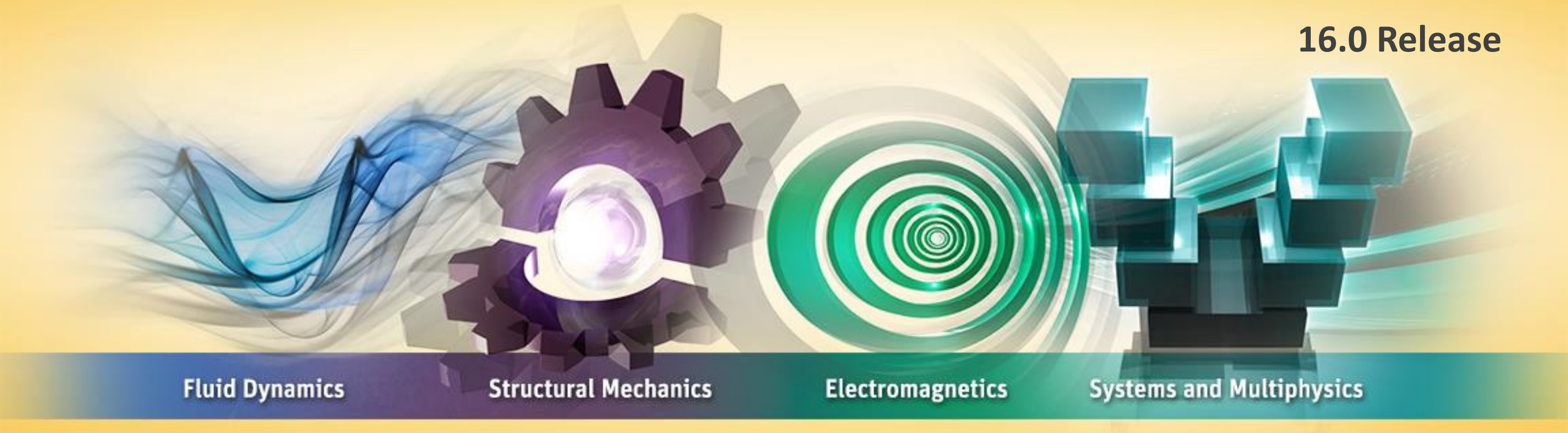
Lecture 5: Post-processing

16.0 Release



Introduction to ANSYS CFX

Introduction

Lecture Theme:

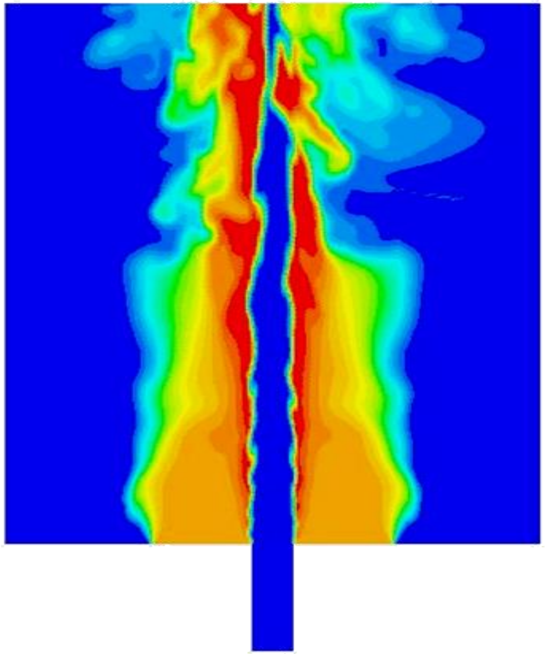
The purpose of CFD analysis is to obtain quantitative and/or qualitative information about fluid flow performance of the system. This lecture will explain how to do this in CFD-Post.

Learning Aims:

You will learn how to perform flow field visualization and quantitative data analysis on your CFD results

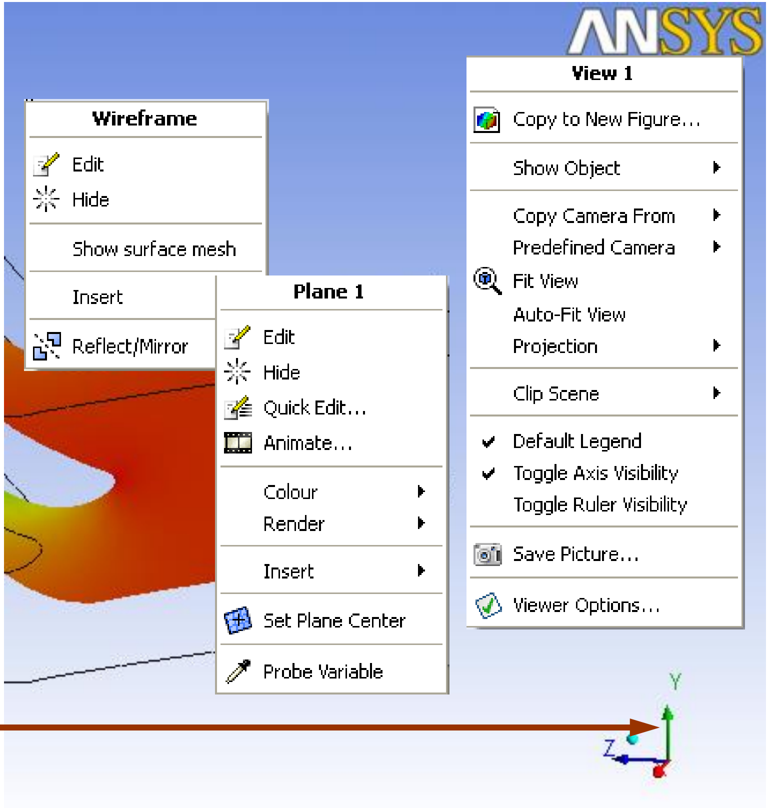
Overview

- CFD- Post include many tools for analyzing CFD results
  - Isosurfaces
  - Vector plots
  - Contour plots (shaded and graded)
  - Streamlines and particle tracks
  - XY plotting
  - Animation creation



