

ANSYS CFD-Post User's Guide



ANSYS, Inc.
Southpointe
275 Technology Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
<http://www.ansys.com>
(T) 724-746-3304
(F) 724-514-9494

Release 12.1
November 2009

ANSYS, Inc. is certified to ISO 9001:2008.
--

Copyright and Trademark Information

© 2009 ANSYS, Inc. All rights reserved. Unauthorized use, distribution, or duplication is prohibited.

ANSYS, ANSYS Workbench, Ansoft, AUTODYN, EKM, Engineering Knowledge Manager, CFX, FLUENT, HFSS and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license. CFX is a trademark of Sony Corporation in Japan. All other brand, product, service and feature names or trademarks are the property of their respective owners.

Disclaimer Notice

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFIDENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS. The software products and documentation are furnished by ANSYS, Inc., its subsidiaries, or affiliates under a software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, compliance with exporting laws, warranties, disclaimers, limitations of liability, and remedies, and other provisions. The software products and documentation may be used, disclosed, transferred, or copied only in accordance with the terms and conditions of that software license agreement.

ANSYS, Inc. is certified to ISO 9001:2008.

ANSYS UK Ltd. is a UL registered ISO 9001:2000 company.

U.S. Government Rights

For U.S. Government users, except as specifically granted by the ANSYS, Inc. software license agreement, the use, duplication, or disclosure by the United States Government is subject to restrictions stated in the ANSYS, Inc. software license agreement and FAR 12.212 (for non-DOD licenses).

Third-Party Software

See the legal information in the product help files for the complete Legal Notice for ANSYS proprietary software and third-party software. If you are unable to access the Legal Notice, please contact ANSYS, Inc.

Published in the U.S.A.

Table of Contents

1. Overview of CFD-Post	1
CFD-Post Features and Functionality	1
Advanced Features	1
Next Steps... ..	2
2. Starting CFD-Post	3
Starting CFD-Post with the CFX Launcher	3
Starting CFD-Post from the Command Line	3
Optional Command Line Arguments	4
Setting CFD-Post Operation Through Environment Variables	5
Running in Batch Mode	8
Example: Pressure Calculation on Multiple Files using Batch Mode	8
3. CFD-Post Graphical Interface	11
Graphical Objects	12
Creating and Editing New Objects	12
Selecting Objects	13
Object Visibility	13
Common Tree View Shortcuts	14
Details Views	14
Outline Workspace	15
Outline Tree View Shortcuts	15
Outline Details View	16
Case Branch	22
User Locations and Plots	23
Report	24
Display Properties and Defaults	36
Variables Workspace	36
Variables Tree View	36
Variables Details View	37
Variables: Example	39
Expressions Workspace	39
Expressions Tree View	40
Expressions Workspace: Expressions Details View	40
Expressions Workspace: Example	41
Calculators Workspace	42
Turbo Workspace	42
4. CFD-Post 3D Viewer	43
Object Visibility	43
3D Viewer Modes and Commands	45
3D Viewer Toolbar	45
CFD-Post 3D Viewer Shortcut Menus	46
Viewer Keys	48
Mouse Button Mapping	49
Picking Mode	50
Views and Figures	51
Creating a Figure	51
Switching to a View or Figure	52
Changing the Definition of a View or Figure	52
Deleting a Figure	52
Views	52
5. CFD-Post Workflow	55
Loading and Viewing the Solver Results	55
Qualitative Displays of Variables	55
Analysis	55
Quantitative Analysis of Results	55
Sharing the Analysis	56

Typical Workflow	56
6. CFD-Post File Menu	57
Load Results Command	57
Close Command	59
Load State Command	59
Save State Command and Save State As Command	59
Save Project Command	60
Refresh Command (ANSYS Workbench only)	60
Import Command	60
Export Command	61
Export: Options Tab	61
Export: Formatting Tab	63
Exporting Polyline Data	64
Exporting Boundary Profile / Surface Data	65
ANSYS Import/Export Commands	65
Import ANSYS CDB Surface	66
Export ANSYS Load File	66
ANSYS Import/Export Example: One-Way FSI Data Transfer	68
Report Command	68
Save Picture Command	69
Loading Recently Accessed Files	70
Quit Command	70
File Types Used and Produced by CFD-Post	70
ANSYS CFX Files	70
ANSYS Meshing Files	71
CFX-4 Dump Files	72
CFX-TASCflow Results Files	72
ANSYS Files	74
CGNS Files	75
ANSYS FLUENT Files	76
7. CFD-Post Edit Menu and Options (Preferences)	81
Undo and Redo	81
Setting Preferences with the Options Dialog	81
CFD-Post Options	82
Common Options	84
8. CFD-Post Session Menu	87
New Session Command	87
Start Recording and Stop Recording Commands	87
Play Session Command	87
9. CFD-Post Insert Menu	89
Location Submenu	89
Point Command	90
Point Cloud Command	92
Line Command	94
Plane Command	95
Volume Command	98
Isosurface Command	101
Iso Clip Command	102
Vortex Core Region	103
Surface of Revolution Command	108
Polyline Command	110
User Surface Command	112
Surface Group Command	115
Turbo Surface Command	116
Turbo Line Command	116
Vector Command	116
Vector: Geometry	117
Vector: Color	118

Vector: Symbol	118
Vector: Render	119
Vector: View	119
Contour Command	119
Contour: Geometry	119
Contour: Labels	120
Contour: Render	120
Contour: View	120
Streamline Command	121
Streamline: Geometry	121
Streamline: Color	123
Streamline: Symbol	123
Streamline: Limits	124
Streamline: Render	125
Streamline: View	125
Particle Track Command	125
Particle Track: Geometry	125
Particle Track: Color	127
Particle Track: Symbol	127
Particle Track: Render	128
Particle Track: View	128
Particle Track: Info	128
Text Command	128
Text: Definition	128
Text: Location	129
Text: Appearance	130
Coordinate Frame Command	130
Coordinate Frame: Definition	130
Legend Command	132
Default Legends	132
User-defined Legends	132
Legend: Definition Tab	132
Legend: Appearance Tab	134
Instance Transform Command	134
Default Transform Object	135
Instance Transform: Definition Tab	135
Instance Transform: Example	137
Clip Plane Command	138
Clip Plane: Geometry	138
Color Map Command	139
Variable Command	140
Expression Command	140
Table Command	140
Editing in the Table Viewer	140
Chart Command	145
Creating a Chart Object	145
Viewing a Chart	153
Example: Charting a Velocity Profile	154
Example: Comparing Differences Between Two Files	154
Comment Command	155
Figure Command	156
10. CFD-Post Tools Menu	157
Timestep Selector	157
Adding Timesteps	158
Multiple Files	158
Animation	159
Quick Animation	159
Keyframe Animation	160

Animation Dialog Box	161
Quick Editor	164
Probe	164
Function Calculator	165
Function Selection	166
Macro Calculator	167
Predefined Macros	167
User-defined Macros	171
Using the Fan Noise Macro	173
Mesh Calculator	179
Mesh Visualization Advice	180
Case Comparison	181
Calculating Difference Variables	182
Command Editor	182
11. Turbo Workspace	185
Visual Representation of Initialization Status	186
Define/Modify Global Rotation Axis	186
Turbo Initialization	186
Requirements for Initialization	186
Initialize All Components	187
Uninitializing Components	187
Individual Component Initialization (Advanced Feature)	187
Details View for Individual Component Initialization	187
Turbo View Shortcuts	190
Turbo Surface	190
Turbo Surface: Geometry	191
Turbo Surface: Common Tabs	193
Turbo Line	193
Turbo Line: Geometry	193
Turbo Line: Common Tabs	194
Turbo Plots	194
Introduction to Turbo Plots	194
Initialization Three Views	195
Blade-to-Blade Object	196
Meridional Object	196
3D View Object	197
Turbo Charts	197
Turbo Macros	203
Calculate Velocity Components	204
Calculating Cylindrical Velocity Components for Non-turbo Cases	210
12. CFX Command Language (CCL) in CFD-Post	213
Object Creation and Deletion	213
13. CFX Expression Language (CEL) in CFD-Post	215
Variables Created by CFD-Post	215

List of Figures

3.1. Sample CFD-Post Interface	11
3.2. A Sample Report, Part 1	25
3.3. A Sample Report, Part 2	26
3.4. A Sample Report, Part 3	27
4.1. Mouse Mapping using Workbench Defaults	49
4.2. Viewport Control	51
9.1. Sample Table Formatting	145
10.1. Relative position of the source and the observer	174
10.2. Example Table and Chart of Sound Pressure Levels Created by the Fan Noise Macro	178
10.3. Example Table and Chart of Sound Power Levels Created by the Fan Noise Macro	179
11.1. Sampling Point Distribution with Include Boundary Nodes Option	200
11.2. Circumferential Averaging by Length	201
11.3. Blade Aligned Linear Coordinates	202
11.4. Blade Aligned Coordinates	202
11.5. Inlet to Outlet Sample Points	203
11.6. Axial, Radial, Circumferential, and Meridional Velocity Components	206
11.7. Velocity Components in Meridional Plane	207
11.8. Streamwise, Spanwise, Circumferential, and Blade-to-Blade Velocity Components	208
11.9. Velocity Components in Blade-To-Blade Plane	209
11.10. Velocity Flow Angle Sign in Each Quadrant on the Blade-to-Blade Plane	210

List of Tables

4.1. Mouse Operations and Shortcuts	50
9.1. Shortcut Menus Toolbar	142
9.2. Table Viewer Tools Toolbar	144
9.3. Octave Band Frequencies and Weightings	151
11.1. Generated Variables	205
13.1. Variables Created by CFD-Post	216

List of Examples

6.1. A Surface Data File for CFD-Post	61
---	----

Chapter 1. Overview of CFD-Post

CFD-Post is a flexible, state-of-the-art post-processor. It is designed to allow easy visualization and quantitative analysis of the results of CFD simulations.

This chapter describes:

- [CFD-Post Features and Functionality](#) (p. 1)
- [Advanced Features](#) (p. 1)
- [Next Steps...](#) (p. 2)

CFD-Post Features and Functionality

CFD-Post has the following features:

- A graphical user interface that includes a viewer pane in which all graphical output from CFD-Post is plotted. For details, see [CFD-Post Graphical Interface](#) (p. 11) and [CFD-Post 3D Viewer](#) (p. 43).
- Support for a variety of graphical and geometric objects used to create post-processing plots, to visualize the mesh, and to define locations for quantitative calculation. For details, see [CFD-Post Insert Menu](#) (p. 89).

You can perform a variety of exact quantitative calculations over objects; for details, see [Quantitative Calculations in the Command Editor Dialog Box](#) (p. 256).

- Scalar and vector user-defined variables.
- Variable freezing (for comparison with other files).
- Post-processing capability for turbomachinery applications. For details, see [Turbo Workspace](#) (p. 185).
- Standard interactive viewer controls (rotate, zoom, pan, zoom box), multiple viewports, stored views/figures.
- Extensive reports, including charting (XY, time plots). For details, see [Report](#) (p. 24).

CFD-Post includes the following functionality:

- Reads CFX-Solver results files (*.res), CFX-Solver input files (*.def), CFX-Mesh files (*.gtm), CFX-Solver Backup results files (*.bak), CFX-Solver Error results files (*.res, *.err), ANSYS files (*.rst, *.rth, *.rmg, *.rfl, *.inn, *.brfl, *.brmg, *.brst, *.brth), ANSYS Meshing files (*.cndb, *.dsdb), CFX-4 dump files (*.d*mp*), CFX-TASCflow files (*.lun, *.grd, *.rso), CGNS files (*.cgns, *.cgs), and ANSYS FLUENT files (*.cas, *.dat, *.msh, *.cdat).

The supported file types are described in [File Types Used and Produced by CFD-Post](#) (p. 70).

Note

CFX-Solver results files are necessary to access some of the quantitative functionality that CFD-Post can provide.

- Supports transient data, including moving mesh. Node locations are repositioned based on the position for the current timestep.
- Imports/exports ANSYS data, generic data, and generic geometry.
- Supports macros through an embedded GUI (see [Macro Calculator](#) (p. 167)).
- Generates a variety of graphical and geometric objects to which you can apply colors and textures. These objects are used to create post-processing plots and to define locations for quantitative calculations. For details, see [CFD-Post Insert Menu](#) (p. 89).
- Outputs to PostScript, JPEG, PNG, various bitmap formats, and VRM, as well as animation (keyframe) and MPEG file output. For details, see [Quick Animation](#) (p. 159).

Advanced Features

CFD-Post also contains advanced features:

CFX Command Language (CCL)

CCL is the internal command language used within CFD-Post. CCL is used to create objects or perform actions. CFD-Post enables command line, session file, or state file input through the CFX Command Language (CCL). For details, see [CFX Command Language \(CCL\) in CFD-Post \(p. 213\)](#).

CFX Expression Language (CEL)

CEL is a powerful expression language used to create user-defined variables, expressions, etc. For details, see [CFX Expression Language \(CEL\) \(p. 133\)](#).

Power Syntax

Power Syntax provides integration of the Perl programming language with CCL to allow creation of advanced subroutines. For details, see [Power Syntax in ANSYS CFX \(p. 259\)](#).

Batch Mode

CFD-Post can be run in batch mode (often using a session file as the basis for a series of actions which will be executed). For details, see [Running in Batch Mode \(p. 8\)](#).

Next Steps...

Now that you have an overview of the capabilities of CFD-Post, you may want to explore:

- [Starting CFD-Post \(p. 3\)](#)
- [CFD-Post Graphical Interface \(p. 11\)](#).

Chapter 2. Starting CFD-Post

This chapter describes how to start CFD-Post and the environment variables that affect how CFD-Post operates:

- [Starting CFD-Post with the CFX Launcher \(p. 3\)](#)
- [Starting CFD-Post from the Command Line \(p. 3\)](#)
- [Setting CFD-Post Operation Through Environment Variables \(p. 5\)](#)
- [Running in Batch Mode \(p. 8\)](#)

Note

You can also start CFD-Post from other ANSYS products; for details, refer to the documentation that comes with those products.

Starting CFD-Post with the CFX Launcher

CFD-Post is installed with the ANSYS CFX Launcher, which makes it easy to run CFD-Post. The launcher enables you to:

- Set the working directory for your project.
- Launch CFD-Post and, if available, other ANSYS products.
- Access various other tools, including a command window that enables you to run ANSYS CFX utilities without having to type the path to the executable.
- Access the online help and other useful information.
- Customize the behavior of the launcher to start your own applications.

You can run the ANSYS CFX Launcher in any of the following ways:

- On Windows:
 - From the **Start** menu, select **All Programs > ANSYS 12.1 > Fluid Dynamics > CFX**.
 - In a DOS window that has its path set up correctly to run ANSYS CFX, enter: `cfx5`
If the path has not been set, you need to type the full path to the `cfx5` command; typically this is:

```
C:\Program Files\ANSYS Inc\v121\CFX\bin\cfx5.exe
```

- On UNIX, open a terminal window that has its path set up to run ANSYS CFX and enter: `cfx5`
If the path has not been set, you need to type the full path to the `cfx5` command; typically this is:

```
/usr/ansys_inc/v121/CFX/bin/cfx5.exe
```

When the launcher starts, set your working directory and click the **CFD-Post** icon.

Note

The CFX Launcher automatically searches for CFD-Post and other ANSYS products, including the license manager.

Starting CFD-Post from the Command Line

You may want to start CFD-Post from the command line rather than by clicking the appropriate button on the launcher for the following reasons:

- You may want to specify certain command-line arguments so that CFD-Post starts up in a particular configuration. For details, see [Optional Command Line Arguments \(p. 4\)](#).
- CFX contains some utilities (for example, a parameter editor) that can be run only from the command line.

- If you are having problems with CFD-Post, you may be able to get a more detailed error message by starting it from the command line than you would get if you started it from the launcher. When you start CFD-Post from the command line, any error messages produced are written to the command-line window.

To start CFD-Post from the command line, enter:

<i>Windows</i>	<CFXROOT>\bin\cfdpost or <CFXROOT>\bin\cfx5post
<i>UNIX</i>	<CFXROOT>/bin/cfdpost or <CFXROOT>/bin/cfx5post

Optional Command Line Arguments

The table that follows summarizes the most common of the optional command line arguments:

Argument	Description
-batch <filename.cse> [<results file 1>] [<results file 2> ...]	Starts CFD-Post in batch mode, running the session file you enter as an argument.
-gui	Starts CFD-Post in graphical user interface (GUI) mode (the default).
-line	Starts CFD-Post in line interface (CFD-Post command line) mode. This interface will start a command line prompt where you can type CCL commands. Typically you would create a session file using the GUI mode, then make modifications to that file as required. To start a CCL section, type “e”. When done typing CCL commands, type “e” to process the CCL. The ability to write and execute CCL is also available in GUI mode through the Command Editor. For details, see Overview of Command Actions (p. 249) .
-remote <host> -port <number> -viewerport <number>	-remote specifies a remote host to run on. -port specifies the port number for GUI-engine communication. -viewerport specifies the port for the viewer. This option also requires the host machine to be running CFD-Post with the -server option.
-report <template> [-name <report name>] [-outdir <dirname>] [<results file 1>] [<results file 2> ...]	Starts CFD-Post in batch mode, loads the results files, then produces a report and exits. Here, <template> may be one of the following: <ul style="list-style-type: none"> The word “auto”. If you use the word “auto” for a template, then CFD-Post will attempt to find the most suitable built-in template. The name of a registered template, wrapped in quotes. Register a template by running CFD-Post in GUI mode. For details, see Report Templates (p. 28). The name of a state or session file. If you provide a state file as a template, the results file indicated in the state file, if there is one, will be used when no results file name is provided on the command line.
-graphics	For UNIX only: specify the graphics system (options are ogl and mesa).

Argument	Description
Alternative form: -gr	
-local-root <path>	Specify the file path of the CFD-Post installation.
-t <file.cst> Alternative form: -state <file.cst>	Start CFD-Post and load the state file <file.cst>.
-results <file.res> Alternative form: -res <file.res>	Start CFD-Post and load the results file <file.res>.
-multiconfig single separate last Alternative form: -m single separate last	<p>Select a multi-configuration load option to control how you load a multi-configuration (.mres) file or a results file (.res) that contains a run history (that is, a file that was produced from a definition file that had its initial values specified from a results file from a previous run and saved to the results file that you are loading). Choose:</p> <ul style="list-style-type: none"> • Single Case to load all configurations of a multi-configuration run as a single case, or all of the results history from a results file that contains a run history. In either case, only one set of results will appear in the viewer, but you can use the timestep selector to move between results. This option is not fully supported. • Separate Cases to load all configurations from a multi-configuration run into separate cases. If a results file with run history is loaded, CFD-Post loads the results from this file and the results for any results file in its run history as separate cases. Each result appears as a separate entry in the tree. • Last Case to load only the last configuration of a multi-configuration results file, or only the last results from a results file that contains a run history.
-s <file.cse> Alternative form: -session <file.cse>	Start CFD-Post and load the session file <file.cse>.
-v Alternative form: -verbose	Display a summary of the currently set environment variables.
-h Alternative form: -help	Display a full list of all the possible arguments with short descriptions.

Setting CFD-Post Operation Through Environment Variables

There are a number of environment variables that can be used to change how CFD-Post behaves:

Environment Variable	Description/Usage
CFXPOST_USER_MACROS	<p>Allows user-defined macros to load at start-up.</p> <p>CFXPOST_USER_MACROS='macro1, macro2, '</p> <p>If the macros contain GUI commands, the appropriate panels will be added to the Macro Evaluator GUI.</p> <p>Example:</p> <p>CFXPOST_USER_MACROS='myMacro1.cse, /home/bob/macros/myMacro3.cse'</p>

Environment Variable	Description/Usage
CFXPOST_ZERO_THETA	<p>Allows adjusting the zero-theta location</p> <p>CFXPOST_ZERO_THETA='x y z'</p> <p>(where 'x y z' is a point not on the rotation axis)</p> <p>This will be used in turbo cases to determine at which position the Theta variable will be equal to zero. By default, CFD-Post will set Theta such that the Theta values in the first encountered domain range from zero to some positive value.</p> <p>Example:</p> <p>CFXPOST_ZERO_THETA='0.05 1.2 0.0'</p>
CFX_COLOR_MAP_FILE	<p>Allows adding custom color maps.</p> <p>CFX_COLOR_MAP_FILE=filepath</p> <p>The maps found in the file will be added to all color map selectors.</p> <p>filepath: color map file name including path</p> <p>The file can contain any number of COLOR MAP object definitions of the following format:</p> <pre>COLOR MAP: <name> Map = <level>, <r>, <g>, , . END</pre> <p>where <level> is the normalized variable value (0-1) for which to assign the rgb values (which are also normalized (0-1)).</p> <p>Example:</p> <p>This will define two maps - one going from red to white, and one similar to the rainbow map, but with the addition of white for maximum values:</p> <pre>COLOR MAP:Red to White Map = 0, 1, 0, 0, \ 1, 1, 1, 1 END COLOR MAP:My Rainbow Map = 0, 0, 0, 1, \ 0.2, 0, 1, 1, \ 0.4, 0, 1, 0, \ 0.6, 1, 1, 0, \ 0.8, 1, 0, 0, \ 1, 1, 1, 1 END</pre> <p>If the above CCL objects were saved to the colormap file '/mymaps/map07.txt', you would use this file by setting</p> <p>CFX_COLOR_MAP_FILE=' /mymaps/map07.txt '</p>
CFX_USER_IMAGE_DATA	<p>Allows you to display a custom logo image in the viewer.</p> <p>CFX_USER_IMAGE_DATA='filepath xLoc yLoc xAttach yAttach scale alphaR alphaG alphaB transparency'</p> <p>filepath: path to the image file</p> <p>Only ppm, png, jpeg, and bmp files are currently supported.</p> <p>xLoc, yLoc: horizontal and vertical location of the image in the viewer (0-1)</p> <p>xAttach: left, center, right or none.</p>

Environment Variable	Description/Usage
	<p>If set to none, xLoc is used.</p> <p>yAttach: top, center, bottom or none.</p> <p>If set to none, yLoc is used.</p> <p>scale: image size relative to viewer size (0-1)</p> <p>If set to 0, original pixels are shown regardless of the viewer size.</p> <p>alphaR, alphaG, alphaB: red/green/blue components (normalized to 0-255) of alpha (the color that will represent 100% transparency)</p> <p>transparency: overall bitmap transparency (0-1)</p> <p>Example:</p> <p>To display image myImage.ppm in the right-bottom corner, occupying 12% of the viewer size, making the pure green color represent 100% transparent, and setting the overall transparency to 60%, use:</p> <pre>CFX_USER_IMAGE_DATA= '/logos/myImage.ppm 0 0 right bottom 0.12 0 255 0 0.6'</pre>
CFX_NO_SPLASH	<p>Turns off startup splash screen.</p> <p>Example:</p> <p>On Windows, set</p> <pre>CFX_NO_SPLASH=1</pre> <p>or, on UNIX, set</p> <pre>CFX_NO_SPLASH</pre> <p>to turn off the splash screen. Deleting this environment variable turns on the splash screen.</p>
VIEWER_EYE_POINT	<p>Allows placing the viewer camera to left/right eye position.</p> <p>It can be used for composing stereo images and movies</p> <pre>VIEWER_EYE_POINT='cameraZ eyeDist mode'</pre> <p>cameraZ: Z location of the camera (must be greater than 1.0; 5.0 is optimum)</p> <p>Smaller numbers bring the camera closer to the scene (and also widen the camera angle), larger numbers move it further.</p> <p>eyeDist: distance between the eyes (0.1 is optimum)</p> <p>mode: 0 = normal, 1 = left eye, 2 = right eye, 3 = left/right eye (two viewports), 4 = right/left (two viewports)</p> <p>Example:</p> <pre>VIEWER_EYE_POINT='5.0 0.1 0'</pre>
CFX_BACKGROUND_ROTATE	<p>Applicable to spherical backgrounds only.</p> <pre>CFX_BACKGROUND_ROTATE='x y z angle'</pre> <p>x,y,z: specifies a direction vector (in the global coordinate system) about which to apply a rotation to the background image</p> <p>angle: specifies the rotation angle, in degrees, of the background image</p> <p>The rotation angle is clockwise looking in the direction of the specified direction vector.</p> <p>Example:</p> <p>If you start CFD-Post with the mountain scenery background, the background will appear upright when the Y axis is “up”. You may find that the geometry of your CFD</p>

Environment Variable	Description/Usage
	<p>mesh has its “top” side pointing in the X axis direction. You can rotate the background image so that it appears upright when the X axis is “up” by rotating the image about the Z axis by -90 degrees, as follows:</p> <pre>CFX_BACKGROUND_ROTATE='0 0 1 -90'</pre>

Running in Batch Mode

All of the functionality of CFD-Post can be accessed when running in Batch mode. When running in batch mode, a Viewer is not provided and you cannot enter commands at a command prompt.

Commands are issued via a CFD-Post session file (*.cse), the name of which is specified when executing the command to start Batch mode. The session file can be created using a text editor, or, more easily, by recording a session while running in line interface or GUI mode. You can leave a session file recording while you quit from GUI or line interface mode to write the >quit command to a session file. Alternatively, you can use a text editor to add this command to the end of a session file.

To run in Batch mode, execute the following command at the command prompt:

<i>Windows</i>	<CFXROOT>\bin\cfdpost -batch <filename.cse>
<i>UNIX</i>	<CFXROOT>/bin/cfdpost -batch <filename.cse>

You can include the name of a results file in your session file, which is described in the example below. However, you can also pass the name of a results file and a session to CFD-Post from the command line. This allows you to apply a generic session file to a series of different results files. To launch CFD-Post in batch file mode, load a results file and execute the statements in a session file using the following command:

<i>Windows</i>	<CFXROOT>\bin\cfdpost -batch <filename.cse> <filename.res>
<i>UNIX</i>	<CFXROOT>/bin/cfdpost -batch <filename.cse> <filename.res>

To load multiple files, you may list the filenames at the end. For example, fluid.res solid.rst fluid1.res

Example: Pressure Calculation on Multiple Files using Batch Mode

This example calculates the value of pressure at a point in each of three results files.

The purpose of this example is for demonstration only. You will deal with only three results files in this case, and it would be faster to produce the output by using the graphical user interface. However, these features can be useful in situations where a large number of results files need to be processed at once.

In order to carry out this procedure, you will make use of session files, power syntax and the **Command Editor** dialog box. You could use the results from any file by making the appropriate substitutions in the following example.

1. Place three results files in your working directory. For this example, in all three results files the location 0, 0, 0 must be in the solution domain.
2. Create a session file based on a results file. This example uses <CFXROOT>/examples/StaticMixer_001.res.
 - a. Start CFD-Post and select **File > Load Results**. Select the static mixer results file StaticMixer_001.res and click **Open**.
 - b. Select **Session > New Session** from the main menu.
For details, see [New Session Command \(p. 87\)](#).
 - c. Enter batchtest.cse as the session file name and click **Save**.
 - d. Select **Session > Start Recording** from the main menu to begin recording the session file commands.

- e. Select **Insert > Location > Point** and accept the default name Point 1.
- f. Click **Apply** to create the point at location 0, 0, 0.
You will now use Power Syntax to find the value of pressure at Point 1, and print it to the command line. In addition to printing the value of pressure, it would be useful to know the name of the results file. You will make use of the DATA READER object to find the name of the current results file.
- g. Select **Tools > Command Editor**.
- h. Enter the following into the command window:

```
! $filePath = getValue("DATA READER","Current Results File");
! $pressureVal = probe("Pressure","Point 1");
! print "\nFor $filePath, Pressure at Point 1 is $pressureVal\n";
```

Note

If you copy the text above into the Command Editor, ensure that the exclamation points are at the beginning of lines.

- i. Click **Process** to process the commands.
 - j. Check the terminal window to make sure the command worked as desired.
 - k. Select **Session > Stop Recording** from the main menu to stop recording the session file.
3. This completes the first part of the example. You may want to close down CFD-Post at this time.
 4. You can now run the session file on any number of results files using the command:

```
<CFXROOT>/bin/cfdpost -batch batchtest.cse <resfile>
```

where <resfile> is the name of your results file.

To load multiple files, you can list the filenames at the end of the command. For example:

```
cfdpost -batch batchtest.cse StaticMixer_001.res fluid.res solid.rst
```

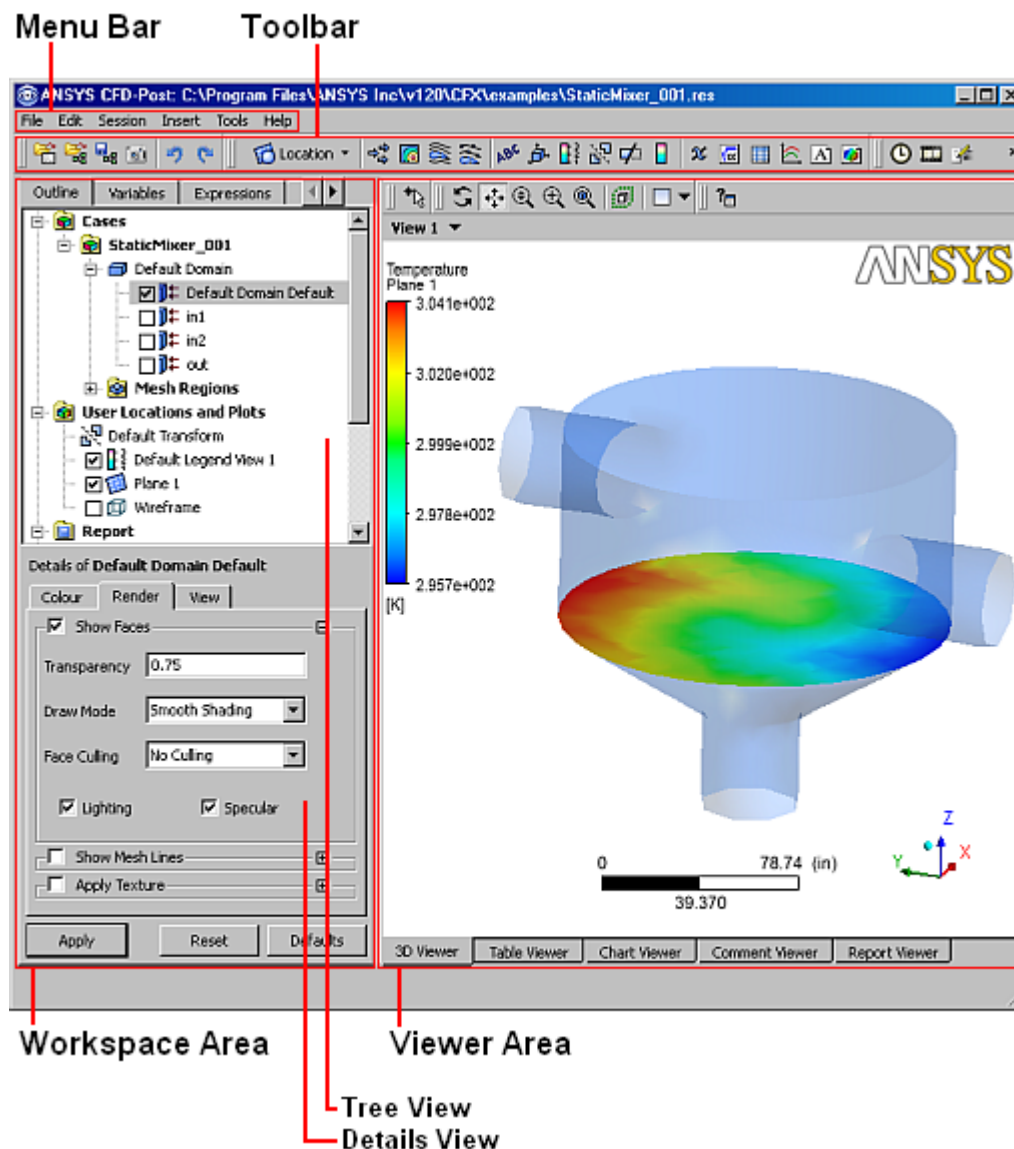
For a very large number of results files, a simple script can be used to pass filenames as command line arguments. As an example, this C shell script would pass arguments as results file names to the CFD-Post command line:

```
#!/bin/csh
foreach file ($argv)
<CFXROOT>/cfdpost -batch batchtest.cse $file
end
```


Chapter 3. CFD-Post Graphical Interface

The CFD-Post interface contains the following areas: the menu bar, the toolbar, the workspace area, and the viewer area.

Figure 3.1. Sample CFD-Post Interface



When CFD-Post starts, the **Outline** workspace area and the 3D Viewer are displayed. The top area of the **Outline** workspace is the *tree view* and the bottom area is the *details view* (the details view is populated only after you edit an item, as described in [Details Views \(p. 14\)](#)).

The viewer displays an outline of the geometry and other graphic objects. In addition to the mouse, you can use icons from the viewer toolbar (along the top of the viewer) to manipulate the view.

The width of the workspace can be adjusted by dragging its right border to the left or right. The width of the viewer is updated to accommodate the new size of the workspace. The dividing line between the tree view and details view can be dragged vertically to re-size the windows. You may want to do this if the details view contains a large amount of information.

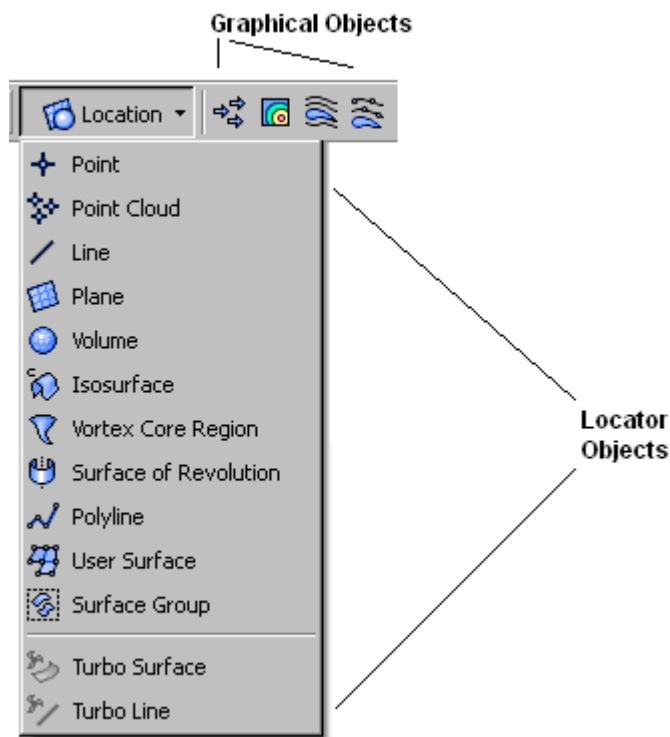
The remainder of this chapter describes:

- [Graphical Objects \(p. 12\)](#)

- [Common Tree View Shortcuts](#) (p. 14)
- [Details Views](#) (p. 14)
- [Outline Workspace](#) (p. 15)
- [Variables Workspace](#) (p. 36)
- [Expressions Workspace](#) (p. 39)
- [Calculators Workspace](#) (p. 42)
- [Turbo Workspace](#) (p. 42)

Graphical Objects

The ANSYS CFD-Post processor supports a variety of *graphical objects* and *locator objects* that are used to create post-processing plots and to define locations for quantitative calculation. In [Figure 3.1, “Sample CFD-Post Interface”](#) (p. 11) a plane has been inserted and configured to display temperature.



The details of all the possible objects and associated parameters that can be defined in CFD-Post are described in the CFD-Post .ccl file available with the installation.

Creating and Editing New Objects

New objects can be created and edited by:

- Right-clicking an object in the tree view area.
- Selecting a command from the **Insert** menu
For details, see [CFD-Post Insert Menu](#) (p. 89).
- Right-clicking in the viewer (not applicable for all object types). In many cases, this is the most convenient way to create locators (such as planes). For details, see [CFD-Post 3D Viewer Shortcut Menus](#) (p. 46).

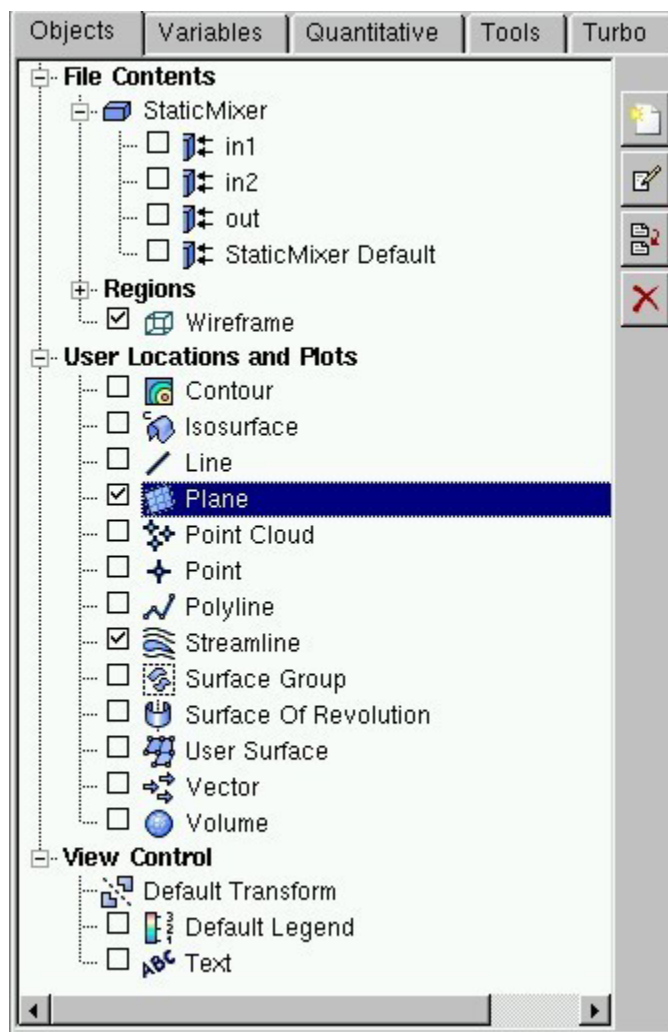
Selecting Objects

You can select multiple objects by holding down the **Ctrl** or **Shift** key as you select each object. Subsequently right-clicking any of the selected objects allows you to perform commands that apply to all of the selected objects (such as **Show** and **Hide**).

Object Visibility

In the **Outline** and **Turbo** workspaces, some objects have a visibility check box beside them. In the graphic that follows, the **Wireframe**, **Plane**, and **Streamline** objects are set to be visible in the viewer.

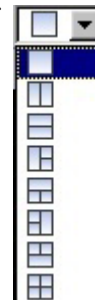
You can change the visibility settings for a group of objects by first selecting a subgroup of objects (using mouse clicks while holding down **Ctrl** (for multiple, independent selections) or **Shift** (to drag over a range of selections)), and then right-clicking on the group and using the appropriate shortcut menu command (for example, **Hide** or **Show**). For details, see [CFD-Post 3D Viewer Shortcut Menus \(p. 46\)](#).



When multiple viewports are used, the state of each check box is maintained separately for each viewport.

Tip

You control the number and layout of viewports with the *viewport* icon in the viewer's toolbar.



Common Tree View Shortcuts


Common tree view shortcuts, accessible by right-clicking an object in the tree view, are listed in the table below:

Command	Description
New (or Insert)	Inserts an object.
Edit	Edits the selected object in a details view.
Edit in Command Editor	Edits the selected object in the Command Editor dialog box.
Duplicate	Creates a new object of the same type, with the same settings, as the selected object.
Delete	Deletes the selected objects.

Details Views

Details view is a generic term used to describe an editor for the settings of a CCL object. A details view for a particular kind of object (such as a plane object) may be referenced by the name of the type of object being edited, followed by the word “details view” (for example, for the Wireframe object, the **Wireframe** details view).

A details view appears after any of the following actions:

- Double-clicking an object in the tree view
- Right-clicking an object in the tree view and selecting **Edit** from the shortcut menu
- Highlighting an object in the tree view and clicking *Edit*  from the shortcut menu
- Clicking **OK** on a dialog box used to begin the creation of a new object
- Clicking an object in the **Viewer** when in pick mode
- Right-clicking on an object in the **Viewer** and selecting **Edit** from the shortcut menu

Details View Controls

Details views contains the following buttons:

- **Apply** applies the information contained within all the tabs of an editor.
- **OK** is the same as **Apply**, except that the editor automatically closes.
- **Cancel** and **Close** both close the editor without applying or saving any changes.
- **Reset** returns the settings for the object to those stored in the database for all the tabs. The settings are stored in the database each time the **Apply** button is clicked.
- **Defaults** restores the system default settings for all the tabs of the edited object.

Outline Workspace

The **Outline** workspace consists of objects in a tree view and a Details view where you can edit those objects; the tree view appears in the top half of that pane. The Details view appears beneath the tree view. For details, see [Details Views \(p. 14\)](#).

You access the **Outline** workspace by clicking the **Outline** tab.

After starting CFD-Post and loading a results file, several special objects will exist in the **Outline** workspace. All of these special objects can have some of their properties edited, but the objects themselves cannot be created or deleted using CFD-Post (without using CCL commands). These objects are described in the following sections:

- [Case Branch \(p. 22\)](#)
- [User Locations and Plots \(p. 23\)](#)
- [Report \(p. 24\)](#)

Objects that do not exist after loading a results file are described in [CFD-Post Insert Menu \(p. 89\)](#).

Shortcuts available to the tree view are described in [Outline Tree View Shortcuts \(p. 15\)](#).

Some of the settings and buttons in a details view of the **Outline** workspace are common for different object types; these are described in [Outline Details View \(p. 16\)](#).

Outline Tree View Shortcuts

Shortcuts for editing and manipulating objects are accessible by right-clicking an object in the tree view. The following table shows commands that are specific to the tree view.

Command	Description
New (or Insert)	Inserts an object.
Edit	Edits the selected object in a details view.
Edit in Command Editor	Edits the selected object in the Command Editor dialog box.
Duplicate	Creates a new object of the same type, with the same settings, as the selected object.
Delete	Deletes the selected objects.
Show	Makes the selected objects visible in the viewer.
Hide	Makes the selected objects invisible in the viewer.
Hide All	Makes all objects, except the wireframe object, invisible in the viewer.
Refresh Preview	Refreshes the report. For details, see Refreshing the Report (p. 35) .
Load '<template>' template	Loads the registered template having the name indicated by <template>. For details, see Report Templates (p. 28) .
Report Templates	Allows you to select a report template. For details, see Report Templates (p. 28) .
Add to Report	Sets the selected report objects to appear in the report the next time the report is generated.
Remove from Report	Sets the selected report objects to not appear in the report the next time the report is generated.
Add All to Report	Sets all report objects to appear in the report the next time the report is generated.
Move Up	Moves the selected objects up one level in the report so as to appear closer to the beginning of the report in relation to the other report objects.
Move Down	Moves the selected objects down one level in the report so as to appear closer to the end of the report in relation to the other report objects.
Show in Separate Window	Displays the selected chart in its own window.

Outline Details View


A details view appears at the bottom of the **Outline** workspace when you open an object in the tree view for editing (which is described in [Details Views \(p. 14\)](#)). Some **Outline** details views have tabs in common.

Geometry Details Tab

The definition of geometry is unique for each graphic object. The basic procedure for geometry setup involves defining the size and location of the object, with most other properties being object specific. For details, see [CFD-Post Insert Menu \(p. 89\)](#).

Selecting Domains

For many objects you can select the **Domains** in which the object should exist.

To select the domain, pick a domain name from the drop-down **Domains** menu. To define the object in more than one domain, you can type in the names of the domains separated by commas or click the *Location editor*  icon.

When more than one domain has been used, most plotting functions can be applied to the entire computational domain, or to a specific named domain.

Color Details Tab


The **Color** details tab controls the color of graphic objects in the **Viewer**. The coloring can be either constant or based on a variable, and can be selected from the **Mode** pull-down menu.

The **Color** details tab enables you to view and/or edit the properties of the tree view's `Display Properties` and `Defaults > Color Maps` definitions; `System` colors are view-only but `Custom` colors can be edited.

The default color map appears in **bold** text.


Mode: Constant

To specify a single color for an object, select the `Constant` option from the **Mode** pull-down menu.

To choose a color, click the *Color selector*  icon to the right of the **Color** option and select one of the available colors. Alternatively, click on the color bar itself to cycle through ten common colors quickly. Use the left and right mouse buttons to cycle in opposite directions.

Mode: Variable and Use Plot Variable

You may want to plot a variable on an object, such as temperature on a plane. To do this, you should select the `Variable` option from the **Mode** pull-down menu. This displays additional options, including the `Variable` pull-down menu where you can choose the variable you want to plot.

The list of variables contains `User Level 1` variables. For a full list of variables, click *More variables* .

For isosurface and vector plots, the **Use Plot Variable** option is also available. This sets the variable used to color your plot to the same as that used to define it.

Range

Range allows you to plot using the `Global`, `Local` or `User Specified` range of a variable. This affects the variation of color used when plotting the object in the **Viewer**. The lowest values of a variable in the selected range are shown in blue in the **Viewer**; the highest values are shown in red.

- The `Global` range option uses the variable values from the results in all domains (regardless of the domains selected on the **Geometry** tab) and all timesteps (when applicable) to determine the minimum and maximum values.
- The `Local` range option uses only the variable values on the current object at the current timestep to set the maximum and minimum range values. This option is useful to use the full color range on an object.
- The `User Specified` range option allows you to specify your own maximum and minimum range values. You can use this to concentrate the full color range into a specific variable range.

Hybrid/Conservative

Select whether the object you want to plot will be based on hybrid or conservative values. For details, see [Hybrid and Conservative Variable Values \(p. 165\)](#).

Color Scale

The color scale can be mapped using a linear or logarithmic scale. For a linear scale, the color map is divided evenly over the whole variable range. For a logarithmic scale, the color scale is plotted against a log scale of the variable values.

Color Map


The colors along the color bar in the legend are specified by this option.

- **Rainbow** uses a standard mapping from blue (minimum) to red (maximum).
- **Rainbow 6** uses an extension of the standard **Rainbow** map from blue (minimum) to magenta (maximum).
- **Zebra** creates six contours over the specified range of values. Between each pair of contour lines, the color scale varies from white (minimum) to black and through to white (maximum) again. The **Zebra** map can be used to show areas where the gradient of a variable changes most rapidly with a higher resolution (five times greater) than the standard **Greyscale** color map
- **Greyscale** changes in color from black (minimum) to white (maximum).

Accessing the CFD-Post Color Map Editor

The CFD-Post Color Map editor enables you to define and name a set of colors (a color map) that you can then apply to an object by using that object's **Color** tab.

To access the Color Map editor, you can:

- From the menu bar, select **Insert > Color Map**.
- From an object's **Color** tab (when the **Mode** is set to **Variable**), click the  icon beside the **Color Map** field and select **Insert**, **Edit**, or **Duplicate**.
- From the **Outline** view under **Display Properties and Defaults > Color Maps**, select a **System** or **Custom** color map, right-click, and select either **Insert**, **Edit**, or **Duplicate**.

A **System** color map can be set as the default, but otherwise cannot be edited directly. However, you can duplicate a **System** color map and use that as a basis for a **Custom** color map (which will be completely editable).


Depending on how you access the Color Map editor, it may appear as a dialog or as a **Details** pane.

To learn how to use the Color Map editor, see [Color Map Command \(p. 139\)](#).

Undefined Color

Undef. Color is the color that is used in areas where the results cannot be plotted because the variable is not defined or variable values do not exist.

For example, a section of an object that lies outside the computational domain will not have any variable value.

Clicking the *Color selector*  icon to the right of this box allows you to change the undefined color. Alternatively, click on the color bar itself to cycle through ten common colors quickly. Values written to the results file as zeros are colored as such and will not be undefined. For example, consider results files containing Yplus/Wall Shear values away from a wall boundary.

Symbol Details Tab

Enables you to configure the appearance of a symbol.

Symbol

Selects the style of the symbol to be displayed.

Symbol size

Specifies the size of the symbol where 0 is the smallest and 10 is the largest.

Render Details Tab

The appearance of the **Render** tab depends on what type of object is plotted in the viewer.

Show Faces

The top half of the tab controls the **Show Faces** options. This toggle is selected by default and draws the faces of the elements that make up an object. The faces are colored using the settings on the **Color** tab. When **Show Faces** is selected, the following options can be set:

Show Faces: Transparency

Set the Transparency value for the faces of the object by entering a value between 0 (opaque) and 1 (transparent), or use the embedded slider.

Show Faces: Draw Mode

Shading properties can be None, Flat Shading, or Smooth Shading.

- **Flat Shading:** Each element is colored a constant color. Color interpolation is not used across or between elements.
- **Smooth Shading:** Color interpolation is applied that results in color variation across an element based on the color of surrounding elements.
- **Draw as Lines:** This option draws lines but uses the color settings defined on the **Color** tab. The color settings that are applied to the lines use Smooth Shading. **You must use the Screen Capture feature to print an image or create an animation containing lines drawn using this option.** (See [Animation Options Dialog Box \(p. 162\)](#) for details on the **Screen Capture** feature.)

Note

Optionally, you can edit the face you want to show as lines to disable **Show Faces** and to enable **Show Mesh Lines**. The resulting display will be similar to **Draw as Lines**, but in constant-color mode only.

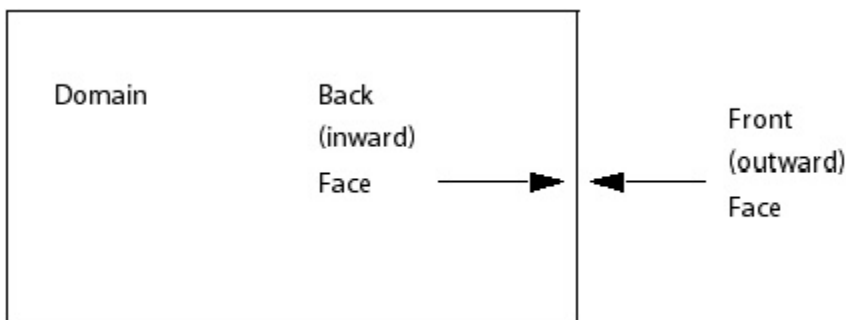
- **Draw as Points:** This option draws points at the intersection of each line, using the color scheme defined on the **Color** tab. **You must use the Screen Capture feature to print an image containing this option.**

Show Faces: Face Culling

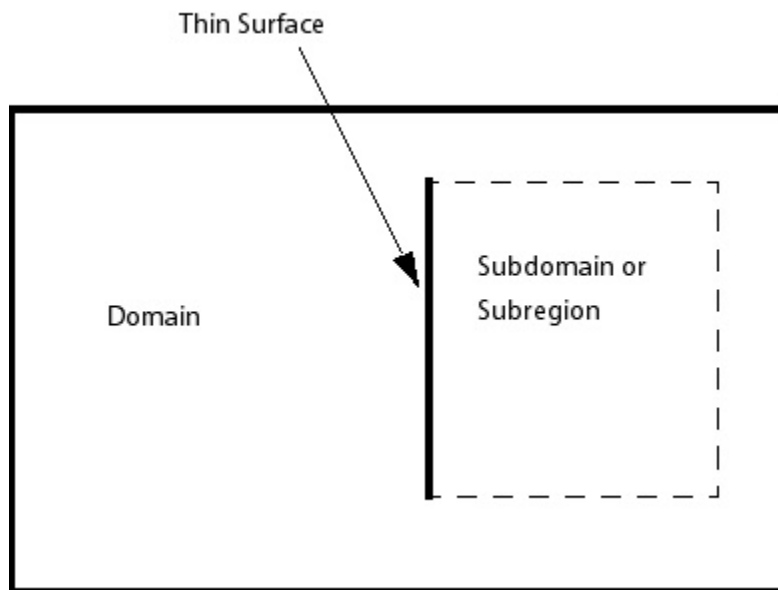
Note

Face Culling affects only printouts performed using **Screen Capture** method.

Toggle **Face Culling** (removal) of the front or back faces of the polygons that form the graphic object. This allows you to clear visibility for element faces of objects that either face outwards (front) or inwards (back). Domain boundaries always have a normal vector that points outwards from the domain. The two sides of a thin surface therefore have normal vectors that point towards each other.



- Selecting **Front** clears visibility for all outward-facing element faces. This would, for example, clear visibility for one side of a plane or the outward facing elements of a cylinder locator. When applied to a volume object, the first layer of element faces that point outwards are rendered invisible. You will also generally need to use face culling when viewing values on thin surface boundaries, which are defined using a wall boundary on two 2D regions that occupy the same spatial location.



If you want to plot a variable on the a thin surface, you will need to select **Front** Face culling for both 2D regions that make up the thin surface to view the plot correctly. As shown by the two previous diagrams, viewing only the back faces means that the data for the inward facing surfaces is always visible.

- Selecting **Back** clears visibility for inward-facing element faces (the faces on the opposite side to the normal vector). When applied to volume objects, the effect of back culling is not always visible in the viewer because the object elements that face outward obscure the culled faces. It can, however, reduce the render time when further actions are performed on the object. The effect of this would be most noticeable for large volume objects. In the same way as for **Front Face** culling, it clears visibility of one side of surface locators.
- **No culling** shows element faces when viewed from either side.

Show Faces: Lighting

When selected, surfaces appear realistically shaded to emphasize shape. Clear this check box when you want to see variable colors that match the legend colors.

Show Faces: Specular Lighting

When selected, objects appear shiny.

Show Mesh Lines

Show Mesh Lines can be selected to show the edge lines of the mesh elements in an object. When selected, the following options are available:

Show Mesh Lines: Edge Angle

This can be altered to enable the same editing features on each object as for the whole wireframe. For details, see [Wireframe \(p. 23\)](#).

Show Mesh Lines: Line Width

Set the line width by entering the width of the line in pixels. Set the value between 1 and 11; you can use the graduated arrows, the embedded slider, or type in a value.


Show Mesh Lines: Color Mode

The line color can be set as **Default** or **User Specified**.

Default sets the line to CFD-Post default color scheme (set using **Options** dialog box in **Edit** menu).

User Specified allows you to pick the color. For details, see [Show Mesh Lines: Line Color \(p. 20\)](#).

Show Mesh Lines: Line Color

Line color can be changed by clicking the *Color selector*  icon to the right of the **Color** box and selecting a color. Alternatively, click on the color bar itself to cycle through ten common colors quickly. Use the left and right mouse buttons to cycle in opposite directions.

Apply Texture

Textures are images that are pasted (mapped) onto the faces of an object. They are used to make an object look like it is made of a certain type of material, or to add special labels, logos, or other custom markings on an object.

Apply Texture: Predefined Textures

To use a predefined texture map, set **Type** to **Predefined** and set **Texture** to the desired material type using the drop-down menu. Options include brick and various types of metal.

Enable **Blend** to blend the texture with the object color specified from the **Color** tab.

Blend allows the colors of the texture to combine with the basic color of the object. For example, if a white object is given the texture **Metal**, the object looks like silver. If the basic color of the object is orange, the object looks like copper. With the **Blend** feature turned off, the basic color of the object has no effect and colors depends only on the texture.

Apply Texture: Custom Textures

To use a custom texture map, set **Type** to **Custom**.

An **Image File** (either a bitmap or ppm file) must be specified. The dimensions of the image, in pixels, should be powers of two. If the texture image has a number of rows not equal to a power of two, some rows are removed (with an even distribution) until the number remaining is a power of two. The same is true for the number of columns. For example, an image with dimensions 65 by 130 is reduced to an image 64 by 128 before it is applied (the file will not be changed, though).

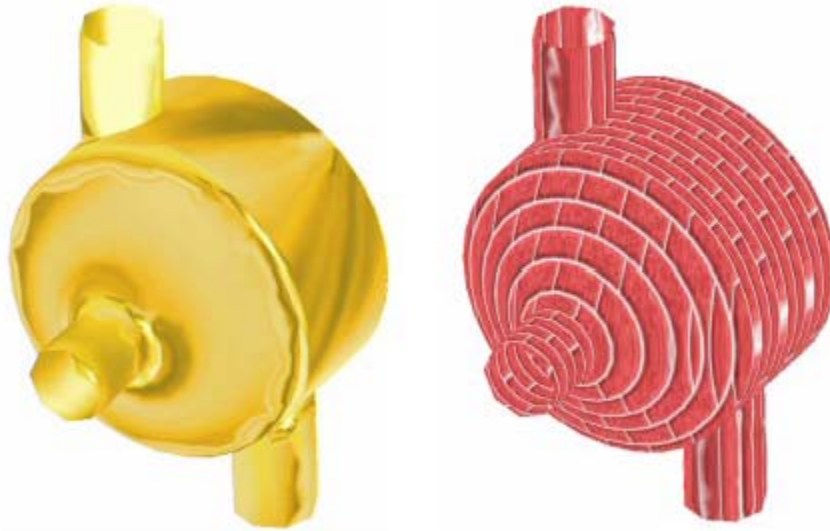
There are two basic kinds of texture mapping available; textures can either move with the object, as if painted on, or textures can “slide” across objects, producing a “shiny metal” effect. The latter kind of texture mapping is activated by turning on the **Sphere-Map** feature.

When **Sphere-Map** is not used, the following additional features apply:

- **Tile**: Causes the texture image to be repeated.
- **Position**: Controls the position of the mapped image relative to the object.
- **Direction**: Controls the direction in which the texture is stamped on the object.
The texture appears undistorted when the object is viewed in this direction.
- **Scale**: Controls the size of the mapped texture relative to the object.
- **Angle**: Controls the texture image orientation about the axis specified by **Direction**.

Apply Texture: Texture Examples

The next two figures show an object with a gold texture map and an object with a brick texture map.



Note that the brick pattern was applied in the direction of the Y axis, which is roughly going from the lower-left corner to the upper-right corner of the figure. The texture is applied to all faces of the object (locator) ignoring the Y coordinate. This results in the texture becoming smeared in the specified **Direction**.

To avoid this, textures can be applied to smaller locators (that is, ones that cover only a portion of the whole object). The **Direction** setting can then be specified using a direction approximately perpendicular to each of the smaller surfaces. Smaller locators can be found in the tree view (for example, under **Regions**).

View Details Tab

The **View** tab is used for creating or applying predefined Instance Transforms for a wide variety of objects. For details on Instance Transforms, see [Instance Transform Command \(p. 134\)](#).

Apply Rotation Check Box

Method, Axis, From, To

These settings specify an axis of rotation. For details, see [Method \(p. 135\)](#).

Angle

The **Angle** setting specifies the angle of rotation about the axis. For details, see [Angle \(p. 136\)](#).

Apply Translation Check Box

Select the **Apply Translation** check box to move the object in the Viewer. For details, see [Apply Translation Check Box \(p. 136\)](#).

Apply Reflection/Mirroring Check Box

Select the **Apply Reflection/Mirroring** check box to mirror an object in the Viewer. For details, see [Apply Reflection/Mirroring Check Box \(p. 136\)](#).

Apply Scale Check Box

Select the **Scale** check box to set values in the X, Y, and Z directions to scale the object about the origin. For example, entering [2 , 1 , 1] would stretch the object to double its size in the X axis direction.

Apply Instancing Transform Check Box

The **Transform** setting specifies a predefined Instancing Transform.

Case Branch

The tree view contains one case branch for each loaded results file or mesh file. The case name is the name of the results file, less the extension.

Tip

To see the full path to the case file, hover the mouse pointer over the case name.

A case branch contains all domains, subdomains, boundaries, and **Mesh Regions** contained in the corresponding results file.

Double-clicking on the case branch name displays the **View** tab in the details view. Select the **Apply Translation** check box to move the object in the Viewer. For details, see [Apply Translation Check Box \(p. 136\)](#).

Double-clicking a domain name displays the domain details view.

Domain Details View

A domain object represents each domain loaded from the results file. You can edit the properties of domains by right-clicking on them in the tree view and selecting **Edit**.

Instancing Details Tab

Instancing affects the display of objects; it allows multiple copies of objects to be displayed with a specified geometric transformation describing the relative positions. For example, a row of turbine blades can be visualized by applying instancing to an object that shows a single blade.

The **Instancing** tab for a domain is the same as the **Instancing** tab for a turbo component. For details, see:

- [Instancing Tab \(p. 189\)](#)
- [Instance Transform Command \(p. 134\)](#).

Instancing information for a domain applies to any instance transform that has the **Instancing Info From Domain** option selected. This applies to the default transform; therefore, any object associated with a domain is affected by a change to the domain's instancing information.

Info Details Tab

Certain information that CFD-Post reads from the results file is displayed on the Info panel. The units that are shown beside some quantities are the default CFD-Post units, which you can change by selecting **Edit > Options** from the main menu bar.

Boundary and Subdomain

All boundaries and subdomains associated with a domain are listed under the domain.

The **Boundary** and **Subdomain** object types are defined during pre-processing and created in CFD-Post when a file is loaded. You cannot create additional boundary or subdomain objects during post-processing, or delete the existing ones.

A boundary object exists for each boundary condition defined in the results file. Any mesh regions that were not specifically assigned a boundary condition appear in a default boundary object for each domain.

If you have a complex geometry where many mesh regions are assigned to the default boundary conditions, it may be worth defining named boundary conditions for some of the regions when they are created, even though you still apply the default wall boundary condition to these named regions. You will then have convenient boundary objects created in CFD-Post upon which you can view variables when you come to view the results.

