageable in the dense areas of the plot. Your final plot should be similar to the one you see below:

At this point, use the zoom and pan tools to explore different regions of the data.

Insert a Georeferenced Image

Step 1: Set Up Coordinate System

Before inserting the image, ensure the coordinate system on your plot is the same as the georeferenced image that you are trying to insert. If a georeferenced image is inserted with a different coordinate system than the dataset, the image will not appear in the correct location. For this example, set the Boston dataset to be in lon/lat coordinates.

۱	Select Variables					
Cho	oose the variables to assign to the axes.					
x [3: Ion	\sim				
Y [4: lat	\sim				
z [5: z	\sim				
	Of Cancel Help					

Select Plot > Assign XYZ and assign lon to the X variable and assign lat to the Y variable. If you are continuing from the previous exercises, this step will already be done.

Step 2: Insert Georeferenced Image

Select Insert > Image/Georeferenced Image and select the 'boston.png' file from the same folder the data was loaded. The map of the Boston harbor will appear alongside the dataset. Click the Play button and notice that the colormap disappears behind the image as we animate through time.

Step 3: Update Image location

Because the sea surface height changes over time, the contour plots disappear behind the georeferenced image. Since we want the contours to always be on top, we will update the height of the georeferenced image. Right click the georeferenced image and select Image Details. The 3D Image Geometry Details dialog appears. Notice that the Z position field defaults to 0. Update this field to -10 and click Accept.

🗕 3D Image Geometry Details 🛛 🗙
File details File name: :ttingStarted\Ocean\boston.png
Resolution:2398 x 1704
Z Position: -10 Z min:-46.9325 Z max:97.981
Misc details:
Arcept Cancel Help

The image will now be moved below the contour flooding. Select the play button again from the Plot Sidebar and notice how the sea height fluctuates but never falls below the image like before.

Understanding Salinity Stratification

The next portion will use an iso-surface to identify the bathymetry of the dataset and use slices to identify layers through the volume. If you are continuing from the previous portion, remove the georeferenced image and turn off the blanking of siglev.

Step 1: Turn off Zone Layers

On the plot sidebar, turn off all of the zone layers. This will revert the plot to an empty workspace with an orange bounding box much like when we first loaded the data.

Step 2: Show the Bathymetry

Turn on the iso-surface checkbox on the plot sidebar. This will create a surface based on a contour variable value. In this case, we want to create an iso-surface to show the bathymetry. To do this, select the button next to the iso-surface checkbox. This will bring up the Iso-surface Details dialog.

			🙋 Contour & Multi-Colorin	ig Details		>
Iso-Surface Details		×	21: siglev	× 1 2	3 4 5 6 7	8
Show group 1 1 2 3	4 5 6 7 8		Levels and Color Bands	Lines Labe	els Legend	
Definition Contour and	Shade Mesh and Vector	Animate	Contour levels			
Define iso-surfaces using:	C3: siglev	× 🔅	1:-0.95	^ Set	t Levels	
Draw iso-surfaces at:	1 Specified Value	\sim	3 : -0.85	🛃 Add	d Level 0	
Value 1:	-1		4:-0.8	Rer	move Selected	
Value 2:	-0.5	×	6 : -0.7	Contour	variable range	
Value 3:	-0.25		7:-0.65	Min: -1	1 Max	: -0
Contour variable	-1		Color map options	~		
Max:	-0		$\rightarrow \diamond \diamond \diamond$	$\diamond \diamond$	 	•
Surface generation method	Auto	\sim	Sequ	uential - Viridis		~ 🎎
Obey source zone blank	ing		Color map distribution method	i	Color cutoff	
			Banded O Continuo	ous	Cutoff above:	-0.25
			Contour values at endpo	ints	Cutoff below:	-0.75
			Min: -1		Color map adjustme	nts
			Max: -0	Reset 👻	Reverse	
	Close	Help	Approximate continue	ous	Repeat cycles: 1	
					Close	Help

Set the Iso-surface definition to C3 and enter the value of -1. Update the Contour variable in group three to **siglev** via the Contour Details dialog. See the above image for the Contour details and Iso-surface details options. Also while you have the Contour details dialog open, update the contour group two to be colored by z using the "cmocean-deep" colormap and with the colormap reversed. Then select C2 in the Contour tab of the Iso-surface details dialog. See the below image for the correct options for how to update the iso-surface contour.