Viewer Right-click Menus

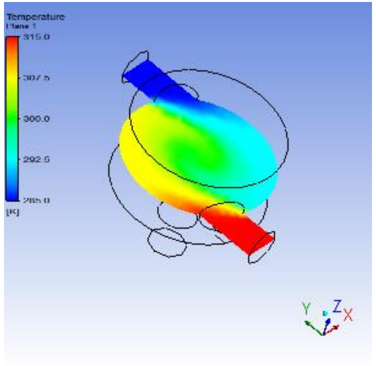
- Right-clicking in the Viewer
  - Right-clicking on an object (e.g. Wireframe, Plane) shows options
  - Insert new objects based on the current location (e.g. a vector plot on a plane, contour plot on a surface)
  - Right-clicking in empty space shows options for the current View
- Click on the axes to orient the view

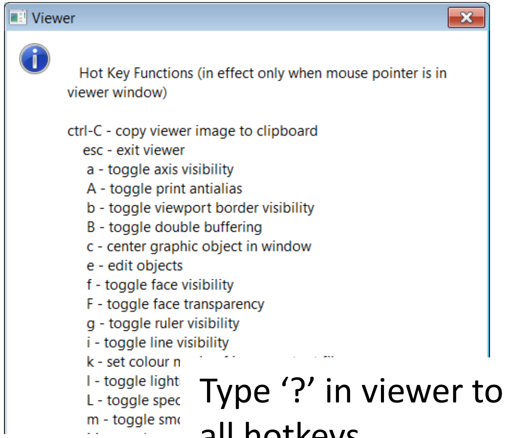


Introduction | Overview | CFD-Post | Summary

3D Viewer Files

- *Save Picture* in the *CFX Viewer State (3D)* file format (.cvf file)
- Can then use the stand-alone Viewer to view the file, rotate, pan, zoom, etc
  - Unlicensed and free to distribute to your customers
  - Can embed 3D Viewer files in PowerPoint and HTML files
  - Should be in your ANSYS CFX 16.0 installation
  - Download from the ANSYS Website (search for “ANSYS CFD Viewer”)
  - Or run ANSYS_CFD-Viewer_160_Setup.exe from your installation (*INSTALL_ROOT*\CFX\viewer)





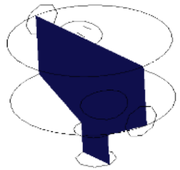
Type ‘?’ in viewer to see all hotkeys

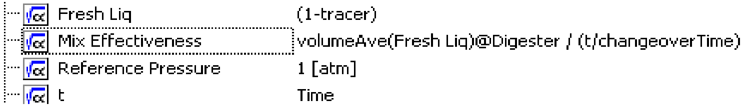
Introduction | Overview | CFD-Post | Summary

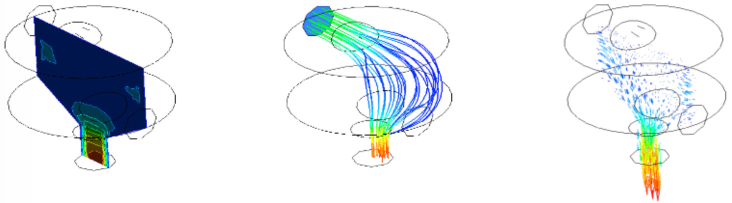
CFD-Post General Workflow

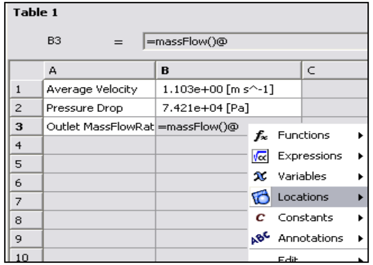
1. Prepare locations where data will be extracted from or plots generated
2. Create variables, expressions which will be used to extract
3. i) Generate qualitative data at locations