Subdomain objects exist only if **Subdomains** are defined during pre-processing.

You can edit both the **Color** and **Render** properties of **Boundary** and **Subdomain** objects. For details, see

- [Color Details Tab \(p. 16\)](#)
- [Render Details Tab \(p. 18\)](#).

Mesh Regions

All of the primitive and composite region names are listed under **Mesh Regions**.

User Locations and Plots

The following objects appear under **User Locations and Plots**:

- **User-defined locators**
You can define a variety of locators, such as points, lines, planes, and volumes; for details, see [Location Submenu \(p. 89\)](#).
- **Transforms**
Instance transforms are used to specify how an object should be drawn multiple times. CFD-Post can create instance transforms using rotation, translation, and reflection; for details, see [Instance Transform Command \(p. 134\)](#).
- **Legends**
Legends can be displayed in the viewer to show the relationship between colors and values for the locators you insert; for details, see [Legend Command \(p. 132\)](#).
- **Wireframe**
The **Wireframe** object contains the surface mesh for your geometry; for details, see [Wireframe \(p. 23\)](#).

Wireframe

The **Wireframe** object contains the surface mesh for your geometry and is created as a default object when you load a file into CFD-Post. You can change how much of the surface mesh you want to see by altering the Edge Angle (see the following section), as well as the line thickness and color.

You toggle the visibility of the wireframe on and off by clicking on the **Wireframe** check box in the **Outline** tree view. To change the way the wireframe displays, double-click **Wireframe**.

Note

You cannot create additional **Wireframe** objects.

Wireframe: Definition Tab

The **Definition** tab contains the settings listed below. After making changes, click **Apply** to make those changes visible.

- **Domains** controls on which domains the wireframe is displayed. For a case with immersed solids, the setting **All Domains** refers to all domains *except* immersed solids; for **Wireframe**, the default setting is **All Domains, All Immersed Solids**.
- **Show Surface Mesh** controls whether you see edges and surfaces, or only edges, when the wireframe is visible.
- **Edge Angle** determines how much of the wireframe is drawn. The *edge angle* is the angle between one edge of a mesh face and its neighboring face. Setting an edge angle in CFD-Post defines a minimum angle for drawing parts of the surface mesh. For example, if an edge angle of 30 degrees is chosen, any edges shared by faces with an angle between them of 30 degrees or more is drawn. 30 degrees is the default edge angle; if you want to see more of the wireframe, reduce the edge angle. To change the wireframe's edge angle, set **Edge Angle** to a new value.
- **Color Mode** determines the color of the lines in the wireframe. To change the wireframe's line color, set **Color Mode** to **User Specified** and click on the color bar to select a new color.
- **Line Width** determines the thickness of the lines in the wireframe. To change the wireframe's line width, set **Line Width** to a new value.

Wireframe: View Tab

The **View** tab is used for creating or applying predefined Instance Transforms for a wide variety of objects.

For details on changing the view settings, see [View Details Tab \(p. 21\)](#).

Report

CFD-Post automatically makes available a report of the output of your simulation. You can control the contents of the report in the **Outline** workspace, see the available sections of the report in the **Report Viewer**, add new sections in the **Comment Viewer**, and publish the report in HTML or in plain text form.

Here is an example of a report that uses the generic template; if you have a RES file loaded in CFD-Post, you can see a similar report by clicking on the **Report Viewer** tab at the bottom of the Viewer area.

Note


The sample report shown in [Figure 3.2, “A Sample Report, Part 1” \(p. 25\)](#), [Figure 3.3, “A Sample Report, Part 2” \(p. 26\)](#), and [Figure 3.4, “A Sample Report, Part 3” \(p. 27\)](#) is taken from a Report.html file like the one that you generate when you click the **Publish** button  **Publish**.

Figure 3.2. A Sample Report, Part 1



Figure 3.3. A Sample Report, Part 2

2. Mesh Report

Table 2. Mesh Information for Buoyancy2D_001

Domain	Nodes	Elements
Buoyancy2D	3322	1560

3. Physics Report

Table 3. Domain Physics for Buoyancy2D_001

Domain - Buoyancy2D	
Type	Fluid
Location	Primitive 3D
<i>Materials</i>	
Air at 25 C	
Fluid Definition	Material Library
Morphology	Continuous Fluid
<i>Settings</i>	
Buoyancy Model	Buoyant
Buoyancy Reference Temperature	40.000 [C]
Gravity X Component	-4.900 [m s ⁻²]
Gravity Y Component	-8.500 [m s ⁻²]
Gravity Z Component	0.000 [m s ⁻²]
Buoyancy Reference Location	Automatic
Domain Motion	Stationary
Reference Pressure	0.000 [Pa]
Heat Transfer Model	Thermal Energy
Turbulence Model	Laminar

Figure 3.4. A Sample Report, Part 3

Table 4. Boundary Physics for Buoyancy2D_001		
Domain	Boundaries	
Buoyancy2D	Boundary - Buoyancy2D Default	
	Type	WALL
	Location	Primitive 2D A, Primitive 2D B
	<i>Settings</i>	
	Heat Transfer	Adiabatic
	Wall Influence On Flow	No Slip
	Boundary - SymP	
	Type	SYMMETRY
	Location	SYMMET1, SYMMET2
	Boundary - cold	
	Type	WALL
	Location	WALLCOLD
	<i>Settings</i>	
	Heat Transfer	Fixed Temperature
	Fixed Temperature	5.000 [C]
	Wall Influence On Flow	No Slip
	Boundary - hot	
	Type	WALL
	Location	WALLHOT
	<i>Settings</i>	
	Heat Transfer	Fixed Temperature
	Fixed Temperature	75.000 [C]
	Wall Influence On Flow	No Slip

A report is defined by the **Report** object and the objects stored under it. The **Report** object, like other objects, can be saved to, and restored from, a state file. For details, see [File Types Used and Produced by CFD-Post \(p. 70\)](#). Only one **Report** object exists in a CFD-Post session.

Omitting Default Report Sections

You can remove major sections of the report by expanding the **Report** section in the **Outline** tree view and clearing the check box beside the section to be omitted. To see the results of such operations, right-click **Report** and select **Refresh Preview**.

Note

The **Title Page** option controls the inclusion of the logo, title, dates, and Table of Contents sections.

Changing the Default Report Sections

To change a default section, click on the section name in the Outline tree view and click **Edit**.

For the Title Page, you can:

- Add a new logo (JPG or PNG only)
- Remove the ANSYS logo


- Change the report's title
- Add the author's name
- Control the display of the date and the table of contents.

After making changes, click **Apply** and **Refresh Preview** to see the results in the Report Viewer.

The other report pages control detailed information about the mesh, physics, and solution. Again, after making changes, click **Apply** and **Refresh Preview** to see the results in the **Report Viewer**.

Adding New Sections to a Report

You can add new sections by using the **Comment Viewer** tab (at the bottom of the viewer pane):

1. In the **Comment Viewer** toolbar, click *New Comment*  to ready the **Comment Viewer** for editing.
2. Add a title for your new section in the **Heading** field.
3. Set the level of the heading in the **Level** field (use "1" for new sections; "2" for subsections, and so on).
4. Type your text in the large, white text-entry field (HTML code is not accepted as it is generated automatically).
5. When your new section is complete, select its name in the **Outline** tree view under **Report**, then press **Ctrl+Up Arrow** (or **Ctrl+Down Arrow**) to move the new section in the report hierarchy.
6. To see how the report will look, right-click **Report** and select **Refresh Preview**. The updated report appears in the **Report Viewer**. To publish the report (that is, to make the report available in a file that others can see), right-click **Report** and select **Publish**.
7. In the **Publish Reports** dialog, you can choose the where to save the report, add an embedded 3D Viewer that can be seen by readers who use the Microsoft Explorer browser and who have ActiveX enabled.
If you click **More Options**, you can change the type of graphics files and charts used and their size.
8. To save the report, click **OK**. The report is written to the file you specified.

Report Templates


Report templates are available for rapidly setting up application-specific reports. Depending on the information contained in a results file, a report template will be selected automatically, and made available as a command in the following places:

- The **File > Report** menu
- The shortcut menu that appears when you right-click the **Report** object.



In the same places, there is a **Report Templates** command that invokes the **Report Templates** dialog box. This dialog box allows you to:

- Browse the list of existing templates.
- Add (register) a template.

To do this:

1. Click *Add template*  on the **Report Templates** dialog box to invoke the **Template Properties** dialog box.
 2. Select a state or session file that contains a report; alternatively, you can choose to use the current state of CFD-Post, and provide a file name to which to save the template.
 3. Provide a name and description for the template. You cannot use the name of an existing template.
 4. If you are loading a state or session file, and the file name does not end in **.cst** or **.cse**, set the **Execution** setting to either **State** or **Session**, as applicable.
- View and/or edit the properties of a template.

To do this:

1. Click *Edit Properties*  on the **Report Templates** dialog box to invoke the **Template Properties** dialog box.
 2. View and/or edit the name, description, and path to the template file, as applicable. You can edit the properties for templates that were added, but not the standard templates.
- Delete templates from the set of available templates.
- You can delete only the templates that were added. To do this:
1. Select a user template in the **Report Templates** dialog box.
 2. Click Delete .

Turbo Report Templates

Post-processing within CFD-Post is fully automated using the turbomachinery report templates. The turbo reports are designed for single-bladerow, single-phase fluid analyses. The turbo reports can be used for individual bladerows of a multi-bladerow analysis by loading each bladerow domain separately into CFD-Post using the domain selector. (To enable the domain selector, click on the **Edit > Options** menu, select **Files** and checkmark the **Show domain selector before load** option.)

Important

- Turbo reports attempt to auto-initialize Turbo mode. However if auto-initialization fails, you must initialize Turbo mode manually and re-run the turbo report.
- CFD-Post cannot automatically detect a solution that is "360 Case Without Periodics", so you need to set this manually.
- Turbo report templates are not designed for multifile usage or comparison mode. In these cases:
 - User charts that contain local variables will not have plots showing the differences in comparison mode.
 - Tables will not show differences in comparison mode.
 - There will be only one picture of the meridional view of the blades (corresponding to the first loaded results file).

These are the variables required for all Release 12.1 turbo reports:

CFX Variables Required for all Release 12.1 Turbo Reports

- Density
- Force X
- Force Y
- Force Z
- Pressure
- Total Pressure
- Total Pressure in Stn Frame
- Rotation Velocity
- Velocity
- Velocity in Stn Frame u
- Velocity in Stn Frame v
- Velocity in Stn Frame w
- Velocity in Stn Frame
- Velocity in Stn Frame Flow Angle
- Velocity Flow Angle

- Velocity in Stn Frame Circumferential
- Velocity Circumferential
- Velocity Meridional

Note

If all of the turbo components in the results file are 'stationary', then variables having names ending with 'in Stn Frame' are not required.

ANSYS FLUENT Variables Required for all Release 12.1 Turbo Reports

- Density
- Static Pressure
- Total Pressure
- X Velocity
- Y Velocity
- Z Velocity

In addition to the variables mentioned above, the following variables are required for compressible flow reports:

CFX Variables Required for all Release 12.1 Compressible Flow Turbo Reports

- Temperature
- Total Temperature
- Total Temperature in Stn Frame
- Static Enthalpy
- Total Enthalpy
- Total Enthalpy in Stn Frame
- Isentropic Total Enthalpy
- Polytropic Total Enthalpy
- Total Density in Stn Frame
- Total Density
- Specific Heat Capacity at Constant Pressure
- Specific Heat Capacity at Constant Volume
- Rothalpy
- Static Entropy
- Mach Number
- Mach Number in Stn Frame
- Isentropic Compression Efficiency
- Isentropic Expansion Efficiency

ANSYS FLUENT Variables Required for all Release 12.1 Compressible Flow Turbo Reports


- Static Temperature
- Total Temperature
- Enthalpy
- Total Enthalpy
- Specific Heat (Cp)
- Rothalpy
- Entropy

- Mach Number

When variables are missing, lines in the turbo report tables that depend on these variables will be missing.

Procedures for Using Turbo Reports when Turbomachinery Data is Missing

Results files from ANSYS FLUENT (and from some other sources) will not have all the turbomachinery data that CFD-Post requires. For turbo results files that lack data about the number of passages, you must do the following:

- For ANSYS FLUENT files, prior to loading a turbo report template, create a new variable that the report expects (but which is not available from ANSYS FLUENT files):
 - From the toolbar, click **Variable** . The **Insert Variable** dialog appears.
 - In the **Name** field, type **Rotation Velocity** and click **OK**. The Details view for **Rotation Velocity** appears.
 - In the **Expression** field, type $\text{Radius} * \text{abs}(\omega) / 1 \text{ [rad]}$ and click **Apply**. This expression calculates the angular speed (in units of length per unit time) as a product of the local radius and the rotational speed.
- When you load a turbo report for a case that is missing some variables, an error dialog appears that describes warnings and errors. Generally this means that some rows in the turbo report will not appear.
Turbo reports for ANSYS FLUENT files will not display information about absolute Mach number. This causes charts of Mach number to display only the relative Mach number.
- For any results file that is missing the number of passages (such as ANSYS FLUENT files and CFX results files not set up using the Turbo Mode in CFX-Pre), after you load the turbo report template, do the following for each domain:
 - A **<domain_name> Instance Transform** appears in the **Outline** view under **User Locations and Plots**. Prior to viewing the report, double-click this name to edit the instance transform. In the **# of Passages** field, ensure that the number of passages matches the number of passages in the domain. If you enter a new number, click **Apply**.
 - On the **Expressions** tab, double-click on the expression **domain_name Components in 360** to edit it. Match the definition to the number of components in the domain. If you enter a new number, click **Apply**.
- In the **Report Viewer**, click **Refresh** to ensure that the contents are updated.

Choosing a Turbo Report

If the model was set up using CFX-Pre Turbo Mode, then CFD-Post will automatically be able to determine which report to load based on the machine type and flow type selected, and will prompt you to load it. The report can also be loaded manually by right-clicking on the **Report** item in CFD-Post. To avoid conflicts with the current CFD-Post state, you should load the report in a clean session.

In the Computed Results table, any values that are shown as “N/A” means that the necessary scalar variables for computing these values were missing from the results file. This might happen if the solution was run using an older version of CFX, or if some of the scalar variables were manually disabled.

The report templates are CFD-Post session files located in the CFX install under the `etc/PostReports` directory. You can edit these reports or make new versions to add to the list of report templates. The following table shows the correspondence between the machine type, the flow type settings, and the report selection.

Machine Type	Fluid Type	Domain Motion (Single Domains Only)	Report Template
Pump	Any	Rotating	Pump Impeller
		Stationary	Stator
Fan	Any	Rotating	Fan
		Stationary	Stator
Fan	Any	Rotating	Fan Noise

Machine Type	Fluid Type	Domain Motion (Single Domains Only)	Report Template
Axial Compressor	Compressible	Rotating	Axial Compressor Rotor
		Stationary	Stator
Centrifugal Compressor	Compressible	Rotating	Centrifugal Compressor Rotor
		Stationary	Stator
Axial Turbine	Compressible	Rotating	Turbine Rotor
		Stationary	Turbine Stator
Radial Turbine	Compressible	Rotating	Turbine Rotor
		Stationary	Turbine Stator
Hydraulic Turbine	Incompressible	Rotating	Hydraulic Turbine Rotor
		Stationary	Stator
Other	Any	Rotating	Pump Rotor
		Stationary	Stator

Several new report templates have been added to CFD-Post for Release 12.0. These reports support post-processing of results that have multiple components/blade rows. The components can be any combination of stationary or rotating types in one or more domains.

The reports attempt to group the components into stages; you can control how the stages are formed by editing the report session file. The new reports include:

AxialCompressorReport.cse

Report template for axial compressors

CentrifugalCompressorReport.cse

Report template for centrifugal compressors

CompressibleTurbineReport.cse

Report template for compressible flow turbines.

HydraulicTurbineReport.cse

Report template for incompressible flow turbines.

PumpReport.cse

Report template for incompressible flow pumps.

Machine Type	Fluid Type	Report Template
Pump	Any	Pump
Axial Compressor	Compressible	Axial Compressor
Centrifugal Compressor	Compressible	Centrifugal Compressor
Axial Turbine	Compressible	Turbine
Radial Turbine	Compressible	Turbine
Hydraulic Turbine	Incompressible	Hydraulic Turbine

Creating, Viewing, and Publishing Reports

To create or modify a report, do the following:

1. Specify the settings for the report that are contained in the **Report** object.
For details, see [Report Object \(p. 33\)](#).
2. Specify the settings for the title page that are contained in the **Title Page** object.
For details, see [Title Page Object \(p. 33\)](#).

3. Decide which predefined tables to use.

For details, see:

- [File Report Object \(p. 34\)](#)
- [Mesh Report Object \(p. 34\)](#)
- [Physics Report Object \(p. 34\)](#)
- [Solution Report Object \(p. 34\)](#)

4. Optionally, create objects that give additional content to the report.

For details, see [Adding Objects to the Report \(p. 34\)](#).

5. Control which objects get included in the report, and the order in which they are included.

For details, see [Controlling the Content in the Report \(p. 35\)](#).

6. Refresh the report.

For details, see [Refreshing the Report \(p. 35\)](#).

You may refresh the report at any time to see the effect of changes you make to the report settings and content. The report appears on the **Report Viewer** tab.

You can publish a report so that it can be loaded into a third-party browser or editor. For details, see [Publishing the Report \(p. 35\)](#).

Report Object

The settings on the **Appearance** tab of the Report object are described next.

Figures: File Type

Choose the image format in which you want the image files to be saved.

Figures: Figure Size

Choose the size for figures that appear in the report. There are preset sizes, and an option for setting a custom size.

Figures: Width and Height

If you set **Figure Size** to Custom, set the figure width and height in pixels.

Figures: Fit All Figures in the Viewport Before Generation Check Box

When this option is selected, each figure is produced with the view centered and the zoom level set automatically.

Charts: File Type

Choose the image format in which you want the chart files to be saved.

Charts: Chart Size

Choose the size for charts that are saved as part of the report. There are options that specify preset sizes, an option for using the same size as figures, and an option for setting a custom size.

Charts: Width and Height

If you set **Chart Size** to Custom, set the chart width and height in pixels.

Title Page Object

The Title Page object is automatically generated and listed under the Report object. The settings of this object determine the content of the title page, and are described next.

Custom Logo Check Box

The **Custom Logo** check box determines whether or not a custom logo is included in the title page. The logo must be available in a file of compatible format.

Custom Logo

The **Custom Logo** setting indicates the image file to use for the custom logo.

ANSYS Logo Check Box

The **ANSYS Logo** check box determines whether or not the ANSYS logo is included in the title page. The ANSYS logo is shown in [Figure 3.2, “A Sample Report, Part 1” \(p. 25\)](#).

Title

The **Title** setting holds the title of the report.

Author

The **Author** setting holds the name of the author of the report.

Current Date Check Box

The **Current Date** check box determines whether or not the date and time are included in the title page.

Table of Contents Check Box

The **Table of Contents** check box determines whether or not a table of contents is included in the title page. An example is shown in [Figure 3.2, “A Sample Report, Part 1” \(p. 25\)](#).

Each entry in the table of contents is an active link to the corresponding section of the report. To follow a link, click the link using the left mouse button.

Table of Contents Check Box: Captions in Table of Contents Check Box

The **Captions in Table of Contents** check box controls the level of detail in the entries in the table of contents. When this check box is selected, the entries in the table of contents that link to objects in the report contain the titles of the objects.

File Report Object

A **File Report** object containing a file information table, is automatically generated for each loaded results file, and listed under the **Report** object. There are no user-adjustable settings except the check box in the tree view, which controls whether or not the file information is included in the report. An example of the file information table is shown in [Figure 3.2, “A Sample Report, Part 1” \(p. 25\)](#).

Mesh Report Object

A **Mesh Report** object is automatically generated and listed under the **Report** object. The **Mesh Report** object contains settings for a mesh information table and a mesh statistics table. Examples of the mesh information table and mesh statistics tables are shown in [Figure 3.3, “A Sample Report, Part 2” \(p. 26\)](#). The data in these tables are the same as given by the mesh calculator. For details, see [Mesh Calculator \(p. 179\)](#).

Physics Report Object

A **Physics Report** object is automatically generated and listed under the **Report** object only when you load a CFX-Solver Results type of file. The **Physics Report** object allows you to control the output of physics summary data for domains and boundaries. Examples of the physics summary tables are shown in [Figure 3.3, “A Sample Report, Part 2” \(p. 26\)](#).

Solution Report Object

A **Solution Report** object is automatically generated and listed under the **Report** object only when you load a CFX-Solver Results type of file. The **Solution Report** object allows you to control the output of boundary flow, force, and torque summaries in the report. CFD-Post uses the summary data contained in the results files.

Adding Objects to the Report

You can create objects of the following types to add additional content to the report:

- Tables

For details, see [Table Command \(p. 140\)](#).

- Charts
For details, see [Chart Command \(p. 145\)](#).
- Comments
For details, see [Comment Command \(p. 155\)](#).
- Figures
For details, see [Figure Command \(p. 156\)](#).

Such objects are listed beneath the **Report** object in the tree view.

Controlling the Content in the Report

Report objects can be shown or hidden in the report by setting the check box next to them in the **Outline** tree view. The changes take effect the next time the report is refreshed or published.

You can control the order of **Report** objects by selecting one or more, then right-clicking on the selection and using the **Move Up** and **Move Down** shortcut menu commands as necessary. You can also press **Ctrl+Up Arrow** and **Ctrl+Down Arrow** to move selected items.

Refreshing the Report

To refresh the report, you can do any of the following:

- Right-click the **Report** object, or any of the report objects under it, then select **Refresh Preview** from the shortcut menu.
- Click the **Refresh Preview** button in the details view for the **Report** object, or any of the report objects under it that have this button.
- Click the **Refresh** button in the **Report Viewer** tab.
- Select **File > Report > Refresh Preview**.

Note

The first time you visit the **Report Viewer** tab after loading a results file, the report will be refreshed automatically.

Viewing the Report

After the report preview has been generated, you can view it in the **Report Viewer** tab.

Publishing the Report

You can publish a report so that it can be loaded into a third-party browser or editor. To publish a report, click the **Publish** button in the **Report Viewer** toolbar to access the **Publish Report** dialog box, adjust settings as appropriate, and click **OK**. You can also access the same dialog box by doing any of the following:

- Right-click the **Report** object, or any of the report objects under it, then select **Publish**.
- Select **File > Report > Publish**.

Format

Set **Format** to one of:

- **HTML**
The **HTML** option causes the report to be written in an **HTML** format.
- **Text**
The **Text** option causes the report to be written in a plain text format.

File

Set **File** to the file name to use for saving the report.

Save Images in Separate Directory Check Box

Selecting the **Save images in separate directory** check box causes all image files to be put in a directory that is beside the main output file.

Generate CFX-Viewer Files (CVF) for Figures Check Box

When this option is selected, each figure is saved to a 3D image file in addition to the 2D image file that is normally saved. The 3D image file has an extension of `cvf`, and can be viewed in the report using Microsoft Internet Explorer with the CFD-viewer plug-in installed. You can find CFD-viewer plug-in in `<install_dir>\viewer` (typically `C:\Program Files\Ansys Inc\V121\CFX\viewer`). Browsers other than Microsoft Internet Explorer display the 2D image file associated with each figure.

More Options Button

The **More Options** button opens the **Publish Options** dialog box. The **Publish Options** dialog box offers the same settings as the Report object, and overrides the latter for the purpose of publishing the report.

Note

The **Publish Options** dialog box settings will be overwritten with the settings of the Report object if you change or otherwise apply the settings of the latter.

Display Properties and Defaults

The **Display Properties and Defaults** branch of the **Outline** tree contains a **Color Maps** area that is divided into **Custom** and **System** area. Initially, only the **System** area has entries; these are the default color map names. You can use any of the default color maps as the basis for color maps that you define, which are stored in the **Custom** color map area.

To learn how customize color maps, see [Color Map Command \(p. 139\)](#).

Variables Workspace

The **Variables** workspace is used to create new user variables and modify existing variables.


The following topics will be discussed:


- [Variables Tree View \(p. 36\)](#)
- [Variables Details View \(p. 37\)](#)
- [Variables: Example \(p. 39\)](#).

Variables Tree View

The following table shows shortcuts that are specific to the **Variables** tree view. For a description of how to access these shortcuts, and a list of commands that appear in most tree views, see [Common Tree View Shortcuts \(p. 14\)](#).

Command	Description
All to Conservative	Makes all variables assume conservative values. For details, see Hybrid and Conservative Variable Values (p. 165) .
All to Hybrid	Makes all variables assume hybrid values. For details, see Hybrid and Conservative Variable Values (p. 165) .
Calculate Velocity Components	Calculates velocity components using the global rotation axis. This can also be done in the Turbo workspace. For details, see Calculate Velocity Components (p. 204) .

Vector variables have a  symbol next to them. Their components are shown beneath them in the tree structure.

Scalar variables have a  symbol next to them.

Variables Details View

The **Variable** details view is used to change the definition of fundamental (system) variables, and to create and edit user variables.

To edit an existing variable, either:

- Double-click the variable in the tree view
- Right-click the variable, then choose **Edit** from the shortcut menu.

The above actions cause the **Variables** details view to appear.

Fundamental Variables

Fundamental variables (variables provided by the solver) can have their units changed. This would allow you to create a legend that uses alternative temperature units (such as degrees Celsius).

Note

These settings override the global units setting (defined in the **Options** dialog box, accessible from the **Edit** menu).

1. Toggle between **Hybrid** and **Conservative** values.
This affects any dependent variables and expressions as well. For details, see [Hybrid and Conservative Variable Values \(p. 165\)](#).
2. Select the units.
3. Click **Reset** to restore the variable settings stored in the database.
Use this to undo changes if you have not yet clicked **Apply**.

Saving Variables Back to the Results File

Fundamental variables can be redefined using expressions and then saved back into the results file for later use. To do this, select the **Replace with expression (write to results)** check box, enter one or more expressions, then click **Apply**. To change a vector variable, you must write three expressions: one expression for each direction (X, Y, and Z). The result file is updated when you click **Apply**. To restore a fundamental variable to its original state, clear the **Replace with expression (write to results)** check box and click **Apply**.

One reason for modifying the variables in a results file is to modify the initial conditions for a new solver run. In this case, you must modify the principal variable for each affected equation.

In CFX-Pre, in most cases, the principal variable for a given equation is the same as the variable used to specify initial conditions, but there are some exceptions, as shown in the table below:

Equation	CFX-Pre Variable	Principal Variable
Thermal Energy	Temperature	Static Enthalpy
Total Energy	Temperature	Total Enthalpy
Mass Fractions	Mass Fraction	Conservative Mass Fraction
Volume Fractions	Mass Fraction	Conservative Volume Fraction
Continuity (with cavitation activated)	Pressure	Solver Pressure

For example, to initialize the mass fractions equation in CFX-Pre, you would set `Mass Fraction`. In order to modify the initial conditions for the same equation in a results file, you would set `Conservative Mass Fraction` instead.

Note

- For the thermal energy and total energy equations, you must set **Temperature** as well as the principal variable.
- When overwriting the mesh **Total Mesh Displacement**, the locations of the mesh nodes in CFD-Post will not be affected, only the variable values.

Radius and Theta

The variables **Radius** and **Theta** are available only when the rotational axis has been defined. The rotational axis can either be defined in the results file or in CFD-Post through the **Initialization** panel in the **Turbo** workspace.

Boundary-Value-Only Variables

Some variables in the CFX results file take meaningful values only on the boundaries of the geometry. Examples of this sort of variable are **Yplus**, **Wall Shear**, **Heat Transfer Coefficient**, and **Wall Heat Flux**. For detail, refer to the *CFX Output File* section in the *CFX-Solver Manager User's Guide*.


To obtain sensible plots when using these variables, use them to color only boundary objects. If, for example, you try to color a slice plane through the center of the geometry with one of these variables, you will see a large area of color that is meaningless; only at the very edges of the geometry will there be useful coloration.

For boundary-value-only variables, only hybrid values exist (as they are undefined away from a boundary).

User Variables

User variables can be defined in any of the following ways:

- As a scalar by defining one expression
- As a vector by defining three expressions
- As a scalar or vector by copying an existing scalar or vector variable (Frozen Copy).

To create a new **User Variable**, click  on the right side of the details view, or right-click a variable in the tree view, then select **New** from the shortcut menu.

- **Method** is set to **Expression**.
- Select **Scalar** or **Vector**.
- Enter an expression or select from the drop-down menu. For vectors, three expressions are required. For details, see [Expressions Workspace \(p. 39\)](#).
- Select **Calculate Global Range** to have range data calculated. It displays after clicking **Apply**.
- **Method** is set to **Frozen Copy**.
- For **Copy From** select scalar variable to copy. Both the hybrid and conservative values copy. Subsequent changes to the original variable will not affect the copied variable (such as changing timestep).

Note

You cannot create a variable with the same name as an existing expression or object.


The details view of an existing **User Variable** looks different than that for a new **User Variable**. In particular:

- The ability to control the type of **User Variable** is no longer available.
- If **Calculate Global Range** was selected, you will see the range limits.
- A variable created as a frozen copy allows you to select **Hybrid** or **Conservative** values. This affects all objects and expressions that depend on the variable.
 - The expression is still adjustable.
 - Units are shown.
 - Range information is shown if **Calculate Global Range** was selected.

- The scalar variable to **Copy From** is still an option. If a different variable is selected, a new copy is made upon clicking **Apply**.
- Toggling **Hybrid/Conservative** selects within the copy. It does not cause data to be copied again from the **Copy From** variable.

Variables: Example

In this example, you will use an expression to create an Isosurface that is a fixed radial distance from an axis or point. For details, see [Expressions Workspace: Example \(p. 41\)](#). Before trying this example, you must first create the expression in the aforementioned example.

1. Copy the <CFX_install_dir>/examples/StaticMixer_001.res file to your working directory and load it into CFD-Post.
2. Click the **Variables** tab.
3. Click  in the **Variables** details view to create a new variable.
4. When the **New Object** window appears, type the name `Radial Distance`, and then click **OK**.
5. In the variable details view, set **Expression** to `radial` (which is the expression you created earlier).
6. Click **Apply** to create the new variable.

This variable appears in the tree view and can be used like any other variable. Notice that the variable is listed as **User Defined**.

You can now create an Isosurface using this variable as follows:

1. Select **Insert > Location > Isosurface**.
2. In the **New Isosurface** dialog box, enter a name and then click **OK**.
3. On the **Geometry** tab for the Isosurface:
 1. Set **Variable** to `Radial Distance`.
 2. Set **Value** to `1 [m]`.
This is a suitable value for results from the `StaticMixer_001.res` file. You may need to alter this value to something sensible depending on the results you are viewing.
4. Click the **Color** tab and set the **Mode** option to **Variable**. Select a sensible variable (such as, **Temperature** or **Velocity**) with which to color the isosurface.
5. Set the **Range** option to **Local** so that the full color range is used on the Isosurface.
6. Click **Apply** to create the isosurface.

You should now see a cylindrical Isosurface centered about the Z-axis. All points on the Isosurface are a distance of 1 m (or the value you used in the **Value** box) from the Z-axis. Note that a cylinder can also be created as a surface of revolution. For details, see [Surface of Revolution Command \(p. 108\)](#). Additional information on expressions is available; for details, see [Further Expressions \(p. 42\)](#).

Expressions Workspace

The **Expressions** workspace is used to select and generate expressions using the CFX Expression Language (CEL), which you can then use in CFD-Post in place of almost any numeric value (as long as the correct units are returned by the expression).

When using expressions in multfile and case-comparison situations, the expression syntax is:

When multiple files are loaded:

```
function()@CASE:case name.location
```

For example, `area()@CASE:newcase.myplane`

For file comparisons:

```
function()@CASE:[1|2].location
```

For example, `area()@CASE:2.myplane`

The following topics will be discussed:

- [Expressions Tree View \(p. 40\)](#)
- [Expressions Workspace: Expressions Details View \(p. 40\)](#)
- [Expressions Workspace: Example \(p. 41\)](#)

You should be aware of the guidelines regarding expressions:

- You cannot create an expression with the same name as an object or variable.
- Within the CFX Expression Language, some variables are known by short names to save typing the full variable name. For example, *p* refers to *Pressure*. Although it is possible to create an expression with the same name as an abbreviated variable, it is ignored. For example, if you define an expression named *p* with the definition 5 [K] , an expression defined as $2 * p$ represents $2 * \text{Pressure}$, not 10 [K] .
- You must always provide units inside square brackets for constant values typed into an expression.

Expressions Tree View

The following table shows shortcuts that are specific to the **Expressions** tree view and are accessed by right-clicking an expression in the tree view. For a list of shortcuts that appear in most tree views, see [Common Tree View Shortcuts \(p. 14\)](#).

Command	Description
Use as Workbench Input Parameter	Specifies the expressions that are to be used as parameters in a Design Exploration session. These parameterized expressions are saved to the CFD-Post state file. To parameterize an expression, right-click the expression and select Use as Workbench Input Parameter or Use as Workbench Output Parameter . The icon next to the expression changes to help identify it as a parameterized expression.
Use as Workbench Output Parameter	

Expressions Workspace: Expressions Details View

The **Expressions** details view contains the following tabs:

- [Expression Definition Tab \(p. 40\)](#)
- [Plot Expression Tab \(p. 41\)](#)
- [Evaluate Expression Tab \(p. 41\)](#)

Expression Definition Tab

You can access lists of variables, expressions, locators, functions and constants by right-clicking in the definition window when defining an expression. Although valid values can be chosen from each of the various lists, the validity of the expression itself is not checked until you click **Apply**. For details, see [CEL Operators, Constants, and Expressions \(p. 135\)](#) and [CFX Expression Language \(CEL\) in CFD-Post \(p. 215\)](#).

Any expressions not containing variables are evaluated when you click **Apply**.

1. Enter the definition of a new expression or edit the definition of an existing expression in the **Definition** text field.

For details, see [CFX Expression Language \(CEL\) \(p. 133\)](#).

2. The value of the expression is shown in the **Value** field.
3. Click **Reset** to restore the expression to the definition stored in the database.
Use this to undo changes that have not yet been applied.
4. Click **Apply** to commit any changes or entries made in the **Definition** box.

After you have defined an expression, you can right-click it to make it a parameter for use with Design Exploration:

- You may choose **Use as Workbench output parameter**.

- If the expression *will not* influence CFX-Pre, you may choose **Use as Workbench input parameter**. Note that this is not a common situation.
- If the expression *will* influence CFX-Pre, you must use the **Expression** shortcut menu in CFX-Pre to make the expression an ANSYS Workbench input parameter.

Plot Expression Tab

The **Plot** tab enables you to plot an expression for a range of one of its variables with the other variables (if there are any others) held constant.

1. If you have multiple cases loaded and an expression that applies to only one case highlighted, specify the **Case**.
2. Choose the number of sample data points (**# of Points**) of the expression that you would like plotted.
3. Select the independent variable (X) of the expression for use in the plot.
4. Specify a **Range** for this variable in the plot.

All other values are constant (their check boxes cannot also be checked). Enter fixed values for them.

5. Click **Plot Expression** to view the plot.

After viewing the chart, you may click **Define Plot** to return the **Plot** tab to its previous state (which shows the plot settings).

Evaluate Expression Tab

The **Evaluate** tab is provided to help you verify that the expression highlighted in the **Expression** tree view is set up correctly. To evaluate an expression:

1. If you have multiple cases loaded and a locator-based function (such as "**areaAve(Pressure)@outlet**") highlighted, specify the **Case** in which you want the expression evaluated.
2. If the expression requires that you provide values, type them in.
3. Click **Evaluate Expression**.

The value of the expression is displayed in the **Value** field.

Expressions Workspace: Example

In this example, you will create an expression that you can use to define a new **User Variable**. For details, see [Variables: Example \(p. 39\)](#).

1. Select **Insert > Expression** to create a new expression.

The **Insert Expression** dialog appears.

2. In the **Insert Expression** dialog, type a name for the expression and click **OK**.
3. In the **Definition** area of the **Expression** details view, enter the expression: $\text{sqrt}(X^2 + Y^2)$

This expression gives the distance of a point from the Z-axis.

4. Click **Apply** to create the expression.

Note that the **Value** field shows that the variable has units of meters. The value is variable so a single number cannot be shown, as indicated by the placeholder: <variable>.

5. Click the **Plot** tab.

Here, you can define a simple 2D plot. Because the function has two independent variables¹, you must select a constant value for one of the variables.

6. Check the check box beside **X**.

This selects X as the variable that varies. All other variables requires a fixed value (for plotting).

7. Leave **Start of Range** and **End of Range** at their default values.

¹CFD-Post automatically finds the variables associated with an expression, even if the expression depends on another expression.

8. For **Y**, set **Fixed Value** to 3 [m] .

9. Click **Plot Expression**.

A plot shows the variation in the expression with values of X ranging from 0 to 1 [m] and the value of Y held constant at 3 [m] .

10. Click the **Evaluate** tab.

11. Set **X** to 0.55 [m] and **Y** to 3 [m] .

12. Click **Evaluate Expression**.

The value 3.05 [m] appears in the **Value** field. This is consistent with the plot and can easily be verified.

Further Expressions

After completing the variable editor example, you can try modifying this expression. You may want to try $\text{sqrt}(X^2+Z^2)$ to define a distance from the Y-axis or $\text{sqrt}(X^2+Y^2+Z^2)$ to define a sphere. Try moving the location of the sphere by adding values to the X, Y, or Z components; for example, $\text{sqrt}(X^2+Y^2+(Z-0.5 \text{ [m]})^2)$ moves the sphere a distance of 0.5 m in the positive Z direction.

Calculators Workspace

The **Calculators** workspace offers access to the function, macro, and mesh calculators. To access the **Calculators** workspace, click the **Calculators** tab.

For details on the functions available from the **Calculators** workspace, see:

- [Function Calculator \(p. 165\)](#)
- [Macro Calculator \(p. 167\)](#)
- [Mesh Calculator \(p. 179\)](#).

Turbo Workspace

The **Turbo** workspace improves and speeds up post-processing for turbomachinery simulations. To access the **Turbo** workspace, click the **Turbo** tab.

For details about using the **Turbo** workspace, see [Turbo Workspace \(p. 185\)](#).

Chapter 4. CFD-Post 3D Viewer

In CFD-Post, the 3D Viewer is accessible by clicking the 3D Viewer tab at the bottom of the panel on the right side of the interface.

After loading a results file into CFD-Post, you can see a visual representation of the geometry in the 3D Viewer. You can create various other objects that can be viewed in the 3D Viewer. For details, see [CFD-Post Insert Menu](#) (p. 89).

Descriptions of the various viewing modes and 3D Viewer commands, including toolbars, shortcut menus, and hotkeys, are given in [3D Viewer Modes and Commands](#) (p. 45).

You can switch between four adjustable “views” that each remember the camera angle and state of visibility of all objects. CFD-Post has “figures”, which are similar to views except that they can be included in reports. For details, see [Views and Figures](#) (p. 51).

Note

In order to see correct colors and accurately displayed objects in the 3D Viewer, some combinations of ATI video cards and ATI graphics drivers on Windows XP require that you set the environment variable VIEWER_CACHE_COLORS to 0:

1. Right-click on **My Computer** and select **Properties**. The **System Properties** dialog appears.
2. Click the **Advanced** tab.
3. Click **Environment Variables**.
4. Under **System variables**, click **New**.
5. In the **Variable name** field, type: VIEWER_CACHE_COLORS
6. In the **Variable value** field, type the number: 0
7. Click **OK**.
8. To verify the setting, open a command window and enter: set
The results should include the line:

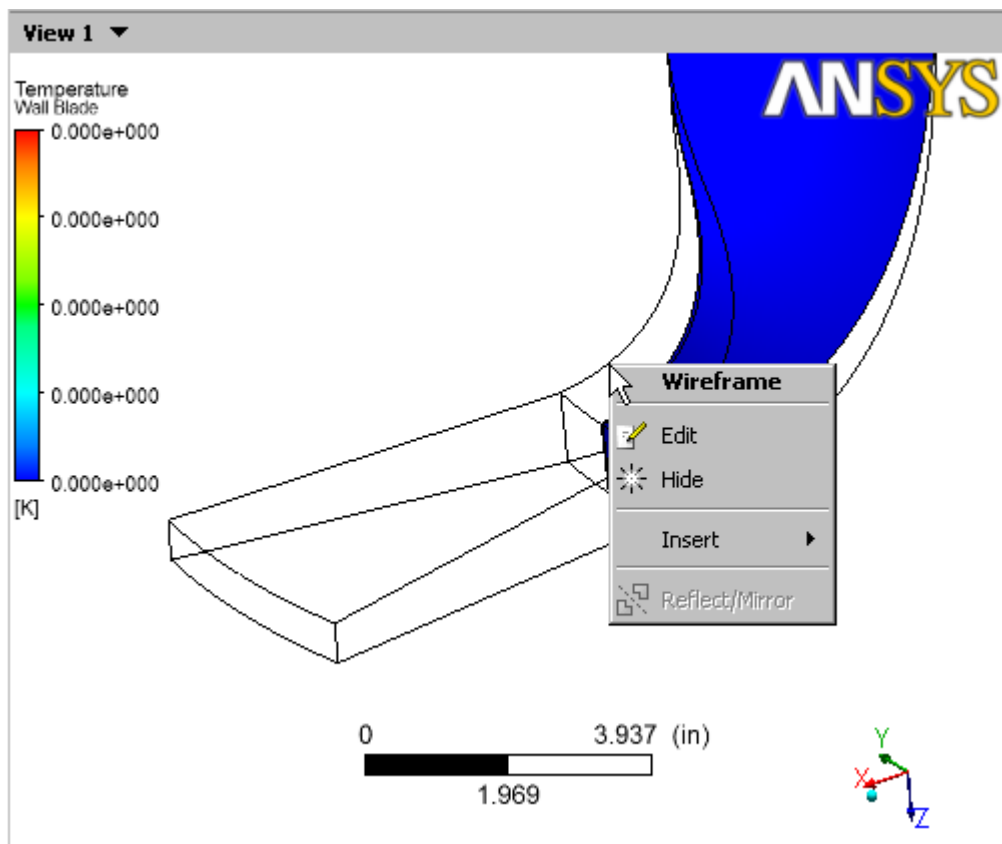
```
VIEWER_CACHE_COLORS=0
```

This setting will fix problems such as:

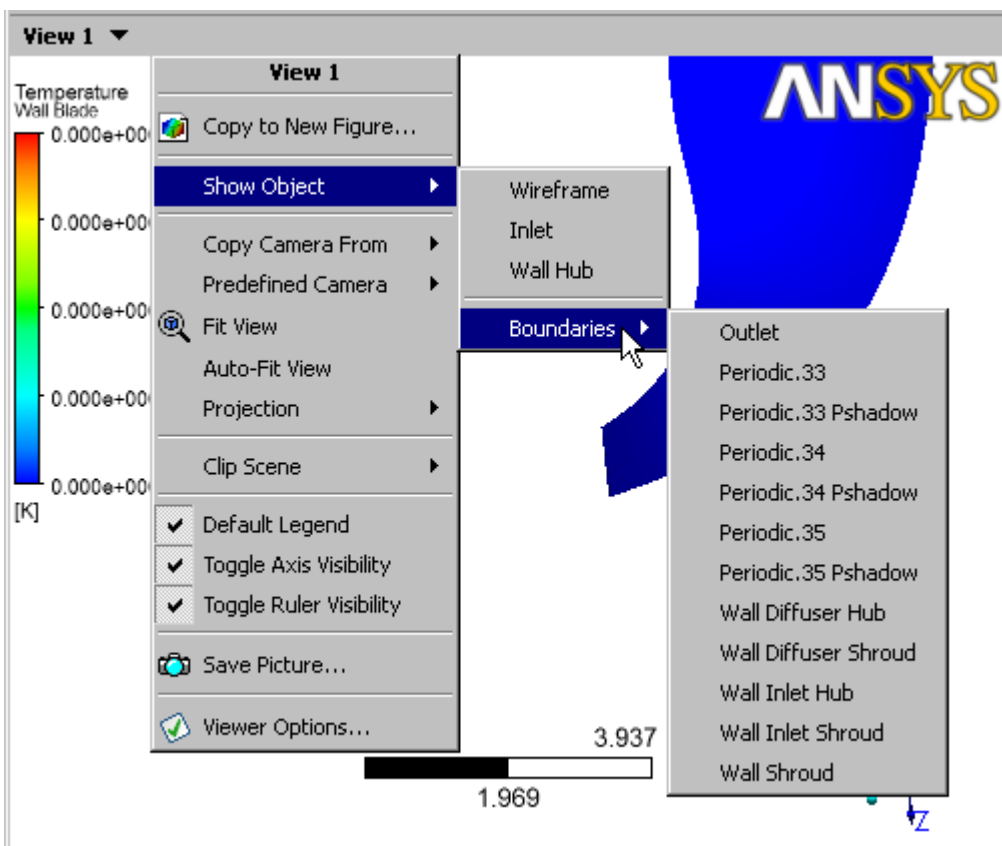
- Boundary condition markers placed incorrectly or rendered in white.
- Regions around the circles are incorrect (rendered as yellow areas marked with blue)
- Mesh lines not displayed properly and with dark patches showing.

Object Visibility

The visibility of each object can be turned on and off using the check boxes in the tree view, as described in [Object Visibility](#) (p. 13). However, you can also hide objects by right-clicking on them and selecting **Hide**. The right-click menu has a title that indicates the object that will be acted upon (**Wireframe** in the figure that follows) so that you do not accidentally hide the wrong object.



Once an object has been hidden, you can show it again by right clicking on the background of the Viewer and selecting **Show Object**:













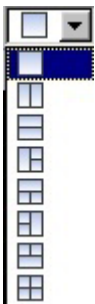
3D Viewer Modes and Commands




This section describes:

- [3D Viewer Toolbar \(p. 45\)](#)
- [CFD-Post 3D Viewer Shortcut Menus \(p. 46\)](#)
- [Viewer Keys \(p. 48\)](#)
- [Mouse Button Mapping \(p. 49\)](#)
- [Picking Mode \(p. 50\)](#)

3D Viewer Toolbar

The 3D Viewer toolbar has the following tools:

Tool	Description
	Activates one of the three picking tools (shown below).
	Selects objects. You can use this tool to drag line, point, plane, and isosurface objects to new locations.
	Selects objects using a box. Drag a box around the objects you want to select.
	Selects objects using an enclosed polygon. Click to drop points around the objects. Double-click to complete the selection. <div data-bbox="483 961 1474 1123"> <p>Note</p> <p>Polygon Select mode will not allow you to create an invalid region, such as would occur if you attempted to move a point such that the resulting line would cross an existing line in the polygon.</p> </div>
	Rotates the view as you drag with the mouse. Alternatively, hold down the middle mouse button to rotate the view.
	Pans the view as you drag with the mouse. Alternatively, you can pan the view by holding down Ctrl and the middle mouse button.
	Adjusts the zoom level as you drag with the mouse vertically. Alternatively, you can zoom the view by holding down Shift and the middle mouse button.
	Zooms to the area enclosed in a box that you create by dragging with the mouse. Alternatively, you can drag and zoom the view by holding down the right mouse button.
	Centers all visible objects in the viewer.
	When enabled, clicking on an object in the tree view causes that object to be highlighted in the 3D Viewer. The style of highlighting is controlled by Edit > Options > CFD-Post > Viewer > Object Highlighting .
	Selects the viewport arrangement. You can perform Independent zoom, rotation and translate options in each viewport.

Tool	Description
	Toggles between locking and unlocking the views of all viewports. When the views are locked, the camera orientation and zoom level of the non-selected viewports are continuously synchronized with the selected viewport. Locking the view for the viewports in this way can be a useful technique for comparing different sets of visible objects between the viewports. This tool is available only when all viewports are using the Cartesian (X-Y-Z) transformation.
	Toggles between synchronizing the visibility of objects in all viewports. When active, any subsequent action to hide or display an object affects all viewports; activating this feature does not affect any existing show/hide states. <div data-bbox="386 457 1377 617" style="border: 1px solid black; padding: 10px; margin-top: 10px;"> <p>Note</p> <p>This toggle will not synchronize the visibility of objects in different cases that have the same name. However, in file comparison mode CFD-Post <i>does</i> synchronize the visibility of objects that have the same name.</p> </div>
	Displays the Viewer Key Mapping dialog box. See Viewer Keys (p. 48) for details.


CFD-Post 3D Viewer Shortcut Menus

You can access the shortcut menu by right-clicking anywhere on the viewer. The shortcut menu is different depending on where you right-click.

Shortcuts for CFD-Post (Viewer Background)

The following commands are available in CFD-Post when you right-click the viewer background:

Command	Description
Deformation	<p>Specifies the deformation scale to be viewed. This option is only available when the <code>Total Mesh Displacement</code> variable exists. When an option is selected, it will be applied to all objects in every view and figure. Select from the following:</p> <ul style="list-style-type: none"> • Undeformed Shows all objects as if they were not deformed • True Scale Displays all objects with their regular deformation values • 0.5x Auto Shows all objects with half of the optimal (Auto) scale • Auto Adjusts the deformation scaling for optimal viewing. Internally, the deformation is scaled so that the maximum deformation results in a viewable displacement of a percentage of the domain extents, regardless of the problem size. • 2x Auto Adjusts the deformation to be double that of regular deformation • 5x Auto Shows all objects with 5 times their regular deformation value. • Custom... Opens the Deformation Scale dialog box and displays the currently applied scale value for the deformation. Specify a new value to change the scale. • Animate...

Command	Description
	Opens the Animation dialog box in Quick Animation mode. For details, see Animating Mesh Deformation Scaling (p. 160) .
Copy to New Figure	Creates a new figure based on the current camera position, zoom level, and object visibility settings. For details, see Views and Figures (p. 51) . The figure appears under the Report object, and can be used in a report. For details, see Report (p. 24) . The Make copies of objects check box controls how the new figure is made: <ul style="list-style-type: none"> When the check box is selected, visible objects are copied for the new figure. Use this option if you want the figure to retain its appearance when the original objects are modified. When the check box is cleared, only the camera position, zoom level, and the object visibility settings are stored in the definition of the figure. Use this option if you want the figure to automatically update with changes to the original objects.
Show Object	Shows hidden objects, boundaries, and regions. See Object Visibility (p. 43) .
Copy Camera From	If you have set a Predefined Camera angle in another view, selecting Copy Camera From > <i>view name</i> will apply that angle to the current view.
Predefined Camera	Displays different views by changing the camera angle to a preset direction.
Fit View	Centers all visible objects in the viewer. This is equivalent to clicking the  icon.
Auto-Fit View	Automatically fits the view while you rotate the camera or resize the 3D Viewer. This disables the manual resizing actions otherwise available from the tool bar or mouse.
Projection	Switches between perspective and orthographic camera angles.
Clip Scene	Controls scene clipping via clip planes. For details, see Clip Plane Command (p. 138) .
Default Legend	Shows or hides the default legend object.
Axis	Shows or hides the axis orientation indicator (known as the triad) in the bottom-right corner of the viewer.
Ruler	Shows or hides the ruler on the bottom of the viewer.
Save Picture	Same as selecting File > Save Picture . For details, see Save Picture Command (p. 69) .
Viewer Options	Opens the Options dialog box with the viewer options displayed. For details, see Viewer (p. 83) .

Shortcuts for CFD-Post (Viewer Object)

The following commands are available in CFD-Post when you right-click an object in the viewer:

Command	Description
Edit	Opens the object for editing.
Hide	Hides the selected object in the 3D Viewer.
Animate	Brings up the Animation dialog box and animates the selected object automatically. For details, see Quick Animation (p. 159) .
Color	Enables you to change the selected object's color.
Render	Enables you to change some of the selected object's render options (such as lighting and face visibility). To change other render options, select Edit and make your changes on the object's Render tab.
Insert	Opens another menu with options to insert planes, contours, streamlines, etc. For details, see CFD-Post Insert Menu (p. 89) .

Command	Description
Set Plane Center	For planes defined using the Point and Normal method, this action moves the point that defines the plane. This changes the focus for plane bounding operations. See Plane Bounds (p. 97) .
Reflect/Mirror	Applies a reflection to the selected domain. To use this command, right-click the corresponding wireframe in the viewer.
Probe Variable	Opens a toolbar at the bottom of the viewer allowing the specification of coordinate points and variable type. After each field is changed, the solution automatically generates to the right of the variable type setting. For details, see Probe (p. 164) .

Viewer Keys

A number of shortcut keys are available to carry out common viewer tasks. These can be carried out by clicking in the viewer window and pressing the associated key.

Key	Action
space	Toggles between picking and viewing mode
arrow keys	Rotates about the screen X and Y axes
Ctrl + up/down arrow keys	Rotates about the screen Z direction (the axis perpendicular to the screen)
Shift + arrow keys	Moves the light source
1	Switches to one viewport
2	Switches to two viewports
3	Switches to three viewports
4	Switches to four viewports
c	Centers the graphic object in the viewer window
n	Toggles projection between orthographic and perspective
r	Resets view to initial orientation
s	Toggles the level of detail between auto, off, and on.
u	Undoes transformation
Shift + U	Redoes transformation
x	Sets view towards -X axis
Shift + X	Sets view towards +X axis
y	Sets view towards -Y axis
Shift + Y	Sets view towards +Y axis
z	Sets view towards -Z axis
Shift + Z	Sets view towards +Z axis





The information in this table is accessible by clicking the *Show Help Dialog*  toolbar icon in the 3D Viewer toolbar.

Table 4.1. Mouse Operations and Shortcuts

Operation	Description	Workbench Mode Shortcuts	CFX Mode Shortcuts
Zoom ObjectZoom Camera Zoom	To zoom out, drag the pointer up; to zoom in, drag the pointer down.	Shift + middle mouse button	Middle mouse button Shift + middle mouse button zooms in a step. Shift + right mouse button zooms out a step.
Translate	Drag the object across the viewer.	Ctrl + middle mouse button	Right mouse button
Zoom Box	Draw a rectangle around the area of interest, starting from one corner and ending at the opposite corner. The selected area fills the viewer when the mouse button is released.	Right mouse button Shift + left mouse button Shift + right mouse button	Shift + left mouse button
Rotate	Rotate the view about the pivot point (if no pivot point is visible, the rotation point will be the center of the object).	Middle mouse button	
Set Pivot Point	Set the point about which the Rotate actions pivot. The point selected must be on an object in the 3D Viewer . When you set the pivot point, it appears as a small red sphere that moves (along with the point on the image where you clicked) to the center of the 3D Viewer . To hide the red dot that represents the pivot point, click on a blank area in the 3D Viewer .	Left mouse button when in rotate, pan, zoom, or zoom box mode (as set by the icons in the viewer's tool bar).	Ctrl + middle mouse button
Move Light	Move the lighting angle for the 3D Viewer . Drag the mouse left or right to move the horizontal lighting source and up or down to move the vertical lighting source. The light angle hold two angular values between 0 - 180.	Ctrl + right mouse button	Ctrl + right mouse button
Picking Mode	Select an object in the viewer.	Ctrl + Shift + left mouse button	Ctrl + Shift + left mouse button

Picking Mode



Picking mode is used to select and drag objects in the viewer. The mesh faces must be visible on an object or region to allow it to be picked. Enter picking mode by selecting the *Single Select*  tool in a pull-down menu of the viewer toolbar. If the *Single Select*  icon is already visible, you can simply click the *New Selection*  icon. You can also pick objects while still in viewing mode by holding down the **Ctrl** and **Shift** keys as you click in the viewer.

Selecting Objects

Use the mouse to select objects (for example, points and boundaries) from the viewer. When a number of objects overlap, the one closest to the camera is picked.

You can change the picking mode by selecting one of the toolbar icons:

-  **Single Select**

-  **Box Select**
-  **Polygon Select**

For details, see [3D Viewer Modes and Commands \(p. 45\)](#).

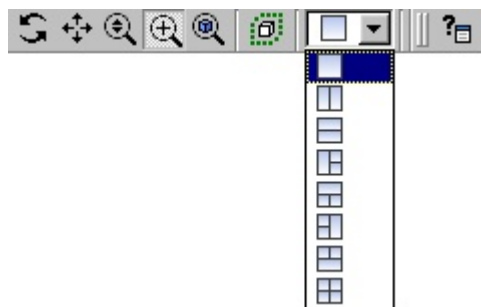
Moving Objects

Point, plane and line objects can be moved in the viewer by dragging and dropping the object to a new location. When an object is moved, its definition is updated in the details view. Any other plots that are located on these movable objects are automatically updated.

Views and Figures

The 3D Viewer opens with a single *viewport*; you can increase the number of viewports to four by using the viewport icon:

Figure 4.2. Viewport Control



The contents of a viewport are a *view*, which is a CCL object that contains the camera angle, zoom level, lighting, and visibility setting of each object in the tree view.

Each viewport contains a different, independent view. By default, four views exist: **View 1**, **View 2**, **View 3**, **View 4**.

When you select an object in the tree view, its information is applied to the active viewport. When you manipulate an object in the viewport, the view's CCL is updated immediately. However if the focus is on that viewport, you can press **u** to revert your change.

In CFD-Post, you can create figures, which are the same as views, except that they are usable in reports. For details, see [Report \(p. 24\)](#).

Creating a Figure

In CFD-Post, figures can be created by selecting **Insert > Figure**, or by selecting **Copy to New Figure** from the viewer shortcut menu (after right-clicking a blank area in the 3D Viewer). The names of views that you create are of the form "Figure *m*" by default, where *m* is an integer that results in a unique name.

A new figure gets its definition from the currently existing view or figure. The latter remains active so that subsequent view manipulations do not affect the new figure.

Copying Objects for Figures

A change made to an object will affect all figures that show that object. This can result in an unwanted change to a figure after it has been created. In order to avoid this problem, you may select the **Make copies of objects** option that is available when creating a new figure. This causes all visible objects to be copied, and the new figure to use the copied objects rather than the original ones.

Any copied objects for a figure will appear in the tree view under `User Locations and Plots > Local Objects for FigureName`, where `FigureName` is the name of the figure.

Switching to a View or Figure

To switch to a view or figure, do one of the following:

- Use the drop-down menu in the upper-left corner of the viewport.
- For figures only: Double-click the figure in the tree view (under the Report object).
- For figures only: Right-click the figure in the tree view (under the Report object), then select **Edit** from the shortcut menu.

Changing the Definition of a View or Figure

To change a view or figure:

1. Switch to the view or figure that you want to change.
For details, see [Switching to a View or Figure \(p. 52\)](#).
2. Change the view or figure (for example, rotate the view) either directly, or, in CFD-Post only, select one of the **Copy Camera From** commands from the viewer shortcut menu after right-clicking a blank area of the viewer.

View and figure objects are saved automatically when you switch to a different view or figure.

Deleting a Figure

The figure objects that you have created can be deleted using the tree view or the viewer shortcut menu. To use the viewer shortcut menu:

1. Switch to the figure that you want to delete.
2. Select the **Delete Figure** command from the viewer shortcut menu after right-clicking a blank area of the viewer.

Views

There are four default views that are handled specially. These are named: View 1, View 2, View 3, and View 4. These views will *not* be included in CFD-Post reports. However, any of these views can be viewed in any of the viewports, and you can create new views or figures that will be shown in reports.

Object Visibility

The visibility of an object is specified by the VIEW that should display the object, rather than the object specifying whether it is visible. That is, the object is made visible in a certain view--it is no longer a property of the object.

The VIEW object has a parameter named `Object Visibility List` that is set to a comma-separated list of object paths that should be visible in the VIEW object.

Here is an example of the VIEW object CCL to define the visibility for the view:

```
VIEW: View 1 Object Visibility List=/PLANE:Plane 1, /VECTOR:Vector 4END
```

Note

The `Object Visibility List` parameter should contain only object paths, and not object names.

Setting the `Visibility` parameter on an object has no effect.

For session files, there are command actions that enable you to change the visibility of objects:

- `>show`
- `>hide`
- `>toggle`

Each of these actions take an object name, path, or list of names and paths for which to show, hide, or toggle the visibility. Also, the actions optionally take a parameter that specifies the view to show the object. The visibility action parameters can alternatively take names or entire paths to specify the objects and the views.

Example 1: The following action will show the object /PLANE:Plane 1 in all existing views, including user figures.

```
>show Plane 1
```

Example 2: The following action will hide both /PLANE:Plane 1 and /PLANE:Plane 2 in view /VIEW:View 1.

```
>hide Plane 1, /PLANE:Plane 2, view=View 1
```

Example 3: If Plane 1 is visible, and Plane 2 is not visible in /VIEW:View 2, the following action will make /PLANE:Plane 1 not visible, and /PLANE:Plane 2 visible in view /VIEW:View 2:

```
>toggle Plane 1, Plane 2, view=/VIEW:View 2
```

Legends

There is a default legend for each VIEW object. The default legend is automatically created and deleted along with the view. By default, the default legend is made visible in the view it is associated with.