	0 Contour & Multi-Coloring Details ×
Iso-Surface Details ×	5: z v 1 2 3 4 5 6 7 8
✓ Show group 1 1 2 3 4 5 6 7 8 Definition Contour and Shade Mesh and Vector Animate ✓ Show contour	Levels and Color Bands Lines Labels Legend Contour levels 1: -40 • • • 1: -40 • • • • 2: -30 • • • • 3: -20 • • • • 4: -10 • • • • 5: 0 • • • •
Contour lines by: C1: Velocity Magnitude Line color: Line thickness (%): O.1 Use lighting effect Show shade	6:10 Contour variable range 7:20 Min: -46.9325 8:30 Min: -46.9325 Color map options
Color: White Use lighting effect Effects Lighting effect Use translucency: 50	cmocean - deep Image: Color map distribution method Color cutoff Image: Color map distribution method Color cutoff Image: Cutoff above: 61.7527 Image: Contour values at endpoints Image: Cutoff below: -10.7041 Image: Max: 97.981 Reset rese
Close Help	Approximate continuous Repeat cycles: 1 Close Help

Setting the iso-surface definition will create an iso-surface of the bottom layer of the dataset. Rotating the plot around, you can see the ridges of the harbor floor through the dataset.

Step 3: Place a Slice

Next, place a slice using the Slice tool and the Slice Details dialog using Y-planes at a position of 42.33. This will place a slice right around the middle half of the Boston dataset.

Slice Details ×
Show group 1 1 2 3 4 5 6 7 8
Definition Contour Vector Other Animate
Slice location: Y-Planes \vee
Show primary slice
42.33
Show start/end slices
Start:
42.2116390228
End:
42.4884189606
Show intermediate slices Number of slices: 1
Slice through: Volume Zones \checkmark
Obey source zone blanking
Show solid plane while dragging (faster)
Extract Slices
Close Help

Remove the z legend by right-clicking the legend edge and selecting "Hide". Rotate the plot to see the slice easier like below.

Step 4: Contour the Slice by Salinity

Now that we have a slice placed in the region of interest, we want to contour by salinity to get the salinity stratification. Click the button next to the Slice tool to bring up the Slice Details dialog. Select the Contour tab and be sure the contour group is set to 1.

If the variable is not already salinity, update the variable in the Contour Details dialog.

🖲 Slice Details		×
Show group 1 1	2 3 4 5 6 7 8	
Definition Contour	Vector Other Animate	
Show contours		
Contour type:	Flood \vee	
Flood by:	C1: salinity 🗸 🔅	
Contour lines by:	C1: salinity 🛛 🗸	
Line color:	Black	
Line thickness (%):	0.1 ~	
Use lighting effect:	Gouraud \sim	
Use translucency:	10	
	Close	Help

Also be sure to turn on the slice mesh in the Other tab. This will show us the final plot. The slice displays the salinity stratification through the Boston harbor. Be sure to use the zoom and pan tools to orient the plot to your liking.

Advanced Topics

This section contains advanced topics that use Tecplot's Python API, PyTecplot, to extend the functions that the GUI provides by pairing it with Python. This allows users to increase the overall capability of Tecplot 360. Each topic will have a video containing a link to the script as well as a transcript for reference.

Vertical Transect (external video)

A vertical transect gives a curved slice extracted through time. We have created a video showing the steps to create the Vertical Transect. It is located here: www.tecplot.com/2018/10/17/vertical-transect.

Time Average (external video)

The Time Average script creates a duplicate zone with the varaibles averaged over time. It is located here: www.tecplot.com/2018/10/17/calculating-average-over-time.

Shapefile conversion (external video)

The Shapefile conversion script converts a shapefile into Tecplot .plt format. It is located here: www.tecplot.com/2018/10/17/converting-shapefiles-plt-using-pytecplot.

Next Steps

This concludes the Ocean Modeling tutorial for Tecplot 360. At this point, you may wish to dig in to the User's Manual or Help (**Help>Tecplot 360 Help**) for more details on the product in general or on specific features you've used in this tutorial.

We regularly create videos to introduce users to features of the product, not just introductory topics for new users, but also for advanced users and to highlight new features in the latest release. You can find these on our Web site at www.tecplot.com/category/tecplot-360-videos/?product=360 or on our YouTube channel at www.youtube.com/user/tecplot360.