   ii) Generate quantitative data at locations
4. Generate Reports



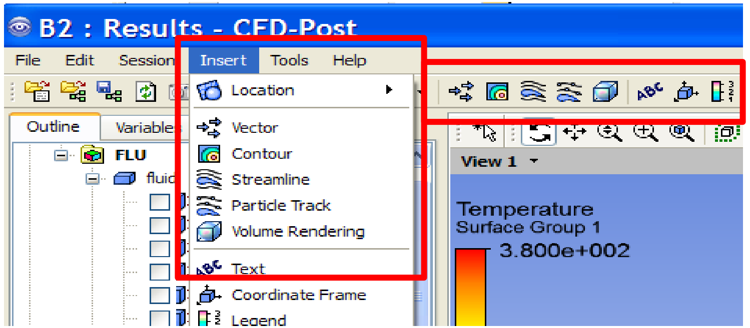


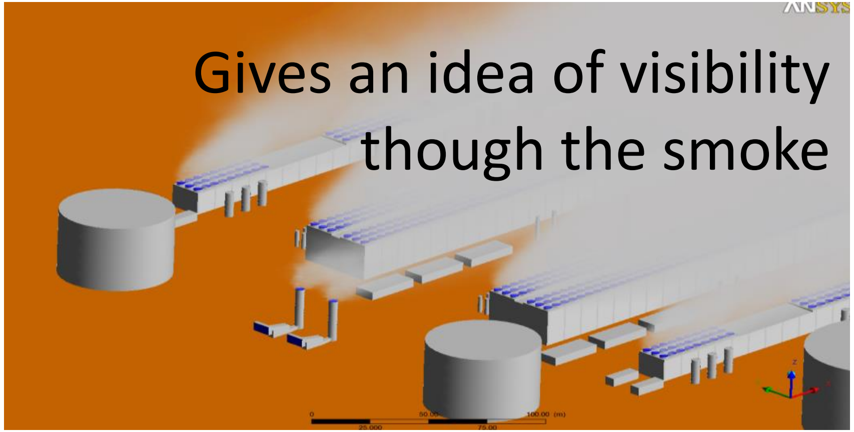




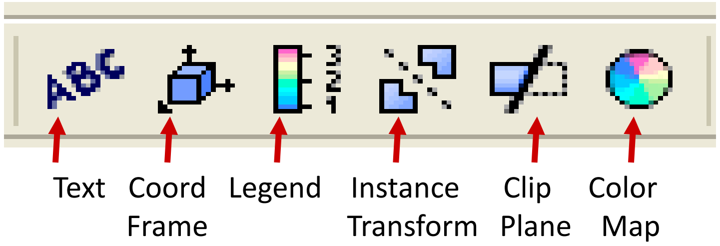
Other Graphics Objects

- Vector Plot → can plot any vector variable; usually velocity
- Streamlines → forwards and/or backwards from a seed
- Vectors, streamlines and contours can use any existing object as a base
- Volume Rendering
  - Shades a series of planes with a transparency based on a variable e.g. concentration

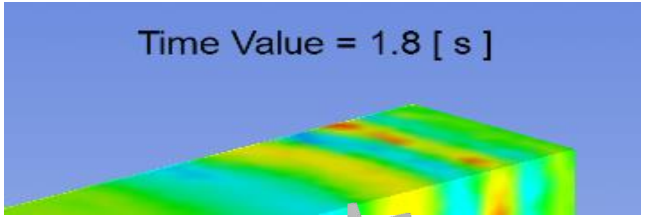




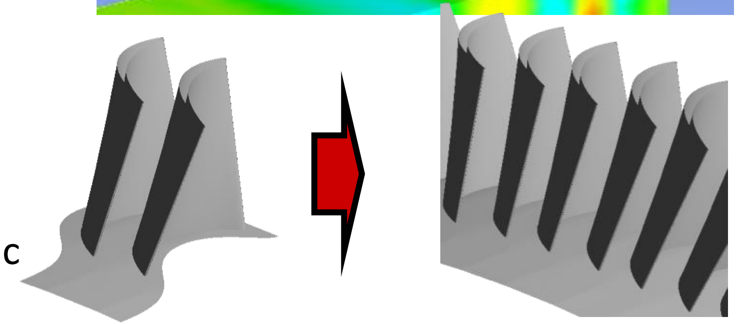
Other Graphics Objects



- **Text**
  - add own labels to the Viewer
  - Auto-text to show time step/values, filenames, etc.

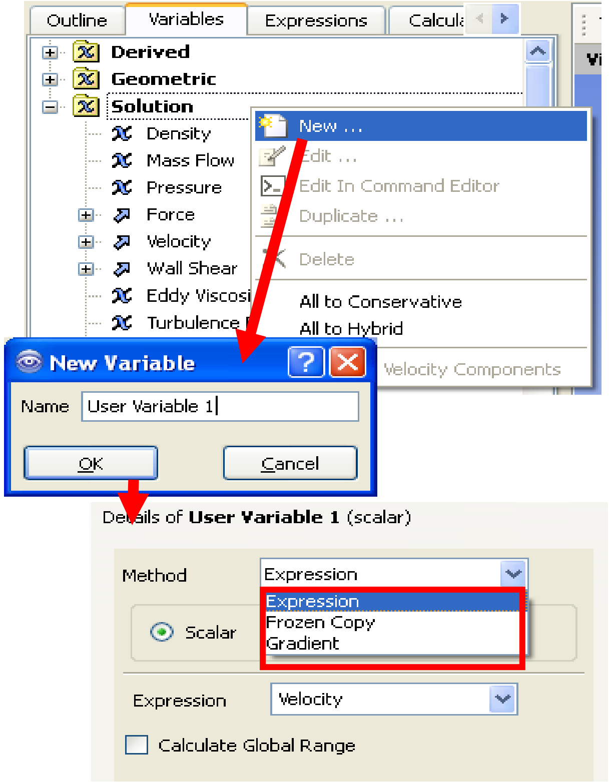


- **Coord Frame**
  - Insert new coordinate frame for use with certain quantitative functions, e.g. components of force
- **Legend**
  - Create additional legends tied to a specific plot
- **Instance Transform**
  - Used to re-create full plots from symmetric/periodic solution data