Chapter 5. CFD-Post Workflow

ANSYS CFD-Post enables you to qualitatively visualize and quantitatively analyze the processes taking place in a simulation; thus, the general workflow is:

1. Planning steps:
 - a. Decide which variables you want to study (your options are constrained by the variables that were solved in the process of creating the solver results file).
 - b. Determine where in the simulation you want to view those variables.
 - c. Decide how you want to display those variables, either qualitative displays (such as contour plots and charts) or quantitative analysis and displays (tables, etc.).
2. Production steps:
 - a. Load the solver results file for the simulation into CFD-Post.
 - b. Create any locations, variables, expressions, or functions required.
 - c. Optionally, publish the report, picture, or animation that shows the findings of the study to best advantage.

Loading and Viewing the Solver Results

When you load a solver results file into CFD-Post, the 3D Viewer displays the wireframe of the geometry, the mesh, the boundaries, and the domains. You can immediately display on any of the walls or boundaries the values of the variables that were imported with the geometry. You can also use the functions and macros that are supplied.

If those elements do not provide enough resolution, you can make use of CFD-Post's features to add:

- Locations where you can display or analyze variable values (points, point clouds, lines, planes, volumes, isosurfaces, vortex core regions, surfaces of revolution, polylines, user surfaces, and surface groups).
- Using selected locations, generate vectors, contours, streamlines and particle tracks to generate qualitative displays of the results.
- Expressions or macros that you can use to make new variables or to perform quantitative evaluation, integration, and averaging.

Qualitative Displays of Variables

The display of graphic objects (locations and qualitative displays) occurs in the 3D Viewer and the **Chart Viewer**. CFD-Post provides a wide range of control over the 3D Viewer, such as:

- How the graphical object is to be colored; either prescribed color or by variable. If colored by variable, how the color is mapped over the range of the selected variable.
- Rendering, including transparency, shading, lighting, specular, and texture.
- Display of lines and faces as well as geometric transformations including rotation, translation, scaling, reflection and instancing.
- Up to four viewports where the orientation of the objects in these miniature viewers can be controlled independently.

The Chart Viewer can display data as lines or as symbols.

Analysis

The quantitative analysis of variables can be displayed in the Table Viewer to enable you to display data and expressions.

Quantitative Analysis of Results

There are a variety of ways you can perform quantitative analysis of results loaded into CFD-Post:

- Use the **Expressions** workspace to make new variables as well numerically processing the results using a variety of math operations including averaging and integration.
- Use the **Calculators** workspace to:
 - Invoke analysis macros supplied for various applications including fan noise, turbomachinery performance, and so on
 - Calculate various measures of mesh quality
 - Probe the value of a function at a given location.
- Use the **Variables** workspace to make new variables.
- Use the **Turbo** workspace to initialize settings for turbomachinery applications.

Sharing the Analysis

There are a variety of ways to output the results of your analysis:

- Save a picture of the contents of the 3D Viewer in a variety of formats, including PNG, PostScript, and VRML.
- Publish a *report*, an HTML publication that includes information about the solver results file, the mesh, and the physics (as well as any other qualitative information, quantitative information, or comments you want to add).
- Produce an animation showing the changes in a variable over a range in the domain.

Typical Workflow

The following is a typical workflow, which you can simplify, reorganize, or extend suit your work patterns and objectives:

1. Start CFD-Post. ([Starting CFD-Post \(p. 3\)](#))
2. Load one or more results files. ([Load Results Command \(p. 57\)](#))
3. Create expressions ([Expressions Workspace \(p. 39\)](#)) and/or invoke macros ([Predefined Macros \(p. 167\)](#)) to perform the desired numerical processing of results.
4. Create any new variables that will be used for qualitative display. ([Variables Workspace \(p. 36\)](#))
5. Examine the existing locations (wireframe and surface boundaries) and create any additional locators required. ([Location Submenu \(p. 89\)](#))
6. For each locator, select visibility, method of coloring, rendering, and transformation.
7. Create any additional objects (such as lines, vectors, or contours) for quantitative display. ([CFD-Post Insert Menu \(p. 89\)](#))
8. For each object, select visibility, method of coloring, rendering, and transformation.
9. Use the 3D Viewer to explore the graphic objects and produce animations as required. ([CFD-Post 3D Viewer \(p. 43\)](#))
10. Create tables of data as required and display in the Table Viewer. ([Table Command \(p. 140\)](#))
11. Create any desired charts and display in the Chart Viewer. ([Chart Command \(p. 145\)](#))
12. Generate or edit any required titles, legends, or labels ([Legend Command \(p. 132\)](#) and [Text Command \(p. 128\)](#))
13. If required, save a picture of the contents in the 3D Viewer. ([Save Picture Command \(p. 69\)](#))
14. Display the report in the Report Viewer and/or modify the report as required. ([Report Command \(p. 68\)](#))
15. Optionally, publish the report to an HTML file. ([Report \(p. 24\)](#))
16. Optionally, save animations. ([Quick Animation \(p. 159\)](#))

For a more detailed illustration of the use of CFD-Post, see the [ANSYS CFX Tutorials \(p. 1\)](#).

Chapter 6. CFD-Post File Menu

This chapter describes the commands that are available from the **File** menu:

- [Load Results Command \(p. 57\)](#)
- [Close Command \(p. 59\)](#)
- [Load State Command \(p. 59\)](#)
- [Save State Command and Save State As Command \(p. 59\)](#)
- [Save Project Command \(p. 60\)](#)
- [Refresh Command \(ANSYS Workbench only\) \(p. 60\)](#)
- [Import Command \(p. 60\)](#)
- [Export Command \(p. 61\)](#)
- [ANSYS Import/Export Commands \(p. 65\)](#)
- [Report Command \(p. 68\)](#)
- [Save Picture Command \(p. 69\)](#)
- [Loading Recently Accessed Files \(p. 70\)](#)
- [Quit Command \(p. 70\)](#)
- [File Types Used and Produced by CFD-Post \(p. 70\)](#)

The file types that you can load and display are described in [File Types Used and Produced by CFD-Post \(p. 70\)](#).

Load Results Command

To load a results file (or files), select **File > Load Results** and browse to the file you want to load. CFX results files and CFX-Solver input files can be loaded from the **Load Results File** dialog box. For information on valid results and CFX-Solver input files, see [File Types Used and Produced by CFD-Post \(p. 70\)](#).

The **Load Results File** dialog presents you with the following options:

Edit case names

Enables you to change the case name as it appears in the **Outline** tree. The default case name is the filename (without the file type extension). Changing the case name does not affect the filename in the file system.

Keep current cases loaded

Controls whether to add to or replace the results that are currently in memory.

If **Keep current cases loaded** is selected, you can choose **Open in new view** to see the two cases side-by-side. If you choose to open the new case the *same* view, the two cases overlap and the title bar of the view displays **All cases**. However, you can use the Viewer's tool bar to manually display two views, then manually display change **All cases** to **Case 1 in View 1** and **Case 2 in View 2**. If you have loaded two cases and you select **Tools > Compare cases**, each case appears in a separate view, with the differences displayed in a third view.

Note

The **Keep current cases loaded** option is particularly useful to perform simultaneous post-processing of both fluid (CFX) and solid (ANSYS) results when a two-way Fluid-Structure simulation has been performed.

Clear user state before loading

Loading a results file causes all domain, boundary, and variable objects associated with the results file to automatically be created or updated by default. This would typically include the wireframe model of the geometry and all the boundary conditions created in CFX-Pre. The data associated with a variable is not loaded until the variable is actually used. Any existing objects (such as planes, vector plots) are plotted using the most recently loaded results, if possible. You can disable this behavior by selecting the **Clear user state before loading** check box.

Failing to clear the user state will cause CFD-Post to apply the state of the current file to the results file being loaded. For Turbo cases, it is important to ensure that settings such as the number of instances in 360 degrees

is correct (or to adjust the setting to be correct after the file is loaded) as CFD-Post does not automatically check to see if the user settings match between files.

Maintain camera position

Controls the loading behavior when you replace one case with another. When selected, the new case loads in the same orientation and size as the initial case; when cleared, the new case opens to fit into the view.

Load particle track data

Controls the loading of the particle tracks that exist in the case.

Load all configurations in a selected file as: | Load only the selected/last configuration

Controls how you load a multi-configuration (.mres) file or a results file (.res) that contains a run history (that is, a file that was produced from a definition file that had its initial values specified from a results file from a previous run and saved to the results file that you are loading).

- Choose **Load only the last results** to load only the last configuration of a multi-configuration results file, or only the last results from a results file that contains a run history.
- Choose **Load complete history as: a single case** to load all configurations of a multi-configuration run as a single case, or all of the results history from a results file that contains a run history. In either case, only one set of results will appear in the viewer, but you can use the timestep selector to move between results. This option is not fully supported.
- Choose **Load complete history as: separate cases** to load all configurations from a multi-configuration run into separate cases. If a results file with run history is loaded, CFD-Post loads the results from this file and the results for any results file in its run history as separate cases. Each result appears as a separate entry in the tree.

Note

To unload a set of results, right-click the case name in the tree view and select **Unload**.

Domain Selector Dialog

If the results file being loaded contains multiple domains, the **Domain Selector** dialog appears and you are prompted to specify which domains to load. Choosing to load only the domains you require will reduce memory usage and can speed processing time.

If you select the **Don't show this panel again** option, all domains will be loaded automatically on subsequent uses of the Load Results command. Note that you can always re-enable this dialog from the **Edit > Options > Files** panel (select **Show domain selector before load**).

Solution Units Dialog

When you load CFX files into CFD-Post, the solution units that were used by the CFX-Solver are automatically read from the file. When you load a file that does *not* store solution units (such as CFX-4 dump files, CFX-TASC files, FLUENT files, or ANSYS results files), by default the **Solution Units** dialog appears and you are prompted to specify the solution units. However, you can enable the **Don't prompt for Solution Units before loading results** toggle to suppress this prompt, in which case the default units of kilograms, meters, seconds, Kelvin, and radians will be used.

Once you have specified the units that were used in the results file, CFD-Post can convert those units to your preferred display units.

You set your preferred display units by selecting **Edit > Options**, then **Common > Units** from the menu bar; for details, see [Setting the Display Units \(p. 85\)](#).

Note

In CFD-Post, the temperature solution units must be an absolute scale (for example, Kelvin [K] or Rankin [R]); you cannot use Celsius and Fahrenheit. Temperature quantities elsewhere in ANSYS CFX can be set in Celsius and Fahrenheit.

Partial Results Files

When a partial results file is loaded, CFD-Post makes available the variables that exist in the full results file, but do not necessarily exist in the partial results file. Variables that do not exist in partial results files are not applicable to the currently loaded time step and are undefined.

You can optionally choose to use variable values that apply to the nearest full results file by changing an option in the **Options** dialog box. For details, see [Turbo \(p. 83\)](#).

Close Command

The **Close** command closes the currently loaded file, prompting you to save if necessary. CFD-Post remains open. To exit CFD-Post, use the [Quit Command \(p. 70\)](#).

Load State Command

Selecting **Load State** opens an existing state file.

Overwrite and Append

You can choose to either replace or append to the current state in CFD-Post. You can also choose to load the results file from which the state file was created. The results of these combinations are outlined below.

- **Replace current state** selected and **Load results** selected: The results file used to create the state file is opened, all existing objects are deleted, and new objects that are defined in the state file are created. The results are plotted on the new objects.
- **Replace current state** selected and **Load results** cleared: All existing objects are deleted and new objects that are defined in the state file are created. The results are plotted on the new objects using the existing results.
- **Add to current state** selected and **Load results** selected: The results file used to create the state file is opened. All objects defined in the state file and all existing objects are plotted with the new results. If objects in the state file have the same name as existing objects, the existing objects are replaced by those in the state file.
- **Add to current state** selected and **Load results** cleared: All objects defined in the state file are created and plotted using the current results. Existing objects are not removed unless they have the same name as an object in the state file, in which case they are replaced.

Results files may contain CEL expressions. If you have one or more results files already loaded and you are about to load a state file, you can prevent overwriting these expressions by clearing the **Load results** check box, then selecting the **Preserve current results expressions** check box.

Save State Command and Save State As Command

When CFD-Post is started from the ANSYS CFX Launcher, the **Save State** command produces a CCL file with a `.cst` file extension. All objects that currently exist in the system are saved to the state file.

Important

A state file is linked to the results file from which it was created by an absolute path. Therefore, do not change the location of the results file. The state file does not contain the geometry, mesh, or any results; these are loaded from the results file into CFD-Post.

If you have not saved a state file during your current CFD-Post session, selecting **Save State** opens the **Save State** dialog box where you can enter a file name.

If you have already saved a previous state, selecting **Save State** overwrites that file. To save a state to a different file name, you should select **Save State As** from the **File** menu.

When CFD-Post is started from ANSYS Workbench, the **Save Project** command writes the current state of the project.

Save Project Command


When CFD-Post is started from ANSYS Workbench, the **Save Project** command writes the current state of the project.

Refresh Command (ANSYS Workbench only)

Reads the upstream data, but does not perform any long-running operation.

Import Command

Using the **Import** dialog box, you can read in data for a polyline or surface.

Click *Browse*  to browse to the file to read the data from, or enter the file name.

Locator Names

If you import a generic file, the locator that is created is named using the locator name stored in the file, with the prefix `Imported`. If a locator with the same name already exists, the lowest integer greater than 1 that creates a unique name is appended. For example, if the imported file specifies a locator called `Line 1`, the locator that is created is called `Imported Line 1`, unless such a locator already exists, in which case the locator is called `Imported Line 1 1`. If the latter were the case, then importing another file with a locator called `Line 1` would cause the creation of a locator called `Imported Line 1 2`.

Importing Experimental Data in a Customized File

You can import experimental data in a customized file; typically this data will be for a user surface boundary profile or a polyline. The file structures are similar, except that the user surface description requires more information to define the boundary. Refer to [USER SURFACE Data Format \(p. 65\)](#) or [POLYLINE Data Format \(p. 64\)](#) as appropriate.

The example that follows shows experimental data that can be imported into CFD-Post.

Example 6.1. A Surface Data File for CFD-Post

```
[Name]
Experimental Data Set 1
[Data]
Node No., X[m], Y[m], Z[m], Press.[Pa], Vel.[m/s], Temp.[R], ...
0, -0.3, -0.3, -1.0, 0.0, 1.0, 0.224,
1, -1.0, -1.0, 1.0, 1.0, 2.0, 1.35987,
2, -1.0, 1.0, 1.0, 1.0, 3.0, -0.45,
3, -0.3, 0.3, -1.0, 0.0, 4.0, -5.82,
4, 0.3, -0.3, -1.0, 2.0, 5.0, 9.6323,
5, 1.0, -1.0, 1.0, 3.0, 6.0, 7.1859,
6, 1.0, 1.0, 1.0, 3.0, 7.0, -4.656234,
7, 0.3, 0.3, -1.0, 2.0, 8.0, 2.1237,
8, 0.0, 0.0, 2.0, 5.0, 9.0, 6.456,
[Faces]
# Faces are defined by their points, represented by the point IDs:
# 3 points for a tri-face and 4 points for a quad-face.
# The face normal is defined by the order of the points, so define
# all points in either a clockwise or counterclockwise direction
# to obtain a uniform face normal

0 - 3
# The face above is created from points 0 through 3
7 - 4
4 1 0
# Tri- and quad-faces may be combined
4 5 1
6 3 2
6 7 3
0 3 7 4
2 1 8
6 2 8
5 6 8
1 5 8
```


Export Command

The **Export** action enables you to export your results to a data file. You may export results for any available variable in CFD-Post on any defined locator. In the export file, data is written in blocks on a per locator basis in the order given by the locator list. Each block starts with lines listing the values of the selected variables at the locator points (one line corresponds to one point). The following two examples on how to export data are given at the end of this section:

- [Exporting Polyline Data \(p. 64\)](#)
- [Exporting Boundary Profile / Surface Data \(p. 65\)](#)

Export: Options Tab

File

The **File** setting specifies a file for the data to be exported to. You may type a filename or click *Browse*  to search for a file to export the results to, or enter a new file name.

Type

The **Type** setting has the following options:

Option	Description
Generic	Exports data to a file, writing the data in blocks for every locator. Each block starts with listing the values of the selected variables at the locator points. The Generic option displays the Export Geometry Information check box. For details, see Export Geometry Information Check Box (p. 62) .
BC Profile	Creates a boundary condition profile to be exported. The BC Profile option enables you to select a Profile Type .
Case Summary	Provides a short summary of the results file in xml format.

Locations

Locations is available only if either the **Generic** or **BC Profile** option is selected. The **Locations** setting specifies the locators for which the results of your variable is written. You can hold down the **Ctrl** key to select more than one locator and the **Shift** key to select a block of locators.

Name Aliases

Name Aliases is available only if either the **Generic** or **BC Profile** option is selected. The **Name Aliases** setting specifies custom naming of locators. To change the names of locators that will appear in the output file, insert a comma-separated list of names in the same order as locators.

Coord Frame

Coord Frame is available only if either the **Generic** or **BC Profile** option is selected. The **Coord Frame** setting specifies the coordinate frame relative to which the data will be exported. Information on creating a custom coordinate frame is available. For details, see [Coordinate Frame Command \(p. 130\)](#).

Unit System

The **Unit System** setting determines the units in which the data will be exported. By default, this will use the global units system selected in **Edit > Options**. For details, see [Setting the Display Units \(p. 85\)](#).

Boundary Vals

Boundary Vals is available only if either the **Generic** or the **BC Profile** option is selected. The **Boundary Vals** setting enables you to select Hybrid or Conservative boundary values. For details, see [Hybrid and Conservative Variable Values \(p. 165\)](#). Setting **Boundary Vals** to **Current** will select Hybrid/Conservative for each variable depending on the current setting. For details, see [Variables Details View \(p. 37\)](#).

Export Geometry Information Check Box

Export Geometry Information is available only if the **Generic** option is selected. Select this check box to export the x, y, z coordinate information of the locator at the beginning of the block.

Line and Face Connectivity Check Box

Line and Face Connectivity is available only if the **Generic** option is selected. Select this check box to export the connectivity information after the coordinate information in the file.

Node Numbers Check Box

Node Numbers is available only if the **Generic** option is selected. Select this check box to export the node numbers after the coordinate information in the file.

Profile Type

Profile Type is available only if the **BC Profile** option is selected. The **Profile Type** setting has the following options:

Option	Description
Inlet Velocity	Exports the Velocity Vector variable.
Inlet Total Pressure	Exports the Total Pressure, Total Temperature, and Velocity Direction variables.
Inlet Direction	Exports the Velocity Direction variable.
Inlet Supersonic	Exports the Velocity Vector, Pressure, and Temperature variables.
Outlet Pressure	Exports the Pressure variable.
Wall	Exports the Velocity Vector and Temperature variables.
Custom	Enables you to select custom variables to export from the Select Variable(s) list box.

Spatial Fields List Box

Spatial Fields is available only if the **BC Profile** option is selected. The **Spatial Fields** list box specifies the coordinate plane axes for the file being exported.

Select Variable(s) List Box

Select Variable(s) is available only if either the **Generic** or **BC Profile** options are selected. This list box is displayed for the **BC Profile** option only if the **Custom** option is selected for the **Profile Type** setting. This list box selects the variables to export. You can hold down the **Ctrl** key to select more than one variable or use the **Shift** key to select a block of variables.

Export: Formatting Tab

Vector Variables

Vector Variables is available only if either the **Generic** or **Case Summary** options are selected for the **Type** setting in the **Options** tab.

Vector Display Options

The **Vector Display** options enable you to select either **Components** or **Scalar**. The **Components** setting writes each component of a vector to the data file. The components appear inside the selected brackets. The **Scalar** option writes only the magnitude of a vector quantity.

Brackets

Brackets is available only if the **Components** option is selected. The **Brackets** setting selects the type of brackets to wrap around the components.

Include Nodes With Undefined Variable Check Box

Select the **Include Nodes With Undefined Variable** check box to write **Null Tokens** to the output file. Select the symbol used to denote undefined variable values. For details, see [Null Token \(p. 63\)](#).

Null Token

Null Token is available only if the **Include Nodes With Undefined Variable** check box is selected. The **Null Token** setting specifies the token to be displayed in the place of an undefined variable value. You may select the item used as a null token from a predefined list. Examples of a variables with undefined values include **Velocity** in a **Solid Domain** and a variable value at a point outside the solution domain, which can be created using a polyline, sampling plane or surface locator.

Some variables, including `Yplus` and `Wall Shear`, are calculated only on the boundaries of the domain and are assigned `UNDEF` values elsewhere.

If the **Line and Face Connectivity** check box is selected in the **Options** tab, then the **Null Token** is automatically exported.

Precision

The **Precision** setting specifies the precision with which your results are exported. The data is exported in scientific number format, and **Precision** sets the number of digits that appear after the decimal point. For example, 13490 set to a precision of 2 outputs `1.35e+04`. The same number set to a precision of 7 yields `1.3490000e+04`.

Separator

The **Separator** setting specifies the character to separate the numbers in each row.

Include File Info Header Check Box

Select the **Include File Info Header** check box to export comments at the top of the export file displaying the build date, date and time, and results file from which it is generated.

Include Header Check Box

Select the **Include Header** check box to include the list locators and a list of variables with their corresponding units. The header should be included for most export applications to ensure successful import into ANSYS CFX products.

Exporting Polyline Data

To save a polyline or line to a file:

1. Select **File > Export**.

The **Export** dialog box appears.

2. On the **Options** tab:

1. Set **Type** to **Generic**.
2. Select **Export Geometry Information** and **Export Connectivity**.

3. On the **Formatting** tab, under **Vector Variables**, ensure that the **Vector Display** option is set to **Scalar**.

Note that, on the **Formatting** tab, there is a **Null Token** field. This is used to indicate the string that should be written to represent values that are undefined.

If you want to make your own polyline file with a text editor, follow the format specified below.

For details, see [Polyline Command \(p. 110\)](#).

POLYLINE Data Format

The following is an abbreviated polyline file:

```
[Name]
Polyline 1
[Data]
X [ m ], Y [ m ], Z [ m ], Area [ m^2 ], Density [ kg m^-3 ]
-1.04539007e-01, 1.68649014e-02, 5.99999987e-02, 0.00000000e+00, ...
-9.89871025e-02, 3.27597000e-02, 5.99999987e-02, 0.00000000e+00,
.
.
.
[Lines]
0, 1
1, 2
```

```

.
.
.
[Name]
Polyline 2
.
.
.

```

The name of each locator is listed under the Name heading. Point coordinates and the corresponding variable values are stored in the Data section. Line connectivity data is listed in the Lines section, and references the points in the Data section, where the latter are implicitly numbered, starting with 0.

Comments in the file are preceded by # (or ## for the CFX-5.6 polyline format) and can appear anywhere in the file.

Blank lines are ignored and can appear anywhere in the file (except between the [`<data>`] and first data line, where `<data>` is one of the key words in square brackets).

Exporting Boundary Profile / Surface Data

Surfaces can be exported and then read into CFX-Pre as a boundary profile (or into CFD-Post as a **User Surface**).

USER SURFACE Data Format

An abbreviated user surface file, that could be read back into CFD-Post, is shown below:

```

[Name]
Plane 1
[Data]
X [ m ], Y [ m ], Z [ m ], Area [ m^2 ], Density [ kg m^-3 ]
-1.77312009e-02, -5.38203605e-02, 6.00000024e-02, 7.12153496e-06, ...
-1.77312009e-02, -5.79627529e-02, 5.99999949e-02, 5.06326614e-06,
.
.
.
[Faces]
369, 370, 376, 367, 375
350, 374, 367, 368, 351
.
.
.
[Name]
Plane 2
.
.
.

```

This is similar to the polyline data format described earlier ([POLYLINE Data Format \(p. 64\)](#)), except for the connectivity information. Instead of defining lines, this file defines faces (small surfaces), each by 3 (triangle) to 6 (hexagon) points. The points must be ordered to trace a path going around the face. For proper rendering, the faces should have consistent point ordering, either clockwise or counterclockwise. Each face is automatically closed by connecting the last point to the first point. Face connectivity data is listed in the Faces section and references the points in the Data section, where the latter are implicitly numbered, starting with 0.

ANSYS Import/Export Commands

The ANSYS **Import/Export** submenu has the following commands:

- [Import ANSYS CDB Surface \(p. 66\)](#)

- [Export ANSYS Load File \(p. 66\)](#)

The main purpose of the ANSYS import/export facility in CFD-Post is to allow fluid-structure interaction (FSI). The facility enables a mapping of boundary data stored in a CFX results file to a surface stored in an ANSYS mesh (.cdb) file.

For more information, see the *Mesh Deformation and Fluid-Structure Interaction* section of the *ANSYS CFX Reference Guide*.

Import ANSYS CDB Surface

The **Import ANSYS CDB Surface** dialog has the following options:

File

The **File** setting specifies the filename of the file to import. You can type the file path of the file, or click the *Browse*



icon to search for the file to import.

Length Units

The **Length Units** setting specifies what units the imported file will be in.

Specify Associated Boundary Check Box

Select the **Specify Associated Boundary** check box to specify an existing boundary to associate with the data in the *.cdb (ANSYS mesh) file. When importing ANSYS files, you should specify an associated existing boundary. If an export of ANSYS data is subsequently performed using the locator from the ANSYS file, data from the associated locator is mapped to, and exported with, the ANSYS file locator.

Boundary

The **Boundary** setting specifies the associated boundary for the imported file.

Maintain Conservative Heat Flows Check Box

Select the **Maintain Conservative Heat Flows** check box to ensure that the total heat flow for the boundary is equal to that of the imported ANSYS surface.

Read Mid-Side Nodes Check Box

Select the **Read Mid-Side Nodes** check box to map the ANSYS classic geometry for side nodes to CFX geometry. Using this feature can greatly extend the time it takes to load a file as reading the mid-side nodes increases the number of nodes that need to be mapped.


Export ANSYS Load File

The selected CFX data that is exported by CFD-Post is interpolated onto the ANSYS surface from the associated CFX boundary. The interpolated data is then exported to an ANSYS load file. For details on how to associate a boundary with an ANSYS surface, see [Specify Associated Boundary Check Box \(p. 66\)](#).

When verifying the load applied to the ANSYS surface, note that the *Pressure* variable available in CFD-Post is not the same as the element stress representing the static structural load; the element stress variables, *Normal Stress*, *Shear Stress*, and *Stress* (the latter being combination of *Normal Stress* and *Shear Stress*) are vector quantities, whereas the *Pressure* variable is a scalar quantity. The element stress variables are computed upon importing the ANSYS surface into CFD-Post, and can be used in plots.

Options Tab

File

The **File** setting specifies the file path and filename of the file to be exported. You may click the *Browse*  icon to select the name and location of the file to be exported. The default name is `export.sfe` or `export.xml` (depending on which File Format is chosen) and the default file path is your current working directory.

Location

The **Location** setting selects the ANSYS surface object to export, which is generated by importing an ANSYS .cdb file. For details, see [Import ANSYS CDB Surface \(p. 66\)](#).

Note

The ANSYS load file does not contain mesh coordinate data, and must be interpreted along with the .cdb file originally imported into CFD-Post.

Unit System

The **Unit System** setting specifies the units for the exported data. By default, this uses the global units system selected in **Edit > Options**. For details, see [Setting the Display Units \(p. 85\)](#).

Boundary Vals

The **Boundary Vals** setting specifies Hybrid or Conservative boundary values. If **Boundary Vals** is set to Current, this setting is picked up from each variable. For details, see [Variables Details View \(p. 37\)](#).

Export Data

The **Export Data** setting has the following options:

Option	Description
Normal Stress Vector	Exports Normal Stress variable data onto imported surf154 surfaces. Normal Stress is a vector variable calculated from the normal component of Force.
Tangential Stress Vector	Exports Shear Stress variable data onto imported surf154 surfaces. Shear stress data is calculated from the tangential component of Force.
Stress Vector	Exports Stress variable data onto imported surf154 surfaces. Stress data is calculated by vector summing the normal stress and shear data.
Heat Transfer Coefficient	Exports convection variable data onto imported surf152 surfaces. When selected, the Specify Reference Temperature check box will appear. See Specify Reference Temperature (p. 67) , below.
Heat Flux	Exports the Heat Flux variable data onto the surf152 surfaces.
Temperature	Exports the Temperature variable data on the nodes of the imported surface.

Fluids

Fluids is available only if either the Tangential Stress Vector or Stress Vector options are selected. The **Fluids** setting specifies which fluids, or All Fluids, that will affect the elements shear or stress values.

Specify Reference Temperature

Specify Reference Temperature is available only if the **Heat Transfer Coefficient** option is selected. Select this check box to enable you to specify a fixed reference temperature value or expression.

1. If you specify a reference temperature, then the exported heat transfer coefficient is calculated based on Heat Flux and Temperature data.

Note that the variable “Surface Heat Transfer Coef” is not recognized by CFD-Post for 1-way FSI.

2. If you do not specify a reference temperature, the exported data is based on the Wall Heat Transfer Coefficient and Wall Adjacent Temperature.

For ANSYS FLUENT Cases: To transfer HTC from ANSYS FLUENT cases without specifying a reference temperature (method 2 above), the following variables have to be exported to the DAT/CDAT file:

- Wall Func. Heat Tran. Coef (which will be converted to the CFX variable Wall Heat Transfer Coefficient)
- Temperature. In ANSYS FLUENT, the wall adjacent temperature is calculated by averaging the adjacent cell temperatures to the wall nodes.

Formatting Tab

The **Formatting** tab enables you to specify only a precision value. This setting is the same for the **Export** command. For details, see [Export Command \(p. 61\)](#).

ANSYS Import/Export Example: One-Way FSI Data Transfer

You can perform one-way FSI operations manually (by exporting CDB files from Mechanical APDL, importing the surface in CFD-Post, and exporting the SFE commands).

To create an ANSYS load file using CFD-Post to transfer FSI data:

1. Load the fluids results file, from which you wish to transfer results, into CFD-Post
2. Select **File > ANSYS Import/Export > Import ANSYS CDB Surface**. The **Import ANSYS CDB Surface** dialog appears.
3. In the **Import ANSYS CDB Surface** dialog, either:
 - Select the CDB file that specifies the surface mesh of the solid object to which to transfer data. Also select the **Associated Boundary** for the surface to map onto, and make other selections as appropriate.
 - Select the XML document that provides all transfer information. Click **OK**, and the surface data is loaded.
4. Select **File > ANSYS Import/Export > Export ANSYS Load File**. The **Export ANSYS Load File** dialog appears.
5. In the **Export ANSYS Load File** dialog, select a filename to which to save the data. For the **Location** parameter value, select the imported ANSYS mesh object. Under **File Format** select **ANSYS Load Commands (FSE or D)**. (Alternatively, you can select **WB Simulation Input (XML)** to get XML output.) Also select the appropriate data to export: Normal Stress Vector, Tangential Stress Vector, Stress Vector, Heat Transfer Coefficient, Heat Flux, or Temperature. Click **Save**, and the data file is created.

Report Command

The **File > Report** menu item has the following options:

Report Templates

Invokes the **Report Templates** dialog, where you can browse the list of existing templates or add (register) a new template. The existing templates are for turbomachinery simulations.

To learn how to use templates, see [Report Templates \(p. 28\)](#).

Load 'Generic Report' Template

Reloads the default template.

Refresh Preview

Updates the report that is displayed on the **Report Viewer**. You need to do this after making changes to your report.

This command is equivalent to clicking on the **Refresh** icon at the top of the **Report Viewer**.

Publish

Displays the **Publish Report** dialog, where you can configure the format and name of your report. See [Publishing the Report \(p. 35\)](#) for details.

This command is equivalent to clicking on the **Publish** icon at the top of the **Report Viewer**.

To learn more about publishing a report, see [Report \(p. 24\)](#).


Save Picture Command

Select **File > Save Picture** to open the **Save Picture** dialog box, which enables you to save the current contents of the viewer to a file.

Options Tab

The **Options** tab has the following settings:

File

Enables you to specify the file name of the file. You may enter the file name and path into the **File** text box, or click the *Browse*  icon and search for the directory in which the file is to be saved.

Format

Can be set to one of the following:

PNG

Portable Network Graphics, a file format intended to replace the GIF format. It was designed for use on the World Wide Web and retains many of the features of GIF with new features added.

CFX Viewer State (3D)

A 3D file format that can be read back directly into a standalone CFX Viewer.

JPEG

A compressed file format developed for compressing raw digital information. File sizes are small and lack detail, so this method is not recommended for line drawings.

Windows Bitmap

A file type (*.bmp) that is usually large and does not adjust well to resizing or editing. This file type does retain all of the quality of the original image and can be easily converted to other formats.

PPM

Stands for Portable Pixel Map, a file format similar to a Windows Bitmap.

PostScript

PostScript (*.ps), a file format recommended for output to a printer.

VRML (3D)

Virtual Reality Modeling Language (*.wrl), a file format used to present interactive three-dimensional views and that can be delivered across the World-Wide Web. The only supported VRML viewer is Cortona from Parallel Graphics (see <http://www.parallelgraphics.com/products/cortona/>).

Use Screen Capture Check Box

Select the **Use Screen Capture** check box to save a screen capture of the 3D Viewer as the output. Note that **Face Culling** affects **Screen Capture** mode only. The **Render** settings **Draw as Lines** and **Draw as Points** are not picked up in screen capture mode.

CFD-Post always attempts to capture only the viewport when this check box is selected. On some systems with particular window systems and OpenGL installations, other objects (such as forms) may also be captured and appear in the image. If you experience this behavior, you may move any other forms away from the viewport before capturing the image, or disable the screen capture method.

White Background Check Box

Select the **White Background** check box to color white objects black, and black objects white, in the image file. All objects are affected by this toggle, so slightly off-white and off-black objects are also inverted. This setting does not work for VRML files.

Enhanced Output (Smooth Edges) Check Box

Smooths the edges of the image.

Use Screen Size Check Box

Select **Use Screen Size** to use the Viewer size for the image. Clear this check box to specify a custom image size in pixels.

Width/Height

These settings are available only if the **Use Screen Size** check box is cleared. These settings specify the width and height of the image in pixels.

Scale (%)

Specifies a scale, in percent, to make the image being saved.

Image Quality

Specifies the JPEG compression number for the output image. For a better quality image, increase the value.

Tolerance

A non-dimensional tolerance value used in face sorting when generating hardcopy output. Larger values are faster, but may cause defects in the resulting output.

Important

When a clip plane is coincident with regions, boundaries, or interfaces that are planes, the results of a **Save Picture** command may not match what you see in the 3D Viewer (depending on the orientation of the case). In this situation, set the **Use Screen Capture** check box.

Loading Recently Accessed Files

CFD-Post saves the file paths of the last six results files, state files, and session files. To re-open a recently used file, select it from the **Recent Results Files**, **Recent State Files**, or **Recent Session Files**, as appropriate.

Quit Command

To exit from CFD-Post, select **File > Quit** from main menu. Objects created during your CFD-Post session are not automatically saved. If there is no state file in memory, the state was changed since the file was opened, or since the last state save, a dialog asks whether you want to save the state before closing. For details, see [Save State Command and Save State As Command \(p. 59\)](#).

File Types Used and Produced by CFD-Post

This section describes the file types used by CFD-Post and the software that outputs those file types:

- [ANSYS CFX Files \(p. 70\)](#)
- [ANSYS Meshing Files \(p. 71\)](#)
- [CFX-4 Dump Files \(p. 72\)](#)
- [CFX-TASCflow Results Files \(p. 72\)](#)
- [ANSYS Files \(p. 74\)](#)
- [CGNS Files \(p. 75\)](#)
- [ANSYS FLUENT Files \(p. 76\)](#)

ANSYS CFX Files

Case Files (.cfx)

A case file is generated when you save a simulation in CFX-Pre. The case file contains the physics data, region definitions, and mesh information for the simulation and is used by CFX-Pre as the 'database' for the simulation setup.

A case file is a binary file and cannot be directly edited.

CFX-Mesh Files (.gtm)

GTM files (.gtm) contain mesh regions that can be used to set up a simulation in CFX-Pre or viewed in CFD-Post.

CFX-Solver Input Files (.def, .mdef)

A CFX-Solver input file is created by CFX-Pre. The input file for a single configuration simulation (.def) contains all physics and mesh data; the input file for multi-configuration simulations (.mdef) contains global physics data only (that is, Library and Simulation Control CFX Command Language specifications). An .mdef input file is supplemented by Configuration Definition (.cfg) files that:

- Are located in a subdirectory that is named according to the base name of the input file
- Contain local physics and mesh data.

Note

Use the `-norun` command line option (described in the *ANSYS CFX-Solver Manager User's Guide*) to merge global information into the configuration definition files, and produce a CFX-Solver input file (.def) file that can be run by the CFX-Solver.

CFX-Solver Results Files (.res, .mres, .trn, .bak)

Intermediate and final results files are created by the CFX-Solver:

- Intermediate results files, which include transient and backup files (.trn and .bak, respectively) are created while running an analysis.
- Final results files for single and multi-configuration simulations (.res and .mres, respectively) are written at the end of the simulation's execution. For multi-configuration simulations, a configuration result file (.res) is also created at the end of each configuration's execution.

Each results file contains the following information as of the iteration or time step at which it is written:

- The physics data (that is, the CFX Command Language specifications)
- All or a subset of the mesh and solution data.

CFX-Solver Backup Results Files (.bak)

A backup file (.bak) is created at your request, either by configuring the settings on the **Backup** tab in **Output Control** in CFX-Pre, or by choosing to write a backup file while the run is in progress in the CFX-Solver Manager.

CFX-Solver Transient Results Files (.trn)

A transient results file (.trn) is created at your request, by configuring the settings on the **Output Control** > **Trn Results** tab in CFX-Pre.

CFX-Solver Error Results Files (.err)

An error results file (.err) is created when the CFX-Solver detects a failure and stops executing an analysis. The .err file can be loaded into CFD-Post and treated the same way as a .bak file, but if the CFX-Solver encounters another failure while writing the .err file, it may become corrupted and accurate solutions cannot be guaranteed.

Session Files (.cse)

Session files are produced by CFD-Post and contain CCL commands. You can record the commands executed during a session to a file and then play back the file at a later date. For details, see [New Session Command \(p. 87\)](#).

You can also modify session files in a text editor.

State Files (.cst)

State files are produced by CFD-Post and contain CCL commands. They differ from session files in that only a snapshot of the current state is saved to a file. You can also write your own state files using any text editor. For details, see [Save State Command and Save State As Command \(p. 59\)](#) and [Load State Command \(p. 59\)](#).

ANSYS Meshing Files

Both .cmdb files (created in the Meshing application) and .dsdb files (created in Simulation) behave the same in CFD-Post as .gtm (and .def) files. For details, see [File Types Used and Produced by CFD-Post \(p. 70\)](#).

Note

- You must have ANSYS Workbench installed in order to be able to load ANSYS Meshing files (cmdb and dsdb) into CFX-Pre or CFD-Post.
- CFD-Post does not support .cmdb files generated by the Meshing application prior to Release 11.0.

CFX-4 Dump Files

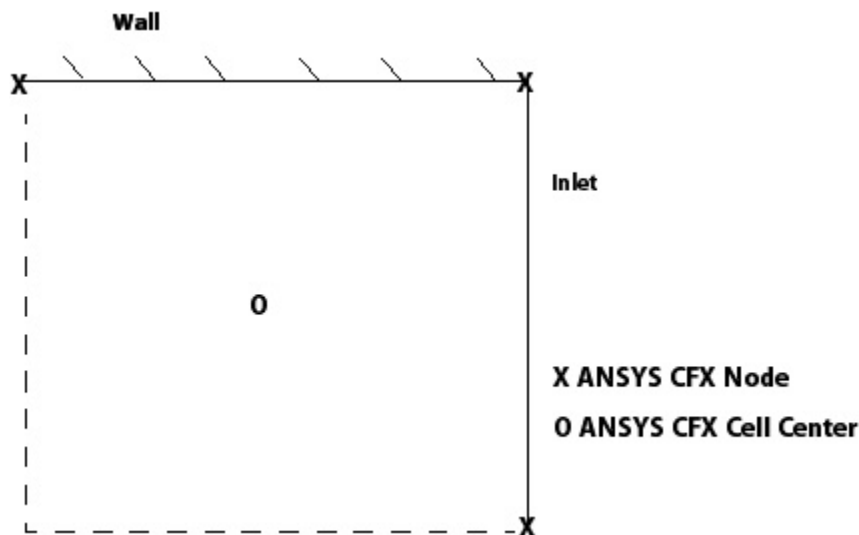
CFD-Post can load dump files (*.d*mp*) created by CFX-4. When you load the results, you may be prompted to provide the solution units that were used in the simulation. For details, see [CFD-Post Solution Units \(p. 82\)](#).

Limitation with CFX-4 Files

There is an important limitation with CFX-4 results files that should be noted: the CFX-4 Solver does not output minimum/maximum ranges for each of the calculated variables. These ranges are calculated when the results file is loaded by CFD-Post. Calculating the range for a very large problem would, however, require prohibitively large amounts of CPU time. As a result, range values are calculated for the loaded time step only. This means that values that appear as global range, are in fact ranges that exist for that time step only.

Interpolation of Results

The CFX-4 Solver uses a cell-based solution method, whereas CFX-Solver uses a node-based solution method. Possible problems can be encountered at the intersection of patches, such as in the following diagram:



When interpolating from cell-centered to node-centered data, the data at a given node is affected by all surrounding cells. In order to get the correct behavior at boundary patches, a priority number is assigned to each patch by CFD-Post. This means that, for example in the above diagram, if the wall has a higher priority number than the inlet, the value of the node is interpolated from the wall value of the CFX solution. When considering a situation in 3D, the priority of all faces is read and interpolation occurs from the face(s) with the highest priority. CFD-Post uses the same default values for every problem, so there are cases in which accuracy can be compromised. These errors can be minimized by refining the grid density in the region around problem areas.

Quantitative calculations can suffer a loss of accuracy due to the limitation described above. The results of mass flow calculations should, therefore, be assumed to be approximations for the purposes of quantitative analysis.

CFX-TASCflow Results Files

CFD-Post can import CFX-TASCflow results files for postprocessing. When you load the results, you may be prompted to provide the solution units that were used in the simulation. For details, see [CFD-Post Solution Units \(p. 82\)](#).

If TBPOST_COMP_X parameters (where X is the component number) are defined in the GCI file, either on their own or within the TBPOST_COMP_LIST macro, they are used to obtain the list of turbo components to load. Each defined component is treated as a separate domain inside CFD-Post, allowing for their individual turbo initialization.

Note

If this list exists but is incomplete, only the defined components are loaded. If you cannot load a turbo file, it may be due to an incompatibility in the component definition. As a workaround, remove TBPOST related parameter and macro definitions from the GCI file.

Limitations with CFX-TASCflow Files

- When loading `rso` or `grd` files, `bcf` and `prm` files are required.
- `bcf` files must be complete (must contain all domain and boundary condition definitions).
- When using the Turbo Post functionality, separate region names are required for the following 2D location types:
 - Hub
 - Shroud
 - Blade
 - Inlet
 - Outlet
 - Periodic1
 - Periodic2

If these regions have not been specified separately (that is, hub and blade comprise of one region), you will either need to recreate them in the CFX-TASCflow pre-processor or specify the turbo regions from line locators. For details, see [Initialize All Components \(p. 187\)](#).

- Mass flow and torque are not written to `rso` files by CFX-TASCflow. These values are approximated in CFD-Post and may not be suitable for use in a formal quantitative analysis.

Variable Translation

By default, CFD-Post does not modify the variable names in the `rso` file. If you want to use all of the embedded CFD-Post macros and calculation options, you need to convert variable names to CFX types. You can convert the variable names to CFX variable names before reading the file by selecting the **Translate variable names to CFX-Solver style names** check box in the **Edit > Options > Files** menu.

Translation is carried out according to the following:

CFX-TASCflow	Translated to CFX Variable
T	Temperature
TKE	Turbulent Kinetic Energy
EPSILON	Turbulence Eddy Dissipation
VISC_TURBULENT	Eddy Viscosity
VISC_MOLECULAR	Molecular Viscosity
CONDUCTIVITY	Thermal Conductivity
SPECIFIC_HEAT_P	Specific Heat Capacity at Constant Pressure
SPECIFIC_HEAT_V	Specific Heat Capacity at Constant Volume
PTOTAL	Total Pressure
PTOTAL_REL	Total Pressure in Rel Frame
PTOTAL_ABS	Total Pressure in Stn Frame
POFF	Pressure Offset
P_CORRECTED	Pressure Corrected
TTOTAL	Total Temperature

CFX-TASCflow	Translated to CFX Variable
TTOTAL_REL	Total Temperature in Rel Frame
TTOTAL_ABS	Total Temperature in Stn Frame
TOFF	Temperature Offset
T_CORRECTED	Temperature Corrected
TAU_WALL	Wall Shear
YPLUS	Solver Yplus
Q_WALL	Wall Heat Flux
P	Pressure
PRESSURE_STATIC	Static Pressure
PRESSURE_REL	Relative Pressure
MACH	Mach Number
MACH_ABS	Mach Number in Stn Frame
MACH_REL	Mach Number in Rel Frame
HTOTAL	Total Enthalpy
HTOTAL_REL	Total Enthalpy in Rel Frame
HTOTAL_ABS	Total Enthalpy in Stn Frame
ENTHALPY	Static Enthalpy
ENTROPY	Static Entropy
FE_VOLUME	FE Volume
CONTROL_VOLUME	Volume of Finite Volumes
DIST_TURB_WALL	Wall Distance

ANSYS Files

ANSYS solver files are created from the ANSYS solver. CFD-Post is able to read results for temperature, velocity, acceleration, magnetic forces, stress, strain, and mesh deformation. The ANSYS solver files may have load-step variables and time steps; CFD-Post will represent both as time steps. The valid file types are *.rst, *.rth, *.rmg, *.rfl, *.inn, *.brfl, *.brmg, *.brst, *.brth, *.inp, *.cdb.

When ANSYS solver files are read together with CFX-Solver files, fluid dynamics and solid mechanics results can be analyzed simultaneously. For details on how to load multiple files, see [Load Results Command \(p. 57\)](#).

The deformations due to change in temperature and stress/strain of the mesh can be amplified by using the **Deformation** option available by right-clicking the viewer background. For details, see [CFD-Post 3D Viewer Shortcut Menus \(p. 46\)](#).

When you load the results, you may be prompted to provide the solution units that were used in the simulation. For details, see [CFD-Post Solution Units \(p. 82\)](#).

Limitations with ANSYS Files

There are some important limitations with ANSYS results files:

- The ANSYS Solver does not output minimum/maximum ranges for each of the calculated variables; these ranges are calculated when the results file is loaded by CFD-Post. Calculating the range for a very large problem would, however, require prohibitively large amounts of CPU time. As a result, range values are calculated for the loaded time step only. This means that values that appear as global range, are in fact ranges that exist for that time step only, at first. As more time steps are added, the global range is extended accordingly. If you want to enable the calculation of true global ranges (and incur the potentially large CPU time each time you load a non-ANSYS CFX file), you can do this by selecting **Edit > Options** and selecting **Pre-calculate global variable ranges**, under Files. For details, see [Files \(p. 82\)](#).

- CFD-Post plots only ANSYS variables that exist in RST files; unlike ANSYS, it will not calculate other variables automatically. Therefore, some variables that you would expect to be able to plot (as in ANSYS) either will be missing or will have all zero values in CFD-Post.
- By default, an ANSYS results file does not contain the definitions of any components that you may have created in the simulation set up and so these will not be available as regions for plotting in CFD-Post. However, it is possible to produce an additional "components" file that does contain these definitions. If CFD-Post finds a file with the name <filename>.cm in the same directory and with the same filename (excluding the file extension) as the ANSYS results file, then it will read component definitions from this file. For instance, if you are post-processing the ANSYS results file *OscillatingPlate.rst*, CFD-Post will look for the file *OscillatingPlate.cm* in the same directory to find component definitions. You can get ANSYS to write the components file by including a command *CMWRITE, <jobname>, cm* in your ANSYS input file, before the *SOLVE* command. If .cm files are to be loaded into CFD-Post, job names need to be consistent across restarts, input file processing, and regular runs. If your ANSYS results file was produced by an ANSYS Multi-field run that had its multi-field commands set up in CFX-Pre, then this command is already added automatically into the resulting ANSYS input file as part of the input file processing.
- Components files (CM files) must have been output in blocked format (which is the default output format). Refer to the ANSYS documentation to learn how to control the ANSYS output format.
All regions from components files are read as surfaces. If a region is volumetric, CFD-Post will read the outer surface only.
CFD-Post will read only nodal components. Components that consist of elements will be ignored.
- CFD-Post can read a limited number of ANSYS results files that contain shell elements only. It depends on the problem set-up details as to whether a file can be successfully read or not. CFD-Post cannot read any ANSYS results files that contain no 3D or shell elements.
- While the ANSYS MFX solver requires nodal and element solution data to be present in the results file for all time steps on the FSI interface, it does not require this data in other places in the geometry. Excluding that data means that the results file size can be reduced when you use ANSYS. When you use CFD-Post, on the other hand, all data must be available at all nodes/elements for all time steps.
- When reading RST files, CFD-Post ignores mid-side nodes and duplicate nodes. The latter situation occurs when a case has multiple bodies with matching mesh on the interfaces. The simulation picks up duplicate nodes and plots accordingly, giving a discontinuous plot. However CFD-Post picks up only one of the nodes, which causes one domain to appear to spill into the next.
- In ANSYS, simulation characteristics such as maximum values are derived from actual local node values. In CFD-Post values need to be presented on global nodes, therefore CFD-Post takes a simple average from all shared elements' local values. When compared the two calculations will be similar, but not exactly the same.

CGNS Files

ANSYS CFD-Post has limited support for reading meshes and solutions from CGNS (CFD General Notation System) Version 2.4 files. Extensions for such files are typically .cgns and .cgs. The following shows the supported and unsupported features of CGNS:

Supported	Not supported
3D problems	1D and 2D problems
Elements of the following types: TRI_3, QUAD_4, TETRA_4, PENTA_6, PYRA_5, HEXA_8 and MIXED	Elements of the following types: TRI_6, QUAD_8, QUAD_9, TETRA_10, PENTA_15, PENTA_18, PYRA_14, HEXA_20, HEXA_27
Base #1	Base selection
Boundary conditions written as collections of nodes	Boundary conditions written as groups of 2D elements (Faces)
Steady State solutions (solution #1)	Transient solution
	Regions

Supported	Not supported
	Zone connections
	Periodic crossings

ANSYS FLUENT Files

CFD-Post can load ANSYS FLUENT version 6 and version 12.0 (preferred) result files (.cas, .cas.gz, .dat, .dat.gz, .cdat, .cdat.gz) for postprocessing. For results generated using FLUENT prior to version 12.0, load the results into ANSYS FLUENT 12.0, run at least one iteration, save the results, and then load these in CFD-Post.

CFD-Post does not calculate derived variables, thus only variables available in the file can be used. However, you can export any variable to the dat or cdat files from ANSYS FLUENT 12.0.

ANSYS FLUENT files store variable values at the cell centers of the mesh. CFD-Post interpolates these values to nodes using an interpolation method similar to the interpolation of CFX-4 files. For details, see [Interpolation of Results \(p. 72\)](#). The valid file extensions are listed in the **File type** setting of the **Load Results File** dialog box.

In CFD-Post, a wall boundary takes precedence over other boundaries, so all wall nodes will have wall values irrespective of whether they are on any other boundary.

Note

You must select only one of the FLUENT files in the **Load** dialog (normally the final timestep's dat or cdat file); other related files are loaded automatically.

Limitations with ANSYS FLUENT Files

ANSYS FLUENT files supported with the following limitations:

General Limitations

- For compressible flows, the values computed by CFD-Post for Total Temperature, Total Pressure, and Total Enthalpy are incorrect; therefore you should export these variables from ANSYS FLUENT.
For incompressible flows, the values computed by CFD-Post for Total Temperature, Total Pressure, and Total Enthalpy are different from the values for these variables from ANSYS FLUENT, but the CFD-Post values are more accurate. If it is important for the values to be the same in CFD-Post and ANSYS FLUENT, export the variables from ANSYS FLUENT; otherwise, let CFD-Post calculate new values.
- An ANSYS FLUENT .dat file does not contain velocity or other face data on symmetry boundaries, which causes vortex cores to be incorrect for those cases. However, .cdat files do contain velocity on symmetry boundaries, so to get correct vortex cores you must use a .cdat file with the .cas file.
- Value ranges reported by the function calculator may differ from the values shown by a contour. The function calculator results are based on cell/face values while contours show values based on node values.
- Plots created in CFD-Post are based on node values and not cell/face values. This results in undesired smoothing of result on the edges where nodes are shared by two objects.
- CFD-Post does not account for surface tension forces.
- Certain real gas properties are not available in CFD-Post for use: gas constant, molecular viscosity, specific heat, and sound speed.
- Wall Heat Flux values reported by CFD-Post for moving and deforming meshes cases will not match those for ANSYS FLUENT. This is because ANSYS FLUENT adds pressure work to get the energy balance.
- For ANSYS FLUENT files using the energy model, Heat Flux is available for all boundaries and Wall Heat Flux is available only for walls. The values of these two variables will be same on walls.
- You need to be careful choosing geometry names in ANSYS FLUENT when the file will be read in CFD-Post. The geometry names must not contain special characters such as '-', '|', ':' and so on. All such characters will be replaced by a space (which is allowed in names in CFD-Post).

- Averaging of vector quantities to nodes differs between CFD-Post and ANSYS FLUENT. In ANSYS FLUENT, vector magnitudes are averaged to nodes explicitly; in CFD-Post, only vector components are averaged to nodes, while the magnitude is calculated from the components at the nodes. The two magnitudes will differ in cases with sharp vector gradients or high face angles (usually due to a coarse mesh).

For example, if a node has four faces attached that have shear stresses in directions radially away from the node, in CFD-Post the shear stress values at the node will be much smaller in magnitude compared with the face stresses because the stresses in opposite directions cancel out. In FLUENT, the direction is ignored and only magnitude is taken into account while calculating the stress magnitude at the node.

- CFD-Post can read files written from ANSYS FLUENT, but the reading of mesh files written from TGrid or GAMBIT is not supported and may lead to a crash.
- Velocity magnitude values for ANSYS FLUENT in CFD-Post are not in good agreement with ANSYS FLUENT results for cases with multiple-frame-of-reference or sliding-mesh models.

For cases solved with relative velocity:

- The "Velocity in Stn Frame" plotted in ANSYS CFD-Post is equivalent to "Velocity Magnitude" in ANSYS FLUENT.
- There is no ANSYS FLUENT equivalent for the CFD-Post variable "Velocity" as this represents a relative velocity in the local reference frame of the domain (which is not available for post-processing in ANSYS FLUENT).
- There is no CFD-Post equivalent for ANSYS FLUENT's "Relative Velocity". In ANSYS FLUENT, "Relative Velocity" is always relative to a global frame of reference (which you can select in ANSYS FLUENT **Reference Values** panel; if no reference frame is selected, an "Absolute Velocity" is used, not a "Reference Velocity").
- The Normalized Force variable is not calculated for Polyline locators in ANSYS FLUENT cases.
- The variable Boundary Heat Flux Sensible is available only for boundary types velocity-inlet, mass-flow-inlet, pressure-inlet, pressure-outlet, pressure-far-field, and outflow.
- Quantitative evaluation of user-defined variables in CFD-Post are not accurate as expressions are evaluated only at nodes instead of cell centers.
To get cell-based user variables, you can explicitly export the custom field function from ANSYS FLUENT to a .cdat or .dat file.
- The loading of particle tracks (either track data in the data file or separate track files) is not supported.
- For transient FLUENT cases, there is no support for adding or removing time steps in the timestep selector.
- There is no support for loading of a subset of domains. All domains are always loaded.
- When loading ANSYS FLUENT results, CFD-Post does not calculate global ranges by default as this would be too time-consuming (there is a warning to this effect when you load a FLUENT case). However, when the variable is used for the first time (for example, when it is plotted), and as timesteps are loaded, the global range should be continually updated.
- For some cases, the fluxes (Mass Flow()@<surface> or AreaInt(Boundary Heat Flux)@<surface>) from CFD-Post are different from the values reported by **Flux Reports** panel from ANSYS FLUENT. This is due to some additional physics model-based calculations done by ANSYS FLUENT that are not available in CFD-Post. However, you can use the ANSYS FLUENT **Surface Integral** or **Volume Integral** panel results for comparison with CFD-Post.
- Grid interfaces from ANSYS FLUENT versions 6.3 and older are not supported by CFD-Post. If your CAS file has old grid interfaces, read the CAS and DAT file into the ANSYS FLUENT Release 12.0, run at least one iteration, and save the file to change to the new grid interfaces. This will convert grid interfaces to use the virtual polygon method and make the file readable in CFD-Post. Attempting to read old grid interfaces may cause CFD-Post to crash.
- For cases with 1:1 interfaces, due to a difference in the handling on nodes at these interfaces, the number of nodes reported by CFD-Post will be different than the number reported by FLUENT. However, the number of cells should match.
- Data generated using the DBNS solver of ANSYS FLUENT versions 6.2 and older use a sign convention for area that is not consistent with CFD-Post. In these instances, read the file into ANSYS FLUENT Release 12.0,

run at least one iteration, and save the file so that the sign convention for area is consistent with CFD-Post. Using data read from older files may cause CFD-Post to generate erroneous results when the results are dependent on area.

- To post-process forces or fluxes using the DBNS solver of ANSYS FLUENT for cases from versions prior to Release 12.0, you must read the case into ANSYS FLUENT Release 12.0, iterate at least once, and then write out the case data.
- A DBNS solver with laminar flow will have zero shear stress on all walls. Force calculations will not include viscous component in such cases.
- In the cavitation model in ANSYS FLUENT, the minimum value for Pressure is limited by the cavitation pressure; (this is not done in CFD-Post).
- For some cases (for example, shell conduction model), the number of cells/elements reported by ANSYS FLUENT is more than that of CFD-Post. This difference is due to the additional cells ANSYS FLUENT creates internally for solving some physics; these are never written into the case file. ANSYS FLUENT reports include these cells as well.
- CFD-Post does not smooth out values across non-conformal interface boundaries; that is, there must be a 1-1 mapping of nodes across the interface. As a result, contour and color plots as well as iso-surfaces are discontinuous across these interfaces.
- CFD-Post and ANSYS FLUENT display contours differently in the vicinity of a hanging node. ANSYS FLUENT takes values from cells only on one side, causing a discontinuity of contours. In CFD-Post, the hanging node is made to be a conformal node and takes values from cells on both sides, making a smoother contour.
- A periodic surface in ANSYS FLUENT is actually a pair of surfaces. In CFD-Post this pair appears as a Periodic object and a corresponding Periodic Shadow. When looking at quantitative results in CFD-Post, you need to look at a surface group that contains the “periodic/periodic-shadow” to see output that is in agreement with ANSYS FLUENT's results.
- CFD-Post will not display any shear stress values on coupled non-conformal interfaces as shear stresses are undefined on such interfaces.
- ANSYS FLUENT .cas, .dat, and .cdat files do not contain the units for user-defined scalars, user-defined memory, or custom field functions, so these will be dimensionless in CFD-Post.
- CFD-Post reads User-Defined Materials (UDM) and User-Defined Scalars (UDS) as follows:
 - When CAS/DAT files are read into CFD-Post, UDM/UDS variables will appear with names as "User Defined Memory 0"/"Scalar 0".

When CAS/CDAT files are read into CFD-Post, CFD-Post will show all UDM/UDS variables that were exported to the CDAT file.

Turbo Limitations

- CFD-Post can initialize turbo space only for domains that are enclosed with inlet, outlet, hub, and shroud regions. For more complex geometries you must set up the problem such that the region of interest is isolated into a separate domain that can be initialized in CFD-Post.
- When choosing a report template for an ANSYS FLUENT turbo report, choose Release 12 templates (which do not have the word “Rotor” in the template name).
Report template that have “Rotor” in the template name are from Release 11 and require variables that are not available from ANSYS FLUENT turbo files.
- For rotating machinery applications, identification of components and ordering, regions, rotation axis, number of passages, and interfaces cannot be done automatically; you must supply this information on the Turbo initialization panel. When generating turbo reports, select variables, instance transforms, and expressions will require manual updates; for details see [Procedures for Using Turbo Reports when Turbomachinery Data is Missing](#) (p. 31).

Quantitative Differences Between FLUENT and CFD-Post

- The Function Calculator may give variable averages on cut planes and isosurfaces that are different from those given by ANSYS FLUENT. These differences may occur when the surface is cutting through a mesh face that

joins two mesh elements. In this situation, CFD-Post may use the element-center data from a different element than ANSYS FLUENT uses. Note that as both elements are equally valid choices, both calculations are correct.

- In CFD-Post, on boundaries that have zero velocity, Total Temperature and Total Pressure will have same values as Temperature and Pressure, respectively (as expected). In ANSYS FLUENT, Total Temperature is different from Temperature for boundaries that have zero velocity; similar differences apply between Total Pressure and Pressure. This is a limitation in ANSYS FLUENT.

Chapter 7. CFD-Post Edit Menu and Options (Preferences)

Undo and **Redo** commands are available in the **Edit** menu. Additionally, there are a variety of options that can be set to customize the software.