Copyright

COPYRIGHT NOTICE

Tecplot 360[™] Getting Started Manual is for use with Tecplot 360[™] Version 2025 R1.

Copyright © 1988-2025 Tecplot, Inc. All rights reserved worldwide. Except for personal use, this manual may not be reproduced, transmitted, transcribed, stored in a retrieval system, or translated in any form, in whole or in part, without the express written permission of Tecplot, Inc., 3535 Factoria Blvd, Ste. 550; Bellevue, WA 98006 U.S.A.

The software discussed in this documentation and the documentation itself are furnished under license for utilization and duplication only according to the license terms. The copyright for the software is held by Tecplot, Inc. Documentation is provided for information only. It is subject to change without notice. It should not be interpreted as a commitment by Tecplot, Inc. Tecplot, Inc. assumes no liability or responsibility for documentation errors or inaccuracies.

Tecplot, Inc. Post Office Box 52708 Bellevue, WA 98015-2708 U.S.A. Tel:1.800.763.7005 (within the U.S. or Canada), 00 1 (425) 653-1200 (internationally) email: sales@tecplot.com, support@tecplot.com Questions, comments or concerns regarding this document: support@tecplot.com For more information, visit www.tecplot.com

THIRD PARTY SOFTWARE COPYRIGHT NOTICES

LAPACK 1992-2007 LAPACK Copyright © 1992-2007 the University of Tennessee. All rights reserved. Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met: Redistribu- tions of source code must retain the above copyright notice, this list of conditions and the following disclaimer. Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer listed in this license in the documentation and/or other materials provided with the distribution. Neither the name of the copyright holders nor the names of its contributors may be used to endorse or promote products derived from this software without specific prior written per-mission. THIS SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE COPYRIGHT OWNER OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE. The University of Tennessee. All Rights Reserved. SciPy 2001-2009 Enthought. Inc. All Rights Reserved. NumPy 2005 NumPy Developers. All Rights Reserved. VisTools and VdmTools 1992-2009 Visual Kinematics, Inc. All Rights Reserved. NCSA HDF & HDF5 (Hierarchical Data Format) Software Library and Utilities Contributors: National Center for Supercomputing Applications (NCSA) at the University of Illinois, Fortner Software, Unidata Program Center (netCDF), The Independent JPEG Group (JPEG), Jean-loup Gailly and Mark Adler (gzip), and Digital Equipment Corporation (DEC). Conditions of Redistribution: 1. Redistributions of source code must retain the above copyright notice, this list of conditions, and the following disclaimer. 2. Redistributions in binary form must reproduce the above copyright notice, this list of conditions, and the following disclaimer in the documentation and/or materials provided with the distribution. 3. In addition, redistributions of modified forms of the source or binary code must carry prominent notices stating that the original code was changed and the date of the change. 4. All publications or advertising materials mentioning features or use of this software are asked, but not required, to acknowledge that it was developed by The HDF Group and by the National Center for Supercomputing Applications at the University of Illinois at Urbana-Champaign and credit the contributors. 