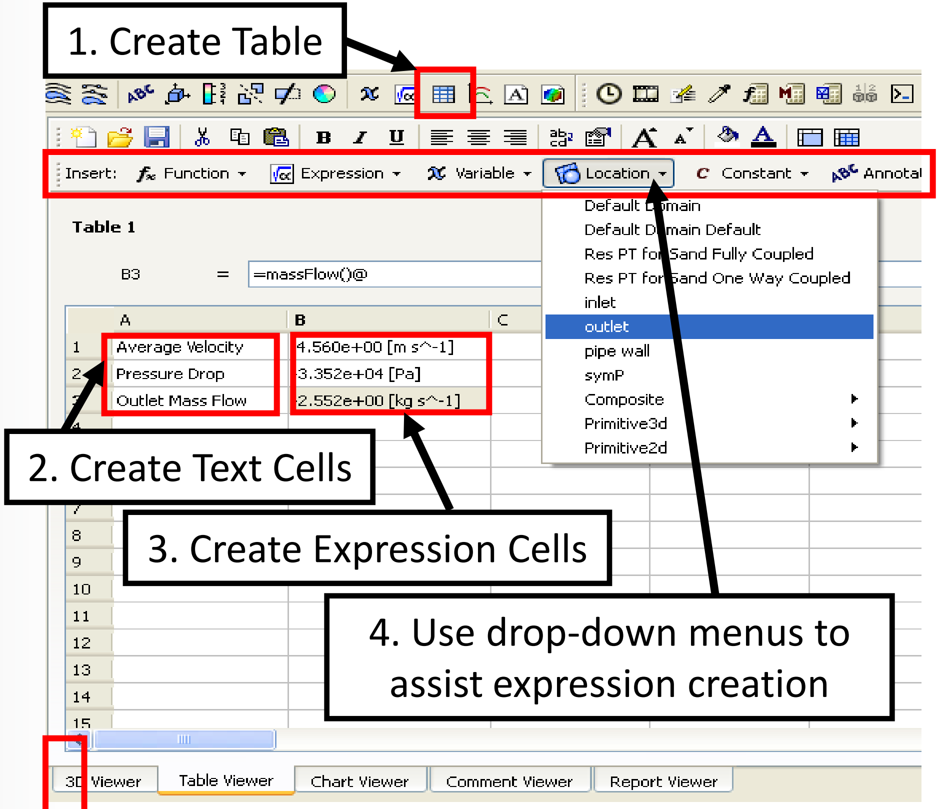
Variables Tab: User Defined Variables

- *Right-click > New…* in the top half of the *Variables* tab
- There are 3 methods for user defined variables:
  - <u>*Expression*</u> defines a variable via an expression, which can be a function of any other variable
  - <u>*Frozen Copy*</u> used to plot values of a variable from one run in the CFD-Post session for another run
  - <u>*Gradient*</u> calculates the gradient of an existing scalar variable
    - Produces a new vector variable

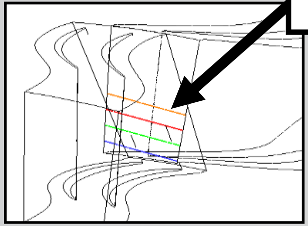


Tables

- ***Insert > Table*** **or use the toolbar icon**
  - *3D Viewer* switches to the *Table Viewer*
- **Display data and expressions in a tabular view**
- **Automatically added to the Report**
- **Cells can contain expressions or text**
  - Begin with "=" to distinguish
  - Expressions are evaluated and updated when variables and/or locations they depend on change
- **Not a spreadsheet**
  - Cannot reference other cells