This chapter describes:

- [Undo and Redo \(p. 81\)](#)
- [Setting Preferences with the Options Dialog \(p. 81\)](#)

Undo and Redo

The undo and redo capability is limited by the amount of available memory. The undo stack is cleared whenever a **New**, **Open**, or **Close** action occurs.

Issue the **Undo** command by doing any of the following:

- Select **Edit > Undo**.
- Click *Undo*  on the toolbar.
- Press **Ctrl + Z**

Note

- You can repeatedly issue the **Undo** command.
- Some viewer manipulations cannot be reverted using the **Undo** command.
- Some commands that you issue have multiple components. For example, when you create some objects the software creates the object and sets the visibility of the object on (in two separate operations). Thus, when you perform an undo operation in such a situation, you are setting the visibility of the object off; you must choose undo a second time to “uncreate” the object.
- **Undo** cannot be used when recording session files.

The redo feature is used to do an action that you have just undone using the **Undo** command. Issue the **Redo** command by doing any of the following:

- Select **Edit > Redo**.
- Click *Redo*  on the toolbar.
- Press **Ctrl + Y**

Setting Preferences with the Options Dialog

The **Options** dialog enables you to set various general preferences. Settings are retained per user.

1. Select **Edit > Options**.

The **Options** dialog box appears.

2. Set options as required. If desired, select **CFX Defaults** to use *all* of the default settings.

If you are using ANSYS Workbench and want to use its default settings, select **Workbench Defaults**.

For descriptions of the available options, see:

- [CFD-Post Options \(p. 82\)](#)
- [Common Options \(p. 84\)](#)

3. Click **OK**. Your changes are implemented immediately.

CFD-Post Options

When the **Options** dialog box appears, CFD-Post options can be configured under **CFD-Post**.

Interpolation Tolerance

The **Interpolation Tolerance** sets the amount of the area outside the domain that will be treated as a part of the domain when interpolating variables. For example, a point that is within this tolerance distance will be given a value that is interpolated from the nearest domain boundary face.

By default the tolerance "layer" is 0.5% of the domain. You can set the value to 0 to turn the Interpolation Tolerance off.

Note that this value should be set to a value less than half the size of the smallest openings or features of the domain geometry. This prevents a point from being detected in two overlapping Interpolation Tolerance regions.

Angular Shift for Transient Rotating Domains

You can set this setting to **Automatic**, **Always rotate**, or **Never rotate**.

Enable Beta Features

Some beta features are hidden in the user interface. You can select this option to "unhide" those beta features. When selected, such Beta features will be identified by "(Beta)" in the user interface.

Files

- Select **Disable region load** if you do not want to have region definitions loaded when you load a file that contains them.
- **Translate variable names to CFX-Solver style names** converts variable names from other results files into CFX variable names. (For example, the variable P in a CFX-TASCflow file will be converted to **Pressure**.)

Important

- By default, CFD-Post will not modify the variable names in the `.rso` file. If you want to use all of the embedded CFD-Post macros and calculation options, you will need to convert variable names to CFX types.
- In order to use the **Turbo Charts** feature with ANSYS FLUENT files, you must have **Translate variable names to CFX-Solver style names** enabled.

The complete list of translated variables is given in [Variable Translation \(p. 73\)](#).

- Clear **Pre-calculate global variable ranges** to turn off the calculation of all variable ranges.
- Select **Don't prompt to auto-load report template** to prevent CFD-Post from automatically asking you if you want to load a report upon loading results files.
- Select **Show domain selector before load** to enable you to choose which domains to load when more than one domain exists in the results file. If this option is turned off, then all domains will be loaded next time you load a results file.
- When **Load missing variables from nearest FULL time step** is cleared, it makes all variables that are not written to the partial results file undefined for the current timestep. When selected, CFD-Post loads the missing variables from the nearest full results file. This option is used when partial transient results files do not contain all of the variables calculated by the CFX-Solver. By default, these variables will be undefined (but still visible in the variables list) for the current timestep.

Important

Take care when using this option because values that are plotted may not apply to the current timestep.

CFD-Post Solution Units

CFD-Post has a **Solution Units** option that is available from the **Options** tab.

The solution units assumed, which are read when the file was loaded, are displayed on the right. When files that do not store solution units (such as CFX-4 dump files, CFX-TASC files, FLUENT files, or ANSYS results files) are loaded, you will be prompted to specify the solution units. You can enable the **Don't prompt for Solution Units before loading results** toggle to suppress this prompt, in which case the default units of kilograms, meters, seconds, Kelvin, and radians will be used.

The units shown on this form are not necessarily those used by CFD-Post, but are the solution units used in the currently loaded file. The units used by CFD-Post are set elsewhere; for details, see [Setting the Display Units \(p. 85\)](#). CFD-Post needs to know the solution units used in the file so that it can convert them to the units specified. When CFX files are loaded into CFD-Post, the solution units that were used by the CFX-Solver are automatically read from the file. For this reason, **Don't prompt for Solution Units before loading results** is ignored when loading CFX files and selected by default for other file types.

When post-processing a results file in CFD-Post, the units used are not necessarily those used in the results file. CFD-Post will convert to your preferred units.

Note

In CFD-Post, the temperature solution units must be an absolute scale (for example, Kelvin [K] or Rankin [R]); you cannot use Celsius and Fahrenheit. Temperature quantities elsewhere in ANSYS CFX can be set in Celsius and Fahrenheit.

Turbo

These settings are related to turbomachinery simulations loaded into CFD-Post, and are mostly self-explanatory.

Viewer

To configure the viewer, right-click on the viewer and select **Viewer Options**.

Object Highlighting

Controls how an object that is generated after a change to the setting of this option is highlighted in the viewer. Such highlighting occurs when in picking mode, when selecting a region in a list, or when selecting items in the tree view.

Under **Type**, select one of the following:

- **Surface Mesh**: Displays the surface mesh for selected regions using lines.
- **Wireframe**: Traces objects that contain surfaces with green lines.
- **Bounding Box**: Highlights the selected objects with a green box.

Note

When you load a state file, the highlighting is dictated by the setting that is stored in the case, rather than by the current preferences setting.

Background

Set **Mode** to Color or Image.

Color

Choose either a constant color or a gradient of colors.

Image

Select one of a list of predefined images or a custom image.

If selecting a custom image, choose an image file and a type of mapping. Image types that are supported include *.bmp, *.jpg, *.png, and *.ppm. Mapping options are Flat and Spherical. Flat maps are stationary while spherical maps surround the virtual environment and rotate with the objects in the viewer.


Custom images have some restrictions: all background images and textures sent to the viewer must be square and must have dimensions that are powers of 2 (for example, 512 x 512 or 1024 x 1024).

If the dimensions of your background image is not a power of 2, the viewer sizes the image to be a power of 2 by doing bicubic resampling.

To make the background image square, transparent pixels are added to the smaller dimension to make it the same as the larger dimension. The transparent pixels enable you to see the regular viewer background, which gives you control over what fill color your background has.

Other Viewer Options

Text/Edge Color

Select a color by clicking in the box, or clicking the *Ellipsis*  icon.

Axis/Ruler Visibility

Select or clear **Axis Visibility** or **Ruler Visibility** to show or hide the axis indicator or ruler in the viewer.

Hide ANSYS Logo

Controls whether or not the ANSYS logo appears in the 3D Viewer.

Advanced

Under **Cmd Timeout**, specify the minimum time between registered mouse clicks, in milliseconds.

Common Options

Auto Save


Select the time between automatic saves.

To turn off automatic saves, set **Auto Save** to **Never**.

Note

This option affects more than one CFX product.

Temporary directory

To set a temporary directory, click *Browse*  to find a convenient directory where the autosave feature will save state files.

Appearance

The appearance of the GUI can be controlled from the **Appearance** options. The default GUI style will be set to that of your machine. For example, on Windows, the GUI has a Windows look to it. If, for example, a Motif appearance to the GUI is preferred, select to use this instead of the Windows style.

1. Under **GUI Style**, select the user interface style to use.
2. For **Font** and **Formatted Font**, specify the fonts to use in the application.

Note

It is important not to set the font size too high (over 24 pt. is not recommended) or the dialog boxes may become difficult to read. Setting the font size too small may cause some portions of the text to not be visible on monitors set at low resolutions. It is also important not to set the font to a family such as Webdings, Wingdings, Symbols, or similar type faces, or the dialog boxes become illegible.

Formatted Font has no function in CFD-Post.

Viewer Setup

1. Select **Double Buffering** to use two color buffers for improved visualization.

For details, see [Double Buffering \(p. 85\)](#).

2. Select or clear **Unlimited Zoom**.

For details, see [Unlimited Zoom \(p. 85\)](#).

Double Buffering

Double Buffering is a feature supported by most OpenGL implementations. It provides two complete color buffers that swap between each other to animate graphics smoothly. If your implementation of OpenGL does not support double buffering, you can clear this check box.

Unlimited Zoom

By default, zoom is restricted to prevent graphics problems related to depth sorting. Selecting **Unlimited Zoom** allows an unrestricted zoom.

Mouse Mapping

The mouse-mapping options allow you to assign viewer actions to mouse clicks and keyboard/mouse combinations. These options are available when running in standalone mode. To adjust or view the mouse mapping options, select **Edit > Options**, then **Viewer Setup > Mouse Mapping**. For details, see [Mouse Button Mapping \(p. 49\)](#).

Setting the Display Units

The settings on the **Edit > Options > Common > Units** dialog control the *preferred units* of the CFX application. Preferred units are the units of the data that CFD-Post uses when information is displayed to you and are the default units when you enter information (as contrasted with units of the data that are stored in results files). For example, if your preferred units are SI and you load a results file that contains data in British Technical units, the values you see in CFD-Post will be in SI.

To set your preferred units:

1. Under **System**, select the unit system to use. Unit systems are sets of quantity types for mass, length, time, and so on.

The options under **System** include SI, CGS, English Engineering, British Technical, US Customary, US Engineering, or Custom. Only Custom enables you to redefine a quantity type (for example, to use inches for the dimensions in a file that otherwise used SI units).

The most common quantity types appear on the main **Options** dialog; to see *all* quantity types, click **More Units**.

2. Select or clear **Always convert units to Preferred Units**.

If **Always convert units to Preferred Units** is selected, the units of entered quantities are immediately converted to those set on this dialog.

For example, if you have set **Velocity** to $[m \ s^{-1}]$ on this dialog to make that the preferred velocity unit, and elsewhere you enter $20 \ [mile \ hr^{-1}]$ for a velocity quantity, the entered value is immediately converted and displayed as $8.94078 \ [m \ s^{-1}]$.

The two sets of units are:

- The units presented on this dialog box, which control the default units presented in the GUI, as well as the units used for mesh transformation.
- The solution units; for details, see [CFD-Post Solution Units \(p. 82\)](#).

Chapter 8. CFD-Post Session Menu

Session files contain a record of the commands issued during a CFD-Post session. The actions that cause commands to be written to a session file include:

- Viewer manipulation performed using the commands available by right-clicking in the viewer window.
- All actions available from the **File** and **Edit** menus.
- Creation of expressions.
- Creation of new objects and changes to an object committed by clicking **OK** or **Apply** on any of the panels available from the **Tools** and **Insert** menus/toolbars.
- Commands issued in the **Tools > Command Editor** dialog box.

This chapter describes:

- [New Session Command \(p. 87\)](#)
- [Start Recording and Stop Recording Commands \(p. 87\)](#)
- [Play Session Command \(p. 87\)](#)

New Session Command

When a session file is not currently being recorded, you can select **Session > New Session**. This opens the **Set Session File** dialog box where you can enter a file name for your session file. Once you have saved the file, it becomes the current session file. Commands are not written to the file until you select **Session > Start Recording**.

1. Browse to the directory in which you want to create the session file, and then enter a name for the file ending with a `.cse` (CFD-Post) extension.
2. Click **Save** to create the file.

This will not start recording to the session file. To start recording, you must select **Session > Start Recording**.

If you create more than one session file during a CFD-Post session, the most recently created file is the current session file by default. You can set a different file to be the current session file by selecting an existing file from the **New Session > Set Session File** window and then clicking **Save**. Because the file exists, a warning dialog appears:

- If you select **Overwrite**, the existing session file is deleted and a new file is created in its place.
- If you select **Append**, commands will be added to the end of the existing session file when recording begins.

Start Recording and Stop Recording Commands

The **Start Recording** action writes into the current session file the CCL commands you issue. A session file must first be set before you can start recording (see [New Session Command \(p. 87\)](#)). **Stop Recording** terminates writing of CCL commands to the current session file. You can start and stop recording to a session file as many times as necessary.

Important

A session file cannot be played if it contains an **Undo** command. To run a session file that contains an **Undo** command, first edit the session file to remove the command.

Play Session Command

Selecting **Session > Play Session** opens the **Play Session File** dialog box in which you can select the session file to play. The commands listed in the selected session file are then executed.

Important

Existing objects with the same name as objects defined in the session file are replaced by those in the session file (for example, if `Plane 1` exists in this CFD-Post session file, playing the session file will overwrite any existing object with the name `Plane 1`).

To play a session file:

1. From the menu bar, select **Session > Play Session**.
2. In the **Play Session File** dialog, browse to the directory containing the session file and select the file you want to play.
3. Click **Open** to play the session file. The commands listed in the selected session file are executed. Existing objects with the same name as objects defined in the session file are replaced by those in the session file.

Note

You can play session files in standalone CFD-Post, but not in CFD-Post in ANSYS Workbench.

Chapter 9. CFD-Post Insert Menu

The **Insert** menu in CFD-Post is used to create new objects (such as locators, tables, charts, etc.), variables, and expressions.

A *locator* is a place or object that another object uses to plot or calculate values. For example, if you were to select a plane from which to start a streamline, the plane would be a locator.

This chapter describes:

- [Location Submenu \(p. 89\)](#)
- [Vector Command \(p. 116\)](#)
- [Contour Command \(p. 119\)](#)
- [Streamline Command \(p. 121\)](#)
- [Particle Track Command \(p. 125\)](#)
- [Text Command \(p. 128\)](#)
- [Coordinate Frame Command \(p. 130\)](#)
- [Legend Command \(p. 132\)](#)
- [Instance Transform Command \(p. 134\)](#)
- [Clip Plane Command \(p. 138\)](#)
- [Color Map Command \(p. 139\)](#)
- [Variable Command \(p. 140\)](#)
- [Expression Command \(p. 140\)](#)
- [Table Command \(p. 140\)](#)
- [Chart Command \(p. 145\)](#)
- [Comment Command \(p. 155\)](#)
- [Figure Command \(p. 156\)](#)

Location Submenu

When you select any of the objects from the **Insert > Location** submenu, an **Insert Object** dialog box appears in which you can either accept the default name for the new object or enter a new one. CFD-Post will not let you create objects that have duplicate names.

Click **OK** on the dialog box to open the relevant details view in the **Outline** workspace. A new object will be created in the database when you click **Apply** in the details view of the location object.

Tip

You can also access locator objects from the *Location* icon  **Location** ▼ on the toolbar.

The following topics will be discussed in this section:

- [Point Command \(p. 90\)](#)
- [Point Cloud Command \(p. 92\)](#)
- [Line Command \(p. 94\)](#)
- [Plane Command \(p. 95\)](#)
- [Volume Command \(p. 98\)](#)
- [Isosurface Command \(p. 101\)](#)
- [Iso Clip Command \(p. 102\)](#)
- [Vortex Core Region \(p. 103\)](#)
- [Surface of Revolution Command \(p. 108\)](#)

- [Polyline Command \(p. 110\)](#)
- [User Surface Command \(p. 112\)](#)
- [Surface Group Command \(p. 115\)](#)
- [Turbo Surface Command \(p. 116\)](#)
- [Turbo Line Command \(p. 116\)](#)

Point Command

A *point* is an object in 3D space that has a set of coordinates. You can use a point to locate the position of a variable minimum or maximum or as an object with which other objects can interact.

The following characteristics of points will be discussed:

- [Point: Geometry \(p. 90\)](#)
- [Point: Color \(p. 91\)](#)
- [Point: Symbol \(p. 91\)](#)
- [Point: Render \(p. 92\)](#)
- [Point: View \(p. 92\)](#)

Note

There are several ways to insert a point:

- From the menu bar, select **Insert > Location > Point**.
- From the toolbar, select **Location > Point**.
- Depending on the context, you may be able to perform an insert from the shortcut menu in the tree view or in the 3D Viewer.

Point: Geometry

Domains

The **Domains** setting selects the domains in which the point will exist.

Note

For a case with immersed solids, the setting **All Domains** refers to all domains *except* the immersed solids. To display all of the domains in a case that contains immersed solids, click the *Location Editor*



icon and hold down the **Ctrl** key while selecting **All Domains** and **All Immersed Solids**.

Variables used for plots or calculations on immersed solid domain boundaries are not taken from the immersed solid domain; instead, they are interpolated from the fluid/porous domain in which the solid is immersed. The accuracy of such interpolation is dependent on the mesh densities of both the fluid/porous domain and the surface of the immersed solid domain. To visualize, or perform computations with, variables that are associated with the immersed solid domain, use cut planes, user surfaces, or other locators that are offset into the immersed solid domain, and set the applicable **Domains** setting to refer to the immersed solid domain.

Definition

Method

The **Method** setting has the following options:

Option	Description
XYZ	Enables you to set a coordinate in 3D space for the Point.
Node Number	Enables you to select a node to which to attach the Point.

Option	Description
Variable Minimum	Places the Point at the selected variable's lowest value. Select whether the object you want to plot will be based on hybrid or conservative values. For details, see Hybrid and Conservative Variable Values (p. 165) .
Variable Maximum	Places the Point at the selected variable's greatest value. Select whether the object you want to plot will be based on hybrid or conservative values. For details, see Hybrid and Conservative Variable Values (p. 165) .

Note

You can move only points that have been specified with the XYZ option.

Point

Point is available only if the XYZ option is selected. The **Point** setting specifies the Cartesian coordinates for the Point object. Once the point is created, you can use the mouse pointer to drag the point around in the domain. For details, see [Picking Mode \(p. 50\)](#).

Node Number

Node Number is available only if the Node Number **Method** is selected. The **Node Number** setting specifies at which node to place the Point object. When more than one domain is selected, a point is created for the specified node number in each domain (if it exists). If the node number does not exist in one domain but exists in another, you should select only the domain in which the node exists or an error message will be displayed.

Location

Location is available only if the Variable Minimum or the Variable Maximum options are selected. The **Location** setting specifies an object for the Point to be located in. When more than one domain is selected, a point is created for the minimum or maximum value of the variable within each domain.

Variable

Variable is available only if the Variable Minimum or the Variable Maximum options are selected. The **Variable** setting selects the variable to be used to find the maximum or minimum point.

Nearest Node Value

Nearest Node appears when any option except the Node Number option is selected. The **Nearest Node** text displays the numerical value of the nearest node to the point's current position.

Point: Color

The **Color** tab controls the color settings. For details, see [Color Details Tab \(p. 16\)](#).

Point: Symbol**Symbol**

The **Symbol** setting has the following options:

Symbol	Description
Crosshair	A 3D “+” sign.
Octahedron	A 3D diamond that has eight faces.
Cube	A box.

Symbol	Description
Ball	A sphere.

Symbol Size

The **Symbol Size** setting specifies the size of the Point symbol. Each **Symbol Size** unit represents 5% of the domain span. The domain span, which is dependent on the geometry, is equal to the largest difference from the X, Y, and Z ranges.

Point: Render

The rendering settings can be changed by clicking the **Render** tab. For details, see [Render Details Tab \(p. 18\)](#).

Point: View

The **View** tab is used for creating or applying predefined Instance Transforms for a wide variety of objects.

For details on changing the view settings, see [View Details Tab \(p. 21\)](#).

Point Cloud Command

To create multiple points, select **Insert > Location > Point Cloud**. You can create uniform vector plots independent of the mesh by using the Point Cloud object. You can also create streamlines that use a Point Cloud as the locator.

The following characteristics of point clouds will be discussed:

- [Point Cloud: Geometry \(p. 92\)](#)
- [Point Cloud: Color \(p. 94\)](#)
- [Point Cloud: Symbol \(p. 94\)](#)
- [Point Cloud: Render \(p. 94\)](#)
- [Point Cloud: View \(p. 94\)](#)

Note

There are two ways to insert a point cloud:

- From the menu bar, select **Insert > Location > Point Cloud**.
- From the toolbar, select **Location > Point Cloud**.

Point Cloud: Geometry

Domains

For details, see [Domains \(p. 90\)](#).

Definition

Locations

The **Locations** setting selects the location or locations in which the point cloud is created.

Tip

Click *Location Editor*  to open the **Location Selector** dialog box, which displays the complete list of available locations.

Sampling

The **Sampling** setting has the following options:

Option	Description
Equally Spaced	Generates points with roughly the same distance between them.
Rectangular Grid	Generates a rectangular grid of points on the surface. This option should be used only on flat surfaces.
Vertex	Generates the points on the vertices of the mesh. The maximum number of points is the total number of vertices in the mesh.
Face Center	Generates the points at the center of the mesh faces. The maximum number of points is the total number of faces in the mesh.
Free Edge	Generates the points on the outer edge at the center of the edge segments.
Random	Generates the points randomly. If the seed is positive, the point distribution can be reproduced.

of Points

of Points is available only when either the `Equally Spaced` or `Random` option is selected. The **# of Points** setting specifies the number of equally spaced points you want generated on the surface of the mesh.

Spacing

Spacing is available only when the `Rectangular Grid` option is selected. The **Spacing** setting specifies a value which represents a fraction of the maximum domain extent. For example, if your domain has a maximum extent of 1 [m] and a **Spacing** of 0.1 was used, a rectangular grid with 0.1 [m] spacing would be created.

Aspect Ratio

Aspect Ratio is available only when the `Rectangular Grid` option is selected. The **Aspect Ratio** setting stretches the rectangle in a direction parallel to the grid axes. If a value less than one is entered, the grid will be stretched in one direction. If a value greater than one is entered, the grid will be stretched in the direction perpendicular to the previous direction.

Grid Angle

Grid Angle is available only when the `Rectangular Grid` option is selected. The **Grid Angle** setting specifies the magnitude and direction of grid rotation.

Reduction

Reduction is available only when the `Vertex`, `Face Center`, or `Free Edge` options are selected. The **Reduction** setting has the following options:

Option	Description
Max Number of Points	Enables the option to specify the maximum number of points allowed to be plotted.
Reduction Factor	Enables the option to specify a reduction factor from the full number of points.

Max Points

Max Points is available only if the `Max Number of Points` option is selected. The **Max Points** setting specifies a value for the maximum number of points allowed. If the maximum number of vertices is greater than that of the specified value, then the points taken will be randomly selected.

Factor

Factor is available only if the `Reduction Factor` option is selected. The **Factor** setting specifies a value by which to decrease the total number of points in the Point Cloud object. The final number of vectors is $total/n$, where $total$ is the total number of seeds, and n is the reduction value entered into the box.

Seed

Seed is available only if the **Random** option is selected. The **Seed** setting generates a different set of random points for each value entered. The distribution cannot be replicated or reproduced for negative seed values. For negative seed values, the random series is based on the system time. Different compilers may generate different distributions for the same positive seed value.

Note

Similar sampling options are also available directly on **Vector** and **Streamline** objects.

Point Cloud: Color

The color settings can be changed on the **Color** tab. For details, see [Color Details Tab \(p. 16\)](#).

Point Cloud: Symbol

For details, see [Point: Symbol \(p. 91\)](#).

Point Cloud: Render

The rendering settings can be changed by clicking the **Render** tab. For details, see [Render Details Tab \(p. 18\)](#).

Point Cloud: View

The **View** tab is used for creating or applying predefined Instance Transforms for a wide variety of objects.

For details on changing the view settings, see [View Details Tab \(p. 21\)](#).

Line Command

A line locator can exist between two points anywhere inside or outside the domain.

The following characteristics of lines will be discussed:

- [Line: Geometry \(p. 94\)](#)
- [Line: Color \(p. 95\)](#)
- [Line: Render \(p. 95\)](#)
- [Line: View \(p. 95\)](#)

Note

There are several ways to insert a line:

- From the menu bar, select **Insert > Location > Line**.
- From the tool bar, select **Location > Line**.
- Depending on the context, you may be able to perform an insert from the shortcut menu in the tree view.

Line: Geometry

Domains

For details, see [Domains \(p. 90\)](#).

Definition

Method

The only available option is the **Two Points** option.

Point 1

The **Point 1** text boxes specify the start point of the line.

Point 2

The **Point 2** text boxes specify the end point of the line.

Line Type**Cut/Sample Options**


Selecting **Cut** will extend the line in both directions until it reaches the edge of the domain. Points on this line exist where the line intersects with a mesh element face.

Selecting **Sample** creates a line existing between the two points entered. It is mesh-independent, and the number of points along the line corresponds to the value you enter in the **Samples** box.

Samples

Samples is available only if the **Sample** option is selected. The **Samples** setting specifies a value for the number of evenly-spaced sampling points along the line.

Line Translation Using Picking Mode

You can use picking mode to select or translate a line in the viewer. To move a line, select picking mode by clicking *Single Select*  in the **Selection Tools** toolbar and drag the line to a new location. The line properties will automatically update in the details view. For details, see [Picking Mode \(p. 50\)](#).

Line: Color

The color settings can be changed by clicking the **Color** tab. For details, see [Color Details Tab \(p. 16\)](#).

Line: Render

You can change the **Line Width** by entering a value corresponding to the pixel width of the line. You can specify the value between 1 and 11 by using the graduated arrows, the embedded slider, or by typing in the value.

Line: View

The **View** tab is used for creating or applying predefined Instance Transforms for a wide variety of objects.

For details on changing the view settings, see [View Details Tab \(p. 21\)](#).

Plane Command

A plane is a two-dimensional area that exists only within the boundaries of the computational domain.

The following characteristics of planes will be discussed:

- [Plane: Geometry \(p. 96\)](#)
- [Plane: Color \(p. 98\)](#)
- [Plane: Render \(p. 98\)](#)
- [Plane: View \(p. 98\)](#)

Note

There are several ways to insert a plane:

- From the menu bar, select **Insert > Location > Plane**.
- From the toolbar, select **Location > Plane**.
- Depending on the context, you may be able to perform an insert from the shortcut menu in the tree view or in the 3D Viewer.

Plane: Geometry

Domains

For details, see [Domains \(p. 90\)](#).

Definition

Method

The **Method** setting has the following options:

Option	Description
YZ Plane	Defines a plane normal to the X axis.
ZX Plane	Defines a plane normal to the Y axis.
XY Plane	Defines a plane normal to the Z axis.
Point and Normal	Enables you to specify a point on the plane and a normal vector to the plane.
Three Points	Enables you to define a plane by providing three points that lie in the plane.

X

X is available only if the YZ Plane option is selected. The **X** setting specifies an offset value from the X axis.

Y

Y is available only if the ZX Plane option is selected. The **Y** setting specifies an offset value from the Y axis.

Z

Z is available only if the XY Plane option is selected. The **Z** setting specifies an offset value from the Z axis.

Point

Point is available only if the Point and Normal option is selected. The **Point** setting specifies the 3D coordinates of the point that lies on the plane.

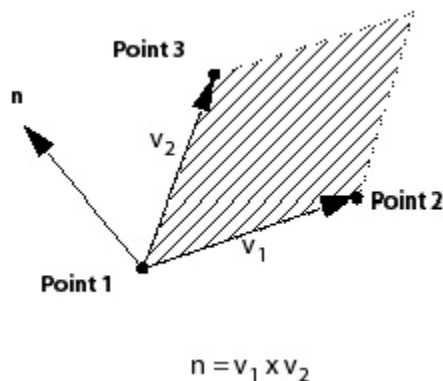
Normal

Normal is available only if the Point and Normal option is selected. The **Normal** setting specifies the normal vector by entering a point along the vector from the **Point** coordinates.

Point 1, Point 2, and Point 3

These options are only available if the Three Points option is selected. The **Point 1**, **Point 2**, and **Point 3** settings specify three points that lie on the plane.

The normal vector to the plane is calculated using the right-hand rule. The first vector is from **Point 1** to **Point 2**, and the second is from **Point 1** to **Point 3**, as shown in the following diagram. For example, the direction of this vector might be important if you are using the plane to define a Clip Plane.




Plane Bounds

Type

The **Type** setting has the following options:

Option	Description
None	Cuts through a complete cross-section of each domain specified in the Domains list. A slice plane is bounded only by the limits of the domain. The Plane Type must be set to Slice for this option (default).
Circular	Causes the boundary of the plane to be in the shape of a circle. The circle is centered at the origin for the YZ, ZX, and XY Planes. For the other two methods, the circle is centered at the first point entered in the Definition frame.
Rectangular	Causes the boundary of the plane to be a rectangular shape. The rectangle is centered at the origin for the YZ, ZX, and XY Planes. For the other two methods, the rectangle is centered at the first point entered in the Definition frame.

Radius

Radius is available only if the **Circular** option is selected. The **Radius** setting specifies a radius for the circular boundary. You can enter a value or select the *Expression*  icon to the right of the **Radius** setting to specify the radius as an expression.

X/Y/Z Size

These settings are available only if the **Rectangular** option is selected. Two of these options will be displayed because a plane is a 2D object. These settings will specify a width and height for the rectangular boundary. The size of the rectangle is determined with reference to the planes origin (that is, the plane is resized around its center).

X/Y/Z Angle

This setting is available only if the **Rectangular** option is selected. Only one of these settings is displayed at once. This setting specifies an angle to rotate the plane counterclockwise about its normal vector by the specified number of degrees.

Invert Plane Bound Check Box

Invert Plane Bound is available only if the **Circular** or the **Rectangular** option is selected. If this check box is selected, the area defined by the rectangle or circle is used as a cut-out area from a slice plane that is bounded only by the domains. The area inside the bounds of the rectangle or circle do not form part of the plane, but everything on the slice plane outside of these bounds is included.

Plane Type

Slice Option

Select the **Slice** option to cut the plane so that it lies only inside the domain.

A slice plane differs from a sampling plane. A sampling plane is a set of evenly-spaced sampling points that are independent of the mesh. When you create a slice plane, the sampling points are placed at locations where the slice plane intersects an edge of the mesh, causing an uneven distribution of the sampling points. The density of these sampling points in a slice plane is related to the length scale of the mesh.

When you use the slice plane for **Vector** plots, the seeds are the points where the plane intersects a point on the edge of three mesh elements. You can view the seeds by turning on the **Show Mesh Lines** option on the **Render** tab for the plane.

Sample Option

Select the **Sample** option to specify the amount of seeds in the plane.

When creating a sampling plane, the **Plane Bounds** must be either **Circular** or **Rectangular**. For the **Circular** option, the density of sampling points is determined by the radius of the plane specified in the **Plane Bounds** tab and the number of radial and circumferential sampling points. For **Rectangular** bounds, you must specify the size of the bounds for your plane in each of the plane directions. The density of sampling points depends on the size of the plane and the number of samples in each of the two coordinate directions that describe the plane.

Certain types of plots will show small differences across GGI interfaces. This is to be expected when the nodes of the computational grids on each side of a GGI connection do not match. For example, contour lines or fringe lines may not match exactly across a GGI interface. This is a very minor effect and is not an indicator of any problem.

Plane Translation using Picking Mode

For details, see [Line Translation Using Picking Mode \(p. 95\)](#).

Plane: Color

The color settings can be changed by clicking the **Color** tab. For details, see [Color Details Tab \(p. 16\)](#).

Plane: Render

The rendering settings can be changed by clicking the **Render** tab. For details, see [Render Details Tab \(p. 18\)](#).

Plane: View

The **View** tab is used for creating or applying predefined Instance Transforms for a wide variety of objects.

For details on changing the view settings, see [View Details Tab \(p. 21\)](#).

Volume Command

A **Volume** is a collection of mesh elements that can be used as a locator for graphic objects or calculations. Volumes will not be displayed as perfect shapes (for example, a perfect sphere) because mesh elements are either included in or excluded from the **Volume** object.

The following characteristics of volumes will be discussed:

- [Volume: Geometry \(p. 99\)](#)
- [Volume: Color \(p. 100\)](#)
- [Volume: Render \(p. 100\)](#)
- [Volume: View \(p. 100\)](#)

Note

There are several ways to insert a volume:

- From the menu bar, select **Insert > Location > Volume**.
- From the tool bar, select **Location > Volume**.

- Depending on the context, you may be able to perform an insert from the shortcut menu in the tree view or in the 3D Viewer.

Volume: Geometry

Domains

For details, see [Domains \(p. 90\)](#).

Element Types

The **Element Types** setting has the following options:

Option	Description
Tet	Displays volume that is connected to a tetrahedral mesh.
Pyramid	Displays volume that is connected to a pyramid-shaped mesh.
Wedge	Displays volume that is connected to a wedge-shaped mesh.
Hex	Displays volume that is connected to a hexagonal mesh.

Definition

Method

The **Method** setting has the following options:

Option	Description
Sphere	Creates a sphere-shaped volume. Enables you to specify a center point and radius for the sphere volume.
From Surface	Creates a volume on a surface. Enables you to select a surface from the Location setting. Some surface types may not be available.
Isovolume	Creates a volume at a specified variables value. Enables you to specify a variable and one or two values (depending on the Mode) to create one or two isosurfaces that bound the isovolume.
Surrounding Node	Creates a volume at a node. Enables you to specify a node by number.

Point

Point is available only if the **Sphere** option is selected. The **Point** setting specifies a center point for the sphere volume. The point can be anywhere in 3D space.

Radius

Radius is available only if the **Sphere** option is selected. The **Sphere** setting specifies a radius for the sphere volume.

Location

Location is available only if the **From Surface** option is selected. The **Location** setting selects from a list of valid locations for the volume to exist on.

Variable

Variable is available only if the **Isovolume** option is selected. The **Variable** setting selects a variable to plot the volume on. A **Value** for the variable must be selected before the volume can be defined.

Hybrid/Conservative Options

These options are available only if the **Isovolume** option is selected. For help on which field to select, see [Hybrid and Conservative Variable Values](#) (p. 165).

Mode (for the Sphere and From Surface options)

The **Mode** setting has the following options:

Option	Description
Intersection	Creates a volume at the specified radius for the Sphere option. For the From Surface option, the volume is created on the surface of the object.
Below Intersection	Creates a volume for all of the radii less than the specified radius for the Sphere option. For the From Surface option, the volume is plotted for all values less than the given value on the location object.
Above Intersection	Opposite to the Below Intersection option.

Mode (for the Isovolume option)

The **Mode** setting has the following options:

Option	Description
At Value	Creates a volume for all the mesh elements in the domain equal to the entered value.
Below Value	Creates a volume for all the mesh elements in the domain above the entered value.
Above Value	Creates a volume for all the mesh elements in the domain less than the entered value.
Between Value	Creates a volume for all the mesh elements in the domain in between the two entered values.

Value Text Boxes

The **Value** text boxes are available only if the **Isovolume** option is selected. The **Value** text boxes specify values to compare to using the **Mode** options. For example, if **Value** is set to 2 and **Mode** is set to **At Value**, the Volume will plot where the variable is equal to 2.

Inclusive Check Box

Select the **Inclusive** check box to add the entered values to an above or below comparison **Mode**. For example, if the **Inclusive** check box is selected with the **Below Intersection** option, the volume will include the radius entered or surface selected.

Volume: Color

The color settings can be changed by clicking the **Color** tab. For details, see [Color Details Tab](#) (p. 16).

Volume: Render

The rendering settings can be changed by clicking the **Render** tab. For details, see [Render Details Tab](#) (p. 18).

Volume: View

The **View** tab is used for creating or applying predefined Instance Transforms for a wide variety of objects.

For details on changing the view settings, see [View Details Tab](#) (p. 21).

Isosurface Command

An *isosurface* is a surface upon which a particular variable has a constant value, called the *level*. For instance, an Isosurface of pressure would be a surface consisting of all the points in the geometry where the pressure took a value of 1.32×10^5 Pa. In CFD-Post, isosurfaces can be defined using any variable. You can also color the isosurface using any variable or choosing a constant color.

The following characteristics of isosurfaces will be discussed:

- [Isosurface: Geometry \(p. 101\)](#)
- [Isosurface: Color \(p. 101\)](#)
- [Isosurface: Render \(p. 101\)](#)
- [Isosurface: View \(p. 101\)](#)

Note

There are several ways to insert an isosurface:

- From the menu bar, select **Insert > Location > Isosurface**.
- From the tool bar, select **Location > Isosurface**.
- Depending on the context, you may be able to perform an insert from the shortcut menu in the tree view.

Isosurface: Geometry

Domains

For details, see [Domains \(p. 90\)](#).

Definition

Variable

The **Variable** setting specifies the variable that you want to plot.

Tip

Click the *Location Editor*  to open the **Variable Selector** dialog box, which displays the complete list of available options.

Hybrid/Conservative Option

For help on which field to select, see [Hybrid and Conservative Variable Values \(p. 165\)](#).

Value

The **Value** setting specifies a numerical value or expression to plot for the given variable.

Isosurface: Color

You can change the color settings by clicking the **Color** tab; for details, see [Color Details Tab \(p. 16\)](#).

You should not select the **Local Range** option when coloring an isosurface with the variable used to define it. In this case, the **Local Range** would be zero by definition, and the plot would highlight only round-off errors.

Isosurface: Render

To change the rendering settings, click the **Render** tab. For details, see [Render Details Tab \(p. 18\)](#).

Isosurface: View

The **View** tab is used for creating or applying predefined Instance Transforms for a wide variety of objects.

For details on changing the view settings, see [View Details Tab \(p. 21\)](#).

Iso Clip Command

An *iso clip* hides the portion of one or more locators subject to one or more constraints (visibility parameters) that you specify.

The following characteristics of iso clips will be discussed:

- [Iso Clip: Geometry \(p. 102\)](#)
- [Iso Clip: Color \(p. 102\)](#)
- [Iso Clip: Render \(p. 103\)](#)
- [Iso Clip: View \(p. 103\)](#)

Note

There are several ways to insert an iso clip:


- From the menu bar, select **Insert > Location > Iso Clip**.
- From the tool bar, select **Location > Iso Clip**.
- Depending on the context, you may be able to perform an insert from the shortcut menu in the tree view.

Iso Clip: Geometry

Domains

For details, see [Domains \(p. 90\)](#).

Location

Click the *Location Editor*  icon to open the **Location Editor** dialog box, which displays the complete list of available options. If you specify multiple locators, they must all have the same dimensionality (for example, all must be planes, rather than a combination of lines and planes).

Visibility Parameters

The **Visibility parameters** area is where you set the variables that hide the values that fail to meet a specified condition on a locator specified in the **Locations** field. For example, if the locator is an X-Y plane and the visibility is restricted to $Y > 0$, $Y \leq .1$, and $X \geq .15$, only areas that have values within those bounds will be displayed.

You create a new clip setting by clicking the *New*  icon or by right-clicking in the **Visibility parameters** area and selecting **New**. These actions cause the **Visibility Parameter Properties** settings to appear:

Variable

Sets the variable that controls where the iso clip regions are placed. Typically you would specify geometric variables.

Visible when [value]

Sets the display of regions (\geq , \leq) or a line ($=$).

Boundary Data

Enables you to set the boundary data to use of hybrid or conservative variable values. For details, see [Hybrid and Conservative Variable Values \(p. 165\)](#).

Iso Clip: Color

You can change the color of the locator or the variable that is colored on the locator by clicking the **Color** tab. For details, see [Color Details Tab \(p. 16\)](#).

Iso Clip: Render

To change the rendering settings, click the **Render** tab. For details, see [Render Details Tab \(p. 18\)](#).

Iso Clip: View

The **View** tab is used for creating or applying predefined Instance Transforms for a wide variety of objects.

For details on changing the view settings, see [View Details Tab \(p. 21\)](#).

Vortex Core Region

A *vortex* is a circular or spiral set of streamlines; a *vortex core* is a special type of isosurface that displays a vortex. The CFD-Post vortex core visualization tools are designed to help you identify and understand vortex regions.

The following characteristics of vortex cores will be discussed:

- [Vortex Core Region: Geometry \(p. 103\)](#)
- [Vortex Core Region: Color \(p. 108\)](#)
- [Vortex Core Region: Render \(p. 108\)](#)
- [Vortex Core Region: View \(p. 108\)](#)

Note

There are several ways to insert a vortex core region:

- From the menu bar, select **Insert > Location > Vortex Core Region**.
- From the tool bar, select **Location > Vortex Core Region**.
- Depending on the context, you may be able to perform an insert from the shortcut menu in the tree view.

Vortex Core Region: Geometry

Domains

The **Domains** setting specifies the domains where the vortex core should be found. Selected domains do not need to be contiguous.

For details, see [Domains \(p. 90\)](#).

Definition Area

The **Definition** area is where you define the type and the strength of the vortex core.

Method

The **Method** setting specifies sets of equations that detect vortices as spatial regions. Click on the drop-down arrow to choose a method:

Q-Criterion	The second invariant of the velocity gradient tensor. For a region with positive values, it could include regions with negative discriminants and exclude region with positive discriminants.
Lambda 2-Criterion	The negative values of the second eigenvalue of the symmetry square of velocity gradient tensor. Derived through the hessian of pressure.
Swirling Discriminant	The discriminant of velocity gradient tensor for complex eigenvalues. The positive values indicate existence of swirling local flow pattern.
Swirling Strength	The imaginary part of complex eigenvalues of velocity gradient tensor. It is positive if and only if the discriminant is positive and its value represents the strength of swirling motion around local centers.
Eigen Helicity	Dot product of vorticity and the normal of swirling plane (that is, the plane spanned by the real and imaginary parts of complex eigen-vectors of velocity gradient tensor).

Real Eigen Helicity	Dot product of vorticity and swirling vector that is the real eigen-vector of velocity gradient tensor.
Vorticity	Curl of velocity vector.
Absolute Helicity	Absolute value of the dot product of velocity vector and vorticity vector.

Note

There no recommended vortex core method; the appropriate choice of vortex core is always case-dependent.

Vortex Core Mathematics

A number of methods are based on eigen analysis in local velocity gradient tensor. The following are the related notations and equations.

For the velocity gradient tensor

$$\underline{D} = [d_{ij}] = \begin{bmatrix} d_{11} & d_{12} & d_{13} \\ d_{21} & d_{22} & d_{23} \\ d_{33} & d_{32} & d_{33} \end{bmatrix} = \begin{bmatrix} \frac{\partial u}{\partial x} & \frac{\partial u}{\partial y} & \frac{\partial u}{\partial z} \\ \frac{\partial v}{\partial x} & \frac{\partial v}{\partial y} & \frac{\partial v}{\partial z} \\ \frac{\partial w}{\partial x} & \frac{\partial w}{\partial y} & \frac{\partial w}{\partial z} \end{bmatrix} \quad (\text{Eq. 9.1})$$

The eigenvalues of the gradient tensor satisfies

$$\lambda^3 + P\lambda^2 + Q\lambda + R = 0 \quad (\text{Eq. 9.2})$$

where

$$P \equiv -\text{tr}(\underline{D}) = -\nabla \cdot \mathbf{u} = -(d_{11} + d_{22} + d_{33}) \quad (\text{Eq. 9.3})$$

$$Q \equiv \frac{1}{2} [P^2 - \text{tr}(\underline{D}\underline{D})] = (d_{22}d_{33} - d_{23}d_{32}) + (d_{11}d_{22} - d_{12}d_{21}) + (d_{33}d_{11} - d_{13}d_{31}) \quad (\text{Eq. 9.4})$$

$$R \equiv \frac{1}{3} [-P^3 + 3PQ - \text{tr}(\underline{D}\underline{D}\underline{D})] = d_{11}(d_{23}d_{32} - d_{22}d_{33}) + d_{12}(d_{21}d_{33} - d_{31}d_{23}) + d_{13}(d_{31}d_{22} - d_{21}d_{32}) \quad (\text{Eq. 9.5})$$

Now let

$$q \equiv Q - \frac{1}{3}P^2 \quad (\text{Eq. 9.6})$$

$$r \equiv R + \frac{2}{27}P^3 - \frac{1}{3}PQ \quad (\text{Eq. 9.7})$$

Then, if the discriminant is

$$\Delta \equiv \left(\frac{1}{2}r\right)^2 + \left(\frac{1}{3}q\right)^3 > 0 \quad (\text{Eq. 9.8})$$

then the tensor has one real eigenvalue λ_r and a pair of conjugated complex eigenvalues $\lambda_{cr} \pm i\lambda_{ci}$

That is, the tensor can be decomposed as

$$[d_{ij}] = [v_r v_{cr} v_{cr}] \begin{bmatrix} \lambda_r & 0 & 0 \\ 0 & \lambda_{cr} & \lambda_{ci} \\ 0 & -\lambda_{ci} & \lambda_{cr} \end{bmatrix} [v_r v_{cr} v_{cr}]^{-1} \quad (\text{Eq. 9.9})$$

We denote

$$\xi_2 = \sqrt{\sqrt{\Delta} - \frac{r}{2}} \quad (\text{Eq. 9.10})$$

and

$$\xi_3 = \sqrt{\sqrt{\Delta} + \frac{r}{2}} \quad (\text{Eq. 9.11})$$

Then

$$\lambda_r = \tilde{\lambda}_r - \frac{P}{3} = \xi_2 - \xi_3 - \frac{P}{3} \quad (\text{Eq. 9.12})$$

$$\lambda_{cr} = -\frac{\xi_2 - \xi_3}{2} - \frac{P}{3} \quad (\text{Eq. 9.13})$$

$$\lambda_{ci} = \frac{\xi_2 + \xi_3}{2} \sqrt{3} \quad (\text{Eq. 9.14})$$

The last one is called *swirling strength*, and represents the strength of the local swirling motion.

The following relationships are useful:

$$\xi_2 \xi_3 = \frac{q}{3} \quad (\text{Eq. 9.15})$$

$$\lambda_{ci}^2 = q + \frac{3}{4} \tilde{\lambda}_r^2 = Q + \frac{3}{4} (\lambda_r + P) \left(\lambda_r - \frac{P}{3} \right) \quad (\text{Eq. 9.16})$$

$$\Delta = \frac{1}{3} \lambda_{ci}^2 \left(\frac{\lambda_{ci}^2}{3} + \frac{3 \tilde{\lambda}_r^2}{4} \right) \quad (\text{Eq. 9.17})$$

$$Q = \frac{1}{4} \left\| \nabla \times \bar{U} \right\|^2 + 2 \left(\text{tr}^2(S) - \text{tr}(SS) \right) \quad (\text{Eq. 9.18})$$

Now the real eigen-vector meets:

$$\left[D - \lambda_r I \right] v_r = 0 \quad (\text{Eq. 9.19})$$

We can calculate the real eigen-vector using one of the non-zero vectors:

$$\begin{bmatrix} d_{12}d_{23} - d_{13}(d_{22} - \lambda_r) \\ d_{13}d_{21} - d_{23}(d_{11} - \lambda_r) \\ (d_{11} - \lambda_r)(d_{22} - \lambda_r) - d_{12}d_{21} \end{bmatrix} \quad (\text{Eq. 9.20})$$

$$\begin{bmatrix} d_{12}(d_{33} - \lambda_r) - d_{32}d_{13} \\ d_{13}d_{31} - (d_{11} - \lambda_r)(d_{33} - \lambda_r) \\ d_{32}(d_{11} - \lambda_r) - d_{31}d_{12} \end{bmatrix} \quad (\text{Eq. 9.21})$$

$$\begin{bmatrix} (d_{22} - \lambda_r)(d_{33} - \lambda_r) - d_{32}d_{23} \\ d_{23}d_{31} - d_{21}(d_{33} - \lambda_r) \\ d_{21}d_{32} - d_{31}(d_{22} - \lambda_r) \end{bmatrix} \quad (\text{Eq. 9.22})$$

The complex eigen-vectors' real and imagery parts meet:

$$\left[D - \lambda_{cr} I \right] v_{cr} = -\lambda_{ci} v_{ci} \quad (\text{Eq. 9.23})$$

$$\left[D - \lambda_{cr} I \right] v_{ci} = -\lambda_{ci} v_{cr} \quad (\text{Eq. 9.24})$$

Therefore, if

$$A \equiv DD - 2\lambda_{cr}D + (\lambda_{cr}^2 + \lambda_{ci}^2)I \quad (\text{Eq. 9.25})$$

then, $Av_{cr}=0$ and $Av_{ci}=0$. That is, all rows of matrix A are normal to both v_{cr} and v_{ci} , therefore they are all proportional to

$$v_n = \frac{v_{cr} \times v_{ci}}{\|v_{cr} \times v_{ci}\|} \quad (\text{Eq. 9.26})$$

So any non-zero row vector of matrix A can be used to calculate v_n .

This is useful to get the eigen-helicity $H_e = v_n \cdot \omega$, where ω is the vorticity vector.

$$\text{On } S \text{ and } S^2 + \Omega^2 \text{ let } S \equiv \frac{(D+D^T)}{2} \text{ and } \Omega \equiv \frac{(D-D^T)}{2}$$

Then $D = S + \Omega$ and $S^2 + \Omega^2 = \text{Sym}(D^2)$ have all real eigen-values ($\lambda_1 \leq \lambda_2 \leq \lambda_3$).

The region with negative of λ_2 is used in the method proposed by F. Hussain. By using the eigen-values and eigen-vectors of velocity gradient tensor D , we have

$$[d_{ij}] = [v_r v_{cr} v_{ci}] \begin{bmatrix} \lambda_r & 0 & 0 \\ 0 & \lambda_{cr}^2 - \lambda_{ci}^2 & 2\lambda_{cr}\lambda_{ci} \\ 0 & -2\lambda_{cr}\lambda_{ci} & \lambda_{cr}^2 - \lambda_{ci}^2 \end{bmatrix} [v_r v_{cr} v_{ci}]^{-1} \quad (\text{Eq. 9.27})$$

So, in the case the second eigenvalue is $\lambda_2(S^2 + \Omega^2) = \lambda_{cr}^2 - \lambda_{ci}^2$

Also, we can express the tensor DD as

$$DD = 2\lambda_{cr}D - (\lambda_{cr}^2 + \lambda_{ci}^2)I + \left[(\lambda_r - \lambda_{cr})^2 + \lambda_{ci}^2 \right] v_r v_n^T \quad (\text{Eq. 9.28})$$

Now when we look into the eigen values and vectors of S , the same should apply to $S^2 + \Omega^2$.

Let

$$S = \begin{bmatrix} s_0 & s_3 & s_3 \\ s_3 & s_1 & s_4 \\ s_5 & s_4 & s_2 \end{bmatrix} \quad (\text{Eq. 9.29})$$

Its eigenvalues meet

$$\lambda^3 - A\lambda^2 - B\lambda - C = 0 \quad (\text{Eq. 9.30})$$

where

$$A \equiv \frac{s_0 + s_1 + s_2}{3} \quad (\text{Eq. 9.31})$$

$$B \equiv (s_0 s_1 + s_1 s_2 + s_2 s_0) - (s_3^2 + s_4^2 + s_5^2) \quad (\text{Eq. 9.32})$$

$$C \equiv s_0 s_1 s_2 + 2s_3 s_4 s_5 - s_0 s_4^2 - s_1 s_5^2 - s_2 s_3^2 \quad (\text{Eq. 9.33})$$

Then the three eigenvalues are:

$$\sigma_1 = A + \rho \cos(\theta) \quad (\text{Eq. 9.34})$$

$$\sigma_2 = A + \rho \cos\left(\theta + \frac{2\pi}{3}\right) \quad (\text{Eq. 9.35})$$

$$\sigma_3 = A + \rho \cos\left(\theta + \frac{4\pi}{3}\right) \quad (\text{Eq. 9.36})$$

where

$$\eta = \frac{(s_0 - s_1)^2 + (s_1 - s_2)^2 + (s_2 - s_0)^2}{2} + 3(s_3^2 + s_4^2 + s_5^2) \quad (\text{Eq. 9.37})$$

$$\rho = \frac{2}{3}\sqrt{\eta} \quad (\text{Eq. 9.38})$$

$$\theta = \frac{1}{3}\cos^{-1}\left(\frac{4}{\rho^3}\left(C + A\left(\frac{\eta}{3} - A^2\right)\right)\right) \quad (\text{Eq. 9.39})$$

Since θ is in the range of $\left(0, \frac{\pi}{3}\right)$, we have $\sigma_2 \leq \sigma_3 \leq \sigma_1$. Therefore, the second eigenvalue for a 3x3 symmetry tensor is $A + \rho\cos\left(\left(\theta + \frac{4\pi}{3}\right)\right)$.

The eigenvector corresponding to an eigenvalue λ can be one of the non-zero vectors

$$\begin{bmatrix} s_3s_4 - s_5(s_2 - \lambda) \\ s_5s_3 - s_4(s_0 - \lambda) \\ (s_0 - \lambda)(s_1 - \lambda) - s_3s_3 \end{bmatrix} \begin{bmatrix} s_3(s_2 - \lambda) - s_4s_5 \\ s_5s_5 - (s_0 - \lambda)(s_2 - \lambda) \\ s_4(s_0 - \lambda)(s_1 - \lambda) - s_3s_5 \end{bmatrix} \begin{bmatrix} (s_1 - \lambda)(s_2 - \lambda) - s_4s_4 \\ s_4s_5 - s_3(s_2 - \lambda) \\ s_3s_4 - s_5(s_1 - \lambda) \end{bmatrix} \quad (\text{Eq. 9.40})$$

Vortex Core References

- M. S. Chong, A. E. Perry, and B. J. Cantwell. Copyright © 1990. Phys. Fluid. *A General Classification of Three Dimensional Flow Fields*. 765-777. A 2.
- U. Dallman, A. Hilgenstock, B. Schulte-Werning, S. Riedelbauch, and H. Vollmers. Copyright © 1991. AGARD Conf. Proc. CP-494. *On the Footprints of Three-Dimensional Separated Vortex Flows Around Blunt Bodies*.
- R. Haimes and D. Sujudi. Copyright © 1995. Dept. of Aeronautics and Astronautics, MIT, Cambridge, MA. *Identification of Swirling Flow in 3D Vector Fields*. Tech. Report.
- J. C. R. Hunt, A. A. Wary, and P. Moin. Copyright © 1988. NASA Ames / Stanford University in Oroc. 1988 Summer Program of the Center for Turbulent Research. *Eddies, Streams, and Convergence Zones in Turbulent Flows*. 193-207.
- J. Jeong and F. Hussain. Copyright © 1995. Journal of Fluid Mechanics. *On the Identification of a Vortex*. 69-94. 285.
- M. Jiang, R. Machiraju, and D. Thompson. Copyright © 2002. Eurographics – IEEE VGTC Symposium on Visualization. *A Novel Approach to Vortex Core Region Detection*.
- S. K. Robinson, S. J. Kline, and P. R. Spalart. Copyright © 1988. In Proc. Zoran P. Zaric Memorial International Seminar on Near Wall Turbulence. *Statistical Analysis of Near-wall Structures in Turbulent Channel Flow*.
- M. Roth and R. Peikert. Copyright © 1998. *A Higher-order Method for Finding Vortex Core Lines*.
- J. Sahner, T. Weinkauff, and H.-C. Hege. Copyright © 2005. Eurographics – IEEE VGTC Symposium on Visualization. *Galilean Invariant Extraction and Iconic Representation of Vortex Core Lines*.
- S. Zhang and D. Choudhury. Copyright © 2006. Phys. Fluids 18. *Eigen Helicity Density: A New Vortex Identification Scheme and its Application in Accelerated Inhomogeneous Flows*.
- J. Zhou, R. J. Adrian, and S. Balachander. Copyright © 1996. Phys. Fluids 8. *Autogeneration of Near Wall Vertical Structure in Channel Flow*. 288-291.
- J. Zhou. Copyright © 1997. Ph.D. thesis, Department of Theoretical and Applied Mechanics, University of Illinois at Urbana-Champaign, Urbana, Illinois. *Self-sustaining Formation of Packets of Hairpin Vortices in a Turbulent Wall Layer*.
- J. Zhou, R. J. Adrian, S. Balachander, and T. M. Kendall. Copyright © 1999. Journal of Fluid Mechanics. *Mechanisms for Generating Coherent Packets of Hairpin Vortices in Channel Flow*. 353-396. 387.

Level

The **Level** setting controls the strength of the vortex core that is displayed. The **Level** setting is normalized between **Method** types so that it is easy for you to compare the output of the different methods.

Actual Value

The **Actual Value** setting displays the isosurface value. This read-only value varies between methods.

Vortex Core Region: Color

To learn how to use color to show how a variable changes through a region or just to change the color of the vortex core regions, see [Color Details Tab \(p. 16\)](#).

Vortex Core Region: Render

To learn how to control the display of mesh lines, textures, and vortex core faces, see [Render Details Tab \(p. 18\)](#).

Vortex Core Region: View

The **View** tab is used for creating or applying predefined Instance Transforms for a wide variety of objects.

For details on changing the view settings, see [View Details Tab \(p. 21\)](#).

Surface of Revolution Command

A **Surface of Revolution** is a surface created by revolving a polyline about an axis. The polyline may be as simple as a single line segment or as complicated as a general curve.

The following characteristics of surfaces of revolution will be discussed:

- [Surface of Revolution: Geometry \(p. 108\)](#)
- [Surface of Revolution: Color \(p. 110\)](#)
- [Surface of Revolution: Render \(p. 110\)](#)
- [Surface of Revolution: View \(p. 110\)](#)

Note

There are several ways to insert a surface of revolution:

- From the menu bar, select **Insert > Location > Surface of Revolution**.
- From the tool bar, select **Location > Surface of Revolution**.
- Depending on the context, you may be able to perform an insert from the shortcut menu in the tree view.

Surface of Revolution: Geometry

Domains

For details, see [Domains \(p. 90\)](#).

Definition

Method

The **Method** setting has the following options:

Option	Description
Cylinder	Creates a cylinder using two axial and one radial coordinate points.
Cone	Creates a cone using two axial and radial coordinate points.
Disc	Creates a disc using one axial and two radial coordinate points.
Sphere	Creates a cylinder using one axial and radial coordinate points.
From Line	Enables you to specify a line or polyline to revolve about the axis (to be specified later).

Point 1 (a,r) and Point 2 (a,r)


These text boxes are not available for the **From Line** option. These text boxes specify axial and radial coordinates to define the surface of revolution.

Only one set of coordinates are available for the **Sphere** option. The axial value offsets the sphere in the direction of the rotational axis, and the radial value is used as the radius of the sphere.

Line

Line is available only if the **From Line** option is selected. The **Line** setting selects a valid line or polyline to use for rotation around the axis.

Tip

Click the *Location Editor*  icon to open the **Location Selector** dialog box, which displays the complete list of available lines.

of Samples

of Samples is not available if the **From Line** option is selected. The **# of Samples** setting sets the amount of sample points in the direction of the rotational axis.

Theta Samples

The **Theta Samples** setting specifies the amount of sample points evenly rotated around the rotational axis. For example, increasing this setting would make a cylinder's curve around its origin more accurate (more like a circle).

Project to AR Plane Check Box

The **Project to AR Plane** check box is available only if the **From Line** option is selected. If **Project to AR Plane** is selected (it is by default), then the **Theta** values will be projected to the plane of constant **Theta**. This produces a more refined mesh.

Rotation Axis

Method

The **Method** setting has the following options:

Option	Description
Principal Axis	Enables you to specify a principal axis to rotate around.
Rotation Axis	Enables you to specify a custom axis to rotate around using a line.

Axis

Axis is available only if the **Principal Axis** option is selected. The **Axis** setting enables you to select from a list the X, Y, or Z axis to rotate around.

From/To Text Boxes

The **From** and **To** text boxes are available only if the **Rotation Axis** option is selected. These text boxes create a line representing the axis about which the Solid of Revolution is created.

Angle Range Check Box

Select the **Angle Range** check box if you want to specify a minimum or maximum angle to rotate to.

Min./Max. Angle

These settings specify a minimum and/or maximum angle to rotate to.

Axial/Radial Offset

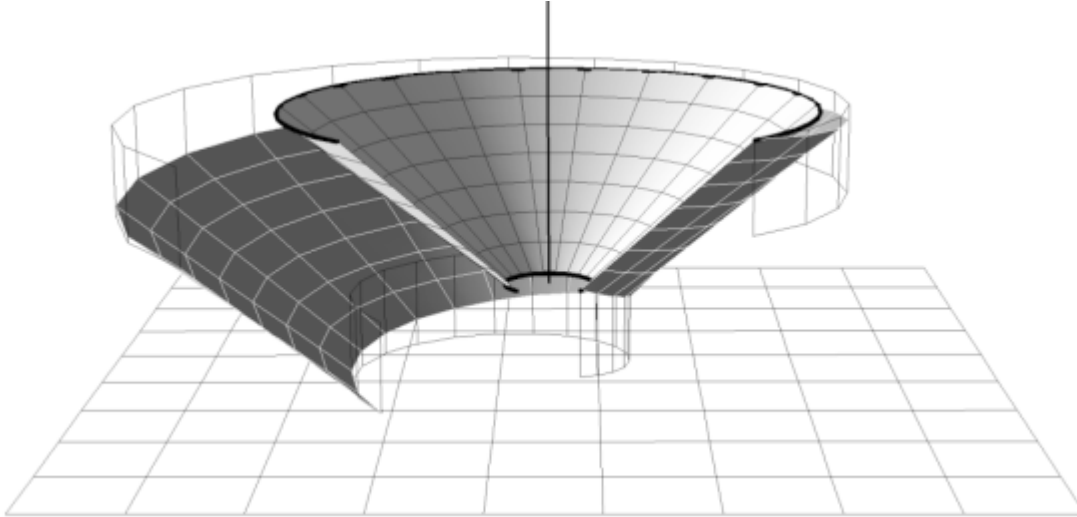
Start/End A

These settings specify a start and end offset along the axis of rotation.