5. Neither the name of The HDF Group, the name of the University, nor the name of any Contributor may be used to endorse or promote products derived from this software without specific prior written permission from the University, THG, or the Contributor, respectively. DISCLAIMER: THIS SOFTWARE IS PROVIDED BY THE HDF GROUP (THG) AND THE CONTRIBUTORS "AS IS" WITH NO WARRANTY OF ANY KIND, EITHER EXPRESSED OR IMPLIED. In no event shall THG or the Contributors be liable for any damages suffered by the users arising out of the use of this software, even if advised of the pos-sibility of such damage. Copyright © 1998-2006 The Board of Trustees of the University of Illinois, Copyright © 2006-2008 The HDF Group (THG). All Rights Reserved. PNG Reference Library Copyright © 1995, 1996 Guy Eric Schalnat, Group 42, Inc., Copy- right © 1996, 1997 Andreas Dilger, Copyright © 1998, 1999 Glenn Randers-Pehrson. All Rights Reserved. Tcl 1989-1994 The Regents of the University of California. Copyright © 1994 The Australian National University. Copyright © 1994-1998 Sun Microsystems, Inc. Copyright © 1998-1999 Scriptics Corporation, All Rights Reserved, https://www.angle.com/ang Rights Reserved. W3C IPR 1995-1998 World Wide Web Consortium, (Massachusetts Institute of Technology, Institut National de Recherche en Informatique et en Automatique, Keio University). All Rights Reserved. Ppmtopict 1990 Ken Yap. All Rights Reserved. JPEG 1991-1998 Thomas G. Lane. All Rights Reserved. Dirent API for Microsoft Visual Studio (dirent.h) 2006-2006 Copyright © 2006 Toni Ronkko. Permission is hereby granted, free of charge, to any person obtaining a copy of this software and associated documentation files (the "Software"), to deal in the Software without restriction, including without limitation the rights to use, copy, modify, merge, publish, distribute, sublicense, and/or sell copies of the Software, and to permit persons to whom the Software is furnished to do so. Toni Ronkko, All Rights Reserved, ICU 1995-2009 Copyright © 1995-2009 International Business Machines Corporation and others. All rights reserved. Permission is hereby granted, free of charge, to any person obtaining a copy of this software and associated documentation files (the "Software"), to deal in the Software without restriction, including without limitation the rights to use, copy, modify, merge, publish, distribute, and/or sell copies of the Software, and to permit persons to whom the Software is furnished to do so, provided that the above copyright notice(s) and this permission notice appear in all copies of the Software and that both the above copyright notice(s) and this permission notice appear in supporting documentation. International Business Machines Corporation and others. All Rights Reserved. QsLog 2010 Copyright © 2010, Razvan Petru. All rights reserved. QsLog Copyright © 2010, Razvan Petru. All rights reserved. Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met: Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer. Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution. The name of the contributors may not be used to endorse or promote products derived from this software without specific prior written permission. THIS SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABI- LITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE COPYRIGHT HOLDER OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUEN- TIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES: LOSS OF USE, DATA, OR PROFITS: OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE. Razvan Petru. All Rights Reserved. VTK 1993-2008 Copyright © 1993-2008 Ken Martin, Will Schroeder, Bill Lorenson. All rights reserved. Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met: Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer. Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution. Neither name of Ken Martin, Will Schroeder, or Bill Lorensen nor the names of any contributors may be used to endorse or promote products derived from this software without spe- cific prior written permission. THIS SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE AUTHORS OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSE- QUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR
SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABIL- ITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE. Ken Martin, Will Schroeder, Bill Lorenson. All Rights Reserved.