Charts



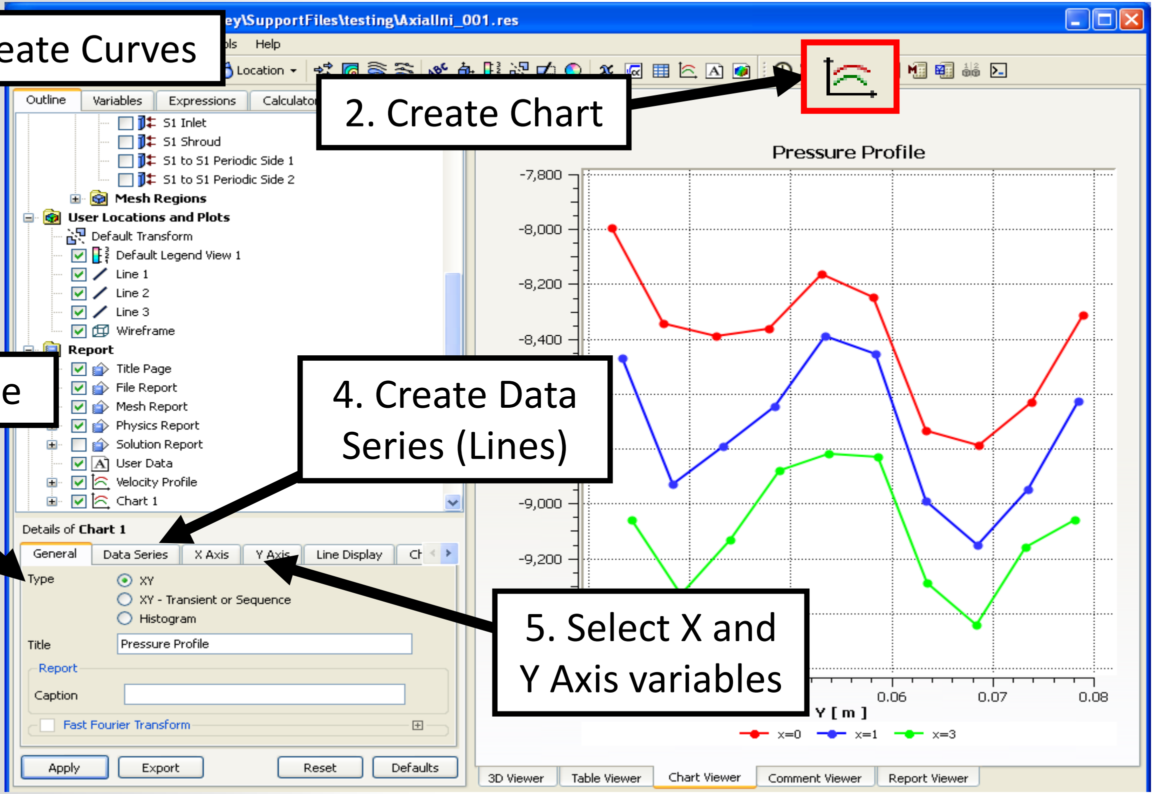
1. Create Curves

2. Create Chart

3. Select Chart Type

4. Create Data Series (Lines)

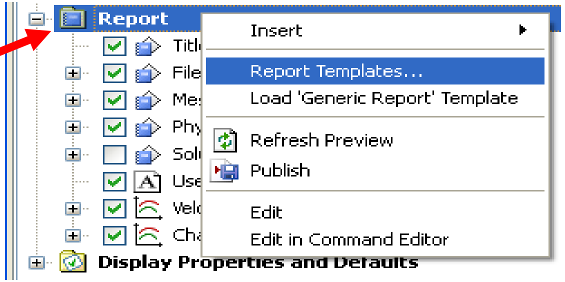
5. Select X and Y Axis variables

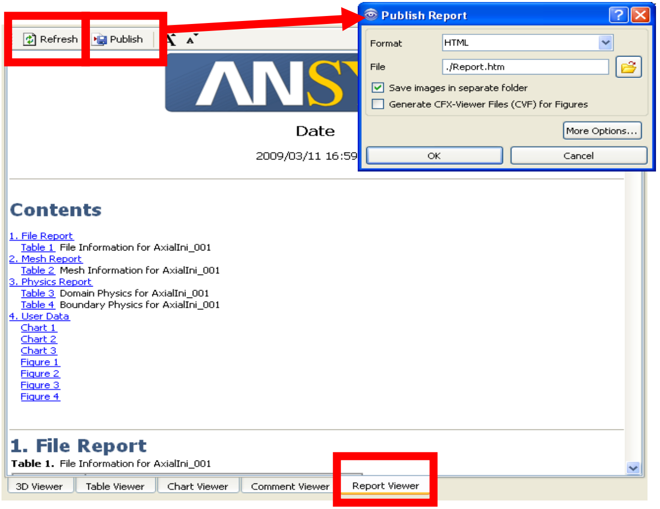


Introduction | Overview | CFD-Post | Summary

Reports

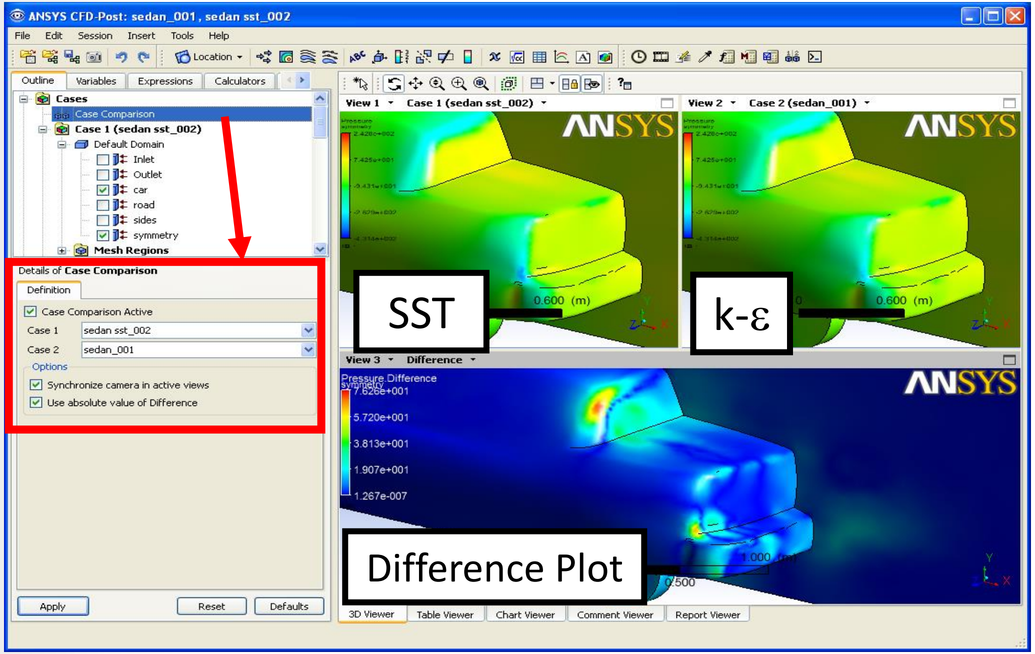
- **CFD-Post has report generation tools for rapid creation of customized reports**
  - To view the report, click the Report *Viewer* tab
  - Use the check boxes to control what is included in the report
- **Reports are template based**
- ***Publish* writes out an HTML or Text copy of the report**





Case Comparison

- When multiple files are loaded you can select Case Comparison from the Outline tree
  - Automatically generates difference variables and plots
- Expression syntax:
  - function()@CASE:#.Location
  - E.g: areaAve(Pressure) @CASE:1.Inlet



Summary and Conclusions

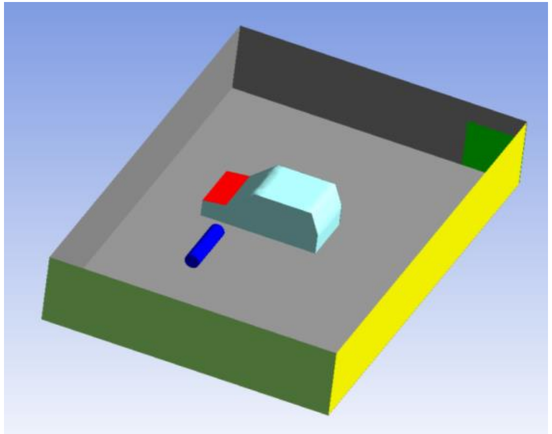
Summary:

- CFD-Post contains many powerful, sophisticated post-processing capabilities including 3D-viewer files, user variables, automatic html report generation and case comparison
- **What Next:**
- **Post-processing is best learned in a hands-on manner. Details of the operations described in the lecture will be covered during the completion of Workshop 2**