Start/End R

These settings specify a start and end offset for the radius.

The following image shows two partial cones with the same profile and theta limits. For the end profile of one of the cones, the radial offset is positive and the axial offset is negative, causing the radius to increase and the axial coordinate to decrease with increasing theta (as determined by the right hand rule with reference to the axis shown). Two other surfaces of revolution were included in the figure to help illustrate axial displacements.



Surface of Revolution: Color

The color settings can be changed by clicking the **Color** tab. For details, see [Color Details Tab \(p. 16\)](#).

Surface of Revolution: Render

The rendering settings can be changed by clicking the **Render** tab. For details, see [Render Details Tab \(p. 18\)](#).

Surface of Revolution: View

The **View** tab is used for creating or applying predefined Instance Transforms for a wide variety of objects.

For details on changing the view settings, see [View Details Tab \(p. 21\)](#).

Polyline Command

A polyline is a line connecting a series of points. The points may have local (path) variables associated with them. The polyline can interact with CFD data and can be colored using path variables or domain variables.

The following characteristics of polylines will be discussed:

- [Polyline: Geometry \(p. 111\)](#)
- [Polyline: Color \(p. 112\)](#)
- [Polyline: Render \(p. 112\)](#)
- [Polyline: View \(p. 112\)](#)

Note

There are several ways to insert a polyline:

- From the menu bar, select **Insert > Location > Polyline**.

- From the tool bar, select **Location > Polyline**.
- Depending on the context, you may be able to perform an insert from the shortcut menu in the tree view.


Polyline: Geometry

Method

The Method setting has the following options:

Option	Description
From File	Enables you to specify a file that has the point data contained within it. The data file format is described in POLYLINE Data Format (p. 64) .
Boundary Intersection	Enables you to select a boundary and an object to intersect it with. The line will then plot on the intersection.
From Contour	Enables you to plot using contour data (for example, a velocity of 5 m/s).

File

File is available only if the **From File** option is selected. The **File** setting specifies the filename of a file to insert. You can type in the filename or click *Browse*  to open the **Import** dialog box and search for the file. The only valid file types to import are *.txt and *.csv.

Tip

This method enables you to read polylines or lines from another case (if that case has the required geometry). First export a polyline or a line from another case, make sure to select **Export Geometry Information**, then use the **From File** method in the other case to import the lines along with any local data. You can also create your own file containing your data, such as experimental data, by using the same format. For a description of the polyline file format, see [POLYLINE Data Format \(p. 64\)](#).


Domains

Domains is available only if the **Boundary Intersection** option is selected. The **Domains** setting selects a domain for the polyline to exist in. For details, see [Domains \(p. 90\)](#).

Boundary List

Boundary List is available only if the **Boundary Intersection** option is selected. The **Boundary List** setting specifies a boundary.

Tip

Click the *Location Editor*  icon to open the **Location Selector** dialog box, which displays the complete list of available boundaries.

Intersect With

Intersect With is available only if the **Boundary Intersection** option is selected. The **Intersect With** setting specifies a graphic object that intersects the boundary.

Note

When intersecting with a thin surface boundary, the resulting polyline will include both sides of the boundary. To intersect only one side, pick the primitive region that defines one side of the thin surface instead of the entire boundary.

Contour Name

Contour Name is available only when the **From Contour** option is selected. The **Contour Name** setting selects a predefined contour plot. If you have not created a contour, see [Contour Command \(p. 119\)](#)

Contour Level

Contour Level is available only when the **From Contour** option is selected. The **Contour Level** setting specifies a contour level. The amount of contour levels is predefined by the [Contour Command \(p. 119\)](#).

Polyline: Color

The color settings can be changed by clicking the **Color** tab. For details, see [Color Details Tab \(p. 16\)](#).

Polyline: Render

The rendering settings can be changed by clicking the **Render** tab. For details, see [Render Details Tab \(p. 18\)](#).

Polyline: View

The **View** tab is used for creating or applying predefined Instance Transforms for a wide variety of objects.

For details on changing the view settings, see [View Details Tab \(p. 21\)](#).

User Surface Command

A user surface can be defined in a number of different ways:

- From a file containing data points.
- From the intersection of a boundary and an existing locator.
- From a contour fringe number.
- By transforming an existing surface.
- Offset from an existing surface. The offset can be uniform or described by a variable.

The following characteristics of user surfaces will be discussed:

- [User Surface: Geometry \(p. 112\)](#)
- [User Surface: Color \(p. 115\)](#)
- [User Surface: Render \(p. 115\)](#)
- [User Surface: View \(p. 115\)](#)

Note

There are several ways to insert a user surface:

- From the menu bar, select **Insert > Location > User Surface**.
- From the tool bar, select **Location > User Surface**.
- Depending on the context, you may be able to perform an insert from the shortcut menu in the tree view.

User Surface: Geometry

Method

The **Method** setting has the following options:

Option	Description
From File	Same as for the polyline object. For details, see Polyline: Geometry: Method (p. 111) .

Option	Description
	The data file format is described in USER SURFACE Data Format (p. 65) .
Boundary Intersection	Same as for the polyline object. For details, see Polyline: Geometry: Method (p. 111) .
From Contour	Same as for the polyline object. For details, see Polyline: Geometry: Method (p. 111) .
Transformed Surface	Enables you to specify a preexisting surface to plot. You may specify a rotation, translation, and uniform scale for the user surface.
Offset From Surface	Enables you to specify a preexisting surface to plot. You may specify different methods of offset for the user surface.
ANSYS	Similar to the From File option, except that this option uses ANSYS files to load into the instance. You may also specify an associated boundary for the file to be loaded onto. For details, see Specify Associated Boundary Check Box (p. 115) .

File

File is the same for the polyline object. For details, see [Polyline: Geometry: File \(p. 111\)](#).

Tip

This method enables you to read surfaces from another case. First export a surface (such as a plane or a boundary) from another case and make sure to select **Export Geometry Information** and **Export Line and Face Data**. Then use the From File method in the other case to import the surface along with any local data. You can also create a file containing your own data, such as experimental data, by using the same format. For a description of the surface file format, see [USER SURFACE Data Format \(p. 65\)](#).

Domains/Boundary List/Intersect With

These settings are the same as for a polyline, except that instead of outlining the intersection, a line of intersection is formed between the boundaries and the location. Each mesh element that the line passes through forms part of the User Surface. For details, see [Domains \(p. 111\)](#).

Contour Name/Contour Level

These settings are the same as for a polyline, except that instead of outlining the contour, the User Surface fills in all of the area above the contour level entered and below the contour level above. Also, when applicable, **Contour Level 1** creates a surface below the first contour line. For details, see [Contour Name \(p. 112\)](#).

Surface Name

Surface Name is available only if either the Transformed Surface or Offset From Surface options are selected. The **Surface Name** setting selects a surface on which to plot the User Surface.

Rotation Check Box

The **Rotation** check box is available only if the Transformed Surface option is selected. Select the **Rotation** check box to specify a rotation for the User Surface. For details, see [Apply Rotation Check Box \(p. 135\)](#).

Translation Check Box

The **Translation** check box is available only if the Transformed Surface option is selected. Select the **Translation** check box to specify a translation for the User Surface. For details, see [Apply Translation Check Box \(p. 136\)](#).

Scale Check Box

The **Scale** check box is available only if the **Transformed Surface** option is selected. Select the **Scale** check box to specify a scale for the User Surface. Use the **Scale** text box to specify a uniform scale factor.

Type

Type is available only if the **Offset From Surface** option is selected. The **Type** setting has the following options:

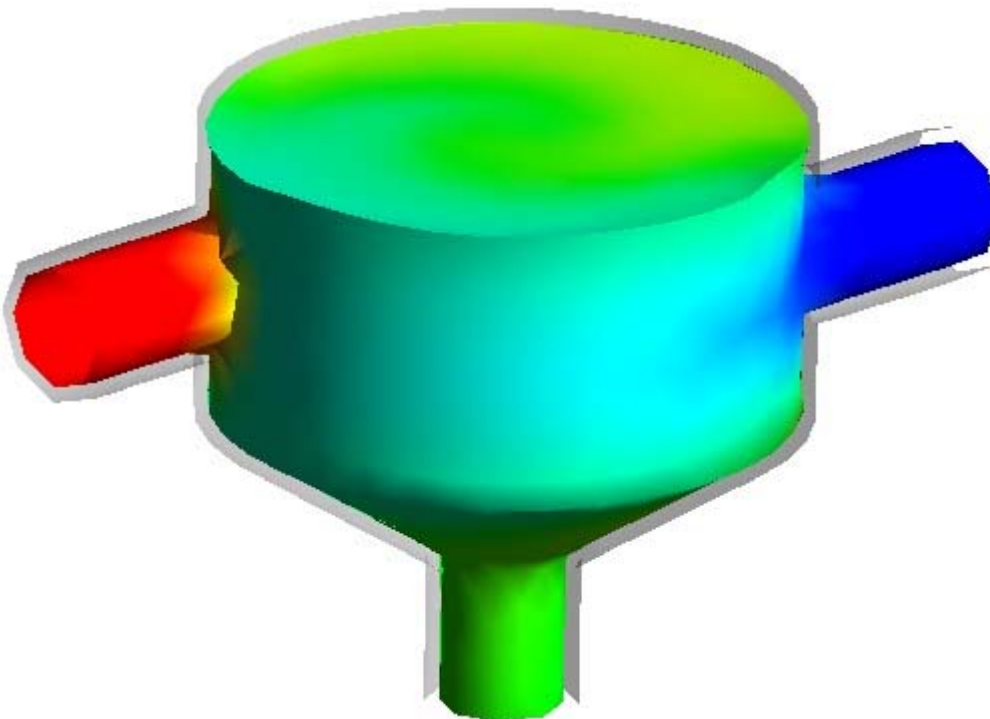
Option	Description
Normal	Enables you to offset the User Surface normal to selected surface.
Translational	Enables you to offset the User Surface from the selected surface by moving the User Surface.

Mode

Mode is available only if the **Offset From Surface** option is selected. The **Mode** setting has the following options:

Option	Description
Uniform	Enables you to specify a uniform offset.
Variable	Enables you to select a variable to plot from the surface.

An example of a uniform normal offset of -0.1 [m] to the **Default** surface of the static mixer, colored by Temperature, is shown in the diagram.



Distance

Distance is available only if the **Uniform** option is selected. The **Distance** setting specifies an offset distance, whether it is translational or normal.

Variable

Variable is available only if the **Variable** option is selected. The **Variable** setting specifies a variable to plot.

When the distance is described by a variable, you can also incorporate the variable into an expression. For example, after you have chosen a variable you can click in the **Distance** box and amend it with valid CFX Expression Language (CEL) (for example, $0.5 * \text{Temperature}$).

Direction

Direction is available only if the **Translational** option is selected. The **Direction** setting selects a direction to offset the User Surface. Increased values do not increase the translational offset, they merely change the ratio that the offset X, Y, and Z directions are placed at. For example, $[2, 3, 1]$ and $[4, 6, 2]$ would identically offset the User Surface.

Specify Associated Boundary Check Box

The **Specify Associated Boundary** check box is available only if the **ANSYS** option is selected. This setting is also available in an import menu. For details, see [Import ANSYS CDB Surface \(p. 66\)](#).

User Surface: Color

The color settings can be changed by clicking the **Color** tab. For details, see [Color Details Tab \(p. 16\)](#).

User Surface: Render

The rendering settings can be changed by clicking the **Render** tab. For details, see [Render Details Tab \(p. 18\)](#).

User Surface: View

The **View** tab is used for creating or applying predefined Instance Transforms for a wide variety of objects.

For details on changing the view settings, see [View Details Tab \(p. 21\)](#).

Surface Group Command

A surface group enables you to create a locator consisting of multiple surface locators.

The following characteristics of user surface groups will be discussed:

- [Surface Group: Geometry \(p. 115\)](#)
- [Surface Group: Color \(p. 116\)](#)
- [Surface Group: Render \(p. 116\)](#)
- [Surface Group: View \(p. 116\)](#)

Note

There are several ways to insert a surface group:

- From the menu bar, select **Insert > Location > Surface Group**.
- From the tool bar, select **Location > Surface Group**.
- Depending on the context, you may be able to perform an insert from the shortcut menu in the tree view.

Surface Group: Geometry

Domains

The **Domains** setting selects the domains in which the surface group will exist. For details, see [Domains \(p. 90\)](#).

Locations

The **Locations** setting specifies a location or locations on which to plot the Surface Group. For details, see [Locations \(p. 92\)](#).

Surface Group: Color

The color settings can be changed by clicking the **Color** tab. For details, see [Color Details Tab \(p. 16\)](#).

Surface Group: Render

The rendering settings can be changed by clicking the **Render** tab. For details, see [Render Details Tab \(p. 18\)](#).

Surface Group: View

The **View** tab is used for creating or applying predefined Instance Transforms for a wide variety of objects.

For details on changing the view settings, see [View Details Tab \(p. 21\)](#).

Turbo Surface Command

Turbo surfaces are graphic objects that can be viewed and used as locators, just like other graphic objects.

Note

There are two ways to insert a turbo surface:

- From the menu bar, select **Insert > Location > Turbo Surface**.
- From the tool bar, select **Location > Turbo Surface**.

For details on working with turbo surfaces, see [Turbo Surface \(p. 190\)](#).

Turbo Line Command

Turbo lines are graphic objects that can be viewed and used as locators, just like other graphic objects.

Note

There are two ways to insert a turbo line:

- From the menu bar, select **Insert > Location > Turbo Line**.
- From the tool bar, select **Location > Turbo Line**.

For details on working with turbo lines, see [Turbo Line \(p. 193\)](#).

Vector Command

A **Vector Plot** is a collection of vectors drawn to show the direction and magnitude (optional) of a vector variable on a collection of points. These points, known as seeds, are defined by a location.


When post-processing a GGI simulation, the velocity vectors can be plotted in the local frame of reference for each domain (**Velocity Field Selection**) or in the absolute frame of reference for each domain (**Velocity in a Stationary Frame**). These two choices produce the same plot in all stationary frame domains, but plot either the rotating frame or absolute frame velocity vectors in domains that are in the rotating frame of reference.

The following characteristics of vectors will be discussed:

- [Vector: Geometry \(p. 117\)](#)
- [Vector: Color \(p. 118\)](#)
- [Vector: Symbol \(p. 118\)](#)
- [Vector: Render \(p. 119\)](#)
- [Vector: View \(p. 119\)](#)

Note

There are several ways to insert a vector plot:

- From the menu bar, select **Insert > Vector**.
- From the toolbar, click the *Vector*  icon.
- Depending on the context, you may be able to perform an insert from the shortcut menu in the 3D Viewer.

Vector: Geometry

Domains

For details, see [Domains \(p. 90\)](#).

Definition

Locations

Locations is the same for the Point Cloud object. For details, see [Locations \(p. 92\)](#).


Sampling

Sampling and all of the settings that correspond to it are the same for the Point Cloud object. For details, see [Sampling \(p. 92\)](#).

Variable

The **Variable** setting selects a variable from the list to plot at the selected location.

Tip

Click the *Location Editor*  icon to open the **Variable Selector** dialog box, which displays the complete list of available variables.

Hybrid/Conservative Options

For details, see [Hybrid and Conservative Variable Values \(p. 165\)](#).

Projection

The **Projection** setting has the following options:

Option	Description
None	Original vectors are plotted without any projection.
Coord Frame	Plots vector components aligned with a principal axis or an axis of a custom coordinate frame.
Normal	Plots vector components normal to the location. Applicable only for surface locations.
Tangential	Plots vector components tangential to the location. Applicable only for surface locations.

When a rotation axis is defined (set in the Turbo tab, or by reading a turbo case), the **Projection** setting has the following additional options:

Option	Description
Axial	Plots vector components along the rotation axis. Available when a rotation axis is defined.
Radial	Plots vector components radially to the rotation axis. Available when a rotation axis is defined.
Circumferential	Plots vector components along the theta direction about the rotation axis. Available when a rotation axis is defined.

Direction

There are two drop-down list boxes for this setting. The first list represents the options for the range of the vector. The second list box represents the available directions to plot the vector in.

Vector: Color

The color settings can be changed by clicking the **Color** tab. For details, see [Color Details Tab \(p. 16\)](#).

Vector: Symbol

Symbol

The **Symbol** setting has the following options to select a shape for the vector:

Option	Description
Line Arrow	Displays the vector as a line arrow. This option takes the least amount of memory and is suggested for large vector field plots.
Arrow2D	Displays a filled line arrow.
Arrow3D	Displays a 3D filled line arrow.
Arrowhead	Displays the tip of the Arrow2D option.
Arrowhead3D	Displays a 3D version of the Arrowhead option.
Fish3D	Displays a 3D fish.
Ball	Displays a sphere at every vector point. This option does not specify a direction, only a scalar value.
Crosshair	Displays a 3D “+” sign. This option, through its natural shape, displays the normal and the tangential vector to the surface automatically. However, the crosshair does not point to the actual direction (does not have an arrow pointing the direction of the actual vector).
Octahedron	Displays a filled Crosshair option.
Cube	Displays a 3D box. One face of the cube lies tangent to the surface and one of the corners points in the direction of the vector.

Symbol Size

The **Symbol Size** setting specifies the scale for the vectors symbol.

Normalize Symbols Check Box

Select the **Normalize Symbols** check box to make all of the vectors the same size.

Vector: Render

The rendering settings can be changed by clicking the **Render** tab. For details, see [Render Details Tab \(p. 18\)](#).

Vector: View

The **View** tab is used for creating or applying predefined Instance Transforms for a wide variety of objects.

For details on changing the view settings, see [View Details Tab \(p. 21\)](#).

Contour Command


A contour plot is a series of lines linking points with equal values of a given variable. For example, contours of height exist on geographical maps and give an impression of gradient and land shape.

The following characteristics of contours will be discussed:

- [Contour: Geometry \(p. 119\)](#)
- [Contour: Labels \(p. 120\)](#)
- [Contour: Render \(p. 120\)](#)
- [Contour: View \(p. 120\)](#)

Note

There are several ways to insert a contour plot:

- From the menu bar, select **Insert > Contour**.
- From the toolbar, click the *Contour*  icon.
- Depending on the context, you may be able to perform an insert from the shortcut menu in the tree view or in the 3D Viewer.

Contour: Geometry

Domains

For details, see [Domains \(p. 90\)](#).

Locations

For details, see [Locations \(p. 92\)](#).

Variable

For details, see [Mode: Variable and Use Plot Variable \(p. 16\)](#).

Range

For details, see [Range \(p. 16\)](#). In addition to the options specified in the link, there is the following option. Value List is a comma-separated list that enables you to specify the actual values at which contours should be plotted. For example, if plotting temperature in a combustor, you might try a value list of 300, 500, 700, 900, and 1100K. It should be noted that entering a value list overrides the number specified in the [# of Contours \(p. 120\)](#) text box.

Hybrid / Conservative Options

For details, see [Hybrid/Conservative \(p. 17\)](#).

Color Scale

For details, see [Color Scale \(p. 17\)](#).

Color Map

For details, see [Color Map \(p. 17\)](#).

of Contours

The **# of Contours** setting specifies the number of contours in the plot. This will not increase the range, it will increase only the number of contours within the range.

Clip to Range Check Box

Select the **Clip to Range** check box to plot values only within the specified **Range**. If selected, you should use this setting in conjunction with the **User Specified range**.

Contour: Labels

Show Numbers Check Box

Select the **Show Numbers** check box to display numbers for the contour lines and edit their appearance. The contour numbers will appear next to the contour values in the legend.

Text Height

The **Text Height** setting specifies a value for the text height. The value corresponds to a ratio of the height of the 3D Viewer. For example, a value of 1 would display the contour numbers to be the full height of the 3D Viewer.

Text Font

The **Text Font** setting specifies a font from the list.

Color Mode

The **Color Mode** setting has the following options:

Option	Description
Default	Displays the text as grey.
User Specified	Enables you to specify a custom color.

Text Color

The **Text Color** setting selects a custom color. You can select a predefined color by clicking the color bar.

Tip

Click the *Location Editor*  icon to open the **Select color** dialog box, which displays the complete range of available colors.

Contour: Render

The **Render** tab for a contour does not contain an **Apply Texture** section, but does contain the other sections described in [Render Details Tab \(p. 18\)](#).

Contour: View

The **View** tab is used for creating or applying predefined Instance Transforms for a wide variety of objects.

For details on changing the view settings, see [View Details Tab \(p. 21\)](#).

Streamline Command

A *streamline* is the path that a particle of zero mass would take through the fluid domain. The path is calculated using a Runge-Kutta method of vector variable integration with variable timestep control. Streamlines start at each node on a given locator.


The assumption of steady state flow is assumed when a streamline is created, even with a transient simulation. Although the CFD-Post streamline algorithm is efficient, the calculation of large numbers of streamlines in a large domain can still take a long period of time. Therefore, when calculating streamlines for a solution for the first time, start by plotting a small number of streamlines and then increase the number of streamlines until the best generation time vs. detail ratio is found.

The following characteristics of streamlines will be discussed:

- [Streamline: Geometry \(p. 121\)](#)
- [Streamline: Color \(p. 123\)](#)
- [Streamline: Symbol \(p. 123\)](#)
- [Streamline: Limits \(p. 124\)](#)
- [Streamline: Render \(p. 125\)](#)
- [Streamline: View \(p. 125\)](#)

Note

There are several ways to insert a streamline:

- From the menu bar, select **Insert > Streamline**.
- From the toolbar, click the *Streamline*  icon.
- Depending on the context, you may be able to perform an insert from the shortcut menu in the tree view or in the 3D Viewer.

Streamline: Geometry

Type

The **Type** setting has the following options:

Option	Description
3D Streamline	Plots the streamline inside a specified domain from a location.
Surface Streamline	Plots the streamline on a surface from a location. A Surface Streamline is defined as a line everywhere tangent to the surface variable component at a given instant of time.

Definition

Domains

Domains is available only if the 3D Streamline option is selected. For details, see [Domains \(p. 90\)](#).

Start From (3D Streamline)

Start From is available only if the 3D Streamline option is selected. The **Start From** setting selects a location or locations to start from. For details, see [Locations \(p. 92\)](#).

If you are starting your streamlines from an inlet, outlet, or slice plane, you are advised to use the **Factor** text box to reduce the number of streamlines. If your solution is likely to contain recirculation areas, or regions of high vorticity, you are advised to reduce the **Max Segments** number to a few hundred streamlines. If the streamlines stop part of the way through the domain, increase the **Max Segments** value until you receive good results.

Surfaces

Surfaces is available only if the **Surface Streamline** option is selected. The **Surfaces** setting selects a location or locations to plot on. For details, see [Locations \(p. 92\)](#).

Start From (Surface Streamline)

Start From is available only if the **Surface Streamline** option is selected. This setting is similar to the **Sampling** setting for a **Point Cloud** object. For details, see [Sampling \(p. 92\)](#). The differences between the settings are that you can not select the **Random** option for this setting and that the **Start From** setting also has the **Locations** option.

Locations

Locations is available only if the **Locations** option is selected. For details, see [Locations \(p. 92\)](#).

Sampling

Sampling is available only if the **3D Streamline** option is selected. The **Sampling** setting is identical to the **Sampling** setting for a **Point Cloud** object, except that you cannot select the **Random** option for this setting. For details, see [Sampling \(p. 92\)](#).

Preview Seeds Button

Click the **Preview Seeds** button to display in the **Viewer** where the streamlines will originate from.

Variable

Use **Variable** to select a variable to plot. Using the **Velocity** variable is recommended. For details, see [Variable \(p. 117\)](#).

Hybrid/Conservative Options

For help on which field to select, see [Hybrid and Conservative Variable Values \(p. 165\)](#).

Direction

The **Direction** setting has the following options:

Option	Description
Forward	Specifies that the streamline goes only in the positive direction from the start point.
Backward	Specifies that the streamline goes only in the negative direction from the start point.
Forward and Backward	Specifies that the streamline goes in both the positive and negative directions from the start point.

Cross Periodics Check Box

Cross Periodics is available only if the **3D Streamline** option is selected. Select the **Cross Periodics** check box to have the streamline cross from one periodic interface to the opposite boundary. A periodic interface can be defined by selecting a periodic option for a domain interface.

Domain interfaces are used for multiple purposes:

- Connecting domains or assemblies Domain Interfaces are required to connect multiple unmatched meshes within a domain (for example, when there is a hexahedral mesh volume and a tetrahedral mesh volume within a single domain) and to connect separate domains.
- Modeling changes in reference frame between domains This occurs when you have a stationary and a rotating domain or domains rotating at different rates.
- Creating periodic interfaces between regions This occurs when you are reducing the size of the computational domain by assuming periodicity in the simulation.

Simplify Streamline Geometry Check Box

Simplify Streamline Geometry is available only if the **Surface Streamline** option is selected. Select the **Simplify Streamline Geometry** check box to interpolate a linear line in between points if the streamline is almost linear. This will have negative effects if you plot a variable on the streamline because the linearly interpolated line will omit the points in between the points that create the line.

Streamline: Color

There are two additional options for coloring streamlines not available on other objects. These are **Time** and **Unique**. The **Time** option colors the streamline by the amount of time a massless particle would take to get to each point of the streamline, starting at the location. The **Unique** option gives each streamline a different color along its whole length, and can be used to track individual streamlines through a domain. For details on how to use the rest of the **Color** tab, see [Color Details Tab \(p. 16\)](#).


Streamline: Symbol

The **Symbol** tab adds markers to each streamline at given time intervals.

Show Symbols Check Box

Select the **Show Symbols** check box to draw symbols at a user-specified time interval along the streamline.

Min Time

The **Min Time** setting specifies a minimum time to start plotting the symbols. The time value can also be an expression. To create an expression, click the *Expression*  icon and enter the expression.

Max Time

The **Max Time** setting specifies a maximum time to stop plotting the symbols. The time value may also be an expression. To create an expression, click the *Expression*  icon and enter the expression.

Interval

The **Interval** setting specifies the time interval at which you want to plot the symbols.

Symbol

The same options are available for the **Symbol** setting for the vector object. For details, see [Symbol \(p. 118\)](#). The symbols are drawn along the vector for the streamline at the given point.

Symbol Size

This setting is identical to the **Symbol Size** setting for the vector object. For details, see [Symbol Size \(p. 118\)](#).

Show Streams Check Box

Select the **Show Streams** check box to display the streamline or streamlines.

Stream Type

The **Stream Type** setting has the following options:

Option	Description
Line	Plots the streamline as a line.
Tube	Plots the streamline in tube shape.
Ribbon	Plots the streamline in a flat tube shape. Ribbons also displays axial rotation of the fluid as it passes through the domain.

Line Width/Tube Width/Ribbon Width

These settings control the width of the streamline.

of Sides

of Sides is available only if the **Tube** option is selected. The **# of Sides** setting specifies the number of sides to the tube. The minimum number of sides is 3 and the maximum is 20.

Initial Direction

Initial Direction is available only if the **Ribbon** option is selected. The **Initial Direction** setting specifies the initial direction of the ribbon streamline.

Streamline: Limits

The **Limits** tab enables modification of the tolerance, segments, and maximum time settings.

Step Tolerance

The values for the streamline (location and direction) are calculated at points determined by the step tolerance mode. You can choose to have streamline elements calculated relative to the mesh (grid) or at absolute increments as shown in the table that follows:

Mode

The **Mode** setting has the following options:

Option	Description
Grid Relative	Specifies that the streamline must lie within the specified fraction of the local grid cell size. Selecting Grid Relative means that the Tolerance is directly proportional to the mesh spacing. In areas where the mesh has been refined (such as areas where the flow pattern changes quickly), the Tolerance setting reduces the distance between streamline points proportionately. This in turn produces more accurate streamlines in these areas.
Absolute	Specifies that the calculation points for streamline elements must lie within the Tolerance distance specified.

Tolerance

The **Tolerance** setting specifies the accuracy of the path. As the **Tolerance** setting becomes finer, the accuracy increases but the calculation time increases.

Upper Limits

Max Segments

The **Max Segments** setting specifies the maximum number of segments allowed for a streamline before it ends.

Max Time

The **Max Time** setting specifies the maximum time allowed to pass before the streamline ends. A time of zero, in this case, represents infinite time (because zero would actually plot nothing).

Max Periods

The **Max Periods** setting is available only if the **Cross Periodics** check box is selected in the **Geometry** tab. This setting sets the number of times a streamline is able to pass through a periodic boundary.

Streamline: Render

The render settings can be changed by clicking the **Render** tab. For details, see [Render Details Tab \(p. 18\)](#).

Streamline: View

The **View** tab is used for creating or applying predefined Instance Transforms for a wide variety of objects.

For details on changing the view settings, see [View Details Tab \(p. 21\)](#).

Particle Track Command

In complex flows, it is often useful to track the flow of discrete particles through the flow field. These particles interact with the fluid, following a path that is determined by the particle properties, as well as by the mean and turbulent flow behavior. The tracking is useful in two ways:

- Particle tracking can trace the mean flow behavior in and around complex geometries.
- The injection of several particles from a point can help to display the turbulence properties of the flow.

Particle tracking information is written to the results file in CFD-Post. The parameters are set in the preprocessor. CFD-Post also provides support for track files created in CFX by allowing the import of particle tracking data from a separate file. If a CFX results file contains particle tracking data, an object will exist in the tree view of type **Res Particle Track**.


To create a new particle track, select **Insert > Particle Track** from the main menu.

The following characteristics of particle tracks will be discussed:

- [Particle Track: Geometry \(p. 125\)](#)
- [Particle Track: Color \(p. 127\)](#)
- [Particle Track: Symbol \(p. 127\)](#)
- [Particle Track: Render \(p. 128\)](#)
- [Particle Track: View \(p. 128\)](#)
- [Particle Track: Info \(p. 128\)](#)

Note

There are several ways to insert a particle track:

- From the menu bar, select **Insert > Particle Track**.
- From the toolbar, click the *Particle Track*  icon.
- Depending on the context, you may be able to perform an insert from the shortcut menu in the tree view.

Particle Track: Geometry

Method

The **Method** setting has the following options:

Option	Description
From Res	Creates the particle track from the current .res file. This option is available only with a valid results file.
From File	Creates the particle track data from the selected file.

Domains

Domains is available only if the `From Res` option is selected. For details, see [Domains \(p. 90\)](#).

Material


Material is available only if the `From Res` option is selected. The **Material** setting selects a material to emulate with the particle track.

File Type

File Type is available only if the `From File` option is selected. The **File Type** setting specifies a file type to load.

File

File is available only if the `From File` option is selected. The **File** setting specifies the filename of a file to load.

You may type in the filename or click *Browse*  to open the **Select <filetype> Particle Track File** dialog box, and search for the file.

Reduction Type

The **Reduction Type** setting has the following options:

Option	Description
Maximum Number of Tracks	Enables you to set the maximum number of tracks to be plotted.
Reduction Factor	Enables you to specify a reduction factor to decrease the number of tracks to be plotted.

Reduction

Reduction is available only if the `Reduction Factor` option is selected. This setting is the same as **Factor** for the Point Cloud object. For details, see [Factor \(p. 93\)](#).

Max Tracks

Max Tracks is available only if the `Maximum Number of Tracks` option is selected. The **Max Tracks** setting specifies the maximum number of tracks to be plotted.

Limits Option

The **Limits Option** setting has the following options:

Option	Description
Up to Current Timestep	Plots the track values up to the current timestep only.
Since Last Timestep	Plots the track values from the previous timestep to the current timestep.
User Specified.	Enables you to specify a beginning and ending time/distance.

Limit Type

Limit Type is available only if the `User Specified` option is selected. The **Limit Type** setting specifies either `Time` or `Distance` as the limiting variable for the plot.

Start/End <variable>

<variable>, in this case, is either **Time** or **Distance**. These settings are available only if the **User Specified** option is selected. These settings specify a start and end value for the selected limiting variable.

Filter Check Box

Select the **Filter** check box to specify filters. The settings included are **Start Region**, **End Region**, **Diameter**, **Track**, and the **Match ALL/Match ANY** options.

Start/End Region Check Boxes

These settings are available only if the **From Res** option is selected. Select the **Start/End Region** check boxes to filter out the tracks that do not start or end in the selected region.

Diameter Check Box

Select the **Diameter** check box and set the corresponding text and list boxes to place restrictions on particles at the injection location.

Track Check Box

Select the **Track** check box and enter numbers corresponding to tracks to display indicated tracks by entering a comma-delimited list of track numbers. You may also enter a range of track numbers. For example, -5 specifies tracks 1 to 5, 40- specifies track numbers above 40 and 10-100 specifies tracks 10 to 100. You may view the **Info** tab to view the **Total Tracks** and **Tracks Shown**. For details, see [Particle Track: Info \(p. 128\)](#).

Match ALL/Match ANY Options

Select **Match All** to display only the tracks that meet all of the specified **Filter** conditions. Selecting **Match Any** draws all tracks that meet one or more of the selected conditions.

Particle Track: Color

The color settings can be changed by clicking the **Color** tab. For details, see [Color Details Tab \(p. 16\)](#).

Particle Track: Symbol

Show Symbols Check Box

This setting and its options are similar to those for the **Show Symbols** check box for the streamline object, as described in [Show Symbols Check Box \(p. 123\)](#). The differences are that a particle track has a **Max Time is** setting and different symbol size options.

The **Size Option** choices are **Constant** and **Particle Diameter**:

- When **Size Option** is **Constant**, the symbol size is constant for all particles. The particle size displayed is a mean particle diameter size multiplied by the value you set with the **Scale** setting.
- When **Size Option** is **Particle Diameter**, the **Scale Type** can be **Absolute** or **Relative**.
 - When **Scale Type** is **Absolute**, the particle size displayed is a mean particle diameter size multiplied by the value you set with the **Scale** setting.
 - When **Scale Type** is **Relative**, symbol sizes are scaled by the domain.

Max Time is

The **Max Time is** setting has the following options:

Option	Description
User Specified	Enables you to enter a custom value for the maximum time value. This is the default for steady-state simulations.

Option	Description
Current Time	Uses the current timestep as the maximum time value. This is the default for transient simulations.

Show Tracks Check Box

The settings for this check box are the same as for the **Show Streams** check box for the streamline object. For details, see [Show Streams Check Box \(p. 123\)](#).

Show Track Numbers Check Box

Select the **Show Track Numbers** check box to display and edit the appearance of track numbers. The track numbers will be displayed at the beginning of each numbered track. This setting and its options are similar to the **Show Numbers** check box for the contour object. For details, see [Show Numbers Check Box \(p. 120\)](#).

Particle Track: Render

The render settings can be changed by clicking the **Render** tab. For details, see [Render Details Tab \(p. 18\)](#).

Particle Track: View

The **View** tab is used for creating or applying predefined Instance Transforms for a wide variety of objects.

For details on changing the view settings, see [View Details Tab \(p. 21\)](#).

Particle Track: Info

The **Info** tab displays information about the current state of the particle tracks. If you have changed settings on another tab menu, you must click **Apply** before the information is updated.

Text Command


Text can be added to the viewer for titles, annotations, or comments in CFD-Post.

The following characteristics of text will be discussed:

- [Text: Definition \(p. 128\)](#)
- [Text: Location \(p. 129\)](#)
- [Text: Appearance \(p. 130\)](#)

Note

There are several ways to insert a text object:

- From the menu bar, select **Insert > Object > Text**.
- From the toolbar, click the *Text*  icon.
- Depending on the context, you may be able to perform an insert from the shortcut menu in the tree view.

Text: Definition

Text String

The **Text String** setting enters text for the object. When `<aa>` appears, auto-annotation will be embedded there.

Embed Auto Annotation Check Box

Select the **Embed Auto Annotation** check box to insert auto-annotation into the text string.

Type

The **Type** setting has the following options:

Option	Description
Expression	Adds an expression, selected from a list, to the text string.
Timestep	Adds the current timestep to the text string.
Time Value	Adds the current time value to the text string.
Filename	Adds the filename or the entire pathname to the text string.
File Date	Adds the date that the file was created to the text string.
File Time	Adds the time that the file was created to the text string.

Expression

Expression is available only if the **Expression** option is selected. The **Expression** setting specifies an expression to enter into the text string.

Format (for Filename option)

Format is available only if the **Filename** option is selected. The **Format** setting selects either **Entire Path** or **Filename Only** to insert into the text string.

Format (for the File Date and File Time options)

Format is available if either the **File Date** or **File Time** options are selected. The **Format** setting selects a time format to enter into the text string.

More/Fewer Buttons

Click the **More** button to create another line of text. Click the **Fewer** button to remove these added lines of text.

Text: Location

Location

Position Mode

The **Position Mode** setting has the following options:

Option	Description
Two Coords	Specifies the text to sit in the Viewer in the 2D plane.
Three Coords	Specifies the text to be fixed to one point in the Viewer and rotate with that point when the view is rotated.

X Justification

This setting is available only if the **Two Coords** option is selected and is the same for the Legend object. For details, see [X Justification \(p. 133\)](#).

Y Justification

This setting is available only if the **Two Coords** option is selected and is the same for the Legend object. For details, see [Y Justification \(p. 133\)](#).

Position (for Two Coords option)

Position is available only if the **Two Coords** option is selected. The **Position** setting specifies a fixed 2D point at which the text will be displayed.

Position (for Three Coords option)

Position is available only if the **Three Coords** option is selected. The **Position** setting specifies a 3D point at which the text will be displayed.

Rotation

The **Rotation** setting specifies a rotation for the text about the bottom-left corner of text in a counterclockwise direction. When the **X/Y Justification** is set to **Center**, the object rotates about the center point of the text.

Text: Appearance

Height

The **Height** setting specifies a text height. The value is equivalent to a fraction of the Viewer size.

Color Mode

This setting and its corresponding settings are the same for the Contour object. For details, see [Color Mode \(p. 120\)](#).

Font


The **Font** setting specifies a font type for the text from a list.

Coordinate Frame Command

In CFD-Post it may be necessary to define a new coordinate frame for certain quantitative operations, which are described in [Function Calculator \(p. 165\)](#).

Note

There are several ways to insert a coordinate frame:

- From the menu bar, select **Insert > Coordinate Frame**.
- From the toolbar, click the *Coordinate Frame*  icon.
- Depending on the context, you may be able to perform an insert from the shortcut menu in the tree view.

For information on how to define a coordinate frame, see [Coordinate Frame Details \(p. 131\)](#).

Coordinate Frame: Definition

Type

The **Type** setting is always set to **Cartesian**.

Origin

The **Origin** setting specifies 3D coordinates corresponding to the location of the new Coordinate Frame.

Z Axis Point

The **Z Axis Point** setting specifies a point on the Z axis from the origin.

X-Z Plane Pt

The **X-Z Plane Pt** setting specifies a point in the XZ plane used to define the positive X axis direction.

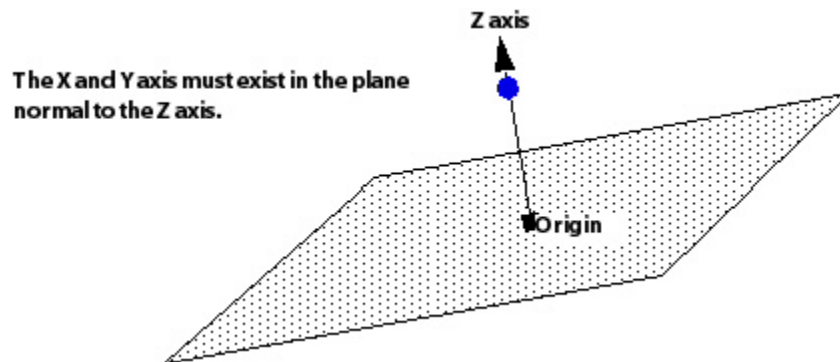
Symbol Size

The **Symbol Size** setting scales the size of the coordinate frame being edited.

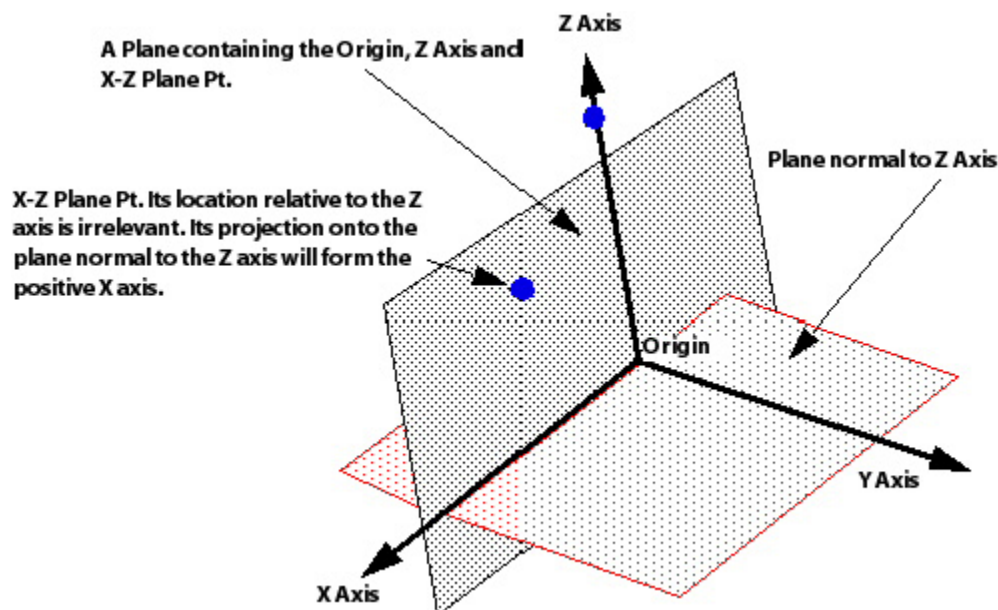
Coordinate Frame Details

A coordinate frame is created by specifying three points. It is important to understand how these three points are used to create a coordinate frame.

The first point is the origin for the new coordinate frame (labelled **Origin** in the **Definition** tab). The second point is used to create a Z axis in the new frame. A vector is calculated from the **Origin** to the point defined in the **Z Axis Point** box and used as the third axis of the new coordinate frame. The plane normal to the Z axis is now set and contains both the X and Y axes.



A third point entered into the **X-Z Plane Pt** box is needed to define the location of the X and Y axis in the plane normal to the Z axis. The **X-Z Plane Pt** point, along with the two points already specified, define a plane that lies in the X-Z plane (see diagram below). Because the X axis must now lie in both the X-Z plane and the plane normal to the Z axis, its location must be the line of intersection between the two planes. The positive direction for the X axis is the same side as the **X-Z Plane Pt** point lies with respect to the Z axis.



Finally, because the Y axis must be perpendicular to both the X Axis and the Z Axis, its positive direction is determined by the right-hand rule.

If **X-Z Plane Pt** is specified such that it lies on Axis 3, an error is displayed. The projection of the **X-Z Plane Pt** onto the plane normal to the Z axis would be on the origin and does not give enough information to define the X axis.

Legend Command


Default and user-defined legends can be plotted in the viewer to show the mapping between colors and quantities for plots that are colored by variable values.

The following characteristics of legends will be discussed:

- [Default Legends \(p. 132\)](#)
- [User-defined Legends \(p. 132\)](#)
- [Legend: Definition Tab \(p. 132\)](#)
- [Legend: Appearance Tab \(p. 134\)](#)

Note

There are several ways to insert a legend:

- From the menu bar, select **Insert > Legend**.
- From the toolbar, click the *Legend*  icon.
- Depending on the context, you may be able to perform an insert from the shortcut menu in the tree view.

Default Legends

Each view/figure has a `Default Legend` object that appears whenever an eligible plot is created or updated. As further objects are added to, or updated in, a viewport, the `Default Legend` updates to show the variable values for the latest plot.

Only the default legend for the selected view/figure is shown in the tree view. The other default legends continue to exist, even when not displayed in the tree view.

User-defined Legends

To create a user-defined legend, select **Insert > Legend**.

Legend: Definition Tab

Plot

The **Plot** setting is available only when creating or modifying a user-defined legend (not the default legends). Select from a list of objects for the legend to act on.

Title Mode

The **Title Mode** setting has the following options:

Option	Description
No Title	Omits the title.
Variable	Sets the title to the name of the variable mapped by the legend.
Variable and Location	<code>Variable and Location</code> is the same as <code>Variable</code> except that the name of the locator is appended to the title.
User Specified	Enables you to specify a custom title.

Title

The **Title** text box is available only after the **User Specified** option has been selected. This setting enters a custom title.

Show Legend Units Check Box

Clearing the **Show Legend Units** check box will hide the legend units. By default, the check box is selected, and so units are displayed.

Note

The legend will always display Temperature in absolute units: if C or K are selected as temperature units, the legend's data will be displayed in K; if F or R are selected, the legend's data will be displayed in R. For details, see [Function Calculator \(p. 165\)](#).

Vertical / Horizontal Options

Selecting **Vertical** or **Horizontal** will display the legend vertically or horizontally in the Viewer.

Location

X Justification

The **X Justification** setting has the following options:

Option	Description
None	Enables you to specify a custom X location using the Position text boxes.
Left	Places the legend on the left side of the viewer.
Center	Places the legend in the center of the viewer.
Right	Places the legend on the right side of the viewer.

Y Justification

The **Y Justification** setting has the following options:

Option	Description
None	Enables you to specify a custom Y location using the Position text boxes.
Top	Places the legend at the top of the viewer.
Center	Places the legend in the center of the viewer.
Bottom	Places the legend at the bottom of the viewer.

Position

The **Position** text boxes specify a custom point at which to position the legend. This setting is available after the **None** option is selected for the **X** and/or **Y Justification** settings. The values entered are fractions of the screen width/height for x and y respectively. For example, 0.2 for the X value would place the legend 1/5 across the screen from the left. A value of 0.2 for the Y direction would place the legend 1/5 up from the bottom of the Viewer. The placement uses the bottom left corner of the legend as a reference.

Legend: Appearance Tab

Sizing Parameters

Size

The **Size** setting scales the legend height to a fraction of the Viewer height.

Aspect

The **Aspect** setting specifies the width of the color range bar.

Text Parameters

Precision

The **Precision** setting specifies the number of significant digits after the decimal place that the legend can hold. You may also choose to display the numbers in a **Fixed** or **Scientific** format.

Value Ticks

The **Value Ticks** text box holds the number of intervals that you want shown by the legend.


Font

The **Font** setting specifies a font for the interval labels.

Color Mode

The **Color Mode** setting specifies whether to use a **User Specified** color or the **Default** color.

Color

The **Color** setting specifies a color for the title and interval labels. You can click the color bar to browse through predefined colors, or click the *Color Selector*  icon and select a color from the **Select color** dialog box.

Text Rotation

The **Text Rotation** setting specifies a value in degrees to rotate the text at (in a counterclockwise direction from horizontal).

Text Height

The **Text Height** setting specifies a value corresponding to the text height of the legend relative to the Viewer size. You may enter a value between 0.005 and 0.1.

Note

When using a legend as the basis for quantitative analysis, you should ensure that lighting is turned off for any objects colored by a variable. This will give you exact matches between object colors and legend colors.

Instance Transform Command

Instance Transforms are used to specify how an object should be drawn multiple times. CFD-Post can create Instance Transforms using rotation, translation, and reflection. For example, if you have a mesh that contains one blade from a blade row that contains 51 blades, you would set **# of Passages** to 51. You could then choose to display 1 through 51 blades by setting **# of Copies**.


To apply an Instance Transform to an object, select the **Apply Instancing Transform** check box on the **View** tab for the object and select the transform from a list.

The following characteristics of instance transforms will be discussed:

- [Default Transform Object \(p. 135\)](#)
- [Instance Transform: Definition Tab \(p. 135\)](#)
- [Instance Transform: Example \(p. 137\)](#)

Note

There are several ways to insert an instance transform:

- From the menu bar, select **Insert > Instance Transform**.
- From the toolbar, click the *Instance Transform*  icon.
- Depending on the context, you may be able to perform an insert from the shortcut menu in the tree view or in the 3D Viewer.

Default Transform Object

By default, an Instance Transform called `Default Transform` (which is set to apply no instancing by default) is applied to all objects where Instance Transforms are possible. As a result, editing the definition of `Default Transform` will cause all plots and objects to be transformed (unless you modify the View properties for a particular object). An example is available for applying Instance Transforms. For details, see [Instance Transform: Example \(p. 137\)](#). Note that instancing is purely geometric (in the **Viewer**). This means that quantitative calculations are carried out for the original geometry.

Instance Transform: Definition Tab

Instancing Info From Domain Check Box

Clear the **Instancing Info From Domain** check box to enable creating a custom Instance Transform object. Selecting **Instancing Info From Domain** will ignore the application of the Instance Transform inside the domain.

of Copies

The **# of Copies** setting specifies the number of copies to be made of the object when it is transformed.

If the Instance Transform object is using more than one of the following check boxes, (**Apply Rotation**, **Apply Translation**, and **Apply Reflection/Mirroring**) the order in which each segments are applied are rotation, translation, then reflection.

Apply Rotation Check Box

Select the **Apply Rotation** check box if you want to apply a rotation.

Method

The **Method** setting has the following options:

Option	Description
Principal Axis	Rotates about a principal axis.
Rotation Axis	Rotates about a user-specified axis.

Axis

Axis is available only if the `Principal Axis` option is selected. The **Axis** setting specifies a principal axis to rotate about.

From/To Text Boxes

These settings are available only if the `Rotation Axis` option is selected. These settings create an axis of rotation.

Full Circle Check Box

Select the **Full Circle** check box to uniformly distribute the copies around 360 degrees of rotation.

Angle From

The **Angle From** setting has the following options:

Option	Description
Instances in 360	Splits 360 degrees into the amount of passages entered and places a copy at each passage, if possible.
Value	Evenly distributes copies from zero to the specified angle.

of Passages

of Passages is available only if the **Instances in 360** option is selected. The **# of Passages** setting specifies a value for the number of passages in 360 degrees.

Passages/Com

Passages/Com (passages/component) is available only if the **Instances in 360** option is selected. The **Passages/Com** setting specifies a value for the number of passages per component.

Angle

Angle is available only if the **Value** option is selected. The **Angle** setting specifies the rotational angle.

Apply Translation Check Box

Select the **Apply Translation** check box if you want to specify a translation.

Translation

The **Translation** setting specifies a 3D translation.

Apply Reflection/Mirroring Check Box

Select the **Apply Reflection/Mirroring** check box to select a reflection method and direction.

Tip

A quick way to define a reflection for your case is to right-click on the **Wireframe** near the reflection plane and select **Reflect/Mirror**.

Method

The **Method** setting has the following options:

Option	Description
YZ Plane	Specifies a reflection about the YZ plane.
ZX Plane	Specifies a reflection about the ZX plane.
XY Plane	Specifies a reflection about the XY plane.
From Plane	Specifies a reflection about a user-specified plane.

X/Y/Z

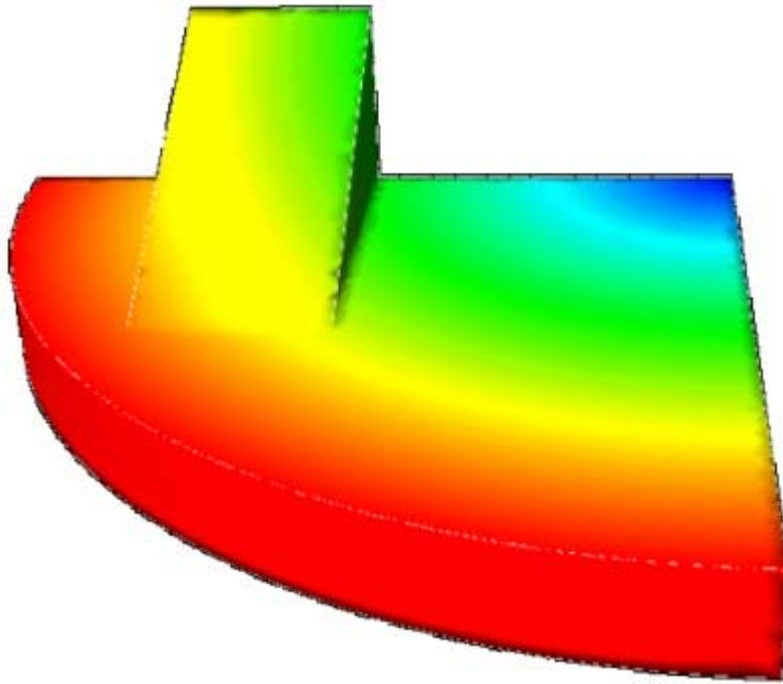
These settings are available only if one of the principal plane options are selected. These settings specify the distance along the normal axis to the plane to reflect by.

Plane

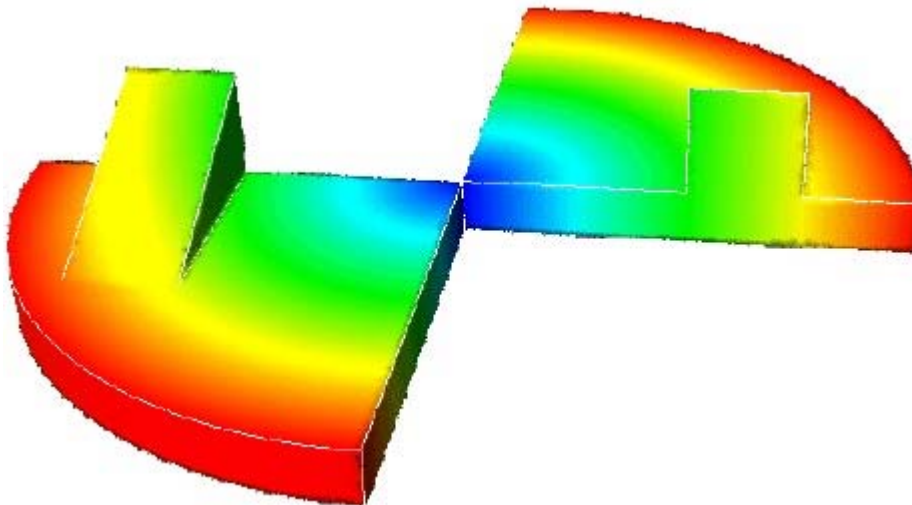
Plane is available only if the **From Plane** option is selected. The **Plane** setting specifies a plane from the list.

Instance Transform: Example

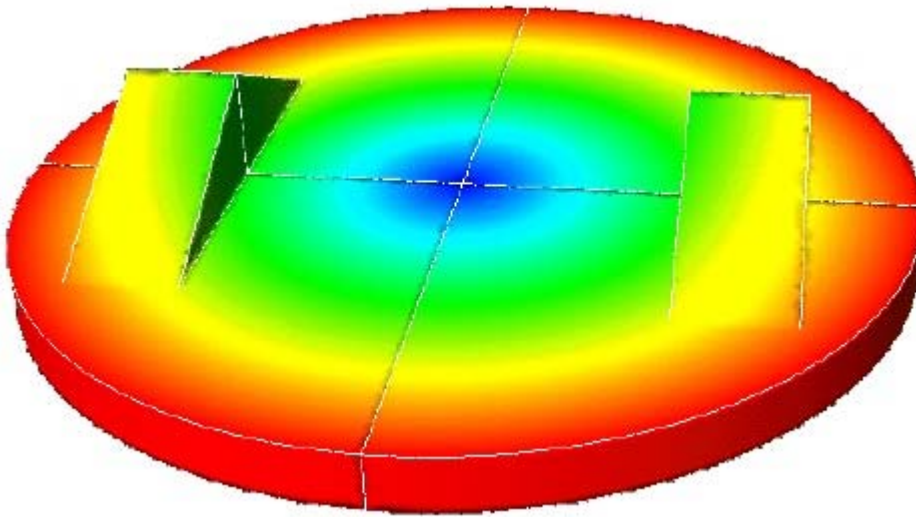
The following example shows how coupling of rotation and reflection instancing can be used to simulate reflection in two planes using a random geometry.



The axis of rotation is defined using the **Rotation Axis** feature on the **Rotation** section of the **Instance Transform Definition** form. An axis parallel to the z-axis was set. Rotation only was applied initially. An angle of 180 degrees was implemented.



The next step involves creating an XY plane (called **Plane 1**) at $X = -1$ and $Z = 1$. For details, see [Plane Command \(p. 95\)](#). After clicking to expand the Reflection/Mirroring submenu, reflection is applied on **Plane 1**.




Clip Plane Command

A clip plane enables you to define a plane that hides all objects displayed in the **Viewer** that lie to one side of the plane. For example, you could use an XY plane and clip it at $Z=1$ so that objects are visible only where Z is less than or equal to 1 (or greater than or equal to 1 if the **Flip Normal** check box is selected).

A clip plane will act on all objects in the Viewer, including the Wireframe, but will not affect other functions such as calculations (that is, a calculation will still use the entire location, whether visible or not).

Note

There are several ways to insert a clip plane:

- From the menu bar, select **Insert > Clip Plane**.
- From the toolbar, click the *Clip Plane*  icon.
- Depending on the context, you may be able to perform an insert from the shortcut menu in the tree view.

Important

When a clip plane is coincident with regions, boundaries, or interfaces that are planes, the results of a **Save Picture** command may not match what you see in the 3D Viewer (depending on the orientation of the case). In this situation, set the **Use Screen Capture** check box.

Clip Plane: Geometry

Definition

Method

Method has the same options and settings as for the **Plane** object, except for the *From Slice Plane* option. For details, see [Method \(p. 96\)](#). *From Slice Plane* enables you to select a predefined slice plane.

Slice Plane

Slice Plane is available only if the *From Slice Plane* option is selected. The **Slice Plane** setting selects a plane to clip by.

Flip Normal Check Box

Select the **Flip Normal** check box to cut all objects in the negative normal direction. If the check box is cleared, the Clip Plane cuts all objects in the positive normal direction.

Note

To enable/disable Clip Planes, you must use a Viewer shortcut menu command. For details, see [CFD-Post 3D Viewer Shortcut Menus \(p. 46\)](#).

Color Map Command

To access the Color Map editor, from the menu bar select **Insert > Color Map**.

The color map editor has the following controls:

Color Map Style

The **Color Map Style** controls whether the Color Map is a **Gradient**, which forms continuous bands of colors between any number of “color points” that you set, or **Zebra**, which forms bands between only two color points, using a number of divisions that you set with the **divisions** counter.

In gradient mode, all Color Map controls other than **divisions** are enabled; in zebra mode, **Insert**, **Delete**, and **Distribute** are all disabled and the **Position** indicator is read-only (and reflects the setting in the **divisions** indicator).

Preview

The **Preview** both shows the results of your edits and enables you to modify your color points. One color point will always be longer than the others; this indicates the color point that you can drag with the mouse or modify with the controls in the **Color Point Properties** area: the **Color** definition bar, the **Transparency** slider, and (in gradient mode) the **Position** indicator.


You can navigate from one color point to the next by:

- Clicking a color point
- Clicking **Next** or **Previous**.

In gradient mode, you can insert a new color point by:

- Clicking **Insert** to add a color point mid way between the current color point and its neighbor.
- Clicking on the **Preview** bar to insert the color point and, if necessary, adjusting its location by typing a value in the **Position** field.

Color

The **Color** control enables you to change the color of the active color point. When you click on the color field it cycles through ten preset colors. To define any color, click the *Color Selector*  icon to the right of the **Color** option and select one of the available colors.

Transparency

The **Transparency** slider enables you to control how opaque each color is.

Insert, Delete, Next, Previous, Distribute

The color point buttons control the number of color points, which color point is active, and the distribution of color points. In Zebra mode, only the buttons that control the active color point are enabled.

Make available in other cases, Set as default

These settings control where the color map is stored; unless you specify otherwise, the color map you define will be available only with the current file.

If you select **Make available in other cases**, the color map will be stored in your preferences file when you click **Apply**.

If you select **Set as default**, when you click **Apply** the color map will be stored in your preferences file and will be the default color map for all future objects in all future files. For this reason **Make available in other cases** will also be enabled automatically.


Note

The default CFD-Post color map is not the same as the default ANSYS FLUENT color map. To use the default ANSYS FLUENT color map for a particular locator (such as a contour):

1. Select **File > Load Results** and double-click the desired file.
2. Select the locator from the **Insert** menu.
 - a. On the **General** tab for the locator, set **Color Map** to **Fluent Rainbow**.
 - b. On the **Render** tab, clear the **Lighting** check box.
 - c. Make any other changes desired and click **OK**.

Variable Command


There are several ways to insert a variable:

- From the menu bar, select **Insert > Variable**.
- From the toolbar, click the *Variable*  icon.
- In the CFD-Post workspace, click the **Variables** tab.