TRADEMARKS

Tecplot®, Tecplot 360[™], the Tecplot 360 logo, Preplot[™], Enjoy the View[™], Master the View[™], and Framer[™] are registered trademarks or trademarks of Tecplot, Inc. in the United States and other countries.

3D Systems is a registered trademark or trademark of 3D Systems Corporation in the U.S. and/or other countries. Macintosh OS is a registered trademark or trademark of Apple, Incorporated in the U.S. and/or other countries. Reflection-X is a registered trade- mark or trademark of Attachmate Corporation in the U.S. and/or other countries. EnSight is a registered trademark or trademark of Computation Engineering International (CEI), Incorporated in the U.S. and/or other countries. EDEM is a registered trademark or trademark of DEM Solutions Ltd in the U.S. and/or other countries. Exceed 3D, Hummingbird, and Exceed are registered trademarks or trademarks of Hummingbird Limited in the U.S. and/or other countries. Konqueror is a registered trademark or trade- mark of KDE e.V. in the U.S. and/or other countries. VIP and VDB are registered trademarks or trademarks of Halliburton in the U.S. and/or other countries. ECLIPSE FrontSim is a registered trademark or trademark of Schlumberger Information Solutions (SIS) in the U.S. and/or other countries. Debian is a registered trademark or trademark of Software in the Public Interest, Incorporated in the U.S. and/or other countries. X3D is a registered trademark or trademark of Web3D Consortium in the U.S. and/or other countries. X Window System is a registered trademark or trademark of X Consortium, Incorporated in the U.S. and/or other countries. ANSYS, Fluent and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS Incorporated or its subsidiaries in the U.S. and/or other countries. PAM-CRASH is a registered trademark or trademark of ESI Group in the U.S. and/or other countries. LS-DYNA is a registered trademark or trademark of Livermore Software Technology Corporation in the U.S. and/or other countries. MSC/NASTRAN is a registered trademark or trademark of MSC Software Corporation in the U.S. and/or other countries. NASTRAN is a registered trademark or trademark of National Aeronautics Space Administration in the U.S. and/or other countries. 3DSL is a registered trademark or trademark of StreamSim Technologies, Incorporated in the U.S. and/or other countries. SDRC/IDEAS Universal is a registered trademark or trademark of UGS PLM Solutions Incorporated or its subsidiaries in the U.S. and/or other countries. Star-CCM+ is a registered trademark or trademark of CD-adapco in the U.S. and/or other countries. Reprise License Manager is a registered trademark or trade-mark of Reprise Software, Inc. in the U.S. and/or other countries. Python is a registered trademark or trademark of Python Software Foundation in the U.S. and/or other countries. Abaqus, the 3DS logo, SIMULIA and CATIA are registered trademarks or trademarks of Dassault Systèmes or its subsidiaries in the U.S. and/or other countries. The Abaqus runtime libraries are a product of Dassault Systèmes Simulia Corp., Providence, RI, USA. © Dassault Systèmes, 2007 FLOW-3D is a registered trademark or trademark of Flow Science, Incorporated in the U.S. and/or other countries. Adobe, Flash, Flash, Player, Premier and PostScript are registered trademarks or trademarks of Adobe Systems, Incorporated in the U.S. and/or other countries. AutoCAD and DXF are registered trademarks or trademarks of Autodesk, Incorporated in the U.S. and/or other countries. Ubuntu is a registered trademark or trademark of Canonical Limited in the U.S. and/or other countries. HP, LaserJet and PaintJet are registered trademarks or trademarks of Hewlett-Packard Development Company, Limited Partnership in the U.S. and/or other countries. IBM, RS/6000 and AIX are registered trademarks or trademarks of International Business Machines Corporation in the U.S. and/or other countries. Helvetica Font Family and Times Font Family are registered trademarks or trademarks of Linotype GmbH in the U.S. and/or other countries. Linux is a registered trademark or trademark of Linus Torvalds in the U.S. and/or other countries. ActiveX, Excel, Microsoft, Visual C++, Visual Studio, Windows, Windows Metafile, Windows XP, Windows Vista, Windows 2000 and PowerPoint are registered trademarks or trademarks of Microsoft Corporation in the U.S. and/or other countries. Firefox is a registered trade- mark or trademark of The Mozilla Foundation in the U.S. and/or other countries. Netscape is a registered trademark or trademark of Netscape Communications Corporation in the U.S. and/or other countries. SUSE is a registered trademark or trademark of Novell, Incorporated in the U.S. and/or other countries. Red Hat is a registered trademark or trademark of Red Hat, Incorporated in the U.S. and/or other countries. SPARC is a registered trademark or trademark of SPARC International, Incorporated in the U.S. and/or other countries. Products bearing SPARC trademarks are based on an architecture developed by Sun Microsystems, Inc. Solaris, Sun and SunRaster are registered trademarks or trademarks of Sun MicroSystems, Incorporated in the U.S. and/or other countries. Courier is a registered trademark of Monotype Imaging Incorporated in the U.S. and/or other countries. UNIX and Motif are registered trademarks or trademarks of The Open Group in the U.S. and/or other countries. Qt is a reg- istered trademark or trademark of Digia PLC in the U.S. and/or other countries. Zlib is a registered trademark or trademark of Jean-loup Gailly and Mark Adler in the U.S. and/or other countries. OpenGL is a registered trademark or trademark of Silicon Graphics, Incorporated in the U.S. and/or other countries. JPEG is a registered trademark or trademark of Thomas G. Lane in the U.S. and/or other countries. SENSOR is a registered trademark or trademark of Coats Engineering in the U.S. and/or other coun- tries. SENSOR is licensed and distributed only by Coats Engineering and by JOA Oil and Gas, a world-wide authorized reseller.All other product names mentioned herein are trademarks or registered trademarks of their respective owners.

NOTICE TO U.S. GOVERNMENT END-USERS

Use, duplication, or disclosure by the U.S. Government is subject to restrictions as set forth in subparagraphs (a) through (d) of the Commercial Computer-Restricted Rights clause at FAR 52.227-19 when applicable, or in subparagraph (c)(1)(ii) of the Rights in Technical Data and Computer Software clause at DFARS 252.227-7013, and/or in similar or successor clauses in the DOD or NASA FAR Supplement. Contractor/manufacturer is Tecplot, Inc., 3535 Factoria Blvd, Ste. 550; Bellevue, WA 98006 U.S.A.

Part Number: 21-360-02-2 Build Revision {CI_PIPELINE_ID}

Released: 10/2021