Introduction | Overview | CFD-Post | Summary

Workshop 02 – Multi-species Flow & Post-processing

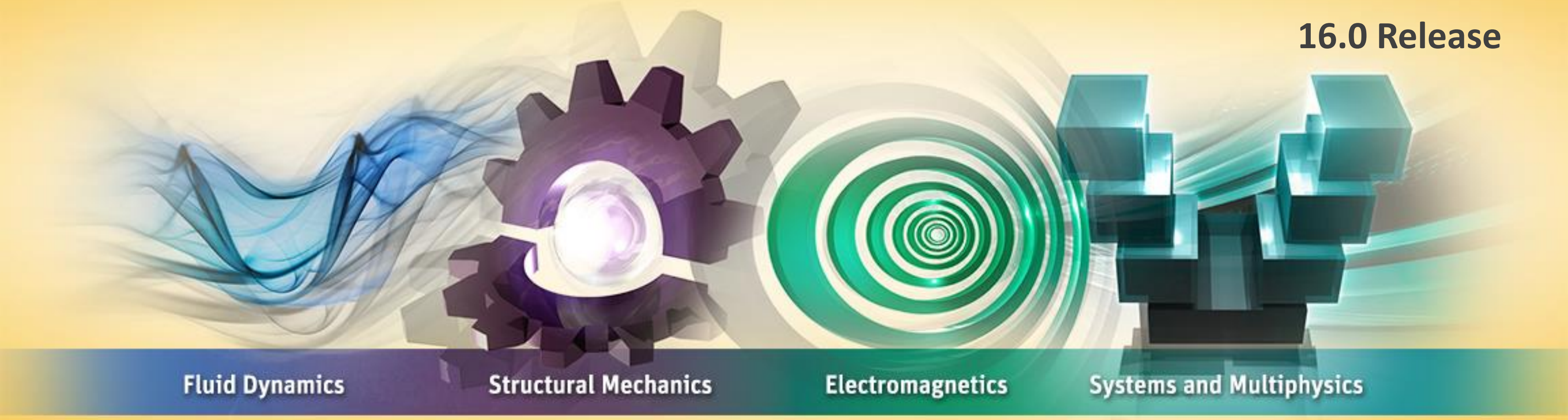
- Use of a multi-component mixture to represent air and combustion gases
- Natural convection
- Momentum source to model a fan
- Charts, user variables, quantitative functions, volume rendering, reports



You can choose whether to work through the entire workshop or the post-processing stage only.

Appendix

16.0 Release

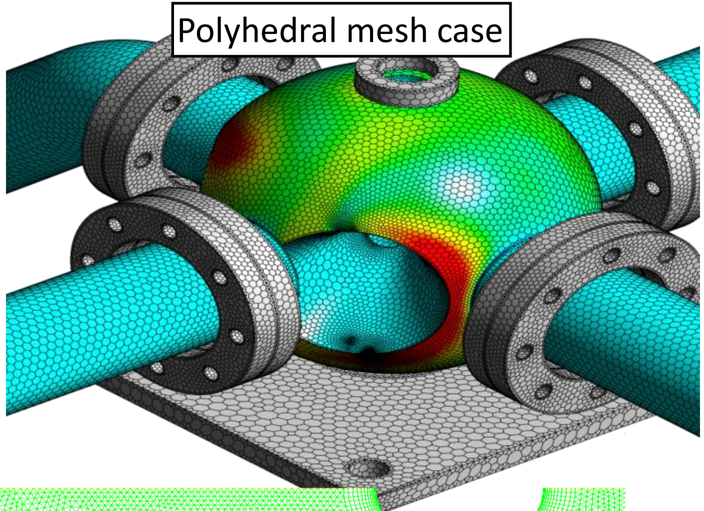


Introduction to ANSYS CFX

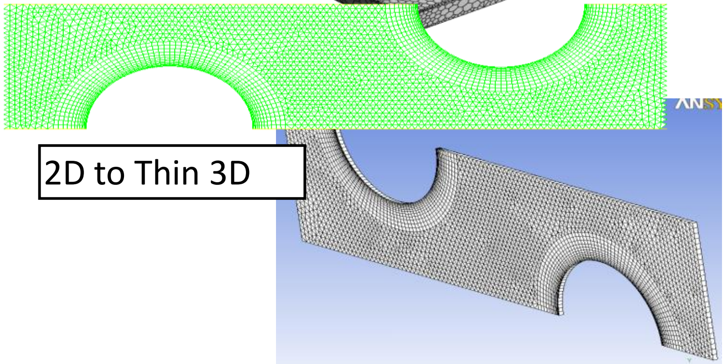
Realize Your Product Promise®

ANSYS

Limitations of CFD Post

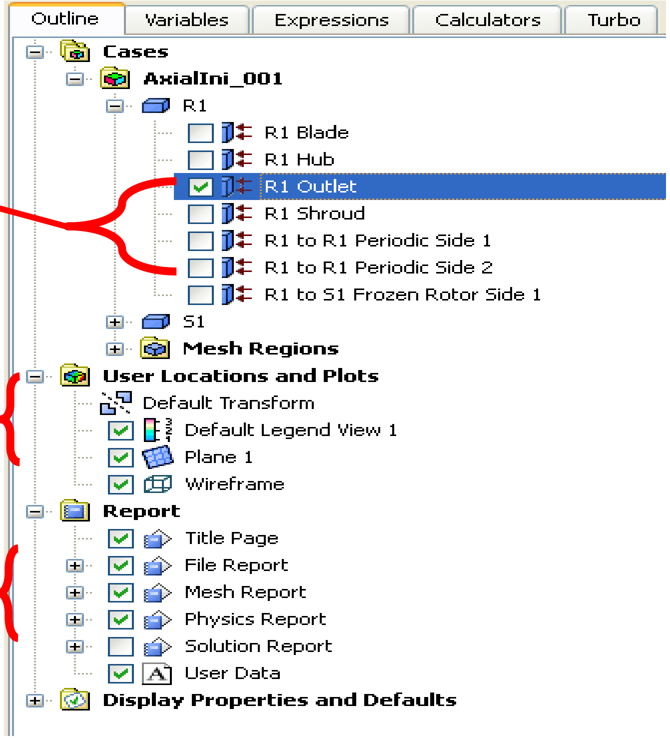


- **All mesh types supported**
  - Polyhedral, non-conformal, adapted, ...
  - 2D FLUENT meshes are extruded to thin 3D domains
  - 2D axisymmetric meshes are converted to 3D wedges
- **Some data may not be in the standard .dat file**
  - Export through the Data File Quantities or the Export to CFD-Post panels
  - Note that for particle tracks this must be done (example in DPM Workshop)
- **CFD-Post is serial, not parallel**