Each of these methods inserts a new variable and opens the **Variables** workspace. For details, see [Variables Workspace \(p. 36\)](#).

Expression Command



There are several ways to insert an expression:

- From the menu bar, select **Insert > Expression**.
- From the toolbar, click the *Expression*  icon.
- In the CFD-Post workspace, click the **Expressions** tab.

Each of these methods inserts a new expression and opens the **Expression** workspace. For details, see [Expressions Workspace \(p. 39\)](#).

Table Command

The **Insert > Table** command opens a table for editing in the **Table Viewer** tab. In addition to that method, you can also create a table as follows:

- From the toolbar, click the *Create Table*  icon.
- In the **Table Viewer** tab, click the *New Table*  icon.

Each of these methods inserts a new table under the Report object. To see the table in the report, you must generate the report. For details, see [Report \(p. 24\)](#).

To learn how to work with tables, see [Editing in the Table Viewer \(p. 140\)](#).

Editing in the Table Viewer

Note

When multiple cases are loaded, the **Default** field enables you to specify which case the table values apply to. If the cases are in case-comparison mode, you have the option of creating a table that uses values from the differences in values between cases 1 and 2.

Changing the **Default** field removes all unsaved values and definitions from the table.

To enter data into a cell, select a cell and type in the information you want. To edit the current contents of a cell in the cell itself (rather than in the cell definition field), double-click on the cell.

The cell contents can be formatted with bold, italic, and underline fonts; left, center, and right justification; word wrapping; font sizes; and text and background colors. Multiple cells can be merged into a single larger cell to allow large items (for example, titles) to span multiple cells. For details, see [Table 9.2, “Table Viewer Tools Toolbar”](#) (p. 144).

To perform a formatting operation on multiple cells, click in the upper-left cell of the group and, while pressing **Shift**, click in the lower-right cell of the group. While the group is highlighted, tool bar operations are applied to all cells in the group.

Numeric data, (that is, numbers alone, numbers with units, and expression results), can be formatted to display in scientific or fixed notation with a specified number of significant digits.







Table contents can be cut (**Ctrl +C**) and pasted (**Ctrl+V**) into Microsoft Excel documents and vice versa.

Shortcut Menu

To access the shortcut menu for a table, type = into a cell and right-click the cell, or right-click the text box for the selected cell above the table. The shortcut menu has all of the commands listed in [Table 9.1, “Shortcut Menus Toolbar”](#) (p. 142), plus an **Edit** submenu that has the standard editing commands.

For faster expression entry, there is also a **Shortcut Menus** toolbar above the table with the following items. Type = into the cell and click on the given menu to display a variety of items that can be inserted automatically at the current cursor location. All, except **Annotation**, are also available in details view for expressions.

Table 9.1. Shortcut Menus Toolbar

Type of Item to Insert	Description
 Function ▼	<p>Select from the following submenus:</p> <ul style="list-style-type: none"> • CFD-Post Select from a list of predefined and user-defined functions from CFD-Post to insert into the cell. For details, see CFX Command Language (CCL) in CFD-Post (p. 213). • CEL Select from a list of predefined CEL functions. For details, see CEL Mathematical Functions (p. 141) in the ANSYS CFX Reference Guide.
 Expression ▼	Enables you to specify CFD-Post expressions or expressions that you have created with the Expressions workspace. For an example of using the Expressions workspace, see Expressions Workspace: Example (p. 41) .
 Variable ▼	Select from a list of existing variables to insert into the cell.
 Location ▼	Select from a list of existing locations to insert into the cell.
 Constant ▼	Select from a list of mathematical constants to insert into the cell.
 Annotation ▼	<p>Select from the following menu items/submenus:</p> <ul style="list-style-type: none"> • Time Step Inserts the value of the current timestep. • Time Value Inserts the value of the current time value. • File Name submenu <ul style="list-style-type: none"> • Name Inserts the name of the current results file, including the extension. • Path Inserts the file path of the current results file. • File Date submenu Select from a list of different date formats to insert into the cell. The inserted value represents the date that the file was created. • File Time submenu Select from a list of different time formats to insert into the cell. The inserted value represents the time of day that the file was created.

Expressions

Tables in CFD-Post have the ability to evaluate and display expression results and update those results when variables and/or locations they depend on change.

To enter an expression, edit a cell and prefix a valid CFD-Post expression with an equals sign (=). For example, you may enter the following into a cell:









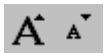


```
=2*areaAve(Pressure)@inlet
```


When the focus leaves the cell, the table displays the evaluated result of that equation in the cell. When selecting a cell containing an expression, the expression is displayed in the cell editor box immediately above the table. You can edit the expression in the cell editor box. Alternatively, you can double-click the cell and edit the equation from the cell itself. For details on how to enter common expressions and functions quickly, see [Shortcut Menu \(p. 141\)](#).

If there is an error in evaluating the expression contained in a table cell, the cell will be colored red.

The toolbar above the **Table Viewer** contains the following icons:

Table 9.2. Table Viewer Tools Toolbar

Icon	Description
	Creates a new table.
	<p>Opens the Load Table from file dialog box.</p> <p>Tables can be loaded from files in two different formats:</p> <ul style="list-style-type: none"> CFD-Post State Files (*.cst) - Loads the table CCL from the given state file. If the file contains tables with names that already exist, numbers will be added to the end of the names of the imported tables to differentiate them from existing tables. Comma Separated Values Files (*.csv) - Loads the values in the CSV file into a new table. You can specify the table name in the Load Table dialog box.
	<p>Opens the Save Table to file dialog box.</p> <p>Tables can be saved to several formats:</p> <ul style="list-style-type: none"> CFD-Post State (*.cst) - Saves the current table to a state file. Tables saved in state files will maintain expressions and formatting and, when reloaded, will exactly reproduce the original table. HTML (*.htm, *.html) - Saves the current table to an HTML file. Note that the saved HTML table will contain expression results, and not the expressions. All formatting will be converted to the HTML equivalent. Word-wrapping is always on. The Save Table dialog box contains additional formatting options including table title, caption, borders, margins, spacing, and gridline visibility. Comma Separated Values (*.csv) - Saves the current table to a CSV file. Note that the saved table will contain expression results, not the expressions. No formatting information is saved to the file. The Save Table dialog box provides the option to clear the output of trailing separators for table rows that have fewer columns than other rows. If this option is on, extra commas will appear on some lines so that all rows in the CSV file will contain the same number of columns. This format can be directly imported to Microsoft Excel. Text (*.txt) - Behaves identically to the CSV option, except that you can specify the separator.
	<p>Edit operations for contents of cells: <i>Cut</i>, <i>Copy</i>, and <i>Paste</i>.</p> <p>To select a rectangle of cells for an operation, click in the cell in the upper-left corner, then Shift-click the cell in the lower-right corner. The cells become highlighted and can be operated upon as a unit.</p>
	Font operations for text in cells: <i>Bold</i> , <i>Italic</i> , and <i>Underline</i> .
	Text-alignment operations: <i>Left</i> , <i>Center</i> , and <i>Right</i> .
	Makes the selected cells wrap text.
	Launches the Cell Formatting dialog where you can specify scientific or fixed notation, the precision, and whether to show the value or the units (at least one of the value or units must appear).
	Changes the size of the font used in the cell.
	Opens the Select color dialog box for setting the background color.
	Opens the Select color dialog box for setting the text color.

Icon	Description
	Causes a cell to span rows or columns (<i>Merge Cells</i>) or reverses that operation (<i>Unmerge Cells</i>).

Here is an example of formatting applied to a table:

Figure 9.1. Sample Table Formatting

	A	B	C	D
1	Proximity to Outlet	Min Temperature	Max Temperature	Difference
2	-6.04 [in]	293.15 [K]	313.15 [K]	20.00 [K]
3	-0.55 [in]	293.15 [K]	305.40 [K]	12.25 [K]
4	2.18 [in]	293.15 [K]	304.35 [K]	11.20 [K]
5	5.04 [in]	293.15 [K]	303.02 [K]	9.87 [K]
6	8.00 [in]	293.15 [K]	301.81 [K]	8.66 [K]

To format the table shown above:

- Cells A1-D1: Applied bold font, background color, and text centering. Manually resized cell widths individually.



- Cell A1: Applied text wrapping and resized cell height manually.



- Cells A2-D6: Right-justified text.



- Cells A2-A3: Manually changed the font color.



Note

To perform a formatting operation on multiple cells, click in the upper-left cell of the group and, while pressing **Shift**, click in the lower-right cell of the group. While the group is highlighted, tool bar operations are applied to all cells in the group.

Chart Command

Charts are graphs that use lines and/or symbols to display data. You can create charts that can be used on their own or in reports.

The following characteristics of charts will be discussed:


- [Creating a Chart Object \(p. 145\)](#)
- [Viewing a Chart \(p. 153\)](#)
- [Example: Charting a Velocity Profile \(p. 154\)](#)

Note

When using the **Turbo** workspace to post-process a turbo-machinery case, several "Turbo Charts" are created by default. For details, refer to [Turbo Charts \(p. 197\)](#).

Creating a Chart Object


To create and view a chart object:

- Click *Create Chart*  or select **Insert > Chart**.

The **Insert Chart** dialog box appears.

2. Enter a name for the new chart object.
3. Click **OK**.

The chart object appears under the **Report** heading in the tree view. A Details view appears for the new chart object and the **Chart Viewer** takes focus.

4. Edit the chart settings as appropriate for each tab:
 - [Chart Details: General Tab \(p. 146\)](#)
 - [Chart Details: Data Series Tab \(p. 149\)](#)
 - [Chart Details: X Axis Tab \(p. 150\)](#)
 - [Chart Details: Y Axis Tab \(p. 151\)](#)
 - [Chart Details: Line Display Tab \(p. 152\)](#)
 - [Chart Details: Chart Display Tab \(p. 153\)](#)
5. Click **Apply** to see the results of your changes displayed in the **Chart Viewer**.
6. Optionally, on the **Data Series** tab click the *Get Information on the Item* icon  to view summary data for the current series.
7. Optionally, click **Export** to save the chart data in a comma-separated values (CSV) file. You can load the values in the CSV file into external programs such as Microsoft Excel.

To see the chart in the report, you must generate the report as described in [Report \(p. 24\)](#).

Chart Details: General Tab

The **General** tab is used to define the chart type, the main title, and the report caption.

Type

The **Type** setting has the following options:

Option	Description
XY	Plots X axis variable vs. the Y axis variable. XY charts use polyline or line locators to plot values that vary in space.
Transient or Sequence	Plots an Expression (typically time) on the X axis and enables you to specify a variable to plot on the Y axis. Transient or Sequence charts use expressions or a point locator to plot the variation of a scalar value vs. time or multi-configuration runs.
Histogram	Plots the number of values or the proportion of values that fall into each specified category.

Title

The **Title** setting specifies a title for the Chart object.

Report: Caption

The **Caption** is the description of the Chart object that appears in the report.

Fast Fourier Transform

The **Fast Fourier Transform** check box can be selected only for Transient or Sequence charts. When the **Fast Fourier Transform** check box is enabled, the following options are available:

Modify Input Signal Filter

Enables you to select the signal filter to be Hanning (default), Barlett, Blackman, Hamming, or None. For details, see [Fast Fourier Transform \(FFT\) Theory \(p. 147\)](#).


Subtract mean

Causes the mean to be subtracted from each value to better show the amplitude of the noise.

Note

This feature applies to *all* loaded files.

Full range of input data vs. Setting Min/Max Limits

You can choose to analyze the **Full range of input data** or to set **Min** and **Max** values. To get the range for the **Min** and **Max** values, click *Get range from FFT output* .

Reference Values

Click **Reference Values** to display the **Reference Values** dialog. There you can set the following values (which will apply to all Fast Fourier Transform charts): **Reference Acoustic Pressure**, **Length**, **Velocity**, and whether to **Save as default**.

Fast Fourier Transform (FFT) Theory

When interpreting time-sequence data from a transient solution, it is often useful to look at the data's frequency attributes. For instance, you may want to determine the major vortex-shedding frequency from the time-history of the drag force on a body recorded during a simulation, or you may want to compute the frequency distribution of static pressure data recorded at a particular location on a body surface. Similarly, you may need to compute the frequency distribution of turbulent kinetic energy using data for fluctuating velocity components.

To interpret some of these time-dependent data, you need to perform Fourier transform analysis. The Fourier transform enables you to take any time-dependent data and resolve it into an equivalent summation of sine and cosine waves.

The CFD-Post FFT module assumes that the input data have been sampled at equal intervals and are consecutive (in the order of increasing time).

The lowest frequency that the FFT module can pick up is given by $1/t$, where t is the total sampling time. If the sampled sequence contains frequencies lower than this, these frequencies will be aliased into higher frequencies.

The highest frequency that the FFT module can pick up is $1/2dt$, where dt is the sampling interval (or time step).

Windowing in Fast Fourier Transforms

The discrete FFT algorithm is based on the assumption that the time-sequence data passed to the FFT corresponds to a single period of a periodically repeating signal. Because in most situations the first and the last data points will not coincide, the repeating signal implied in the assumption can have a large discontinuity. The large discontinuity produces high-frequency components in the resulting Fourier modes, causing an aliasing error. You can condition the input signal before the transform by "windowing" it, in order to avoid this problem.

Suppose there are N consecutive discrete (time-sequence) data that are sampled with a constant interval Δt :

$$\phi_k \equiv \phi(t_k), \quad t_k \equiv k\Delta t, \quad k=0, 1, 2, \dots, (N-1) \quad (\text{Eq. 9.41})$$

Windowing is done by multiplying the original input data (ϕ_j) by a window function W_j :

$$\tilde{\phi}_j = \phi_j W_j \quad j=0, 1, 2, \dots, (N-1) \quad (\text{Eq. 9.42})$$

There are four different window functions:

Hamming's window:

$$W_j = \begin{cases} 0.54 - 0.46 \cos\left(\frac{8\pi j}{N}\right) & j \leq \frac{N}{8}, j \geq \frac{7N}{8} \\ 1 & \frac{N}{8} < j < \frac{7N}{8} \end{cases} \quad (\text{Eq. 9.43})$$

Hanning's window:

$$W_j = \begin{cases} 0.5 \left[1 - \cos\left(\frac{8\pi j}{N}\right) \right] & j \leq \frac{N}{8}, j \geq \frac{7N}{8} \\ 1 & \frac{N}{8} < j < \frac{7N}{8} \end{cases} \quad (\text{Eq. 9.44})$$

Barlett's window:

$$W_j = \begin{cases} \frac{8j}{N} & j \leq \frac{N}{8} \\ 8 \left(1 - \frac{j}{N} \right) & j \geq \frac{7N}{8} \\ 1 & \frac{N}{8} < j < \frac{7N}{8} \end{cases} \quad (\text{Eq. 9.45})$$

Blackman's window:

$$W_j = \begin{cases} 0.42 - 0.5 \cos\left(\frac{8\pi j}{N}\right) + 0.08 \cos\left(\frac{16\pi j}{N}\right) & j \leq \frac{N}{8}, j \geq \frac{7N}{8} \\ 1 & \frac{N}{8} < j < \frac{7N}{8} \end{cases} \quad (\text{Eq. 9.46})$$

These window functions preserve 3/4 of the original data, affecting only 1/4 of the data the ends.

Using Fast Fourier Transforms

The Fourier transform utility enables you to compute the Fourier transform of a signal, $\phi(t)$, a real-valued function, from a finite number of its sampled points.

The discrete Fourier transform of ϕ_k is defined by:

$$\phi_k = \sum_{n=0}^{N-1} \hat{\phi}_n e^{2\pi i k n / N} \quad k=0, 1, 2, \dots, (N-1) \quad (\text{Eq. 9.47})$$

where $\hat{\phi}_n$

are the discrete Fourier coefficients, which can be obtained from:

$$\hat{\phi}_n = \frac{1}{N} \sum_{k=0}^{N-1} \phi_k e^{-2\pi i k n / N} \quad n=0, 1, 2, \dots, (N-1) \quad (\text{Eq. 9.48})$$

The previous two equations form a Fourier transform pair that enables you to determine one from the other.

Note that when you vary n from 0 to $N-1$ in [Equation 9.47 \(p. 148\)](#) or [Equation 9.48 \(p. 148\)](#) instead of from $-N/2$ to $N/2$, the range of index $1 \leq n \leq N/2-1$ corresponds to positive frequencies, and the range of index $N/2+1 \leq n \leq N-1$ corresponds to negative frequencies. $n=0$ still corresponds to zero frequency.

For the actual calculation of the transforms, the CFD-Post adopts the Fast Fourier transform (FFT) algorithm, which significantly reduces operation counts in comparison to the direct transform. Furthermore, unlike most FFT algorithms in which the number of data should be a power of 2, the FFT utility in CFD-Post employs a prime-factor algorithm. The number of data points permissible in the prime-factor FFT algorithm is any products of mutually prime factors from the set 2,3,4,5,7,8,9,11,13,16, with a maximum value of 720720=5x7x9x11x13x16. Thus, the prime-factor FFT preserves the original data better than the conventional FFT.

Just prior to computing the transform, CFD-Post determines the largest permissible number of data points based on the prime factors, discarding the rest of the data.

Refresh Settings

Performing a refresh means re-reading files; therefore refreshing data from a series of files (such as transient files for a transient case) is potentially a time-consuming operation. If necessary, CFD-Post automatically performs a refresh of all charts when printing a chart and when generating or refreshing a report. At other times, you have

control over when to perform a "refresh" of the data. When the **General** tab is set to create an XY transient or sequence chart, settings appear that control how charts are refreshed:

Refresh chart on Apply

Causes only the currently displayed chart to be refreshed when you click **Apply**.


Refresh all charts on Apply

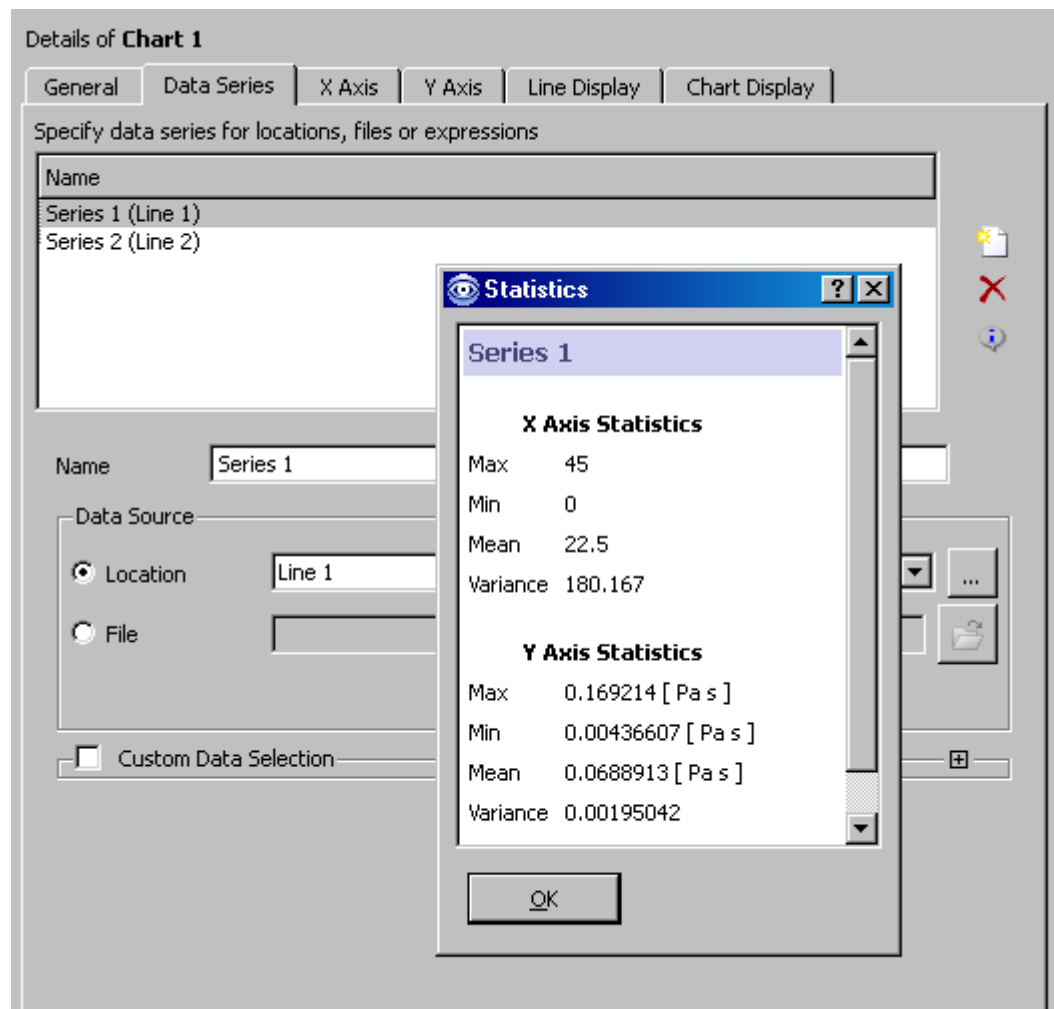
Causes all loaded charts to be refreshed when you click **Apply**. This is provided as a convenience; it enables you to work on a series of charts, refreshing all of them when you are done your editing (rather than clicking **Apply** on each chart individually).

Chart Details: Data Series Tab

The **Data Series** tab is used to specify the data series to be plotted and to define the lines and fills used on the chart.

Name Controls

The **Name** list is where you add and delete data series on the chart. You can also click *Statistics*  to display a dialog that shows statistics about the data set. You perform all of these tasks by clicking the icons beside the list area (New, Delete, Statistics). All the icons become enabled after you create an initial series; in addition, all functions are available when you right-click on a data series name.



Data Source

The fields in the **Data Source** area become enabled after you create an initial series.

- For a **General > XY** chart or a **General > Histogram** chart, choose either **Location** or **File** to define the source of the data for the series you are creating. A typical location would be a line or a streamline; a typical data file would be a .csv file.
- For a **General > XY - Transient or Sequence** chart, a typical **Location** would be a point. However, such a chart will also accept a **File** or an **Expression** as the data source. For example, you could use an expression to plot `areaAve(Temperature)@Outlet` as a function of time.

Custom Data Selection Controls

When enabled (which occurs when you have multiple lines to compare and when **General > Fast Fourier Transform** is disabled), the **Custom Data Selection** controls enable you to override the settings on the X Axis and Y Axis tabs for each series individually. See [Chart Details: X Axis Tab \(p. 150\)](#) and [Chart Details: Y Axis Tab \(p. 151\)](#) for more details on what each setting means.

You can also specify the use of **Hybrid** or **Conservative** values or to use the absolute value of data points. For help on the use of Hybrid or Conservative values, see [Hybrid and Conservative Variable Values \(p. 165\)](#).

Other available settings depend on the chart type; see [X Axis Data Selection \(p. 150\)](#) for details.

Chart Details: X Axis Tab

The **X Axis** tab is used to set properties for all data series that do not have custom data selection (which is set on the **Series** tab). The options available on the X Axis tab vary according to the **General** tab's **Type** setting.

X Axis Data Selection

The **Data Selection** settings control which variable is used as the data source and how the data is processed:

When **General > Type** is **XY** or **Histogram**, the **Variable** field can be set to any variable. **Hybrid** vs. **Conservative** sets how conservation equations for the boundary control volumes are solved. See [Hybrid and Conservative Variable Values \(p. 165\)](#) for details. **Take absolute value of data points** controls whether the values of data points are always positive.

When **General > Type** is **XY - Transient or Sequence**, the **Expression** field can be set to expressions provided in CFD-Post or to expressions that you have defined. For example, you could use this option to plot `areaAve(Temperature)@Outlet` as a function of time.

When **General > Type** is **XY - Transient or Sequence** and **Fast Fourier Transform** is enabled, you can define an **X Function**.

Specifying an X Function

The options for the X Function are related to the discrete frequencies at which the Fourier coefficients are computed. You can apply the following specific analytic functions:

Frequency is defined as:

$$f_n = \frac{1}{N \Delta t} n \quad n=0, 1, 2, \dots, N/2 \quad (\text{Eq. 9.49})$$

where N is the number of data points used in the FFT.

Strouhal Number is the nondimensionalized version of the frequency defined in the equation for **Frequency**:

$$St_n \equiv \frac{f_n L_{\text{ref}}}{U_{\text{ref}}} \quad (\text{Eq. 9.50})$$

where L_{ref} and U_{ref} are the reference length and velocity scales. Note that you can set L_{ref} and U_{ref} by clicking **Reference Values** on the **General** tab.

Fourier Mode is the index in

$$\phi_k = \sum_{n=0}^{N-1} \hat{\phi}_k e^{2\pi i k n / N} \quad k=0, 1, 2, \dots, N-1 \quad (\text{Eq. 9.50})$$

and/or

$$\hat{\phi}_n = \frac{1}{N} \sum_{k=0}^{N-1} \hat{\phi}_k e^{-2\pi i k n / N} \quad n=0, 1, 2, \dots, N-1 \quad (\text{Eq. 9.50})$$

which represents the n th or k th term in the Fourier transform of the signal.

Octave Band Full is a range of discrete frequency bands for different octaves within the threshold of hearing. The range of each octave band is double to that of the previous band (see [Table 9.3, “Octave Band Frequencies and Weightings”](#) (p. 151)).

One Third Full is a range of discrete frequency bands within the threshold of hearing. Here the range of each band is one-third of an octave, meaning that there are three times as many bands for the same frequency range.

Table 9.3. Octave Band Frequencies and Weightings


Lower Freq. (Hz)	Center Freq. (Hz)	Upper Freq. (Hz)	dB A	dB B	dB C
11	16	22	-56.7	-28.5	-8.5
22	31.5	45	-39.4	-17.1	-3.0
45	63	90	-26.2	-9.3	-0.8
90	125	180	-16.1	-4.2	-0.2
180	250	355	-8.6	-1.3	0.0
355	500	710	-3.2	-0.3	0.0
710	1000	1400	0.0	0.0	0.0
1400	2000	2800	1.2	-0.1	-0.2
2800	4000	5600	1.0	-0.7	-0.8
5600	8000	11200	-1.1	-2.9	-3.0
11200	16000	22400	-6.6	-8.4	-8.5

Category Divisions


These controls are enabled when the chart type is **Histogram**.

If **Category Divisions** are set to **Automatic**, you are able to specify the **Number of Divisions**. If **Category Divisions** are set to **User Defined**, you are able to specify the **Division Values**.

The **Division Values** field allows you to type points where you want to create histogram boundaries. You can either enter user-defined category divisions by typing a comma-separated ordered list directly into the **Division Values**

text box, or click *More*  to open up an editor for the division values (which includes the ability to set the values in a particular unit). If you use the editor, then the values do not need to be entered in order as you will be offered the chance to sort the values when you close the editor.

Axis Range

You can choose to **Determine ranges automatically** or to set **Min** and **Max** values. To get the range for the **Min** and **Max** values, click *Get range from existing chart* .

The default X-axis scale is linear but can be set to be a **Logarithmic scale**. Select **Invert Axis** to reverse the direction of the scale.

Axis Labels

You can choose to **Use data for axis labels**, or to use a **Custom Label**.

Chart Details: Y Axis Tab

The **Y Axis** tab is used to define the characteristics of the Y axis of the chart you are going to produce. For descriptions of many of the fields on this tab, see [Chart Details: X Axis Tab](#) (p. 150). Fields unique to this tab are described below.

Y Axis: Data Selection

When the **General** tab is set to create an XY or an XY transient or sequence chart, the **Data Selection** includes a **Variable** that you can set; you can also control whether the boundary data is **Hybrid** or **Conservative**, and whether or not to take the absolute value of data points.

When the **General** tab is set to create a histogram, the Y axis **Value** can be **Count** (which displays the number of times a set of values occurs for a given range) or **Percentage** (where the sum of the histogram values = 100). The **Weighting** can be **None**, **Geometrical**, or **Mass Flow**. The weighting setting changes the shape of the histogram by removing mesh dependencies (for example, if mesh density varies along a line, counts are biased towards areas of higher density; the **Geometrical** setting removes that bias).

When the **General** tab is set to **Fast Fourier Transform**, a **Y Function** field appears where you can choose one of the following settings:

A Weighted | B Weighted | C Weighted

These options are available only when the **X-Axis** tab's **X Function** is set to **Octave Band Full** or **One Third Full**.

Weighted scales characterize the loudness of a noise source and are designed to quantify sounds by one single value in dB(Y) where Y is the weighted scale used. Four weighted scales are used in industry: A, B, C and D; the A weighted scale is used almost exclusively in measurements.

Power Spectral Density

The distribution of signal power in the frequency domain. It has units of the signal magnitude squared (Pa²) and is defined as:

$$E(f_0) = |\hat{\phi}_0|^2 \quad (\text{Eq. 9.51})$$

$$E(f_n) = 2|\hat{\phi}_n|^2 \quad n = 1, 2, \dots, N/2 \quad (\text{Eq. 9.52})$$

Sound Amplitude

Sound Amplitude (dB) is exactly one-half of the sound pressure level in the equation for Sound Pressure Level. This quantity is also applicable for acoustics analysis.

$$A_{sp}(f_n) = 10 \log \sqrt{\frac{p'^2(f_n)}{p_{\text{ref}}^2}} \quad (\text{Eq. 9.53})$$

Sound Pressure Level

Sound Pressure Level (dB) is the decibel level. For either general or acoustic data, when the sampled data is pressure (for example, static pressure or sound pressure), you can compute the power in decibel units using

$$L_{sp}(f_n) = 10 \log \left(\frac{p'^2(f_n)}{p_{\text{ref}}^2} \right) = 20 \log \left(\frac{p'(f_n)}{p_{\text{ref}}} \right) \quad (\text{Eq. 9.54})$$

where $p'(f_n)$ is the acoustic pressure and p_{ref} is the reference acoustic pressure. Note that you can set p_{ref} by clicking **Reference Values** on the **General** tab.

Magnitude


The square root of the power spectral density.

$$A(f_n) = \sqrt{E(f_n)} \quad n = 0, 1, 2, \dots, N/2 \quad (\text{Eq. 9.55})$$

Chart Details: Line Display Tab

On the **Line Display** tab you can set the **Line Style** to a variety of settings, including **Automatic**, **Solid**, **Dash**, **Dot**, and so on.

You can select the **Use series name for legend name** check box to derive the name of the line (as it appears in the legend) from the series' name (as defined on the **Data Series** tab and, if more than one case is loaded, from the case name). Alternatively, you can clear that check box and type in a new **Legend Name**.

You can have CFD-Post **Automatically generate Line Color** or you can click on the bar beside **Line Color** to cycle through 10 basic colors. (Click the right-mouse button to cycle backwards.) Alternatively, you can choose any color by clicking *Color Selector*  to the right of the **Line Color** option.

Use the **Symbols** drop-down menu to place a graphic at every data point of the series. Use the **Symbol Color** control to set a color for the graphic the same way you did for the **Line Color**.

Note

Line width and symbol size can be set on the **Chart Display** tab for the chart as a whole, but cannot be set for each line individually.

Fill Area Controls

When **Fill Area** is enabled, you can choose to have a fill color generated automatically or at all times (**Always On**). The **Automatic** setting generates a fill when the chart's **General** tab's **Type** is set to **Histogram**.

Note

Plotting fill areas for graphs that have multiple y values for a given x (such as streamlines) does not produce useful results.

Chart Details: Chart Display Tab

The following sections describe the **Appearance** tab.

Display Legend Area

When **Display Legend** is enabled, the chart's legend appears, either **Outside Chart** or **Inside Chart**, and positioned at one of **Bottom**, **Right**, **Top**, or **Left**.

Sizes Area

Here you set the width of the line and the size of the symbol (if any) that you defined on the **Line Display** tab. These sizes apply to all lines and all symbols (you cannot set sizes for individual lines or symbols).

Fonts Area

Here is where you control the font type and size of the **Title** (which you defined on the **General** tab), the **Axes Titles**, the **Axes** values, and the **Legend**.

Note

By default, the titles of the axes are derived from the variables used in the line definition (not necessarily from the **X Axis** and **Y Axis** tabs because a transient chart that uses an expression and any chart that uses custom data selection will set the variables used directly). You can override these default titles by going to the **X Axis** and **Y Axis** tabs, clearing the **Use data for axis labels** check box, and typing in a **Custom Label** name.

The legend text is defined by default as a combination of the series definitions on the **Series** tab and, when more than one case is loaded, the case names, but can be specified on a line-by-line basis directly on the **Line Display** tab by clearing the **Use series name for legend name** check box and typing in a **Legend Name**.

Grid Area

Here is where you configure the background grid (if any) and the thickness of its major and minor lines.

Viewing a Chart

After a chart object has been created, you can view it in the **Chart Viewer** tab after selecting it in the tree view, or by including the chart in a report and viewing the report. For details, see [Viewing the Report \(p. 35\)](#).


Time charts, which depict transient runs, have a **Refresh** button at the top of the page. When CFD-Post determines that the chart requires updates, a note appears beside the **Refresh** button. Refreshes are generally not automatic in order to ensure that you can make a series of changes without having to wait through the update required by each change. However, time charts are updated automatically when you print the chart, when a report is previewed, or when a report is generated (HTML/text).

Note

As time charts are compute-intensive, they are generated only after user action. And because time chart data is not included in a state file, loading a state file will show an empty chart until you click **Apply** in the chart **Details** view or **Refresh** in the **Chart Viewer**.

Example: Charting a Velocity Profile

This example demonstrates how a polyline locator can be used to create a chart of a velocity profile.

1. Load the following results file, which is provided with your installation:
<CFXROOT>/examples/StaticMixer_001.res.
2. Insert a plane (**Insert > Location > Plane**) and define its location using the point and normal method.
Define the point to be (0,0,0) and the normal to be (0,1,0) so that the plane is normal to the Y axis; click **Apply** when you are done. For details, see [Plane: Geometry \(p. 96\)](#).
3. Insert a polyline (**Insert > Location > Polyline**) and define its location using the Boundary Intersection method.
Set **Boundary List** to out and **Intersect With** to the plane you just created; click **Apply** when you are done. For details, see [Polyline: Geometry \(p. 111\)](#).
4. Create a new chart by clicking *Create chart* .
5. In the **Insert Chart** dialog box, enter a name for the chart, and then click **OK**.
The details view for the chart appears.
6. On the **General** tab, set **Title** to Velocity Profile at Outlet.
7. Click the **Data Series** tab.
8. Set data source **Location** to the name of the polyline you just created.
9. Click the **X Axis** tab.
10. Set the x-axis Variable to X.
The x-coordinate direction is parallel to the polyline in this example so the plot shows a variable profile across the outlet.
11. Click the **Y Axis** tab.
12. Set the y-axis Variable to Velocity.
13. Click **Apply**.
A chart showing **Velocity** versus **X** is displayed on the **Chart Viewer** tab.

Example: Comparing Differences Between Two Files

You can use Case Comparison mode with the **Chart Viewer** to automatically see the differences in values between the two files:

1. Load the following results files, which are provided with your installation:
<CFXROOT>/examples/elbow1.cdat and <CFXROOT>/examples/elbow3.cdat. (Press the **Ctrl** key while selecting the two files, then click **Open**.)
Two viewports open, one with **elbow1** and the other with **elbow2** loaded.
2. Insert a line (**Insert > Location > Line**). Accept the default values for **Geometry > Method**, but set **Line Type** to **Cut**. On the **Color** tab, set **Mode** to **Variable** and **Variable** to **Temperature**. Set the **Range** to **Local**.

Click **Apply** when you are done.

3. In the **Outline** view, double-click **Case Comparison**. The **Case Comparison** details view appears. Select **Case Comparison Active** and click **Apply**.

A third viewport opens that displays the temperature difference between the two cases.

4. Click **Create chart** .

5. In the **Insert Chart** dialog box, enter a name for the chart, and then click **OK**.

The details view for the chart appears.

6. On the **General** tab, set **Title** to Comparison of Temperatures in the Elbow.

7. Click the **Data Series** tab.

8. Set data source **Location** to the name of the line you just created.

9. Click the **X Axis** tab.

10. Set the x-axis **Variable** to X.

The x-coordinate direction is parallel to the polyline in this example so the plot shows a variable profile across the outlet.

11. Click the **Y Axis** tab.

12. Set the y-axis **Variable** to Temperature.

13. Click **Apply**.

A chart showing **Temperature** versus **X** is displayed on the **Chart Viewer** tab. Three lines are there: one for each of the sets of temperature values, and a third line that shows the difference between those values.

Note

- You can change some of the properties of each line individually (including turning them on and off) by using the **Line Display** tab.
- The **Difference** line plots only the variable difference on the y-axis. For example, if you defined a chart of Velocity (y-axis) against Pressure (x-axis), then the difference line will plot Velocity Difference against Pressure, not Velocity Difference against Pressure Difference.

Comment Command

You can create comment objects to include in the report. Comments are used to add text to a report in the form of titles and paragraphs.

To define a comment object:

1. From the toolbar, click the **Comment**  icon or select **Insert > Comment**.

The **Insert Comment** dialog box appears.

2. Enter a name for the comment object.

3. Click **OK**.

The **Comment Viewer** tab is displayed.

The comment object appears in the tree view, under the **Report** object.

4. Enter a heading and/or a paragraph of text.

A heading is entered into the **Heading** box. The **Level** setting controls the level of the heading text in the report.


Paragraph text is entered into the large text box below the **Heading** box. Some *Rich Text* features are supported using toolbar icons that appear at the top of the **Comment Viewer** tab. Pictures can be inserted in the paragraph

text area. External hyperlinks can be included in the paragraph text, but will not work in the **Report Viewer** tab of CFD-Post. External hyperlinks will work when the report is viewed in a web browser.

To see the comment in the report, you must generate the report. For details, see [Report \(p. 24\)](#).

Figure Command

You can create a figure (an image of the objects in the 3D Viewer) to include in the report. There are two ways to create a figure:

- From the menu bar, select **Insert > Figure**.
- From the toolbar, click the *Figure*  icon.

To see the new figure, you must open the **Report Viewer** and refresh or publish the report. For details, see [Report \(p. 24\)](#).

Chapter 10. CFD-Post Tools Menu

The **Tools** menu offers access to quantitative analysis utilities, the animation editor, and the timestep selector. The **Command Editor** dialog box is also available so that you can enter CFX Command Language (CCL) directly.

This chapter describes:

- [Timestep Selector \(p. 157\)](#)
- [Animation \(p. 159\)](#)
- [Quick Editor \(p. 164\)](#)
- [Probe \(p. 164\)](#)
- [Function Calculator \(p. 165\)](#)
- [Macro Calculator \(p. 167\)](#)
- [Mesh Calculator \(p. 179\)](#)
- [Case Comparison \(p. 181\)](#)
- [Command Editor \(p. 182\)](#)

Timestep Selector

For a transient results file, the **Timestep Selector** dialog box enables you to load the results for different timesteps by selecting the timestep and clicking **Apply**.

When reading transient cases, CFD-Post re-reads and re-imports the mesh, if the transient file contains them. This feature allows CFD-Post to support transient rotor/stator problems, as well as moving-mesh cases.

Note

All variables will always appear in the variables list for all transient files, even if the transient file does not contain some of the variables. If you have the **Load missing variables from nearest FULL timestep** option selected (**Edit > Options > Files > Transient Cases**), then the missing variable data will be loaded from the nearest full timestep. Otherwise, the data will be colored with the undefined color in these cases.

The following list describes the column headings in the list box.

- The **Configuration** column indicates the configuration name as set in CFX-Pre. This column appears when you have a multi-configuration (.mres) file loaded.
- The **#** column displays the index number for the timestep. These values always begin at 1 and increase by 1.
- The **Step** column displays the timestep number, which is used for synchronization by time step. These values always increase; because they are unique, they can be used in scripts.




For most cases, the values in the **Step** column are the same as those in the **Solver Step** column. However, if you have a multi-configuration case or a case with run history, loaded using the **Load complete history as: A single case** option (described in [Load Results Command \(p. 57\)](#)), then the **Step** is calculated to give a unique, increasing value through all the configurations. It differs from index because it can maintain a consistent value even though (for example) some transient files (.trn) that were present when the run completed are no longer available. (For example, suppose that a case has transient files at three timesteps and these appear in CFD-Post as steps 1, 2, and 3. If you delete the middle transient file, CFD-Post will show entries in the timestep selector at steps 1 and 3, but not 2. If a script was loading step 3, it will load the same results as previously.) Note, however, that if an entire results file (.res) that is referenced by the multi-configuration results file (or the run history) is no longer available, **Step** cannot maintain a consistent value for the remaining entries in the timestep selector. For example, if you load just Step 10, you will not necessarily get the same results loaded at the same timestep as you would have if you had loaded Step 10 before you deleted the .res file.

- The **Solver Step** column displays the solver timestep or outer iteration number. In multi-configuration cases, the solver step may not always increase across different cases and may not be unique.


Solver Step can be used in expressions. Timestep-related expressions such as Current Time Step and Accumulated Time Step refer to the **Solver Step**.

- The **Time [s]** column shows the real time duration corresponding to the timestep. The units are always seconds.
- The **Type** list displays the `Partial` or `Full` results file corresponding to that timestep.

The following icons/commands appear on the right side of the dialog box and/or the shortcut menu accessible by right-clicking on a timestep in the list box.

Icon/Command	Description
Switch To	Selects the timestep. Same as double-clicking the timestep.
 Add timesteps	Opens the Add Timestep Files window, which allows you to select one or more results files and load them into the memory.
 Delete	Only applies when there are added timesteps. This command allows you to delete added timesteps from the timestep selector.
 Animate	Automatically animates the timesteps using Quick Animation mode. For details, see Animating Timesteps (p. 160) .

Adding Timesteps

After you load a results file, the **Timestep Selector** dialog box shows the set of timesteps from that file. You can add to the set by clicking *Add Timesteps*  and selecting a file of type `res`, `bak`, or `trn`, or a directory containing files of type `trn`.

Select **Ignore duplicate timesteps** to avoid loading duplicate timesteps when loading a new file or directory. If this option is not selected, duplicate timesteps will appear at the end of the list, and will be given a unique timestep number.

Note

Adding timesteps to steady-state runs that contain particle tracks causes particles to be displayed up to the current time (which is zero for steady-state runs). To see the full particle track:

1. Open the particle track in question for editing.
2. On the **Geometry** tab, set **Limits Option** to `User Specified` and **End Time** to the maximum time value for the simulation.

Multiple Files

When multiple files are loaded, they appear on separate tabs on the top of the **Timestep Selector** dialog box. The **Sync Cases** setting is available to synchronize the cases in the following ways:

- `Off`

The `Off` option causes each set of results to be independent in terms of the selected timestep.

- `By Time Step`

The `By Time Step` option causes each set of results to switch to “match” the timestep you select for any set of results. All sets of results are therefore synchronized by timestep. The **Match** setting controls the matching criterion. The `Same Step` option causes results with identical timesteps to be synchronized, and results without identical timesteps to remain at their current timestep. The `Nearest Available` option causes the closest timestep to be selected for each set of results if there is not an exact match.

- `By Time Value`

The `By Time Value` option causes each set of results to switch to “match” the time value you select for any set of results. All sets of results are therefore synchronized by time value. The **Match** setting controls the matching criterion. The `Same Value` option causes results with identical time values to be synchronized, and results without identical time values to remain at their current time value. The `Nearest Available` option causes the closest time value to be selected for each set of results if there is not an exact match. The remaining **Match** options allow different degrees of matching; they are: `Within 1%`, `Within 5%`, and `Within 10%`.

- `By Index`

The **By Index** option causes each set of results to switch to “match”, as closely as possible, the index number you select for any set of results. All sets of results are therefore synchronized by index.

Animation

There are the following types of animation:

- [Quick Animation \(p. 159\)](#), which is a means to automatically sweep objects across their defined range
- [Keyframe Animation \(p. 160\)](#), in which you define the start and end points of each section of animation using *keyframes*, then link these end points together by having CFD-Post create a number of intermediate frames.


Selecting **Tools > Animation** produces the **Animation** dialog, where you can choose the type of animation you want. The animation options are described in the following sections.


Quick Animation


The Quick Animation option in CFD-Post provides a means to automatically sweep objects across their defined range to visualize the data throughout the domain. Planes, isosurfaces, turbo surfaces, streamlines, and particle tracks can all be animated with the Quick Animator.

To activate the Quick Animator, right-click on an object in the 3D Viewer and select **Animate**, or open **Tools > Animation** and select **Quick Animation**.

Use the slider to select the number of frames per animation loop. The more frames you add, the more positions the animating object will go through. The number of frames increases logarithmically as you move the slider toward the **Slow** end.

You can animate as many objects, of any type, as you want. Just select the objects in the list and click **Play** ; all selected objects will animate.

To stop an animation in progress, click **Stop** . The objects will return to the state they were in before the animation began.

By default, the animation will repeat infinitely until you click **Stop**. You can also specify a number of repetitions (raise the *Repeat forever*  button to enable the **Repeat** field).

You can create an animation in any of a variety of formats by selecting the **Save Movie** option, specifying the **Format**, and providing a filename. Select the **Options** button to select video creation and quality options, just as for keyframe animations.

Important

- Some combinations of graphics card type, operating system, and MPEG resolution fail to play MPEGs properly. You may be able to get normal playback results simply by changing the MPEG settings. Alternatively, you can upgrade your graphics card.
- Timestep animation cannot be combined with other types of animations because they have different loop cycles.
- When timestep animation is used in multi-file mode, CFD-Post loops over the timesteps specified in the primary (first loaded) case, as shown in the first tab of the Timestep Selector. If that case is steady-state, there will be no animation.

Animating Planes

An animated plane will be shifted in a direction normal to its surface.

If the **Bounce** option is selected (default), the plane will move to the positive limit, and then in reverse to the negative limit, and then repeat, moving to the positive limit again. If the **Loop** option is selected, the plane will move to the positive limit, and then jump to the negative limit (in one frame), and then start moving to the positive limit again.

Depending on the shape of the domain relative to its bounding box and the plane orientation, the animating plane may disappear for a number of frames at the ends of its ranges.

Animating Isosurfaces

Isosurface value is modified to traverse through the entire variable range.

If the **Bounce** option is selected (default), the isosurface value is increased to its maximum value, and then decreased to its minimum value, and repeated. If the **Loop** option is selected, the isosurface value is increased to its maximum value, then set to its minimum value (in one frame), and then increased to its maximum value again.

Animating Turbo Surfaces

Depending on the surface type, `Span`, `Streamwise Location` or `Theta` value will be modified to sweep through its respective range.

Animating Streamlines & Particle Tracks

`Streamlines` and `Particle Tracks` symbols are shifted along the lines.

By default, the symbol shape, interval, and size are overridden by the animation routines. If you want to change these settings, click the **Options...** button. To use symbol settings from the original object, clear the **Override Symbol Settings** check box.

The **Spacing** option specifies the interval at which to start a new batch of symbols. For example, with a spacing of 0.6, the symbols animating on the object will move 60% of the way along the lines, at which point another group of symbols will start at the beginning of the line. With a spacing of 1.0, all symbols will travel to the end of their lines before a new group of symbols starts at the beginning.

Animating Timesteps

Timestep animations play from the first to the last, and then repeat from the beginning.

Unlike other quick animations, Timestep animation does not automatically start when selected from the Timestep Selector, you need to click **Play**. This gives you an opportunity to configure the animation, which takes longer to load than other types of quick animations.

Note that the **Bounce** setting and the **Frame Count** (fast/slow) setting have no effect on Timestep animations.

Animating Mesh Deformation Scaling

The Deformation scaling factor is animated from **Undeformed** to the current scaling factor. If the current scaling is set to **Undeformed**, the animation goes from **Undeformed** to **True Scale**.

Keyframe Animation

In CFD-Post, you can make animations based on keyframes. Keyframes define the start and end points of each section of animation. Keyframes are linked together by drawing a number of intermediate frames, the number of which is set by the **# of Frames** field in the **Animation** dialog box.

The basic approach to creating an animation sequence is to configure the problem in a particular state and then save this state as a keyframe. Next, change one or more aspects of the problem state. For example, change the viewer orientation by rotating the viewer object. You can then save this state as a second keyframe.

Animations are created by linearly interpolating the change in state of the viewer position between keyframe states. By default, 10 frames are created between keyframe states, but this is easily adjustable. If the camera position changes between keyframes, the view is interpolated between the two positions at each frame.

Every option and button that is accessible when **Animation** is active will increment by one linearly for each of the frames between the two states. For example, if one keyframe has 10 contour levels, and the next has 20 contour levels, then the number of contour levels will increment by one for each of the ten frames between the two states. Objects that are binary in state are toggled at the end of the keyframe sequence (for example, the visibility of an object). Animations can also be created using the different timesteps in a transient run.

Note






If you have 2 keyframes with 10 frames between them, there are a total of 11 steps from one keyframe to the next.

Important

Some combinations of graphics card type, operating system, and MPEG resolution fail to play MPEGs properly. You may be able to get normal playback results simply by changing the MPEG settings. Alternatively, you can upgrade your graphics card.



Creating an Animation

The basic steps to creating an animation are as follows:

1. Once you have manipulated the GUI into a chosen start position, click *New*  to set the current state as **Keyframe 1**.
2. The keyframe becomes visible in the **Keyframe Creation and Editing** window.
3. Change the viewer and/or object parameters to obtain the second required state and click *New*  to create **Keyframe 2**.
4. When you click a keyframe to highlight it, the other options to the right of the keyframe list become active.
5. To display the highlighted keyframe in the viewer, click *Edit Keyframe*  or double-click on the keyframe itself. To apply changes in the viewer to the highlighted keyframe, click *Set Keyframe* . If more than 2 keyframes exist and you want to change their order, you can move a Keyframe up and down by clicking on the blue arrows. To delete a keyframe, click *Delete* .
6. To set the number of intermediate interpolated frames, click on a keyframe and set the value in the **# of Frames** box.

After a second keyframe has been created, additional playback options are made available.

7. The looping option allows you to specify whether you want the animation to play in one direction during each repeat or play forwards and backwards. For example, selecting **Repeat** of 3 on the **Loop** setting will play the animation 3 times, jumping from the last Keyframe back to the first at the end of the first two cycles. Selecting **Bounce** for the same number of repeats will cause the animation to play forwards, and then backwards before playing forwards once more.

With the **Repeat** option, you specify how often the animation repeats before stopping. By default the *Repeat forever*  button is selected, so the animation will repeat continuously until you click *Stop* .



8. The **Animate Camera** feature toggles whether the camera position is moved (interpolated) with the animation. If it is switched off, all objects, except for the camera positions, are animated.

















Animating Expressions

There is a limitation with respect to the animating of expressions. If the value of a parameter of an object is set to an expression, that expression is evaluated at the keyframes, but those values are not interpolated to obtain values at the frames between the keyframes. Thus, after the value of the parameter is determined for the first keyframe, that value does not change for the intermediate frames until it is recalculated at the next keyframe, which causes the animation to be unexpectedly discontinuous.


Animation Dialog Box

The following is a general explanation of the icons in the **Animation** dialog box:

Icon	Description	Icon	Description
	Create a new keyframe		Go to beginning

Icon	Description	Icon	Description
	Edit a keyframe		Go to previous keyframe
	Set the keyframe		Go to previous frame
	Move the keyframe up		Go to next frame
	Move the keyframe down		Go to next keyframe
	Delete the keyframe		Go to end
	Load animation state		Play forward
	Save animation state		Stop the animation
	More animation options		Repeat forever

Animation Options Dialog Box

The **Animation Options** dialog box is opened by expanding *More animation options*  at the bottom of the **Animation** dialog box, then clicking **Options**.

Animation Speed

The **Animation Speed** settings enable you to scale the animation to speed it up or slow it down without having to manually adjust the number of frames between keyframes in the animation.

The Approximate Animation Time is calculated with the following information: total number of frames in the animation, the number of repetitions, the frame rate (regardless of whether you are saving to a movie or not), and any animation speed adjustments.

Selecting an animation speed of **Normal** does not scale the animation by any factor.

Selecting an animation speed of **Slower** slows down the animation by adding sufficient additional frames to achieve the specified factor. Selecting *Generate more frames, spread evenly* automatically and transparently adds additional frames between keyframes. You will see the effect of this the next time you play the animation. This results in higher quality animations, but will take longer to compute because of the additional frames to interpolate. Selecting *Duplicate frames when saving movie* duplicates existing frames when generating the final movie output. The effect of this will be visible only when playing back the movie; you will see no effect when playing the animation in CFD-Post. This option is faster, but the quality of the movie may suffer: it may look a little jerky.

Selecting an animation speed of **Faster** speeds up the animation by removing sufficient frames to achieve the specified factor. Selecting *Generate fewer frames, spread evenly* automatically and transparently removes some of the frames between keyframes. You will see the effect of this next time you play the animation. The fewer frames between keyframes will be interpolated smoothly, as if you had reduced the number of frames manually. Selecting *Skip frames when saving movie* skips existing frames only when generating the final movie output. The effect of this will only be visible when playing back the movie file; you will see no effect when playing the animation in CFD-Post. This option is slower because all frames will be played in CFD-Post, but only some of the frames will be used to generate the movie.

Transient Case

The **Transient Case** setting is effective only for transient simulations and controls the way in which timesteps are selected. A particular frame is calculated. **Sequential Interpolation** evenly distributes frames over each

transient output file. **Timestep Interpolation** evenly distributes frames based on the timestep number associated with each transient output file. **TimeValue Interpolation** evenly distributes frames based on the time value associated with each transient output file.

Print Options

Image Format

Select either a JPEG or PPM format for creation of the movie.

Use Screen Capture

By default, all image files are created using an internal (software-based) rendering engine. In some situations, it is desirable to create an exact copy of the screen image, as rendered by the graphics hardware on the computer. This is possible by selecting the **Use Screen Capture** toggle. For very complex images, this option can be significantly faster. It is important to note that this is a true screen capture, and on some machines and graphics cards, it will be important to make sure that there are no other undesired windows or screen objects overlapping the viewer window at the time that the image file is created. These unwanted windows can, in some cases, become part of the movie.

White Background

Toggles between a white/black background.

Enhanced Output (Smooth Edges)

Allows you to select higher quality output for the generated images.

Image Size

Allows you to specify the resolution of the resulting movie. By default, 640 × 480 is selected, but you can select any of the values in the combo, including NTSC or PAL standard resolutions. You can also select **Custom** to specify the pixel resolution in the **Width** and **Height** fields, or select **Use Screen Size** and specify a scale factor in the **Scale (%)** field.

Tolerance

Controls the amount of depth calculated for the creation of an image, where smaller values represent more accurate images. The benefit of relatively high values is that less processing is required. However, if the **Tolerance** value is too high (for instance, a value of 1), the back faces in an image may be displayed on top of near faces.

Advanced Tab

Save Frames As Image Files

If you have enabled **Save Movie** (see [Quick Animation \(p. 159\)](#)), selecting **Save Frames As Image Files** will prevent the deletion of the animation frame files from the temporary directory, where they are stored by default.

Output To User Directory

Selecting this option enables you to specify where you want the animation files to be saved by entering a path in the **Directory** field.

Frame Rate

The rate (in frames per second) at which the movie will be generated. The movie viewer may also dictate the playback rate.

Quality


Select a **Quality** from: High, Medium, Low, Custom. With the Custom setting, you may specify the **Variable Bit Rate** by unchecking the **Variable Bit Rate** toggle and entering a bit rate. Reduce the **Bit Rate** value to lower the file size (and the file playback quality).

Don't Encode Last MPEG Frame

A single cycle of an animation loop starts and ends at the same frame. If you repeat a loop, that frame is encoded twice at the end of each cycle, leading to a brief pause at that point in the animation. Enable this setting to smooth the playback of repeated loop animations.

Saving an Animation

When **Save Movie** is selected and a filename is specified, the animation is saved to a file when the animation is played.

1. To select a file, click *Browse*  and browse to a convenient location.
2. Enter the name of the file; the extension is taken from the setting of the **Format** field.

Note

The **Windows Media Video**, **AVI**, and **MPEG4** format options all use MPEG-4 encoding. You will need a player that supports MPEG-4 to view videos in those formats. If you are using Windows Media Player, you can download an MPEG-4 codec from ffdshow.

3. Click **Save** to save the file.

Saving the Animation State (*.can file)

You can load or save your animation state as a `.can` (CFX Animation) file. It saves the current status of all of the animation settings.

To open:

1. Browse to the correct directory to load the file.
2. Enter the name of the file or select it by using the mouse.
3. Click **Open**.

To save:

1. Browse to a convenient directory to save the file.
2. Enter or select the name of the file.
3. You should save the file as a `.can` (CFX Animation) type.

Playing an MPEG file

Non-Windows installations of CFX include an MPEG player, called `mpeg_play`, which can be run from the operating system with the name of an MPEG file provided as an argument. The MPEG player is located in `<CFXROOT>/tools/share/bin/`.

Quick Editor

The Quick Editor in CFD-Post lets you easily perform repetitious modifications to certain objects. You can move planes along their normals to a specified location, set the value of isosurfaces, and set turbo surface locations.




To specify a value, you can enter a number in the value editor, move the slider, or click left/right mouse buttons to increment/decrement the value by a portion of the range. All changes are applied immediately; there is no need to click **Apply**.

Probe

Probe in CFD-Post allows you to determine exact variable values at specified points within a domain.

1. Select **Tools > Probe** or click *Probe* , or right-click on an object in the viewer and select **Probe Variable**.

The **Probe** tool appears at the bottom of the viewer.

2. You can manually input the probe coordinates in the **Probe At** boxes or select a point in the viewer.
If *Probe only this variable*  is not selected, the probe variable will be automatically chosen. (For example, Temperature will be selected if you select a point on a plane that is colored by Temperature).
3. The probe variable can also be selected manually from the variable list.
4. If the desired variable does not appear in the list, select **Other...** and choose the variable from the **Variable Selector**.
5. If *Probe only this variable*  is selected, the probe variable will not change automatically when new coordinates are entered.
6. Select *Hide Probe tool*  to close the **Probe** tool.
The probed value appears in the box adjacent to the variable list and automatically updates every time a new coordinate or probe variable is selected.

If you probe on a **Point** object, the probe position will use the position coordinates of the **Point** object, not necessarily exactly where you chose.

Limitation:

Probe locations will be more accurate when you zoom in tight on the probe location when picking in the viewer. The smaller the object in the viewer is, the less accurate the picked location will be. A consequence is that you may get an undefined value on an outer boundary, since the point location will be slightly outside the domain. This problem may disappear if you zoom in on the boundary and probe again. Note that you can also adjust the probe location by typing in the coordinates.

Function Calculator

The function calculator is used to provide quantitative information about the results. To use the function calculator:

1. Select the function to evaluate from this list.
2. Choose the location for the calculation.
Only locations valid for the selected function will be available.
3. If multiple cases are open, choose which cases the function calculator should act upon.
4. If applicable to the function, choose a variable from the list.
For most functions, you can click in the **Variable** box and enter an expression. The expression can include variables and any valid CEL (CFX Expression Language) function. For example, `abs (Velocity u)` could be entered so that the calculation is performed using the absolute values of the variable `Velocity u`. For details, see [CEL Operators, Constants, and Expressions \(p. 135\)](#) and [CFX Expression Language \(CEL\) \(p. 133\)](#).
User variables are also available. For details, see [Variables Workspace \(p. 36\)](#).
5. Select the direction if applicable to the function.
For some functions **None** is available.
For details, see [Quantitative Function List \(p. 146\)](#).
6. If applicable to the function, select the appropriate fluid.
For multiphase results, you can select which fluids to use in your calculation for selected functions. The **All Fluids** option can be selected to perform the calculation using all of the fluids in the results.

Note

When calculating mass flow rate for an ANSYS FLUENT file, the option **Mixture** gives the same results as **All Fluids**. These two options appear because have different origins (ANSYS FLUENT and CFD-Post respectively); you may choose either for your calculations.

Click **Calculate** to calculate the result. Choose whether to base the calculation on hybrid or conservative values. Most quantitative calculations are best performed using conservative variable values. For details, see [Hybrid and Conservative Variable Values \(p. 165\)](#).

Note

If the function result is a temperature, and if C or K are selected as temperature units, the result's units will be K. If F or R are selected, the temperature will be returned in R.

This has an implication for calculations of temperature differences measured in C or F. Expressions are always evaluated in absolute temperature units (K or R) and then, if necessary, the result is converted to the user-selected units. For example, if you evaluate $1[C] - 1[C]$, internally it is evaluated as $274[K] - 274[K]$, which is $0[K]$ and is reported as such (with the units forced to be in an absolute scale). In plots (where CFD-Post cannot force the units to be K), the software cannot tell whether the result is a temperature difference or just the temperature, so the result is converted to user-selected units (in this case, $-273[C]$) and a value of -273 is reported in the plot legend. Thus when analyzing temperature differences, set the preferred temperature units to be in an absolute scale (K or R) in the **Edit > Options > Units** dialog box.

Important

There are some important limitations concerning calculations performed on CFX-4 results files. For details, see [CFX-4 Dump Files \(p. 72\)](#).

For additional information on the function calculator see [Quantitative Calculations in the Command Editor Dialog Box \(p. 256\)](#).

Function Selection

The quantitative functions available from the function calculator in CFD-Post are integrated into CEL and can be used in any expression.

The available quantitative functions are outlined in the table below.

Function Name	Operation
area (p. 150)	Area of location
areaAve (p. 151)	Area-weighted average
areaInt (p. 151)	Area-weighted integral (can be projected to a direction)
ave (p. 152)	Arithmetic average
count (p. 153)	Number of Nodes
countTrue (p. 153)	Number of nodes at which the logical expression evaluates to true.
force (p. 154)	Force on a surface in the specified direction
forceNorm (p. 155)	Length normalized force on a curve in the specified direction
length (p. 156)	Length of a line
lengthAve (p. 156)	Length-weighted average
lengthInt (p. 157)	Length-weighted integration
massFlow (p. 157)	Total mass flow
massFlowAve (p. 158)	Mass Flow-weighted average
massFlowAveAbs (p. 159)	Mass Flow-weighted average with absolute values of mass flow used in numerator and denominator of formula for averaging

Function Name	Operation
massFlowInt (p. 160)	Mass Flow-weighted integral
maxVal (p. 161)	Maximum Value
minVal (p. 161)	Minimum Value
probe (p. 162)	Value at a point
sum (p. 162)	Sum over the calculation points
torque (p. 163)	Torque on a surface about the specified axis
volume (p. 163)	Volume of a 3D location
volumeAve (p. 163)	Volume-weighted average
volumeInt (p. 164)	Volume-weighted integral

For details on each of the functions listed in the table above, see [Quantitative Function List \(p. 146\)](#) in the ANSYS CFX Reference Guide.

Macro Calculator

A *macro* is a set of Perl and Power Syntax¹ commands that are stored in a session file. The commands create an extension to the user interface in the **Macro Calculator** that prompts you for the arguments that are required. For example, for the macro that calculates the noise generated by a low-speed fan, you need to specify the number of blades, the number of harmonics, the position of the observer, and so on.

You can run one of the pre-defined macros provided with CFD-Post or run one of your own:

- [Predefined Macros \(p. 167\)](#)
- [User-defined Macros \(p. 171\)](#)

To run a macro (assuming that a case is already loaded):

1. From the menu bar, select **Tools > Macro Calculator**.
2. Select an appropriate macro function from the list or open a file that contains a user-defined macro definition. (In the latter case, opening the file both loads the macro into the **Macro Calculator** and adds that macro to the macro list.)
3. Fill in the fields that appear in the **Macro Calculator**. The values that you must specify vary for each macro.
4. Click **Calculate** to perform the calculations defined in the macro.
5. Click **View Report** to open the **Report Viewer**.

This displays results and charts of the calculations performed.

Predefined Macros

Some predefined macros are provided as CFD-Post. These are session files that contain a subroutine. The options available in the **Macro Calculator** are simply the arguments with which the subroutine is called.

The predefined macros are:

- [Comfort Factors Macro \(p. 168\)](#)
- [Cp Polar Plot Macro \(p. 168\)](#)
- [Gas Compressor Performance Macro \(p. 168\)](#)
- [Gas Turbine Performance Macro \(p. 169\)](#)
- [Liquid Pump Performance Macro \(p. 169\)](#)

¹For details on power syntax, see [Power Syntax in ANSYS CFX \(p. 259\)](#).

- [Liquid Turbine Performance Macro \(p. 169\)](#)
- [Fan Noise Macro \(p. 170\)](#)

You execute a macro by clicking the **Calculate** button. Any objects, user variables, and expressions defined in the macro are created. A report is also generated in HTML format and can be viewed by clicking on **View Report**. The **Report Viewer** window displays the results; you must close this window by clicking **OK** before you can continue using CFD-Post.

For more details on the macro definitions, refer to [Macro Details \(p. 171\)](#).

Comfort Factors Macro

This macro can be used to calculate values for Mean Radiant Temperature and Resultant Temperature in HVAC simulations.

In order to use the macro, the radiation intensity, velocity, and temperature are required. The created expressions are visible on the **Expressions** tab. The variables Mean Radiant Temperature (evaluated using the expression meanTemp) and Resultant Temperature (evaluated using the expression resultTemp) are created.

As an alternative to calculating comfort factors in CFD-Post, the comfort factors may be calculated during the solution process; this would be required, for example, when the model simulates a ventilation system in which the control system depends dynamically on derived comfort factors.

Cp Polar Plot Macro

The **Cp Polar** macro produces a polar plot of the pressure coefficient (Cp) along a polyline. The macro creates the polyline using the Boundary Intersection method. For details, see [Polyline Command \(p. 110\)](#). The boundary and intersecting slice plane are defined in the macro calculator and passed to the subroutine as arguments. The boundaries selected as the **Boundary List** in the macro calculator make up one surface for the intersection. The second surface is a slice plane created using the X, Y, or Z normal axis to the plane (**Slice Normal**) and a point on that axis (**Slice Position**).

The cp user variable is created by the macro from the cp expression. The cp expression can be defined as:

```
(Pressure - $pref [Pa]) / dynHead
```

where \$pref is the **Ref. Pressure** set in the macro calculator and dynHead is a reference dynamic head (evaluated at the inlet) that can be defined as:

```
0.5 * areaAve(Density)@inlet * areaAve(Velocity)@inlet^2
```

The **Inlet Region** selected in the macro calculator is used as the inlet location in the calculation of dynHead.

Next, a Chart line of the cp variable versus the **Plot X Axis** value is created. The generated report contains the chart and the settings from the macro calculator.

The following information must be specified:

- Boundary List: A list of boundaries used in the simulation.
- Slice Normal: The axis that will be normal to the slice plane.
- Slice Position: The offset of the slice plane in the direction specified by the normal axis.
- Inlet Region: The locator used to calculate inlet quantities.
- Ref. Pressure: The reference pressure for the simulation.
- Plot Axis: The axis on which the results will be plotted.

Gas Compressor Performance Macro

The **Compressor Performance** macro performs a series of calculations using the data set in the macro calculator. The following information must be specified:

- Inlet Region: The locator used to calculate inlet quantities.
- Outlet Region: The locator used to calculate outlet quantities.
- Rotor Blade(s): The locator used to calculate torque (one blade row) about the machine axis.

- Machine Axis: The axis of rotation of the compressor.
- Rot. Speed: The rotational speed of the compressor.
- Num. Blade Rows: Some quantities calculated for a single blade row are multiplied by the number of blades to produce total (all blade) values.
- Ref Pressure: The reference pressure for the simulation.
- Ref Radius: A reference radius between the hub and tip.
- Fluid Gamma: The ratio of specific heat capacity at constant pressure to specific heat capacity at constant volume (C_p / C_v).

Gas Turbine Performance Macro

The following information must be specified:

- Inlet Region: The locator used to calculate inlet quantities.
- Outlet Region: The locator used to calculate outlet quantities.
- Rotor Blade(s): The locator used to calculate torque (one blade row) about the machine axis.
- Machine Axis: The axis of rotation of the turbine.
- Rot. Speed: The rotational speed of the turbine.
- Num Blade Rows: Some quantities calculated for a single blade row are multiplied by the number of blades to produce total (all blade) values.
- Ref Pressure: The reference pressure for the simulation.
- Ref Radius: Reference radius between the hub and tip.
- Fluid Gamma: The ratio of specific heat capacity at constant pressure to specific heat capacity at constant volume (C_p / C_v).

Liquid Pump Performance Macro

The following information must be specified:

- Inlet: The locator used to calculate inlet quantities.
- Outlet: The locator used to calculate outlet quantities.
- Rotor Blade(s): The locator(s) used to calculate torque (one blade row) about the machine axis.
- Machine Axis: The axis of rotation of the pump.
- Rot. Speed: The rotational speed of the pump.
- Num Blade Rows: Some quantities calculated for a single blade row are multiplied by the number of blades to produce total (all blade) values.
- Ref Pressure: The reference pressure for the simulation.
- Ref Radius: Reference radius between the hub and tip.
- Ref Height: Cross-section height (that is, the height of the outlet region, or the height of the blade at the trailing edge).
- Ref Density: The reference density for the simulation.
- Gravity Accel.: The acceleration due to gravity.

Liquid Turbine Performance Macro

The following information must be specified:

- Inlet: the locator used to calculate inlet quantities.
- Outlet: the locator used to calculate outlet quantities.
- Rotor Blade(s): the locators used to calculate torque (one blade row) about the machine axis.
- Machine Axis: the axis of rotation of the turbine.
- Rot. Speed: the rotational speed of the turbine.

- Num Blade Rows: some quantities calculated for a single blade row are multiplied by the number of blades to produce total (all blade) values.
- Ref Pressure: The reference pressure for the simulation.
- Ref Radius: reference radius between the hub and tip.
- Ref Height: Cross-section height (that is, the height of the outlet region, or the height of the blade at the trailing edge).
- Flow Rate: The volume flow rate.
- Head Rise: The pressure head at the inlet.
- Ref Density: The reference density for the simulation.
- Gravity Accel.: The acceleration due to gravity.

Fan Noise Macro

This macro calculates the noise levels of the turbomachinery as observed at a specific location. The following information must be specified:

- Domain: The domain in which the blade is located.
- Blade Selection: Set to Automatic for a single blade passage or Custom for a multiple blade passage. If this is set to Custom, you will need to specify the 2D region for the blade (**Custom Blade**) as well as the number of blades (**Custom # Blades**).
- # of Harmonics: The number of harmonics used in the calculation.
- Observer (r) and Observer (theta): The distance and location of the observer, relative to the blade.
- Theta Sectors: The number of sampling points (sectors) equally spaced over 360° at a given radius around the fan, used to calculate the noise values. A higher number leads to a more accurate solution, but takes more time to calculate.
- Directivity Harm. #: The harmonic level at which the sound pressure levels will be calculated.
- Loading Coeff.: A coefficient between 2 and 2.5.

The loading coefficient parameter defines the decay (or decrease) of the sound-pressure level vs the frequency. In general, the sound-pressure level decreases when the frequency increases. In his experiments, Lowson replaced the unsteady loading by a steady one multiplied by a decay function. Based on these experiments, this decay follows an exponential law with a negative slope. Lowson found that a loading coefficient between 2 and 2.5 gives a sound-pressure level close to the experimental data. In other words, the loading coefficient defines the slope of the exponential law.

In general and for highly loaded blades, the decay of the sound-pressure level is very quick (one or two peaks in the sound-pressure level spectrum) and therefore a higher value of the loading coefficient will be appropriate.

- Acou. Ref. Pressure: Acoustic reference pressure (P_{ref}) is the international standard for the minimum audible sound of 2.10-5 [Pa].

The acoustic reference pressure is used to convert the acoustic pressure into Sound Pressure in dB using the following equation:

$$SPL_m (dB) = 20 \log_{10} \left(\frac{P'_m}{P_{\text{ref}}} \right) \quad (\text{Eq. 10.1})$$

where P_{ref} is the acoustic reference pressure. The reference pressure depends on the fluid.

- Acou. Ref. Power: Acoustic reference power (W_{ref}) is used to convert the sound power SW_m from units of $[W m^{-3}]$ to units of dB.

The equation used is:

$$LW_m (dB) = 10 \log_{10} \left(\frac{SW_m}{W_{\text{ref}}} \right) \quad (\text{Eq. 10.2})$$

where:

- W_{ref} is the value of the acoustic reference power
- SW_m is the sound power and is defined by:

$$SW_m = \frac{\pi r_1^2}{\rho c_0} \int_0^\pi (P'_m)^2 \sin\phi \, d\phi \quad (\text{Eq. 10.3})$$

For air, the acoustic reference power is: $1 \, e^{-11} \left[W \, m^{-3} \right]$

- Sound Speed: The speed of the sound in the fluid at rest.

For details on completing this dialog, see [Using the Fan Noise Macro \(p. 173\)](#).

Macro Details

You can view the macro definitions in a text editor: they are located in <CFXROOT>/etc/ and have a .cse file extension. For details on the input parameters and output expressions for any given macro, you may view the details on the **Expressions** tab in CFD-Post once the macro has been executed.

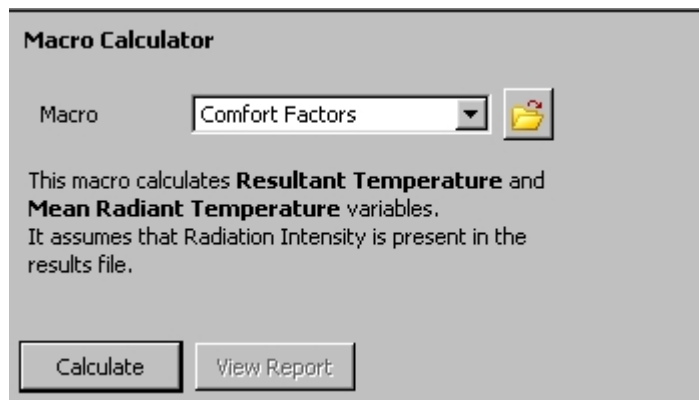
User-defined Macros

You can load macros from a file and have them available in the **Calculators** workspace. To add your own macros to the list, set the CFXPOST_USER_MACROS environment variable to a comma-separated list of the path of each macro you want to add. You can also view the existing macros in <CFXROOT>/etc/* .cse and study the definitions in order to understand how to create your own macro. The macro file must contain at least one power syntax subroutine. For details on power syntax, see [Power Syntax in ANSYS CFX \(p. 259\)](#).

The following is an example of a short subroutine:

```
! sub Hello1 {
! print "Hello !\n";
! }
! sub Hello2 {
! ($title, $name) = @_ ;
! print "Hello $title $name\n";
! }
```

Loading the file containing the subroutines makes each subroutine available for execution. An example of the **Comfort Factors** subroutine is shown.



Use quotation marks for string entries, and separate each argument with a comma.

You can also embed GUI controls into the macro using lines with special comments. In the following example, the name of the macro, the types of options and the subroutine to call are all specified. This is done by adding macro GUI parameters between # Macro GUI begin and # Macro GUI end lines.

```

# Macro GUI begin
#
# macro name = A simple macro
# macro subroutine = mySub
#
# macro parameter = Var
#   type = variable
#   default = Y
#
# macro parameter = Location
#   type = location
#   location type = plane
#
# Macro GUI end
! sub mySub {
! ( $variable, $plane) = @_ ;
!
! print "variable = $variable, plane = $plane\n";
!}

```

where the macro controls are as described in the following table:

Value	Definition
# macro name = <name>	The macro identifier to appear in the macro combo
# macro subroutine = <subname>	The subroutine to call
# macro report file = <filename>	The file generated by the macro (if any). This enables the View Report button, which attempts to load the file in a text/html browser.
# macro related files = <file1>, <file2>	Other related files to load when loading this macro. This is useful when your macro uses subroutines from other files.
# macro parameter = <name> #type = <type> #<option1> = <val> #<option2> = <val> #..	Specifies a GUI option for a subroutine parameter. The type of widget for the option is determined by the type of parameter. For each type there can be several possible options, as described in the table below. The order of the GUI options must match the order of the arguments for the subroutine.

Type	Option	Example	Notes
string	default	My String	
integer	default range	10 1, 100	
float	default range quantity type	0.1 [s] 0.1 [s], 0.4 [s] Time	units are optional controls units ^a
triplet	default range quantity type	0.5[m], 0[m], 1[m] -1[m], 1[m] Length	units are optional controls units [*]

Type	Option	Example	Notes
location	default location type location category	Inlet Boundary point	either type or category can be specified ^{bc}
list	default list	orange apple, orange, fig	
variable	default	Pressure	
domain	default	Stator	

^aA full list of quantity types can be found in <CFXROOT>/etc/<version>/common_units.cfx

^bFull list of types and categories include: **Types**: any valid object type; **Categories**: point, line, surface, plane, volume, variable, geometry, viewer-viewable, selectable.

^cReturns the object's path. To get the object's name, use [getObjectName\(\)](#) (p. 267).

Loading this macro adds an entry `A simple macro` in the macro combo, with two options:

- **Variable** - A combo with all loaded variables listed, defaulting to Y
- **Location** - A combo with all objects of type plane listed

When it is executed, it just prints what is selected in the combos.

Macro Example: Output Path

This is an example macro that explicitly defines the output path using the Perl Print command output redirection.

```
! open(FH, ">myOut.txt");
! $val = ave("Pressure", "Point 1");
! $time = getValue("DATA READER", "Current Timevalue");
! print FH "$time $val\n";
! close(FH);
```

Using the Fan Noise Macro

The Fan Noise macro calculates the tonal noise levels generated by a low-speed fan (primarily axial-flow fans). Tonal noise, or discrete-frequency noise, is due mainly to periodic forces exerted on fluid passing a fan. The Fan Noise macro can be applied to low speed fans having a tip Mach number less than 0.45. For a higher tip Mach number, the accuracy of the results is questionable. The fan must radiate in the free field where the observer can see the fan blades (the Fan Noise macro does not take into account the reflection effect). Thus, the Fan Noise macro cannot be applied to ducted fans.

Fan Noise Theory in Brief

Several methods have been developed to predict tonal noise; the Lowson Model is described here.

In the low-speed regime, the main noise component is a dipolar source. Lowson [Lowson, M. V., 1970, “*Theoretical analysis of compressor noise*”, The Journal of Acoustics So. Am., Vol. 47 (1), 1970, pp. 371-385.] showed that the noise generated by a fan is directly related to the aerodynamic forces exerted on the fixed and rotating blades. First, in a semi-empirical way, he calculated these forces; then he took into account the distance between the source and the observer. In this case, the fan is considered as a noise source for which the frequency depends on the rotational speed and other parameters. In 1962, Lighthill established the acoustic pressure expression produced by a punctual force, F_i , in rectilinear motion.

$$p' \left(x_i, y_i \right) = \frac{x_i - y_i}{4 \pi c_0 r^2 (1 - M_r)^2} \frac{\partial F_i}{\partial t} \quad (\text{Eq. 10.4})$$

where:

$$x_i = (x, y, 0)$$

$$y_i = (0, R \cos \theta, R \sin \theta)$$

$$M_r = \frac{-1}{c_0} \frac{dr}{dt}$$

$$\tau = t - \frac{r}{c_0}$$

$$F_i = (-F, F_y \sin \theta, F_y \cos \theta)$$

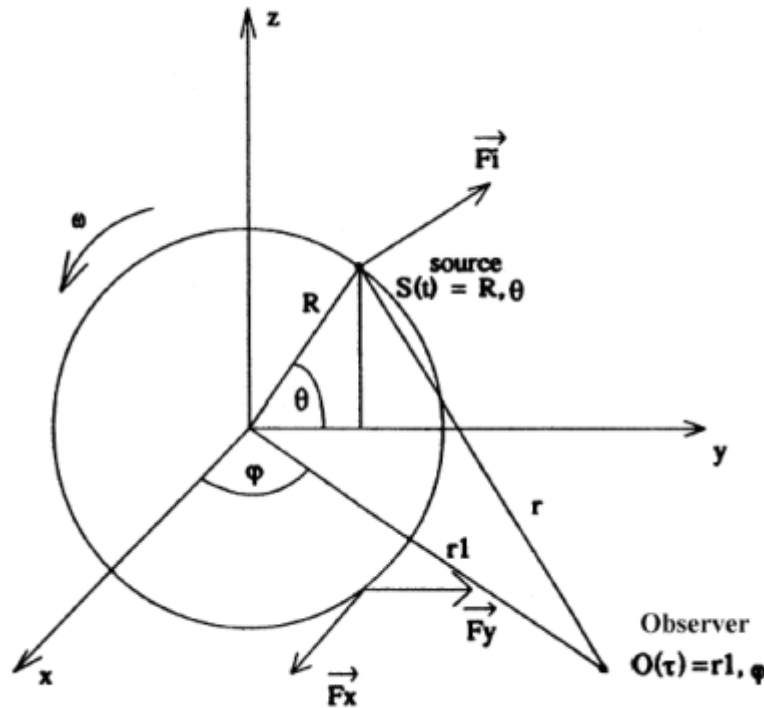
As shown in Figure 10.1, “Relative position of the source and the observer” (p. 174), x_i and y_i are the coordinates of the Observer O (r, ϕ, τ) and of the Source S (R, θ, t), respectively. M_r is the convective component of the rotational Mach number in the r direction. F_x and F_y are respectively the thrust and the drag (torque) forces exerted on the blade. According to Equation 10.4 (p. 173), when the force F_i is constant, the acoustic pressure is equal to zero.

Lowson extended Equation 10.4 (p. 173) to create a more general equation:

$$p' (x_i, y_i) = \frac{x_i - y_i}{4 \pi c_0 r^2 (1 - M_r)^2} \left[\frac{\partial F_i}{\partial t} + \frac{F_i}{1 - M_r} \frac{\partial M_r}{\partial t} \right] \quad (\text{Eq. 10.5})$$

This relation describes the contribution of the convective phenomenon due to the term $\partial M_r / \partial t$. Note that Equation 10.5 (p. 174) must be evaluated at retarded time τ . This equation can be used to find an expression for the sound from a point force in arbitrary harmonic motion.

Figure 10.1. Relative position of the source and the observer



The Lowson model allows the calculation, at the observer position, of the acoustic pressure generated by steady and unsteady efforts. The latter are considered as punctual sources and correspond to the loads exerting by the z blades of the rotor. Lowson integrated Equation 10.5 (p. 174) in time and space to get the m^{th} harmonic of the acoustic pressure generated by a periodic rotating loading:

$$p'_m = a_m + j b_m = \frac{\omega}{\pi} \int_0^{\frac{2\pi}{\omega}} \left[\frac{x_i - y_i}{4\pi a_0 r^2 (1 - M_r)^2} \left[\frac{\partial F_i}{\partial t} + \frac{F_i}{1 - M_r} \frac{\partial M_r}{\partial t} \right] \right] \exp(j m \omega t) dt \quad (\text{Eq. 10.6})$$

using the following equation:

$$dt = (1 - M_r) d\tau \quad (\text{Eq. 10.7})$$

and integrating Equation 10.6 (p. 175) by parts gives:

$$p'_m = -\frac{\omega}{4\pi^2 r} \int_0^{\frac{2\pi}{\omega}} \left\{ \frac{j m \omega F_r}{a_0} + \frac{F_i}{1 - M_r} \left[-\frac{M_i}{r} + \frac{(x_i - y_i)}{r^2} \right] \right\} \exp \left[j m \omega \left(\tau + \frac{r}{a_0} \right) \right] d\tau \quad (\text{Eq. 10.8})$$

as shown in Figure 10.1, “Relative position of the source and the observer” (p. 174) with x being the axis of rotation and the fluctuating loading and observer position being defined as:

$$F_i = \begin{cases} -F_x \\ -F_y \sin \theta \\ F_y \cos \theta \end{cases}$$

$$x_i - y_i = \begin{cases} x \\ y - R \cos \theta \\ -R \sin \theta \end{cases}$$

$$r = |x_i - y_i| \approx r_1 - \left(\frac{yR}{r_1} \right) \cos \theta$$

In

$$F_r = -\frac{x F_x}{r} - \left(\frac{y F_y}{r} \right) \sin \theta \quad (\text{Eq. 10.9})$$

- F_x and F_y are respectively the thrust and drag (torque) components of the aerodynamic unsteady force represented by a global force exerted on the blade.
- The terms in $1/r$ and $1/r^2$ are important only in the acoustic near field. Thus, in the acoustic far field, Equation 10.8 (p. 175) becomes:

$$p'_m = -\frac{\omega}{4\pi^2 r} \int_0^{\frac{2\pi}{\omega}} \left\{ \frac{j m \omega F_r}{a_0} \right\} \exp \left[j m \omega \left(\tau + \frac{r}{a_0} \right) \right] d\tau \quad (\text{Eq. 10.10})$$

Taking into account of the thrust and drag periodicities, Lowson proposed the following formulation:

$$\begin{pmatrix} F_x \\ F_y \end{pmatrix} = \sum_{\lambda=-\infty}^{\lambda=\infty} \begin{pmatrix} F_x(\lambda) \\ F_y(\lambda) \end{pmatrix} \exp(-i\lambda\omega t) \quad (\text{Eq. 10.11})$$

where λ is the effort harmonic order or the mode.

Substituting the results obtained from Equation 10.9 (p. 175) and Equation 10.11 (p. 175) into Equation 10.10 (p. 175) gives:

$$p'_m = -\frac{j m \omega}{4\pi^2 r c_0} \int_0^{2\pi} \left(\sum_{\lambda=-\infty}^{\lambda=\infty} \left\{ \frac{x F_x(\lambda)}{r_1} + \frac{y F_y(\lambda)}{r_1} \sin \theta \right\} \right) \exp \left[j(m - \lambda) \theta - j m \left(\frac{yM}{r_1} \right) \cos \theta \right] d\theta \quad (\text{Eq. 10.12})$$

where the rotational Mach number is $M = \omega R / c_0$

The integrals in [Equation 10.12 \(p. 175\)](#) can be identified as Bessel functions, and, using the expressions:

$$\int_0^{2\pi} \exp[j(m\theta - z \cos\theta)] d\theta = 2\pi j^{-m} J_m(z) \quad (\text{Eq. 10.13})$$

$$\int_0^{2\pi} \exp[j(m\theta - z \cos\theta)] \sin\theta d\theta = (-2)\pi j^{-m} \left(\frac{m}{z}\right) J_m(z)$$

[Equation 10.12 \(p. 175\)](#) can be evaluated directly to give the sound level radiated from z rotor blades:

$$p'_m = \frac{j m z^2 \omega}{2 \pi c_0 r_1} \sum_{\lambda=-\infty}^{\lambda=\infty} (-j)^{mz-\lambda} \left[\cos\phi \left(F_x(\lambda) - \left(\frac{mz-\lambda}{mzM} \right) F_y(\lambda) \right) \right] J_{mz-\lambda}(mzM \sin\theta) \quad (\text{Eq. 10.14})$$

where:

- $J_{mz-\lambda}$ is a first Bessel Function of order $mz-\lambda$
- z is the number of blades
- $j^2 = -1$

The interest of this relation is the knowledge of the components of the fluctuating efforts $F_x(\lambda)$ and $F_y(\lambda)$.

Following the experimental work done on helicopter blades by Scheiman [Scheiman, J., 1964, “*Sources of noise in axial flow fans*”, Journal of Sound and Vibration, Vol. 1, (3), 1964, pp. 302-322.], Lowson extended

[Equation 10.14 \(p. 176\)](#) to an equation that relates the steady-state components of the force to the acoustic pressure.

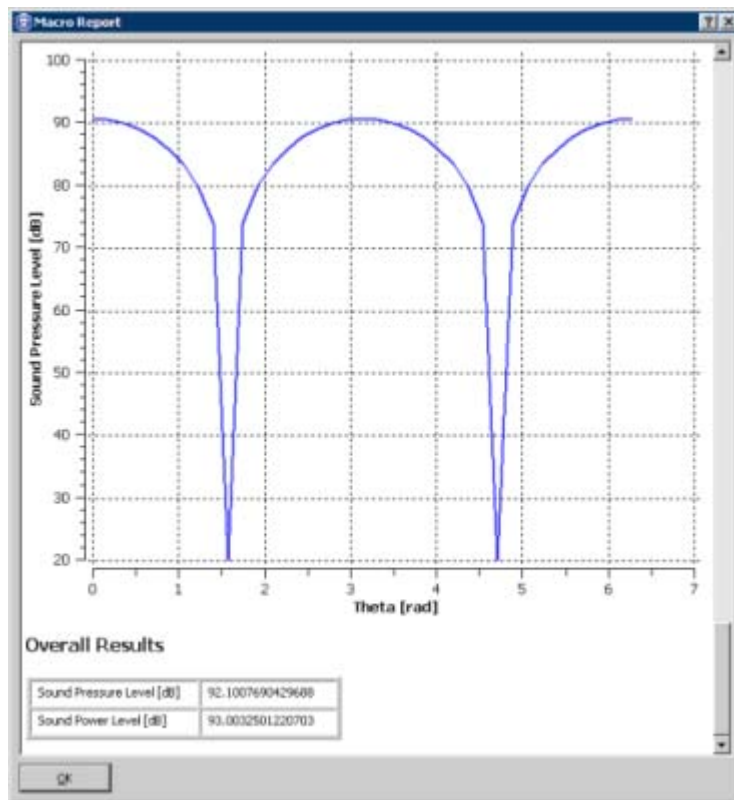
Fan Noise Macro Input

The Fan Noise macro calculates the tonal noise levels generated by a fan as heard at a specific location. To access the Fan Noise macro:

1. Load the .res file into CFD-Post.
2. Click on the **Calculators** tab.
3. In the **Macro** field, select **Fan Noise**.
4. In the Macro Calculator, specify the information described in [Fan Noise Macro \(p. 170\)](#).
5. When the Macro Calculator fields are filled in, click **Calculate**.

Fan Noise Output (Reports)

The Fan Noise macro outputs a report; to view it, click **View Report**. The report displays the input values, the sound pressure levels, the sound power levels, the directivity of harmonic 1, and the overall results. Here is a partial sample:



The turbo noise report will be created in your working directory as `turboNoise_report.html` along with the tables (`turboNoise_*.csv`) and graphics (`turboNoise_*.png`) included in the report. This enables you to reuse these elements in other documents, if required.

Fan Noise Examples

There are two ways to perform turbo noise calculations; you can have:

- A case with a single blade passage (the Lowson model is based on this)
- A case with a multiple blade passage, including a 360° case.

As the following examples show, the only necessary differences in the two cases are the settings for **Blade Selection** and the custom blade fields.

Fan Noise Macro Values	Single Blade Passage	Multiple Blade Passage
Domain	Fan Block	Fan Block
Blade Selection	Automatic	Custom
> Custom Blade		Blade
> Custom # of Blades		9
# of Harmonics	6	6
Observer (r)	1	1
Observer (theta)	0	0
Theta Sectors	36	36
Directivity Harm. #	1	1
Loading Coeff.	2.2	2.2
Acou. Ref. Pressure	2e-005	2e-005
Acou. Ref. Power	1e-011	1e-011
Sound Speed	340	340

To view the report, click **Calculate** and then **View Report**. The report will contain graphs and charts similar to the following:

Figure 10.2. Example Table and Chart of Sound Pressure Levels Created by the Fan Noise Macro

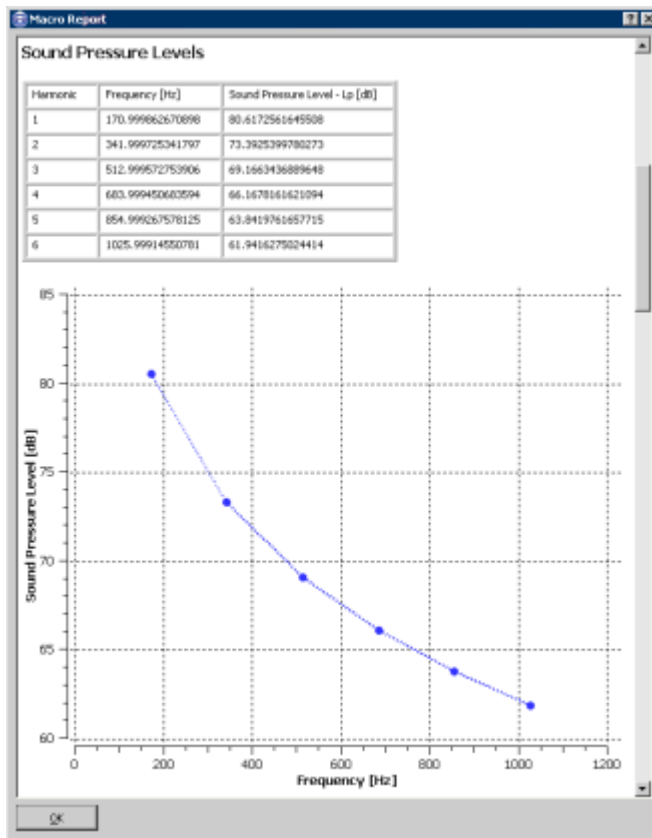
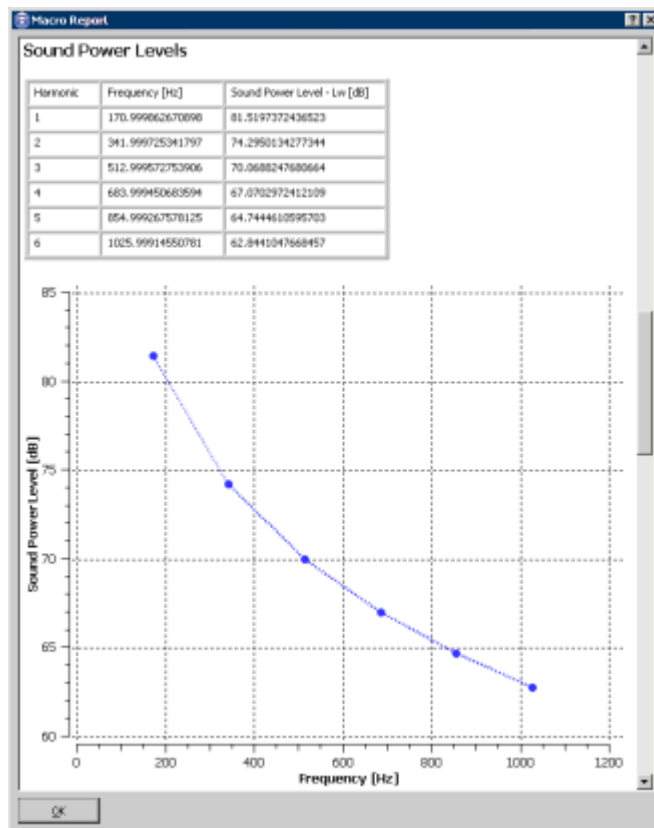


Figure 10.3. Example Table and Chart of Sound Power Levels Created by the Fan Noise Macro



Mesh Calculator

The **Mesh Calculator** (**Tools > Mesh Calculator**) offers a variety of tools to check the quality of your mesh. The results of each calculation are performed over all domains² and printed to the output window. Each calculated variable is also added to the list of available variables, which enables you to use them as a basis for creating new plots. It is important to note that these variables are evaluated on nodes rather than elements, based on the criteria described below.

You can select the following functions to calculate:

Maximum Face Angle

This calculates the largest face angle for all faces that touch a node. For each face, the angle between the two edges of the face that touch the node is calculated and the largest angle from all faces is returned for each node. Therefore, there is one maximum value for each node. The values that are reported are the smallest and largest of these maximums.

The maximum face angle can be considered to be a measure of skewness. For details, see [Mesh Visualization Advice](#) (p. 180).

Minimum Face Angle

This calculates the smallest face angle for all faces that touch a node. For each face, the angle between the two edges of the face that touch the node is calculated and the smallest angle from all faces is returned for each node. Therefore, there is one minimum value for each node. The values that are reported are the smallest and largest of these minimums. For details, see [Mesh Visualization Advice](#) (p. 180).

Edge Length Ratio

This is a ratio of the longest edge of a face divided by the shortest edge of the face. For each face:

²If multiple cases are loaded, the results of each calculation are performed over all domains in the specified cases.

$$\frac{\max(l_1, l_2)}{\min(l_1, l_2)} \quad (\text{Eq. 10.15})$$

is calculated for the two edges of the face that touch the node. The largest ratio is returned.

Connectivity Number

Connectivity number is the number of elements that touch a node.

Element Volume Ratio

Element Volume Ratio is defined as the ratio of the maximum volume of an element that touches a node, to the minimum volume of an element that touches a node. The value returned can be used as a measure of the local expansion factor.

Mesh Information

The Mesh Information option returns the number of nodes and elements in your volume mesh. It also lists the number of elements of each element type. As an example, the mesh for the following output contains two domains: one using hexahedral elements and the other containing tetrahedral elements. The domains were connected using a domain interface:

```
Number of Nodes: 71680
Number of Elements: 139862
  Tetrahedra: 75265
  Wedges: 31395
  Pyramids: 0
  Hexahedra: 33202
```

When you click **Calculate**, the result window displays the results of the specified calculation. If the calculated variable does not already exist, it will be created. This enables you to create plots of the calculated variable.

Note

When you compare the mesh information for an ANSYS FLUENT file in ANSYS FLUENT and in CFD-Post, the reported number of nodes (ANSYS FLUENT's "cells") will differ. In ANSYS FLUENT, each domain can have nodes at its boundaries that are not acknowledged as being shared with other domains. This causes ANSYS FLUENT mesh reports to contain duplicated nodes; however, the actual number of cells is the same as reported by CFD-Post.

Mesh Visualization Advice

The following table gives some guidelines for checking mesh quality. If there are elements which have mesh quality parameters greater or less than those listed, you may find problems with using the mesh in the CFX-Solver.

Element Type	Elements may be a problem if they have any of:
Tetrahedrons (4 nodes)	Edge Length Ratio > 100 Max Face Angle > 170° Min Face Angle < 10° Element Volume Ratio > 30 Connectivity Number > 50
Pyramids (5 nodes)	Edge Length Ratio > 100 Max Face Angle > 170° Min Face Angle < 10° Element Volume Ratio > 5
Prisms (6 nodes)	Edge Length Ratio > 100 Max Face Angle > 170° Min Face Angle < 10°

Element Type	Elements may be a problem if they have any of:
	Element Volume Ratio > 5 Connectivity Number > 12
Hexahedrons (8 nodes)	Edge Length Ratio > 100 Max/Min Edge Length > 100 Min Face Angle < 10° Element Volume Ratio > 5 Connectivity Number > 24

In many cases, the robustness of the CFX-Solver will not be adversely affected by high element volume ratios. However, you should be aware that accuracy will decrease as the element volume ratio increases. For optimal accuracy, you should try to keep the element volume ratio less than the value suggested in the above table.

Case Comparison

The **Compare Cases** command enables you to compare results from two distinct cases, or between two steps of a single case. The **Compare Cases** command is available in the **Tools** menu if:

- you have loaded two or more cases using the **Load Results File** dialog box option **Keep current cases loaded**, or
- you have loaded a single transient case (with results available for at least two time steps), or
- you have loaded a multi-configuration case, or a case with run history, using the **Load Results File** dialog box option **Load complete history as** (either as a single case or as separate cases), so that results for two or more steps are available through the timestep selector.

Selecting **Compare Cases** displays the **Case Comparison** details view.

The following options are available:

Case Comparison Active

Enables the Case Comparison function; the comparison occurs when you click **Apply**.

In **Case Comparison** mode:

- Difference variables are computed as the variable values from Case 1 minus the variable values from Case 2. The latter are interpolated onto the mesh from Case 1 before the subtraction. As a result, the difference variables are located on the mesh from Case 1.
To reverse the order of subtraction, swap the specifications for Case 1 and Case 2 in the **Case Comparison** details view.
- A **Difference** view is shown in a new view (in addition to the **Case 1** (`<case_name>`) view and the **Case 2** (`<case_name>`) view). In that view, differences are shown on the mesh from Case 1.
- Each difference variable is named by appending “.Difference” to the end of the variable name from which it was derived. For example, the difference variable for the variable Pressure is Pressure.Difference.
- The difference variables can be used anywhere that variables can normally be used. The function calculator and **Table Viewer** have special support for the difference variables, enabling you to easily see functions and tables (respectively) of difference values. In addition, a chart that is based on locators which exist in both Case 1 and Case 2 will have a “Difference” chart line. See [Example: Comparing Differences Between Two Files](#) (p. 154).
- CFD-Post refers to the cases as “Case 1” and “Case 2” rather than as the original case names (which are usually based on the results file name).

Case 1 and Case 2

Enables you to select the cases to be compared. If you want to compare two steps from within the same case (that is, two time steps from a transient case) then you should select the same case for both **Case 1** and **Case 2**. The timestep selector that is embedded into the **Case Comparison** details view then enables you to select

which steps you want to compare. In this circumstance, CFD-Post needs to load the results from the selected case a second time, so you will see a second case appearing in the tree view. After the comparison has been initialized, the steps used for the comparison can be changed either by using the embedded timestep selector on the **Case Comparison** details view, or by using the usual timestep selector (which now has separate entries for each of the two copies of the case being compared).

Options: Synchronize camera in displayed views

Causes changes in orientation of one view to be duplicated in the other. If the views are initially in different orientations, the first movement of any view will align all views to the same orientation.

Options: Use absolute value of Difference

Causes all values to be reported as positive numbers.

Note

- If you run a case comparison on a file that contains solver-generated difference variables (such as `Volume Porosity.Difference`), these variables will become unviewable when you enter case comparison mode. However, the variables will be viewable again if you reload the results file.
- When using expressions in case-comparison situations, the expression syntax is:
`function()@CASE: [1 | 2] .location`
 For example, `area()@CASE:2.myplane`
- Case comparison is supported only for general mode. As a result, case comparison initiated from the **Turbo** tab will revert to general mode.

Calculating Difference Variables

There are two ways of creating difference variables:

- You can use the CFX Interpolator.
- You can use CFD-Post in comparison mode.

In each case you can then view variables such as "<vector variable>.Difference" (such as `Velocity.Difference`) and "<scalar variable>.Difference" (such as `Temperature.Difference`). For a description of the general variable syntax, see [Quantitative CEL Functions in ANSYS CFX \(p. 143\) in the ANSYS CFX Reference Guide](#).

The magnitude of a difference variable "<vector variable>.Difference" is always calculated as:

$$\sqrt{(\text{<vec var>.Difference } X)^2 + (\text{<vec var>.Difference } Y)^2 + (\text{<vec var>.Difference } Z)^2} \quad (\text{Eq. 10.16})$$

This is *not* the difference of the vector magnitudes between file 1 and file 2.

If you plot a vector plot such as `Velocity.Difference`, it is obvious that a real vector is being plotted. However, if you plot "<vector variable>.Difference" in plots that use a scalar variable, how the difference variable is calculated is an issue. For example, suppose in one file you have a velocity vector (1, 0, 0), so the velocity magnitude is 1 [m/s], and in the second file you have a velocity vector of (-1, 0, 0), so the velocity magnitude is also 1 [m/s]. The vector variable `Velocity.Difference` variable is (2, 0, 0), and the scalar variable that CFD-Post calls "`Velocity.Difference`" is equal to the magnitude of this vector variable (that is, it is 2 [m/s]). You might expect `Velocity.Difference` to be equal to "velocity magnitude in file 2" - "velocity magnitude in file 1", which would give a value of 0 [m/s], but this is incorrect.

Command Editor

To start the **Command Editor**:

1. Select **Tools > Command Editor**. Alternatively, right-click any object that can be modified using the **Command Editor** and select **Edit in Command Editor**.
 - If you select **Tools > Command Editor**, the **Command Editor** opens and displays the current state regardless of any selection.
 - If the **Command Editor** dialog box has not been used previously, it will be blank.

- If the **Command Editor** dialog box has been used previously, it will contain CCL commands. If you do not want to edit the CCL that appears, click **Clear** to erase all content.
 - If you right-click an object and select **Edit in Command Editor**, the CCL definition of the specific object populates the **Command Editor** automatically. Modify or add parameters as required, then process the new object definition to apply the changes.
2. Click in the **Command Editor**.
 3. Prepare the content of the **Command Editor** by adding new content, modifying the existing content, or both. The types of content that may be prepared are CCL, action commands, and power syntax. Combinations of these types of content are allowed. For details, see:
 - [CFX Command Language \(CCL\) Syntax \(p. 127\)](#)
 - [Command Actions \(p. 249\)](#)
 - [Power Syntax in ANSYS CFX \(p. 259\)](#).

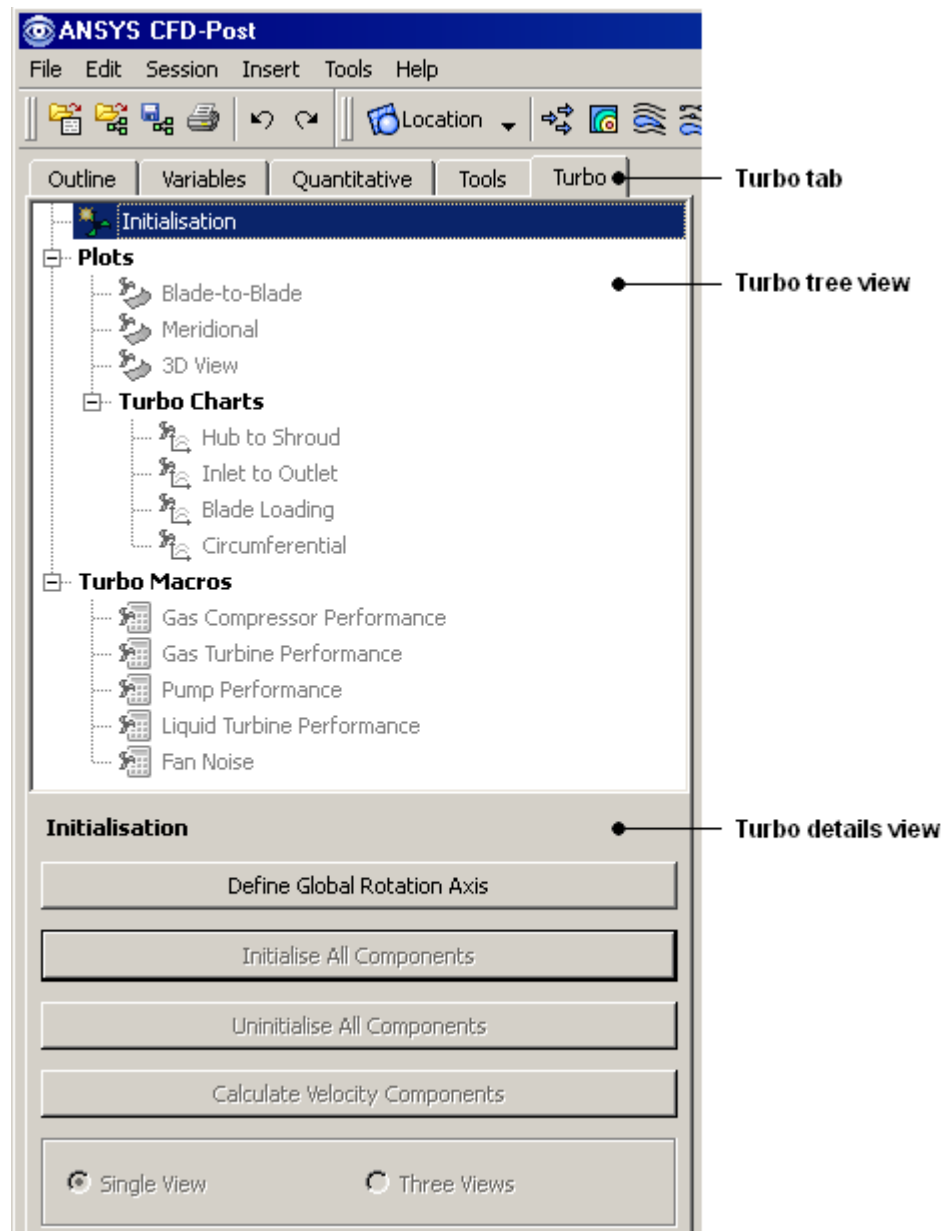
Right-click in the **Command Editor** to access basic editing functions. These functions include **Find**, which makes a search tool appear at the bottom of the **Command Editor** dialog box. Enter a search term and click either **Next** or **Previous** to search upwards or downwards from the insertion point or text selection. To hide the search tool, press **Esc**.

4. Click **Process**.

The contents are processed: CCL changes will affect CCL object definitions, actions will be carried out, and power syntax will be executed.

Chapter 11. Turbo Workspace

The **Turbo** workspace improves and speeds up post-processing for turbomachinery simulations. To access the **Turbo** workspace, click the **Turbo** tab. The two main parts of the **Turbo** workspace interface are the **Turbo** tree view and the **Turbo** details view.



This chapter describes:

- [Visual Representation of Initialization Status \(p. 186\)](#)
- [Define/Modify Global Rotation Axis \(p. 186\)](#)
- [Turbo Initialization \(p. 186\)](#)
- [Turbo View Shortcuts \(p. 190\)](#)
- [Turbo Surface \(p. 190\)](#)
- [Turbo Line \(p. 193\)](#)
- [Turbo Plots \(p. 194\)](#)
- [Turbo Macros \(p. 203\)](#)

- [Calculate Velocity Components \(p. 204\)](#)

Visual Representation of Initialization Status

Tip

You can copy the sample file `AxialIni_001.res` from the installation directory for your software (`<CFXROOT>/examples/`) to your working directory, then load that file into CFD-Post so that the descriptions in this chapter are easier to follow.

When in the **Turbo** workspace, a wireframe representation of each component appears in the viewer. The currently selected turbo component appears as a green wireframe. If it also happens to be initialized, it will be accompanied by a visual depiction of the background mesh, shown as a transparent green surface with white mesh lines.

The **Turbo** tree view also indicates which components are initialized and which are not; if the component is uninitialized, the symbol next to a component name is greyed out.

After entering the **Turbo** workspace and initializing the turbo components, you are ready to start using the turbo-specific features offered in the **Turbo** workspace.

Define/Modify Global Rotation Axis

The **Define/Modify Global Rotation Axis** button is found on the **Turbo** workspace's **Initialization** view. When you click that button, the **Define/Modify Global Rotation Axis** dialog box appears.

Exactly one axis of rotation method must be specified. The axis definition can come from the results file, or it can be specified manually as either a **Rotation Axis** (six Cartesian coordinates) or a **Principal Axis** (X, Y, or Z). Upon changing the axis definition, the axial, radial, and Theta coordinates (and their dependent objects and expressions) are automatically updated. For details, see [Theta \(p. 195\)](#).

Turbo Initialization

Before using the **Turbo** workspace features, the components of the loaded case (such as rotor, stator, etc.¹) need to be initialized. Initialization causes, among other things, span, m' , a (axial) and r (radial) and Theta coordinates to be generated for each component.

The topics in this section include:

- [Requirements for Initialization \(p. 186\)](#)
- [Initialize All Components \(p. 187\)](#)
- [Uninitializing Components \(p. 187\)](#)
- [Individual Component Initialization \(Advanced Feature\) \(p. 187\)](#)
- [Details View for Individual Component Initialization \(p. 187\)](#)

Requirements for Initialization

Initialization of a turbo component requires the following:

- Input for calculating a background mesh. For details, see [Purpose of Background Mesh \(p. 188\)](#).
- Specification of the number of instances of each turbo component (such as stator, rotor, etc.) required to represent the full geometry around the rotation axis.

Note

CFD-Post can initialize turbo space only for domains that are enclosed with inlet, outlet, hub, and shroud regions. For more complex geometries, you must set up the problem to isolate the region of interest into a separate domain that has these regions.

¹Available components depend on the turbo setup in the preprocessor. There is a minimum of one component available for each domain.

Initialize All Components

To access the initialization options, double-click **Initialization** in the **Turbo** tree view. The **Initialize All Components** button that appears is used to set the region and instancing information for each of the domains contained in your results file.

Correctly defined turbo spaces, as described in [Requirements for Initialization \(p. 186\)](#), can be automatically initialized. To automatically initialize all components using the default (best guess) region assignment, you can do one of the following:

- Choose to auto-initialize all components when a message prompts you upon entering the **Turbo** workspace for the first time (after loading a case).
- Right-click a component in the **Turbo** tree view and select **Initialize All**.
- Use a CCL instruction; for details, see [Initializing all Turbo Components \(p. 257\)](#).

Uninitializing Components

After a turbo component has been initialized, it is possible to change or even remove its initialization settings. An uninitialized component still has axial, radial and Theta coordinates generated for it, as long as the rotation axis is defined.

The **Uninitialize All Components** button is accessible in the **Turbo** details view after double-clicking **Initialization** in the **Turbo** workspace. A right-click menu associated with a turbo component in the **Turbo** tree view allows uninitialization for that component, or for all components.

Uninitializing all turbo components can be followed by initializing only the components that will be studied. Keeping the number of initialized components to a minimum saves computer memory. It also saves computational effort when generating plots that span multiple components. For example, having only one component initialized in a domain with many components restricts calculations and plots to just the initialized component.

Uninitialization does not cause graphic objects to be deleted. A graphic object that disappears due to the uninitialization of a turbo component reappears if the component is initialized.

Individual Component Initialization (Advanced Feature)

To manually initialize or modify the initialization of a turbo component, double-click the component in the **Turbo** tree view. A details view for the component appears with two tabs: **Definition** and **Instancing**.

1. Select the boundary names that correspond to the required turbo regions. To select multiple regions, click the icon to the right of the drop-down list and hold the **Ctrl** key while selecting the regions.
2. In the **Background Mesh** frame for each of the hub, shroud, inlet, and outlet curves, choose to specify each to be **From Turbo Region** or **From Line** (that is, from a predefined line). If **From Line** is chosen, choose the line locator.
3. Set the mesh **Method** to either **Linear** or **Quasi Orthogonal**.
4. Click **Apply** (or **Initialize**, for subsequent initializations).

Additional information on Individual Component Initialization is available in the [Details View for Individual Component Initialization \(p. 187\)](#) section; for details, see:

- [Turbo Regions Frame \(p. 188\)](#)
- [Background Mesh Frame \(p. 188\)](#)

Details View for Individual Component Initialization

The Individual Component Initialization view contains the following tabs:

- [Definition Tab \(p. 187\)](#)
- [Instancing Tab \(p. 189\)](#)

Definition Tab

The **Definition** tab is used to specify:

- The hub, shroud, inlet, and outlet curves and other regions for a turbo component (such as a rotor or stator). For details, see [Turbo Regions Frame \(p. 188\)](#).
- The parameters controlling the component's associated background mesh.
The background mesh is a mesh generated on a constant-Theta projection of the passage, used to define spanwise and meridional coordinates for the 3D geometry. For details, see [Background Mesh Frame \(p. 188\)](#).

Turbo Regions Frame

The **Turbo Regions** frame is used to assign 2D regions to the Hub, Shroud, Blade, Inlet, Outlet, and Periodic regions of a turbo component. These regions are not always required, but when provided, may be used in the following ways:

- The Blade region specification is used to enable macros and plots that deal with blades (for example, a blade loading macro).
- The intersections of the Hub, Shroud, Inlet and Outlet regions with Periodic 1 may be used in order to generate internal polylines that are then collapsed in the Theta direction to form the boundaries of the background mesh. Alternatively, or if any of these intersections are not possible, polylines/lines may be specified explicitly in the **Background Mesh** frame. For details, see [Background Mesh Frame \(p. 188\)](#).

In the special case of a turbo component that wraps 360 degrees around the rotation axis, there may be no periodic regions available. In this case, you may do one of the following:

1. Select the **360 Case Without Periodics** check box.
2. Specify the hub, shroud, inlet, and outlet regions. Create a rectangularly-bounded slice plane, using the point-and-normal method, such that it intersects the turbo component on only one side of the rotation axis. In this case, it may be helpful to temporarily set the plane type to *Sample* so that you can see the entire plane. After the plane is in the correct position, set the type to *Slice*. Finally, specify this slice plane as Periodic 1. You do not need to set Periodic 2.
3. Specify polylines for the hub, shroud, inlet, and outlet in the **Background Mesh** frame (described next).

Background Mesh Frame

Purpose of Background Mesh

In order to calculate **Streamwise Location** (m') and **Span** coordinates for a turbo component, a separate 2D mesh is created as an intermediate step. The mesh, here referred to as a *background mesh*, is formed by taking the 3D passage boundaries (hub, shroud, inlet, outlet) and collapsing them in the Theta direction, forming a 2D passage outline on an axial-radial plane. The outline is then filled in with a mesh consisting of lines of constant span and meridional coordinate. The resulting mesh is then used to associate **Streamwise Location** and **Span** coordinates with any 3D position in the passage.

Requirements for Setting Up a Background Mesh

The background mesh frame requires you to specify how the hub, shroud, inlet, and outlet curves will be obtained. The two available options are:

- **From Turbo Region**
When **From Turbo Region** is specified for a particular curve, that curve is automatically extracted by intersecting the corresponding turbo region (specified in the **Turbo Regions** frame) with the specified **Periodic 1** region (also specified in the **Turbo Regions** frame).
- **From Line**
When **From Line** is specified for a particular curve, you must provide a polyline/line locator for that curve. You must use the latter method for every curve that cannot be derived by the first method (for example, because one or more **Turbo Regions** are not specified).

A line or polyline used to generate a background mesh must follow the entire surface it represents (along the component). One way in which a polyline can be created is by using the intersection between a bounded plane (such as a slice plane or a turbo surface of constant Theta) and the appropriate surface (for example, the hub surface). Before the polyline is used for initialization, is transformed by adjusting all Theta coordinates to the same value.

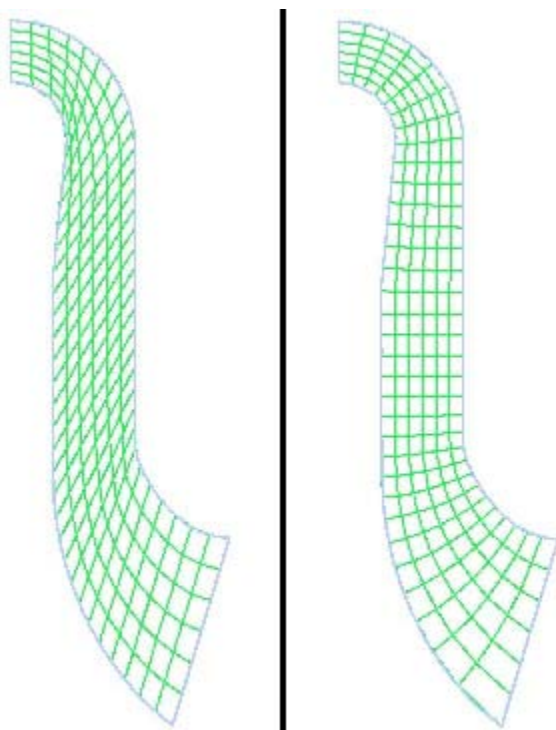
The Theta coordinates of the polyline, therefore, have no effect; polylines obtained by intersection with a plane need not use a constant-Theta plane. If you cannot form the polyline easily, you can save pieces of the polyline to a series of files, use an editor to consolidate the parts, and then reload the edited file. For details, see:

- [Line Command \(p. 94\)](#)
- [Polyline Command \(p. 110\)](#).

Types of Background Mesh

Two methods are offered for creation of the background mesh:

- Linear
- Quasi Orthogonal



The figure on the left shows a background mesh (for clarity, **Density** was set to 200) using the **Linear** method, while the figure on the right shows the mesh using the **Orthogonal** (default) method. As can be seen from pictures, the **Quasi Orthogonal** method offers a higher-quality meridional space representation, especially in highly curved passages.

Density of the Background Mesh

The density of the background mesh influences the accuracy of the representation of the meridional space and, therefore, the accuracy of the nonlinear coordinate transformations. The default offered for **Density** should be sufficient in most cases.

Instancing Tab

The Instancing tab for a turbo component is shown below:

1. Set the **# of Copies**.
2. **Apply Rotation** should be used when the number of copies is more than 1, otherwise, all copies will be drawn in the same location.
3. **Axis Definition from File**: By default, the Axis definition can be automatically determined. To set your own, disable the toggle and enter a **Principal** or **Rotation Axis** (specified using a **From/To Line**).
4. For **Angle From**, set the rotation angle using one of two methods: Instances in 360 degrees or a specified value.

- Between **# of Passages** and number of passages per component (**Passages/Com**), the number of components per 360 degrees is determined.
- Translation is set by entering three vector components.
- By selecting **Apply Reflection/Mirroring**, a reflection is set using an existing plane. You may need to create a plane before you apply the reflection.

The instancing information is used to display multiple instances of the geometry. For example, if there are two components, with the instancing information for component 1 specifying one copy, and component 2 specifying ten copies, a turbo surface of constant span that covers both components will show, by default, one copy of the portion generated for component 1, and ten copies of the portion generated for component 2.

The instancing specified for a component applies to objects (or parts thereof) generated over the component, in order for this instancing information to apply to a graphic object:

- At least part of the graphic object must be generated using data from the component (that is, there must be an association between the graphic object and the component).
- The graphic object must have **Apply Instancing Transform** selected and **Transform** set to an **Instancing Transform** that has **Instancing Info From Domain** selected.

This is because, in the current version of CFD-Post, the instancing information for a component is actually the instancing information for the component's domain. Consequently, changes to the domain instancing, or instancing for any other component in the domain, also alters the instancing information for the component.

Additional information on instance transforms is available in [Instance Transform Command \(p. 134\)](#).

Turbo View Shortcuts

The following table shows commands that are specific to the **Turbo** tree view. To access these commands, right-click the appropriate elements in the **Turbo** tree view.

For a list of the other commands that appear in the **Turbo** tree view (and in most tree views), see [Common Tree View Shortcuts \(p. 14\)](#).

Command	Description
Initialize	Initializes the selected turbo components. For details, see Individual Component Initialization (Advanced Feature) (p. 187) .
Uninitialize	Uninitializes the selected turbo components. For details, see Uninitializing Components (p. 187) .
Initialize All	Initializes all turbo components. For details, see Initialize All Components (p. 187) .
Uninitialize All	Uninitializes all turbo components. For details, see Uninitializing Components (p. 187) .
Show in Separate Window	Displays the selected plot or chart in its own window.
Promote to General Mode	Copies the selected plot object and any required supporting objects (for example, a line locator) to the Outline workspace. This would allow, for example, the selected plot to be included in a report.