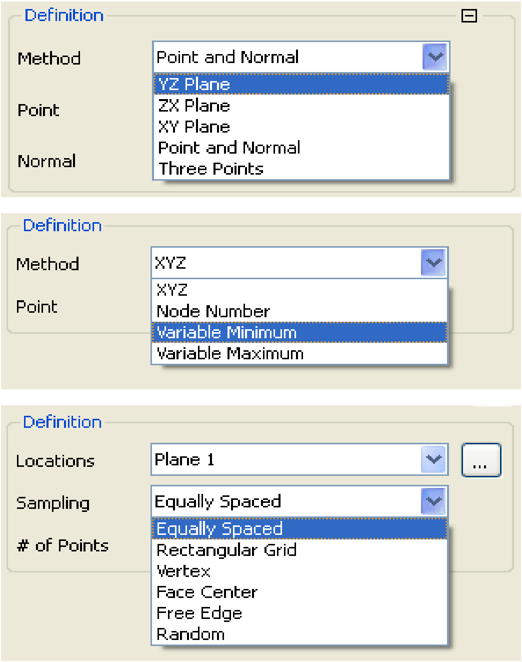
Creating Locations

- **Domain, Subdomain, Boundary and Mesh Regions are always available**
  – Boundary and Mesh Regions can be edited and coloured by any variable
  – Mesh Regions provides all available interior/exterior 2D/3D regions from the mesh
- **All Locations you create are listed under User Locations and Plots**
- **All items contained in the Report are listed here**



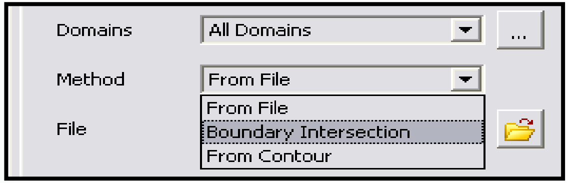
Location Types

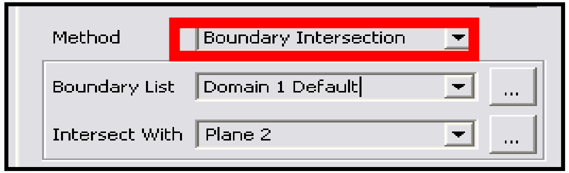
- **Planes**
  - XY Plane, Point and Normal, etc.
  - Can define a circle or rectangle to bound the plane, otherwise it's bounded only by the domains
- **Point**
  - XYZ: At coordinates. Can pick from Viewer
  - Node Number: Some solver error messages give a node number
  - Variable Max / Min: Useful to locate where max / min values occur
- **Point Cloud**
  - Create multiple points

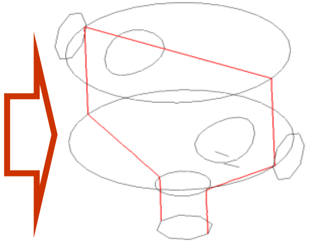


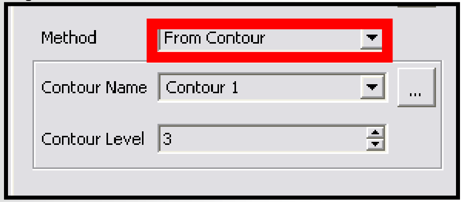
Location Types

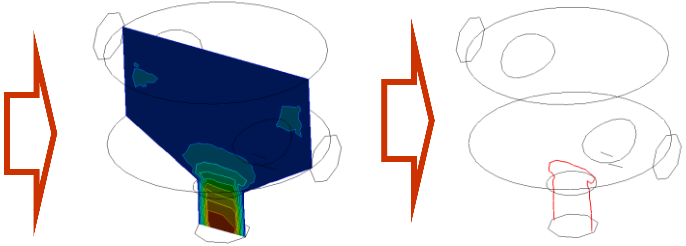
- **Lines**
  - Straight line between two points
- **Polylines**
  - Used for Charts
  - Read points from a file
  - Line of intersection
    between a boundary and another location
  - Extract a line from a contour plot







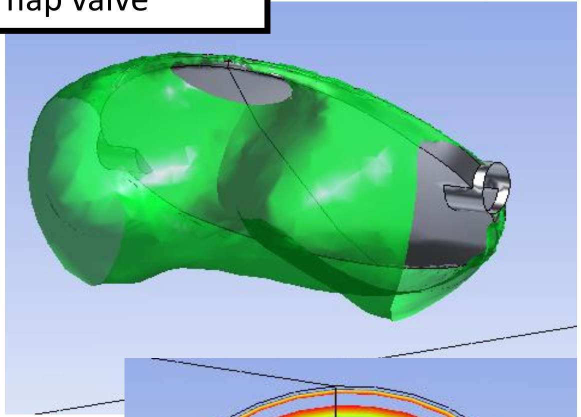


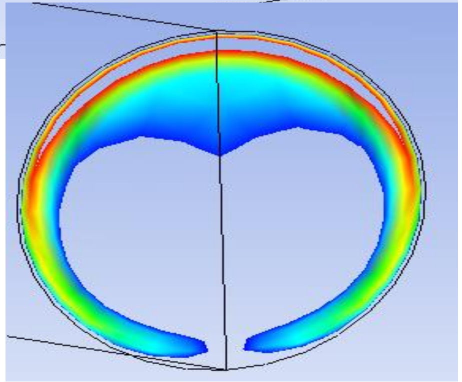


Location Types

- **Isosurfaces**
  - **Surface of a variable at a specified value**
- **Iso Clip**
  - **An Iso Clip takes a copy of any existing location and then clips it using one or more criteria**
    - **e.g. a outlet boundary plot clipped by Velocity >= 10 [m/s] and Velocity <= 20 [m/s]**
  - **Can clip using any variable, including geometric variables**