Turbo Surface

Turbo surfaces are graphic objects that can be viewed and used as locators, just like other graphic objects. To create a turbo surface, select **Insert > Location > Turbo Surface** from the menu bar. After you enter a name in the new **Turbo Surface** dialog box and click **OK**, the details view for the turbo surface will appear.

Note

Blade Aligned Turbo Surfaces can fail due to the following limitations:

- The extraction of leading and trailing edges of the blade is sensitive to tip clearance and to the curvature of the edges.

- The normalization of coordinates is sensitive to blade extend comparing to inlet and outlet extend (that is, when the edges are too close to inlet/outlet).

You can always use the Streamwise Location coordinate when the quality of the blade aligned coordinates are in doubt.

Turbo Surface: Geometry

Options available for **Definition** are:

- Constant Span
- Constant Streamwise Location
- Constant Blade Aligned
- Constant Blade Aligned Linear
- Constant Theta
- Cone

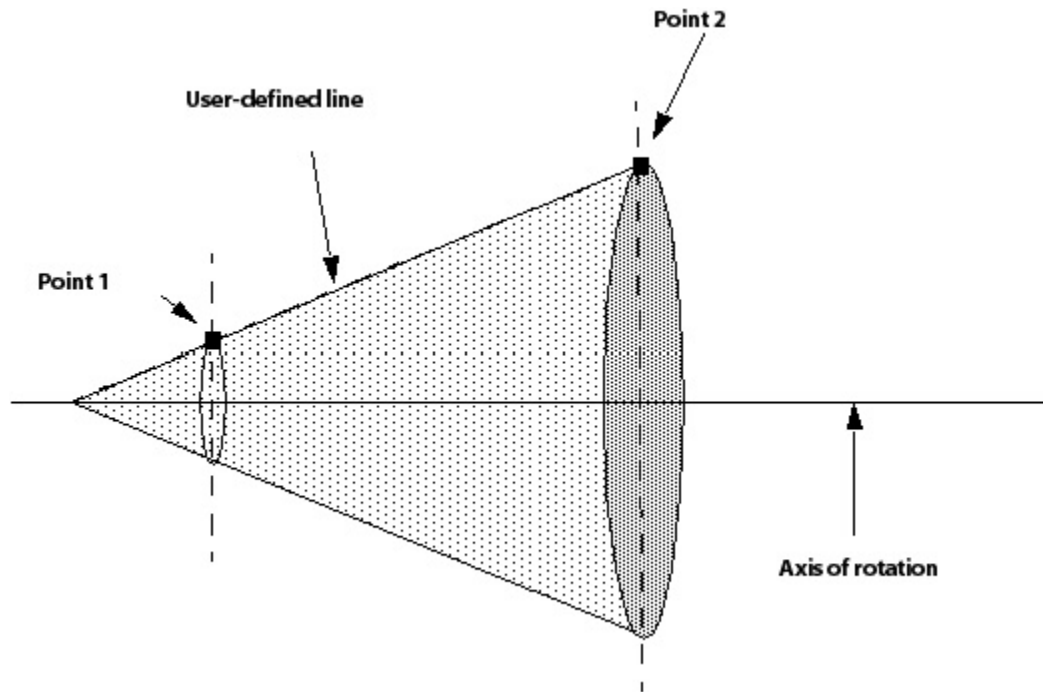
The Constant Span, Constant Streamwise Location, and Constant Theta options are similar to planes in that they can be bounded and have Slice or Sample types. For details, see [Type \(p. 192\)](#).

Domains

See [Selecting Domains \(p. 16\)](#).

Definition

- Constant Span creates a surface at a fractional span value between the hub and shroud. For details, see [Span \(p. 195\)](#).
- Constant Streamwise Location creates a surface at a fractional streamwise distance between the inlet and outlet. For details, see [Streamwise Location \(p. 195\)](#).
- Constant Blade Aligned create surfaces that is aligned with the leading and trailing edges of the blade. If the blade is curved, the surfaces will also be curved.
- Constant Blade Aligned Linear create surfaces that is aligned with the leading and trailing edges of the blade. If the blade is curved, the surfaces will be flat and aligned to run through the middle of the curves.
- Constant Theta creates a surface at a specific Theta value. For details, see [Theta \(p. 195\)](#).
- Cone uses the two supplied points to create a line. The cone is created where the user-defined line intersects the axis of rotation and Point 2:




The user-defined line is then rotated about the axis of origin to create the cone. If the line is parallel to the axis of rotation, a cylinder is created. If the line is normal to the axis of rotation, a disc is created.

The line can be described by Cartesian or cylindrical components. When entering cylindrical coordinates, only the axial distance and radius are required. The points can be entered or picked directly from the viewer.

Note

Constant Theta and Cone methods are available even before turbo initialization has been performed because these methods do not depend on span or streamwise coordinates.

Bounds

The available types of **Bounds** for the **Turbo Surface** to be created can be seen by clicking  next to the Type box.



- When None is selected, the **Turbo Surface** cuts through a complete cross-section of each domain specified in the **Domains** list. The **Turbo Surface** is bounded only by the limits of the domain.
- Using Rectangular, you can enter the maximum and minimum value for the two dimensions on the **Turbo Surface**. The **Turbo Surface** is undefined in areas where the rectangle extends outside of the domains specified in the **Domains** list.

The **Invert Surface Bounds** check box reverses the effect of the surface bound. The surface is defined only in regions outside the bounding constraints.

Type

You can set the **Type** to either *Slice* or *Sample*.

Slice extends the **Turbo Surface** in all directions until it reaches the edge of the domain. Points on the **Turbo Surface** correspond to points where the **Turbo Surface** intersects an edge of the mesh. As a result, the number of points in a slice **Turbo Surface** is indirectly proportional to the mesh spacing.

Sample creates the **Turbo Surface** with rectangular bounds. The density of points on the **Turbo Surface** corresponds to the size of the bounds for your **Turbo Surface** in each of the **Turbo Surface** directions, and the value in the **Samples** box for each of the two directions that describe the **Turbo Surface**. You can type in the value in the **Samples** box, increase or decrease the value by 1 by clicking  or  respectively, or use the embedded slider

(which has a maximum value of 998 and a minimum value of 2). A sample **Turbo Surface** is a set of evenly-spaced points which are independent of the mesh spacing.

Turbo Surface: Common Tabs

You can adjust the other Turbo Surface settings on tabs that are common to other features in CFD-Post:

Turbo Surface: Color

To change the color settings, click the **Color** tab. For details, see [Color Details Tab \(p. 16\)](#).

Turbo Surface: Render

To change the rendering settings, click the **Render** tab. For details, see [Render Details Tab \(p. 18\)](#).

Turbo Surface: View

To change the view settings, click the **View** tab. For details, see [View Details Tab \(p. 21\)](#).

Turbo Line

Turbo lines are graphic objects that can be viewed and used as locators, just like other graphic objects. To create a turbo line, select **Insert > Location > Turbo Line** from the menu bar. After entering a name in the new **Turbo Line** dialog box, the details view for the turbo line will appear.

Turbo Line: Geometry

- See [Selecting Domains \(p. 16\)](#).
- Options available for Method are:
 - Inlet to Outlet
 - Hub to Shroud
 - Circumferential
- See [Definition \(p. 193\)](#).
- 1. Theta: Define the first and second locations which describe the line.
Options depend on the method chosen. For details, see [Definition \(p. 193\)](#).
- 2. Set the **Span** direction.
For details, see [Span \(p. 195\)](#).
- 3. Select the number of points along the line per component with the **Samples/Comp** setting.
- 4. Set **Bounds > Type** to choose how to limit the line.
For details, see [Bounds \(p. 194\)](#).

Domains

See [Selecting Domains \(p. 16\)](#).

Definition

- **Inlet to Outlet** creates a line at specific span and Theta value, over a range of streamwise values. The number of samples is required. The number of points along the line will correspond to the value you enter in the **Samples/Comp** box. The sample line is a set of evenly-spaced sampling points which are independent of the mesh spacing. For details, see:
 - [Span \(p. 195\)](#)
 - [Theta \(p. 195\)](#)

- **Hub to Shroud** creates a line at a specific Theta value. For details, see [Theta \(p. 195\)](#). The method for creating a line in this way is the same as for the locator line in a hub to shroud turbo chart. For details, see [Hub to Shroud \(p. 197\)](#).
- **Circumferential** creates a line at specific streamwise and span values, over a range of Theta values. The number of samples is required. The number of points along the line will correspond to the value you enter in the **Samples** box. The sample line is a set of evenly-spaced sampling points which are independent of the mesh spacing. For details, see:
 - [Streamwise Location \(p. 195\)](#)
 - [Span \(p. 195\)](#).

Bounds

- When **None** is selected, the **Turbo Line** is restricted to only the parameters specified in the Definition section of the form. The **Turbo Line** is not bounded by the limits of the domain if the conditions you specify describe locations outside of the domain.
- When **End Points** is selected, you can define the ends of the **Turbo Line** by entering the maximum and minimum for the dimension making up the line. The **Turbo Line** is visible but will be colored with an undefined color in areas where the line extends outside of the domains specified in the **Domains** list.

Turbo Line: Common Tabs

You can adjust the other Turbo Surface settings on tabs that are common to other features in CFD-Post:

Turbo Line: Color

To change the color settings, click the **Color** tab. For details, see [Color Details Tab \(p. 16\)](#).

Turbo Line: Render

To change the rendering settings, click the **Render** tab. For details, see [Render Details Tab \(p. 18\)](#).

Turbo Line: View

To change the view settings, click the **View** tab. For details, see [View Details Tab \(p. 21\)](#).

Turbo Plots

The following topics will be discussed:

- [Introduction to Turbo Plots \(p. 194\)](#)
- [Initialization Three Views \(p. 195\)](#)
- [Blade-to-Blade Object \(p. 196\)](#)
- [Meridional Object \(p. 196\)](#)
- [3D View Object \(p. 197\)](#)
- [Turbo Charts \(p. 197\)](#)

Introduction to Turbo Plots

Each turbo plot appears in the Turbo tree view under **Plots**, and can be edited.

Show Faces/Show Mesh Lines

Double-click on **3D View** and choose to view the faces (**Show Faces**) or the edges of the mesh elements (**Show Mesh Lines**) by enabling the appropriate toggles.

Instancing

The instancing information has already been entered during the initialization phase. You can opt to show instancing for the plots in each domain by changing the **# of Copies**. For details, see [Instancing Tab \(p. 189\)](#).

Turbo Measurements

Span

Span defines the dimensionless distance (between 0 and 1) from the hub to the shroud. For example, if the distance between the hub and shroud in a straight duct is 0.1 m, a span of 0.9 would describe a location 0.09 m from the hub and 0.01 m away from the shroud.

Streamwise Location

Streamwise location is the dimensionless distance from the inlet to the outlet. It ranges from 0 to 1 for the first component, 1 to 2 for the second, and so on. For example, in a single domain case, if the distance between the inlet and the outlet in a straight duct is 1m, a streamwise location of 0.4 would describe a location 0.4 m from the inlet and 0.6 m away from the outlet. If the same duct were the second component in a multi-component case, the same location would then be expressed as a streamwise location of 1.4.

Theta

Theta is the angular coordinate measured about the axis of rotation following the right-hand rule. When looking along the positive direction of the axis of rotation, theta is increasing in the clockwise direction. Note that the theta coordinate in CFD-Post does not increase over 360°, even for spiral geometries that wrap to more than 360°.

The Theta variable is intentionally generated by CFD-Post to have the following two properties:

- A minimum Theta value of zero (at the inlet).
- Continuously increasing values of Theta independent of the total blade wrap. This is particularly useful for high-wrap blades.

Because of these properties, the Theta variable generated in CFD-Post is most likely different than that of a user-defined expression based on the Cartesian coordinates.

Advanced: Position of Zero Theta

The position of zero Theta ($\text{Theta} = 0^\circ$) relative to the global coordinate system depends on the loaded case. For geometries that define a partial machine (not full 360°), zero Theta is at the geometry point with the lowest angle following the right-hand rule. For full 360° geometries, zero Theta is generally at an arbitrary position. You can specify zero Theta via an environment variable. For details, see [Setting CFD-Post Operation Through Environment Variables \(p. 5\)](#).

Initialization Three Views

If you double-click **Initialization** in the Turbo tree view, the Initialization editor appears and a **Three Views** toggle will be available. Selecting this toggle causes the viewer to show three viewports that have the following views:

- A **3D View**. This is the same as the standard viewer, with 3D manipulation available using the rotate, translate and zoom functions.
- A **Blade-to-Blade** 2D view, which is described in [Blade-to-Blade Object \(p. 196\)](#). The horizontal axis shows streamwise location and the vertical axis shows Theta. The 2D view allows translation, zoom and rotation around the axis normal to the blade-to-blade view. Other rotations are not possible.
- A **Meridional** 2D view, which is described in [Meridional Object \(p. 196\)](#). The horizontal axis shows axial distance and the vertical axis shows the radius. The view will allow the same transformations as the blade-to-blade view, with rotation possible around the axis normal to the meridional view.

The three views listed above are also listed in the **Turbo** tree view under **Plots**.

These default objects can be edited in the same way as other objects created in CFD-Post. For details, see [Turbo Surface \(p. 190\)](#).

You can copy these two objects into the **Object** tree view by right-clicking each one and selecting **Promote to General Mode**.

Blade-to-Blade Object

The Blade-to-Blade object is used to view plots on a surface of constant span. The surface is displayed in the Cartesian (X-Y-Z) and Blade to Blade views.

1. Select the **Domain(s)**.
To select more than one domain, click the multiple select icon and pick the entities.
2. Choose the fractional **Span** (0 to 1) where the plot is located. The **Plot Type** can be one of the following:
 - [Color \(p. 196\)](#)
 - [Contour \(p. 196\)](#)
 - [Vector \(p. 196\)](#)
 - [Stream \(p. 196\)](#)
3. Select a color if **Color By Variable** is not chosen as the **Plot Type**.
For details, see [Instancing \(p. 195\)](#).

Span

Set the fractional distance between the hub and shroud. For details, see [Span \(p. 195\)](#).

Angular Shift

The **Angular Shift** parameter moves the blade-to-blade plot along the Theta coordinate. This is useful to control the point of splitting in high wrap turbo cases. It does not affect the data; this is purely a rendering feature.

Plot Type

Color

The **Color** option displays variable values using a color legend. It requires the specification of a variable, range and the option of using hybrid or conservative values. For details, see:

- [Mode: Variable and Use Plot Variable \(p. 16\)](#)
- [Instancing \(p. 195\)](#).

Contour

Contour lines are drawn on the location described by the surface plot. Additional information on the option is available in [Contour Command \(p. 119\)](#).

Vector

A vector plot is created on the location described by the surface plot. For details, see [Vector Command \(p. 116\)](#).

Stream

A plot of streamlines are drawn on the location described by the surface plot. For details, see [Streamline Command \(p. 121\)](#).

Meridional Object

The **Meridional** object is used to view plots on an axial-radial plane. A surface of constant Theta at 0 degrees is created. The surface is displayed in the **Cartesian (X-Y-Z)** and **Meridional (A-R)** viewports.

1. Select the **Domain(s)**.
To select more than one domain, click the multiple select icon and pick the entities.

2. Choose the number of **Stream Samples** and **Span Samples**.
3. Choose from: Outline, Color, Contour, or Vector plot types.

In order to obtain values for variables on the meridional surface, circumferential averaging is used. The types of circumferential averaging are:

- Length
For details, see [Circumferential Averaging by Length \(p. 201\)](#).
- Area
For details, see [Circumferential Averaging by Area: Hub to Shroud Turbo Chart \(p. 201\)](#).
- Mass
For details, see [Circumferential Averaging by Mass Flow: Hub to Shroud Turbo Chart \(p. 201\)](#).

Area averaging is default.

Toggles are available to show the following:

- Blade wireframe
- Sample mesh
- Chart location lines

3D View Object

3D View is used to draw regions of the turbo assembly for visualization purposes. It is not intended to be the basis for quantitative calculations. Select the regions that you want to draw.

After creating the blade-to-blade object (select the **Three Views** toggle in the Initialization object), you can view the blade-to-blade object in the **3D View** object by setting the appropriate option in the **3D View** object.

Note that you can view chart location lines in the **3D View** object by setting the appropriate option in the **3D View** object.

Turbo Charts

The following turbo charts are created by default:

- [Hub to Shroud \(p. 197\)](#)
- [Inlet to Outlet \(p. 202\)](#)
- [Blade Loading \(p. 203\)](#)
- [Circumferential \(p. 203\)](#)

To see a chart, double-click it in the **Turbo** tree view.

Type

Hub to Shroud

Mode

Set **Mode** to one of the following options:

- Two Points Linear
The Two Points Linear option causes the hub-to-shroud line to be a straight line, specified by two points: one on the hub and one on the shroud. The **Point Type** setting (described below) specifies the coordinate system for interpreting the specified points.
- Blade Aligned Linear
The Blade Aligned Linear option causes the hub-to-shroud line to be specified by a curve of constant *Linear BA Streamwise Location* coordinate. For details, see [Linear BA Streamwise Location Coordinates \(p. 201\)](#).

- Blade Aligned

The **Blade Aligned** option causes the hub-to-shroud line to be specified by a curve of constant *BA Streamwise Location* coordinate. For details, see [BA Streamwise Location Coordinates \(p. 202\)](#).

- Streamwise Location

The **Streamwise Location** option causes the hub-to-shroud line to be specified by a curve of constant streamwise coordinate. Here, the streamwise coordinate system is derived from a “background mesh”. For details, see [Background Mesh Frame \(p. 188\)](#).

Note

Blade Aligned coordinates may not always be available, depending on the case geometry. In particular, if the blade tip clearance is large or uneven between the leading and trailing edges, CFD-Post may not be able to detect the blade edge lines. In this case you will not be able to use Blade Aligned coordinates in turbo surface or turbo chart specification.

Point Type

The **Point Type** setting is applicable when **Mode** is set to **Two Points Linear**. It controls the coordinate system for defining the specified hub and shroud point coordinates. The options for **Point Type** are:

- AR

When the **AR** option is selected, the hub and shroud points are specified in AR (axial, radial) coordinates.

- Blade Aligned Linear

When the **Blade Aligned Linear** option is selected, the hub and shroud points are specified, each by a single *Linear BA Streamwise Location* coordinate. For details, see [Linear BA Streamwise Location Coordinates \(p. 201\)](#).

- Streamwise Location

When the **Streamwise Location** option is selected, the hub and shroud points are specified, each by a single streamwise coordinate. Here, the streamwise coordinate system is derived from a “background mesh”. For details, see [Background Mesh Frame \(p. 188\)](#).

Samples

The **Samples** setting controls the number of sampling points between the hub and shroud.

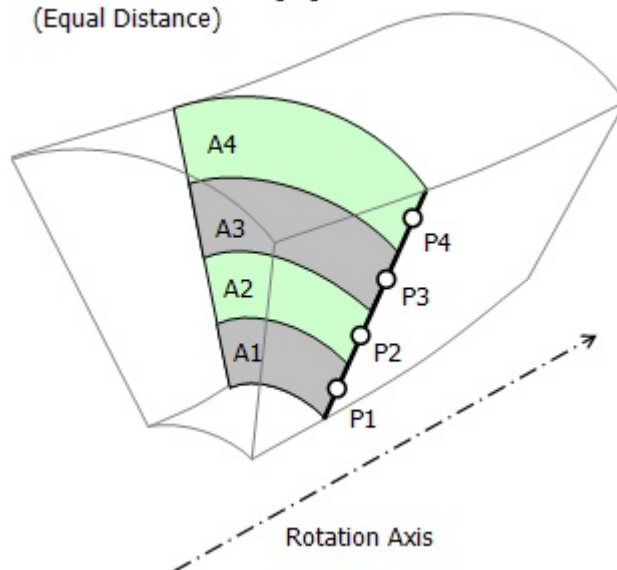
Distribution

The **Distribution** setting controls the method used to distribute sampling points from hub to shroud (at the same streamwise coordinate).

Set **Distribution** to one of:

- Equal Distance

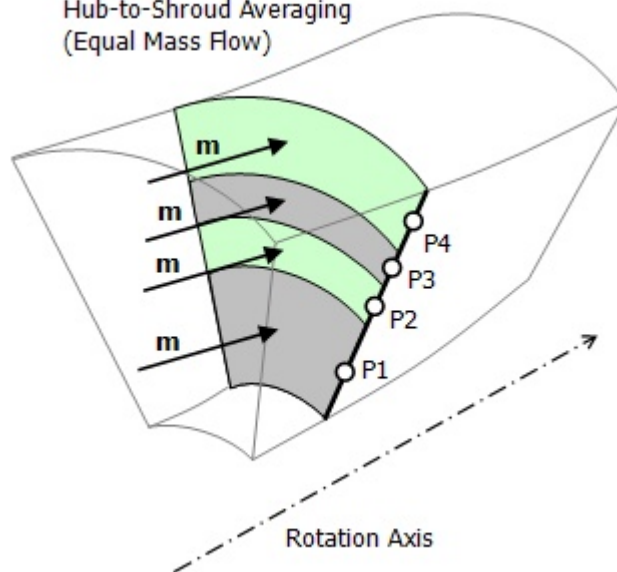
Hub-to-Shroud Averaging (Equal Distance)



The `Equal Distance` option (default) causes the sampling points to be distributed at uniform distances along a hub-to-shroud path. For circumferential averaging purposes, contiguous circular bands are internally constructed, one for each sampling point, concentric about the rotation axis, width-centered (in the spanwise direction) about each sampling point, each band having the *same width* or spanwise extent.

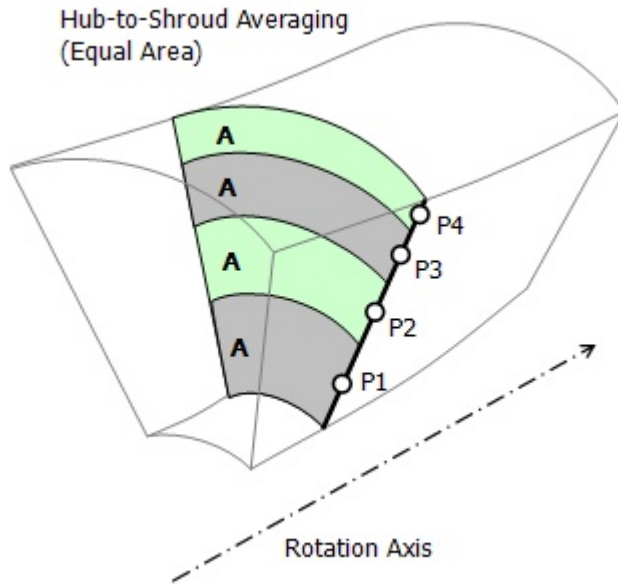
- `Equal Mass Flow`

Hub-to-Shroud Averaging (Equal Mass Flow)



The `Equal Mass Flow` option causes the sampling points to be distributed along a hub-to-shroud path such that contiguous circular bands can be internally constructed, one for each sampling point, concentric about the rotation axis, width-centered (in the spanwise direction) about each sampling point, *with an equal mass flow through each band* (except possibly the first and last bands). See **Include Boundary Points**, below.

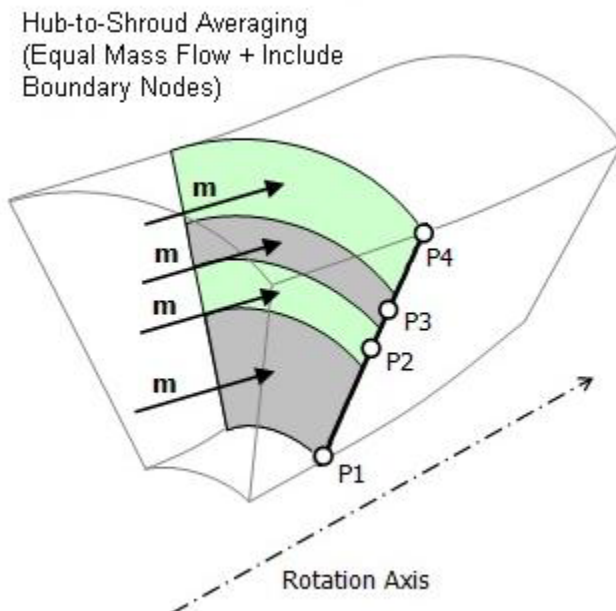
- `Equal Area`



The **Equal Area** option causes the sampling points to be distributed along a hub-to-shroud path such that contiguous circular bands can be internally constructed, one for each sampling point, concentric about the rotation axis, width-centered (in the spanwise direction) about each sampling point, *with an equal area for each band* (except possibly the first and last bands). See **Include Boundary Points**, below.

When either **Equal Mass Flow** or **Equal Area** is set, there is a check box, **Include Boundary Points**, which, if selected, will shift all bands (and consequently the sampling points) by “half” the band width (in the spanwise direction) so that sampling points appear on the hub and shroud. (See [Figure 11.1, “Sampling Point Distribution with Include Boundary Nodes Option”](#) (p. 200).) The first and last bands are then half the size of the other bands in terms of the particular measure used in the initial construction: distance, mass flow, or area.

Figure 11.1. Sampling Point Distribution with Include Boundary Nodes Option



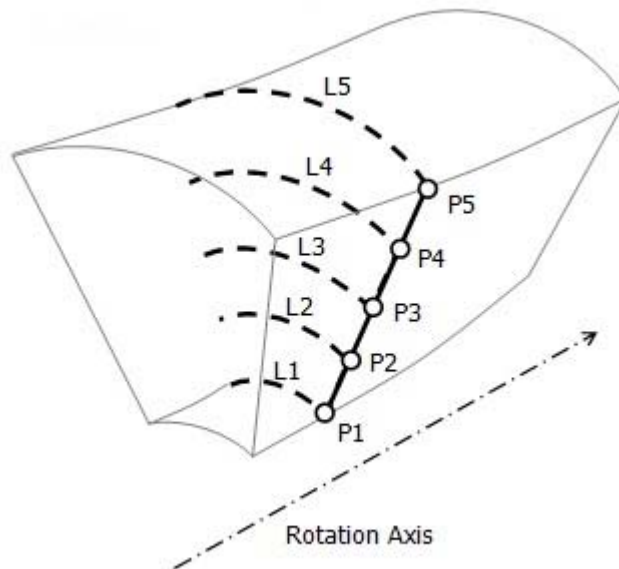
X/Y Variable

Choose X and Y variables for the chart axes from the list.

Circumferential Averaging by Length

When the **Circ. Average** setting is set to **Length**, circumferential averaging of values at a sampling point is carried out internally by forming a circular arc, centered about the rotation axis, passing through the sampling point. Values are interpolated to n equally-spaced locations along the arc, using values from nearby nodes, where n is a number that is inversely proportional to the mesh length scale, and limited by the **Max. Samples** setting. The n values are then averaged in order to obtain a single, circumferentially-averaged value for the sampling point.

Figure 11.2. Circumferential Averaging by Length



Circumferential Averaging by Area: Hub to Shroud Turbo Chart

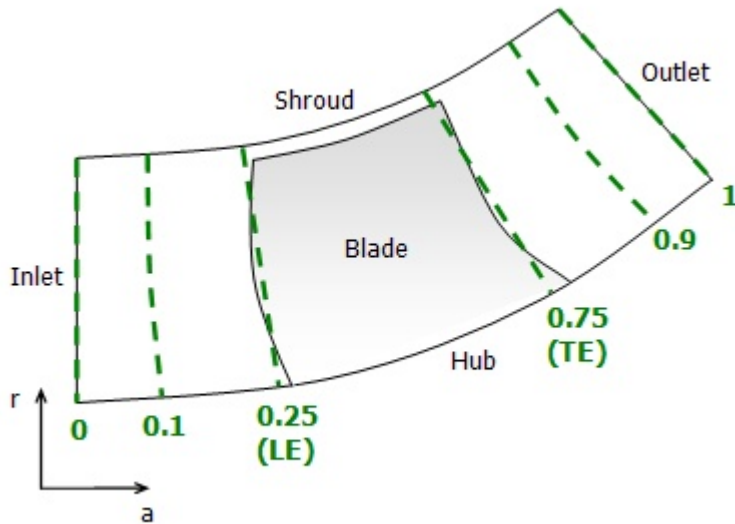
When the **Circ. Average** setting is set to **Area**, a variable value at each sampling point is calculated as an area average over the corresponding circular band that was internally constructed as part of the process of distributing the sampling points. For details, see [Distribution \(p. 198\)](#).

Circumferential Averaging by Mass Flow: Hub to Shroud Turbo Chart

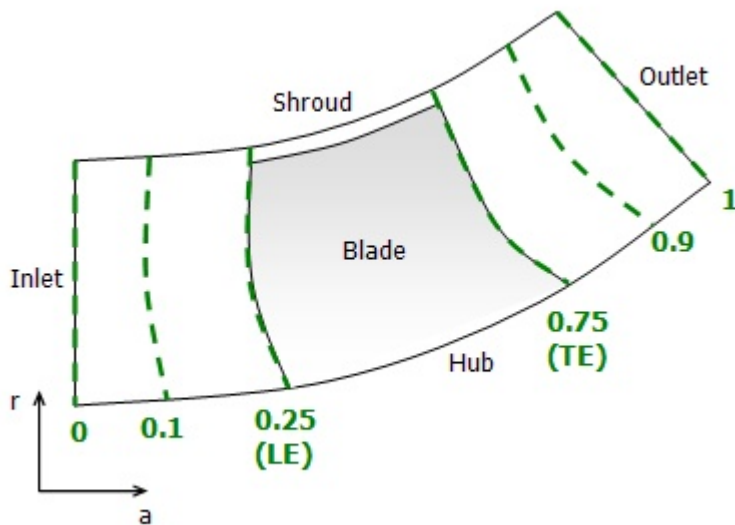
When the **Circ. Average** setting is set to **Mass**, a variable value at each sampling point is calculated as a mass flow average over the corresponding circular band that was internally constructed as part of the process of distributing the sampling points. For details, see [Distribution \(p. 198\)](#).

Linear BA Streamwise Location Coordinates

The **Linear BA Streamwise Linear** coordinates are defined as 0 (zero) at the inlet, 0.25 at a straight line that approximates the blade leading edge, 0.75 at a similar line for the trailing edge, and 1.0 at the outlet, adding 1.0 for each successive turbomachinery component downstream of the first. Dashed lines in [Figure 11.3, “Blade Aligned Linear Coordinates” \(p. 202\)](#) show constant values of **Linear BA Streamwise Linear** coordinate.

Figure 11.3. Blade Aligned Linear Coordinates**BA Streamwise Location Coordinates**

The **BA Streamwise Location** coordinates are defined as 0 (zero) at the inlet, 0.25 at the blade leading edge, 0.75 at the trailing edge, and 1.0 at the outlet, adding 1.0 for each successive turbomachinery component downstream of the first.

Figure 11.4. Blade Aligned Coordinates**Inlet to Outlet**

The distance between sampling points between the inlet and outlet is controlled by the number you enter in the **Samples** box. Choose X and Y variables for the chart axes from the list.

Circumferential Averaging by Length: Inlet to Outlet Turbo Chart

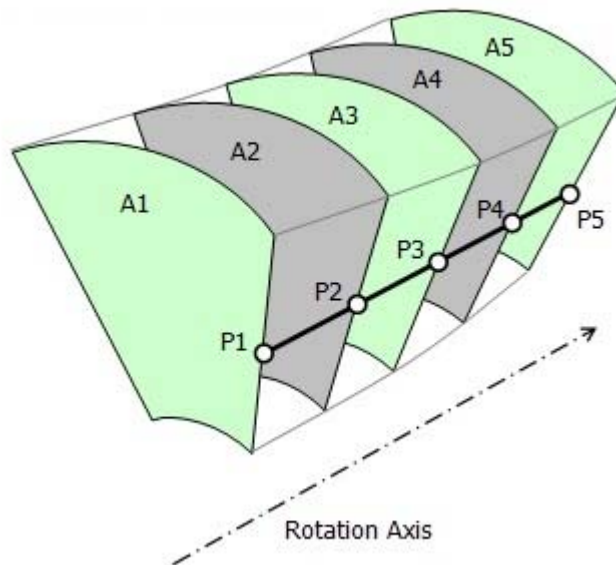
When the **Circ. Average** setting is set to **Length**, circumferential averaging of values is carried out internally by creating arcs through sampling about the rotation axis. Values are interpolated to n equally-spaced locations along the arc, using values from nearby nodes, where n is a number that is inversely proportional to the mesh length scale,

and limited by the Max. Samples setting. The n values are then averaged in order to obtain a single, circumferentially-averaged value for the sampling point.

Circumferential Averaging by Area or Mass: Inlet to Outlet Turbo Chart

When performing area average or mass-flow average calculations, surfaces of constant-streamwise coordinate are used to carry out the averaging. Each surface passes through its associated sampling point, as shown in [Figure 11.5, “Inlet to Outlet Sample Points”](#) (p. 203).

Figure 11.5. Inlet to Outlet Sample Points



Blade Loading

The **Blade Loading** feature plots pressure (or another chosen variable) on the blade at a given spanwise location. A polyline is created at the given spanwise location.

A special variable, `Streamwise` (0-1) is available as the `X Variable` used in blade loading plots. This is a streamwise coordinate that follows the blade surface; it can be used as a substitute for the axial coordinate (for example, `X`) or the variable `Chart Count`. The streamwise coordinate is based on the meridional coordinate, and is normalized so that it ranges from 0 at the leading edge to 1 at the trailing edge of the blade.

Circumferential

Select a streamwise and spanwise location and a number of sampling points.

Note

The Theta extents of the chart line are set to the Theta extents of the domain. For this reason, some of the sample points may fall outside the domain. To see the circumferential chart line, edit the `Plots > 3D View` object and turn on `Show chart location lines`.

Turbo Macros

Select the macro of choice from the **Turbo** tree view.

Note

Turbo initialization automatically sets up the performance macros in such a way that you have to define only a limited number of parameters. For details, see:

- [Gas Compressor Performance Macro](#) (p. 168).

- [Gas Turbine Performance Macro \(p. 169\)](#)
- [Liquid Pump Performance Macro \(p. 169\)](#)
- [Liquid Turbine Performance Macro \(p. 169\)](#)
- [Fan Noise Macro \(p. 170\)](#).

Calculate Velocity Components

The **Calculate Velocity Components** button on the `Initialization` object can be used to calculate velocity component (and other) variables pertinent to turbo simulations. These variables are listed in [Table 11.1, “Generated Variables” \(p. 205\)](#) and illustrated in [Figure 11.6, “Axial, Radial, Circumferential, and Meridional Velocity Components” \(p. 206\)](#), [Figure 11.7, “Velocity Components in Meridional Plane” \(p. 207\)](#), [Figure 11.8, “Streamwise, Spanwise, Circumferential, and Blade-to-Blade Velocity Components” \(p. 208\)](#), [Figure 11.9, “Velocity Components in Blade-To-Blade Plane” \(p. 209\)](#), and [Figure 11.10, “Velocity Flow Angle Sign in Each Quadrant on the Blade-to-Blade Plane” \(p. 210\)](#). The relationship between the velocity components is described in [Equation 11.1 \(p. 206\)](#), [Equation 11.2 \(p. 208\)](#), and [Equation 11.3 \(p. 209\)](#).

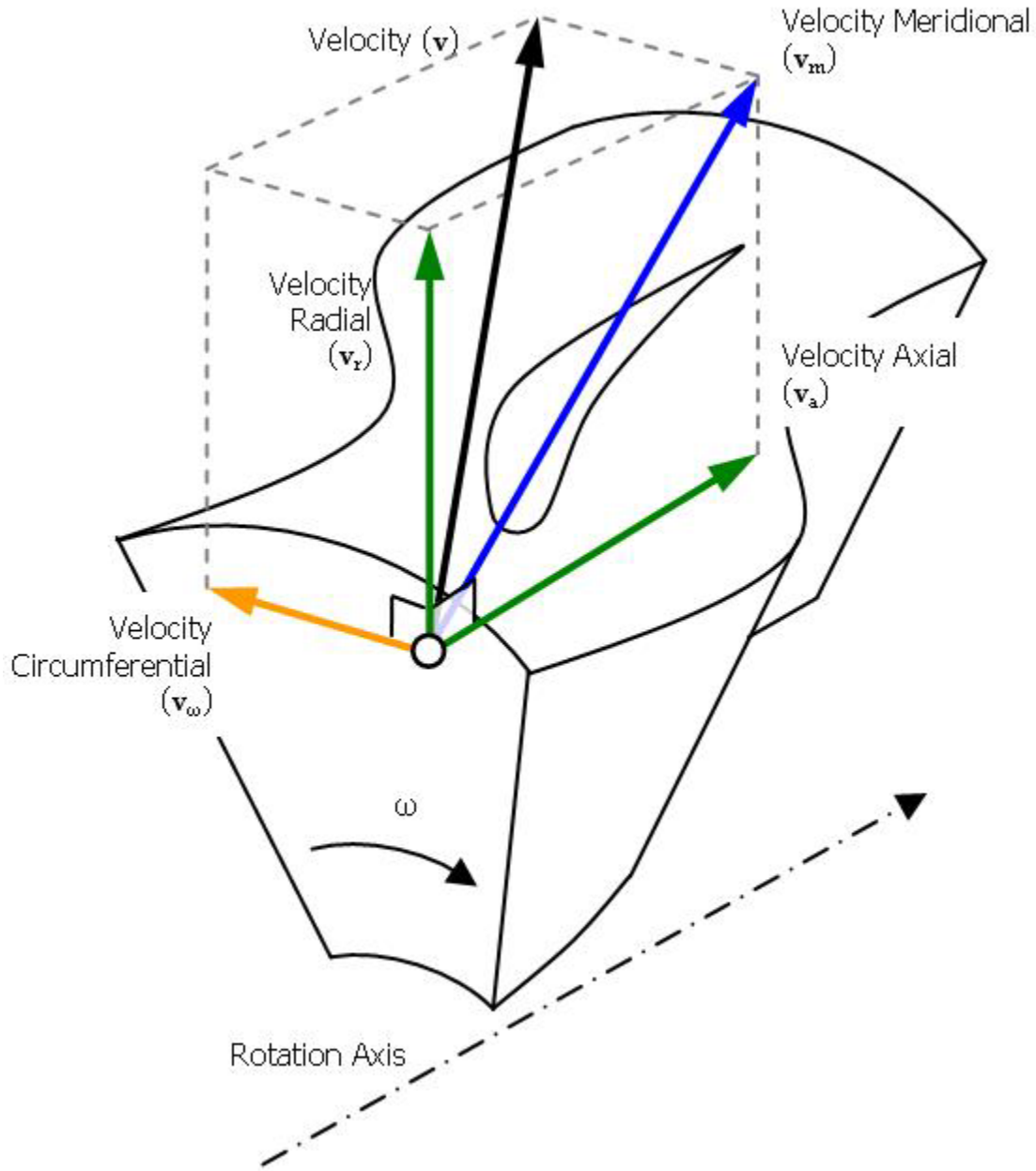
Note

To get velocity units for tip speed derived from `R` and `Omega` quantities, you can divide the expression by 1 [rad] to eliminate the angle units from the expression. For example, use:

```
tipVel = Radius * omega / 1 [rad]
```


Table 11.1. Generated Variables

Variable Name	Type	Description
Velocity Axial	Scalar	The velocity component in the axial direction. It is positive when the velocity is in the direction of increasing axial coordinate. For details, see Figure 11.6, “Axial, Radial, Circumferential, and Meridional Velocity Components” (p. 206) and Figure 11.7, “Velocity Components in Meridional Plane” (p. 207).
Velocity Radial	Scalar	The velocity component in the radial direction. It is positive when the velocity is in the direction of increasing radial coordinate. For details, see Figure 11.6, “Axial, Radial, Circumferential, and Meridional Velocity Components” (p. 206) and Figure 11.7, “Velocity Components in Meridional Plane” (p. 207).
Velocity Circumferential	Scalar	The velocity component in the Theta direction. It is positive when the velocity is in the direction of increasing Theta (for details, see Theta (p. 195)). For details, see Figure 11.6, “Axial, Radial, Circumferential, and Meridional Velocity Components” (p. 206).
Velocity Spanwise	Scalar	The velocity component in the spanwise direction. It is positive when the velocity is in the direction of increasing spanwise coordinate. For details, see Figure 11.7, “Velocity Components in Meridional Plane” (p. 207) and Figure 11.8, “Streamwise, Spanwise, Circumferential, and Blade-to-Blade Velocity Components” (p. 208).
Velocity Streamwise	Scalar	The velocity component in the streamwise direction. It is positive when the velocity is in the direction of increasing streamwise coordinate. For details, see Figure 11.7, “Velocity Components in Meridional Plane” (p. 207), Figure 11.8, “Streamwise, Spanwise, Circumferential, and Blade-to-Blade Velocity Components” (p. 208), and Figure 11.9, “Velocity Components in Blade-To-Blade Plane” (p. 209).
Velocity Meridional	Vector	The vector sum of the axial and radial vector components of velocity. It lies in the meridional plane. For details, see Figure 11.6, “Axial, Radial, Circumferential, and Meridional Velocity Components” (p. 206), Figure 11.7, “Velocity Components in Meridional Plane” (p. 207), and Equation 11.1 (p. 206).
Velocity Blade-to-Blade	Vector	The vector sum of the circumferential and streamwise vector components of velocity. It lies in the blade-to-blade plane. For details, see Figure 11.8, “Streamwise, Spanwise, Circumferential, and Blade-to-Blade Velocity Components” (p. 208), Figure 11.9, “Velocity Components in Blade-To-Blade Plane” (p. 209), Figure 11.10, “Velocity Flow Angle Sign in Each Quadrant on the Blade-to-Blade Plane” (p. 210), and Equation 11.2 (p. 208).
Velocity Flow Angle	Scalar	The angle between the blade-to-blade and circumferential velocity vector components. For details, see Figure 11.10, “Velocity Flow Angle Sign in Each Quadrant on the Blade-to-Blade Plane” (p. 210).

Figure 11.6. Axial, Radial, Circumferential, and Meridional Velocity Components

The velocity in the meridional plane can be represented by axial and radial components or streamwise and spanwise components:

$$\begin{aligned} \mathbf{v}_m &= \mathbf{v}_a + \mathbf{v}_r \\ &= \mathbf{v}_{st} + \mathbf{v}_s \end{aligned} \quad (\text{Eq. 11.1})$$

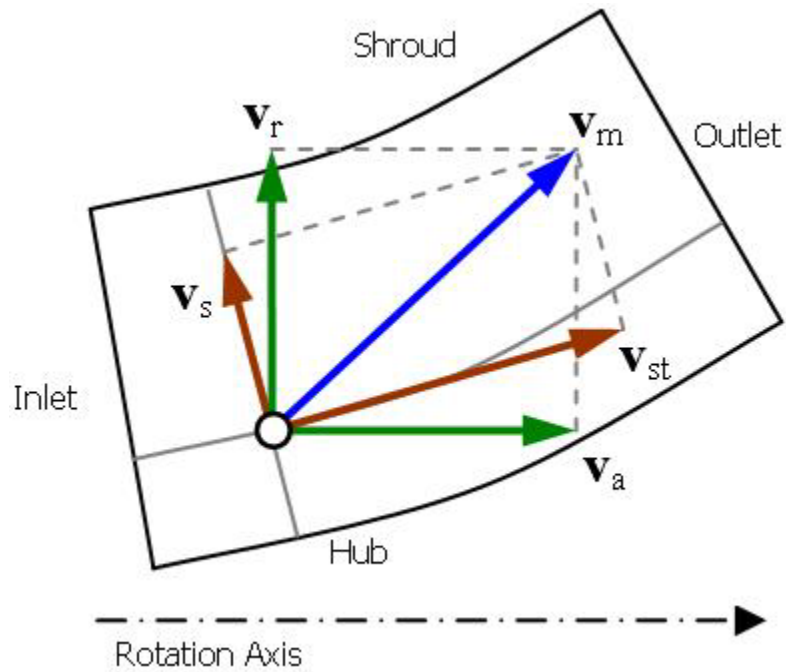
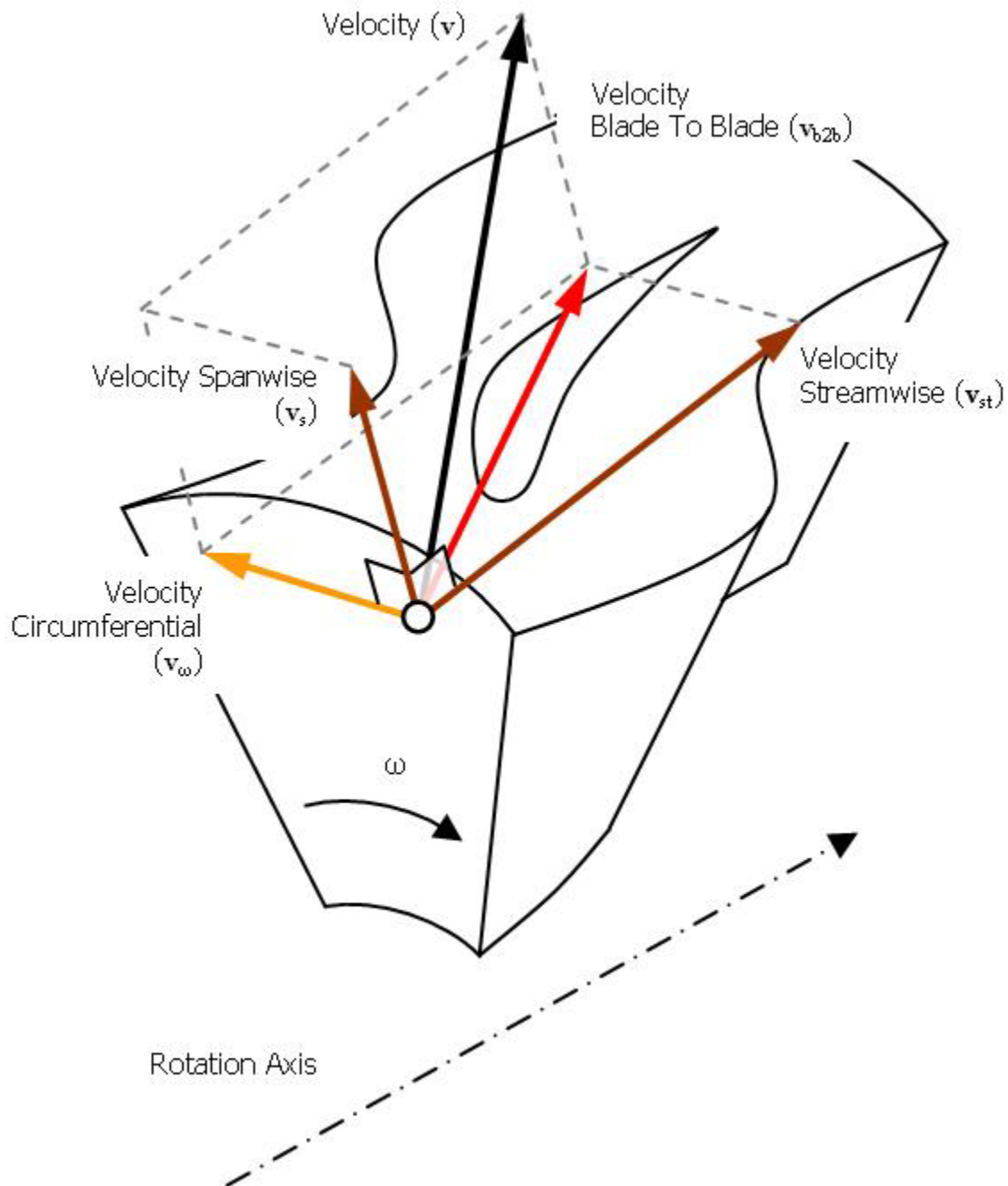
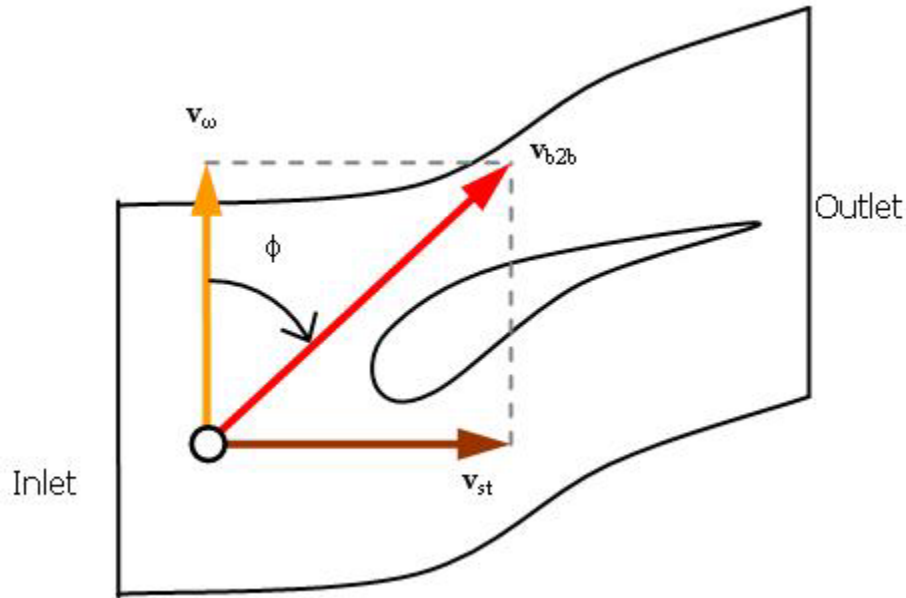
Figure 11.7. Velocity Components in Meridional Plane

Figure 11.8. Streamwise, Spanwise, Circumferential, and Blade-to-Blade Velocity Components



The velocity in the blade-to-blade plane can be represented by streamwise and circumferential components:

$$\mathbf{v}_{b2b} = \mathbf{v}_{st} + \mathbf{v}_{\omega} \quad (\text{Eq. 11.2})$$

Figure 11.9. Velocity Components in Blade-To-Blade Plane

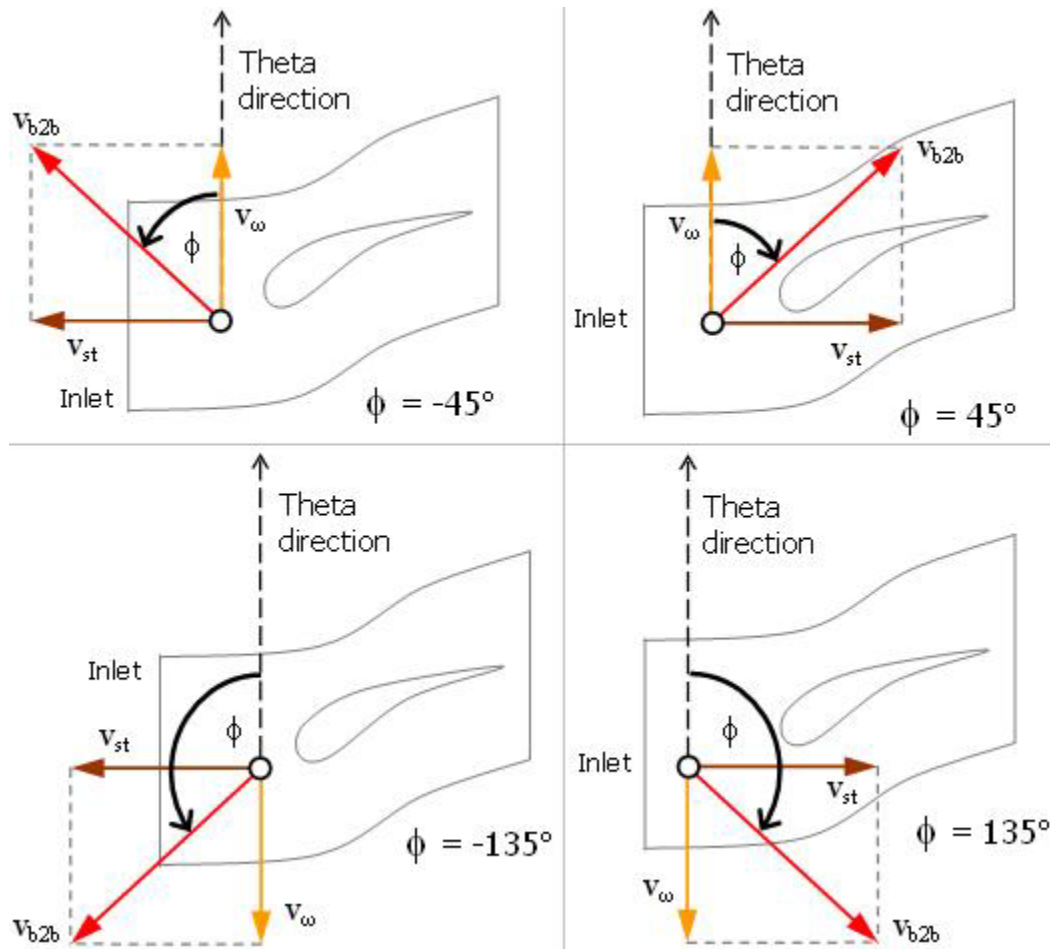
The velocity components are related as follows:

$$\begin{aligned}
 \mathbf{v} &= \mathbf{v}_a + \mathbf{v}_r + \mathbf{v}_\omega \\
 &= \mathbf{v}_{st} + \mathbf{v}_s + \mathbf{v}_\omega \\
 &= \mathbf{v}_m + \mathbf{v}_\omega \\
 &= \mathbf{v}_{b2b} + \mathbf{v}_s
 \end{aligned}
 \tag{Eq. 11.3}$$

Axial, radial and meridional velocities are not calculated for **Velocity in Stn. Frame** because these components are not different from the regular **Velocity** components.

Information on calculating velocity components using CCL is available. For details, see [Calculating Velocity Components](#) (p. 257).

The range of **Velocity Flow Angle** is from -180° to $+180^\circ$. Four examples are shown in [Figure 11.10, “Velocity Flow Angle Sign in Each Quadrant on the Blade-to-Blade Plane”](#) (p. 210).

Figure 11.10. Velocity Flow Angle Sign in Each Quadrant on the Blade-to-Blade Plane

Calculating Cylindrical Velocity Components for Non-turbo Cases

If you want to calculate cylindrical velocity components for non-turbo cases, such as for swirling flows in axi-symmetric geometries and mixing tank calculations, you can use the **Calculate Velocity Components** button on the **Turbo** workspace:

1. Load a results file for an axi-symmetric simulation. (You can load a copy of `<CFXROOT>/examples/StaticMixer_001.res` to work through this example.)
2. Select the **Turbo** tab to open the Turbo workspace. A dialog asks if you want to auto-initialize all components, but as this is unnecessary click **No**.
3. On the **Turbo** workspace's **Initialization** area, click **Define Global Rotational Axis**.
4. In the **Define Global Rotational Axis** dialog, select the appropriate axis and click **OK**. (For the static mixer example, set **Axis** to Z.)
5. In the **Initialization** area, click **Calculate Velocity Components**. New variables such as **Velocity Circumferential** become available. (You can see these new variables in the **Variables** workspace.)
6. Create a plane so that you can display a cylindrical velocity component:
 - a. From the menu bar, select **Insert > Location > Plane**. In the dialog that appears, accept the default name and click **OK**.
 - b. In the **Details** view for Plane 1 on the **Geometry** tab, ensure that **Method** is YZ Plane.
 - c. On the **Color** tab, set **Mode** to Variable and **Variable** to Velocity Circumferential.
 - d. Click **Apply**. The plane is colored to show the velocity at each point.

- e. Right-click on the viewer background and select **Predefined Camera > View Towards +X** so that the plane is easier to see.

Important

Not all axi-symmetric cases can have velocity components calculated in this way. In particular, cases that involve particles (such as smoke) will fail.

Chapter 12. CFX Command Language (CCL) in CFD-Post

[CFX Expression Language \(CEL\) \(p. 133\)](#) is an interpreted, declarative language that has been developed to enable CFX users to enhance their simulations without recourse to writing and linking separate external Fortran routines. You can use CEL expressions anywhere a value is required for input in ANSYS CFX.

The CFX Command Language (CCL) is the internal communication and command language of CFD-Post. It is a simple language that can be used to create objects or perform actions in the post-processor. All CCL statements can be classified into one of three categories:

- Object and parameter definitions, which are described in [Object Creation and Deletion \(p. 213\)](#).
- CCL actions, which are commands that perform a specific task (such as reading a session file) and which are described in [Command Actions \(p. 249\)](#).
- Power Syntax programming, which uses the Perl programming language to allow loops, logic, and custom macros (subroutines). Power Syntax enables you to embed Perl commands into CCL to achieve powerful quantitative Post-processing. For details, see [Power Syntax in ANSYS CFX \(p. 259\)](#).

State files and session files contain object definitions in CCL. In addition, session files can also contain CCL action commands. You can view and modify the CCL in these files by using a text editor, and you can use CCL to create your own session and state files to read into CFD-Post.

Tip

Advanced users can interact with CFD-Post directly by entering CCL in the **Command Editor** dialog box (see [Command Editor \(p. 182\)](#)), or by running CFD-Post in Line Interface mode (see [Line Interface Mode \(p. 273\)](#)).

For more information, see:

- [CFX Command Language \(CCL\) Syntax \(p. 127\)](#)
- [Object Creation and Deletion \(p. 213\)](#)

Object Creation and Deletion

You can create objects in CFD-Post by entering the CCL definition of the object into the **Command Editor** dialog box, or by reading the object definition from a session or state file. The object will be created and any associated graphics shown in the viewer.

You can modify an existing object by entering the object definition with the modified parameter settings into the **Command Editor** dialog box. Only those parameters that are to be changed need to be entered. All other parameters will remain unchanged.

There may be a significant degree of interaction between objects in CFD-Post. For example, a vector plot may depend on the location of an underlying plane, or an isosurface may depend on the definition of a CEL expression. If changes to one object affect other objects, the other objects will be updated automatically.

To delete an object, type `>delete <ObjectName>`. If you delete an object that is used by other objects, warnings will result, but the object will still be deleted.

Chapter 13. CFX Expression Language (CEL) in CFD-Post

This chapter provides information that is specific to CFX Expression Language (CEL) use in CFD-Post. For details on the CFX Expression Language, see [CFX Expression Language \(CEL\) \(p. 133\)](#). A list of variables available for use in CEL expression is available in [Variables and Predefined Expressions Available in CEL Expressions \(p. 175\)](#).

CEL Variables In CFD-Post

Within CFD-Post, you can:

- Create new expressions.
- Set any numeric parameter in a CFD-Post object based on an expression (and the object will update if the expression result changes).
- Create user-defined variables from expressions.
- Directly use the post-processor quantitative functions in an expression.
- Specify units as part of an expression.
- Use the variables x, y, and z in general CEL expressions. Additionally, you can use user-defined coordinate frames with the CEL functions. For details, see [Quantitative CEL Functions in ANSYS CFX \(p. 143\)](#).

However, you *cannot*:

- Automatically read CEL expressions from pre-processing set up in the results file.
- Use a user-defined coordinate frame as part of a general CEL expression, for example, `radius = sqrt(x_myAxis^2 + y_myAxis^2)` is not valid.

All expressions in the post-processor are defined in the EXPRESSIONS singleton object (which is also a sub-object of LIBRARY:CEL). Each expression is a simple `name = expression` statement within that object. New expressions are added by defining new parameters within the expressions object (the EXPRESSIONS object is special in that it does not have a predefined list of valid parameters).

Important

Because Power Syntax uses Perl mathematical operators, you should exercise caution when combining CEL with Power Syntax expressions. For example, in CEL, 2^2 is represented as `2^2`, but in Perl, would be written `2**2`. If you are unsure about the validity of an operator in Perl, please consult a Perl reference guide.

Variables Created by CFD-Post

CFD-Post derives the following variables from the results file; you can use them in expressions or as plot variables:

Table 13.1. Variables Created by CFD-Post

Name	Description
Area	This is meaningful only for surface locators (user surface, plane, isosurface, boundary). The value at each node is equal to the sum of sector areas associated with the node (a sector area is the portion of area of a face touching a node that can be associated with that node). There is a function to sum this variable over a 2D locator to obtain the area of the locator; for details, see area (p. 150) in the ANSYS CFX Reference Guide .
Force	There is a function for calculating force; for details, see force (p. 154) in the ANSYS CFX Reference Guide .
Length	This is meaningful only for polyline and line objects. The value on each line node is equal to the sum of halves of the two line segments joined at the node. There is a function to sum this variable over a line locator to obtain the length of the locator; for details, see length (p. 156) in the ANSYS CFX Reference Guide .
Mass Flow	There is a function for calculating mass flow; for details, see massFlow (p. 157) in the ANSYS CFX Reference Guide .
Normal	This is meaningful only for surface locators (user surface, plane, isosurface, boundary). It is a vector variable defining the surface unit normal at each node in the locator.
Volume	This is defined only on volume locators (volume, domain, subdomain). The value at each node is equal to the sum of the sector volumes associated with the node (a sector volume is the portion of volume of an element touching a node that can be associated with that node). There is a function to sum this variable over a 3D locator to obtain the volume of the locator; for details, see volume (p. 163) in the ANSYS CFX Reference Guide .