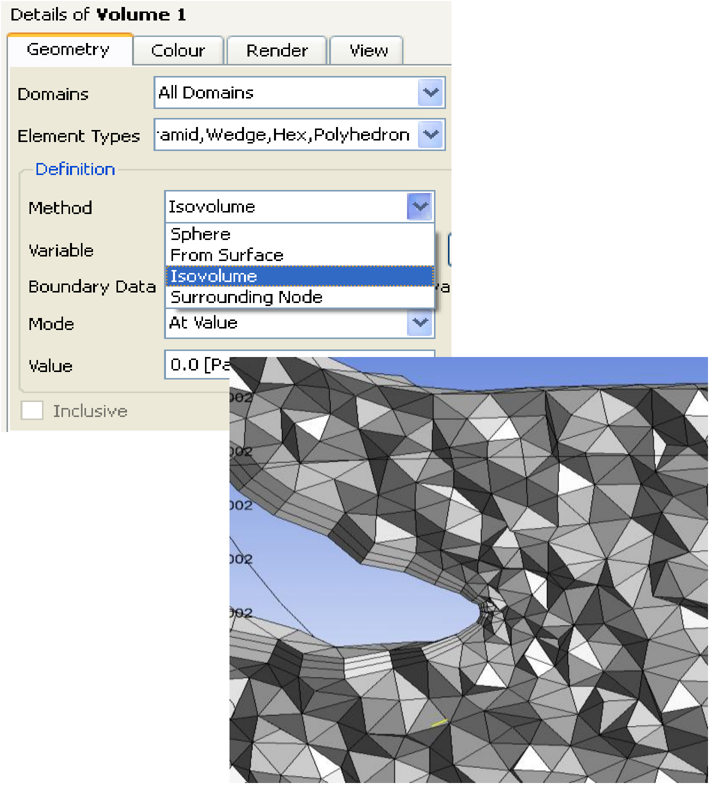
Isosurface of pressure behind a flap valve





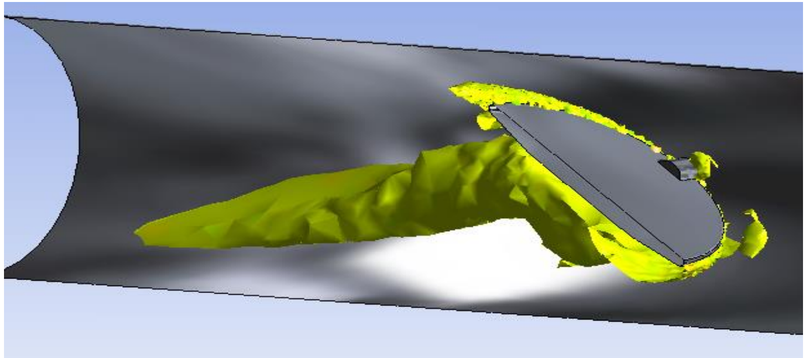
Location Types

- **Volumes**
  - Elements are either in or out
    - No cut volumes
  - From Surface
    - A volume is formed from all elements touching (or above / below) the selected location
    - Useful for mesh checking
  - Isovolume
    - Base on a variable at, above below a given value, or between two values

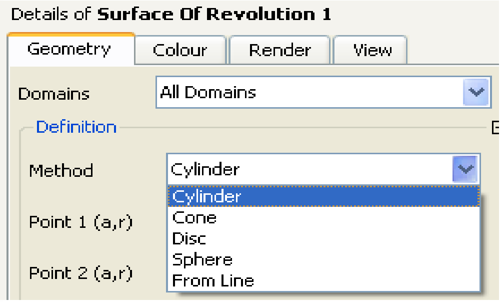


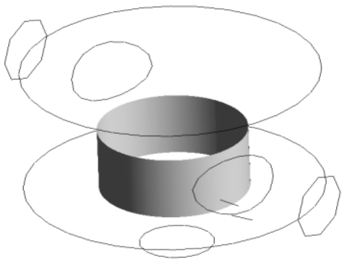
Location Types

- **Vortex Core Region**
  - Used to automatically identify vortex regions
  - Best method is case dependent
  - See documentation for details on the different methods



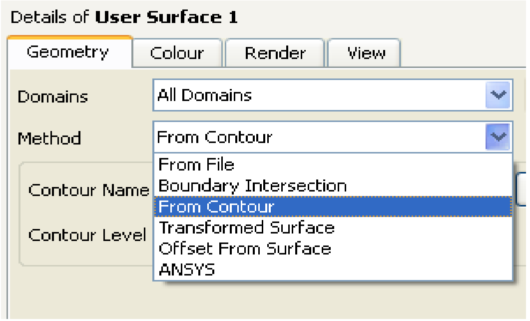
- **Surface of Revolution**
  - Predefined options for Cylinder, Cone, Disc and Sphere
  - From Line is much more general
    - Any existing Line, Polyline, Streamline, Particle Track is rotated about an axis

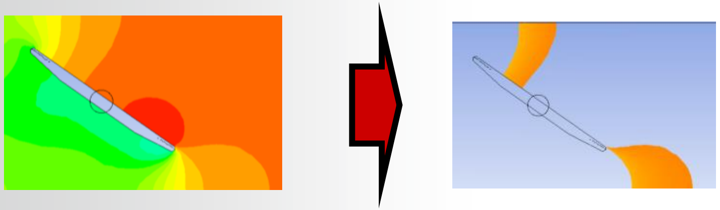




Location Types

- User Surface
  - Additional surface creation options including:
    - From File: reads point data from a text file; usually export this file from a different case
    - From Contour: extract a contour level
    - Transformed Surface: rotate, translate, scale an existing surface
    - Offset From Surface: either in the Normal direction or by Translating

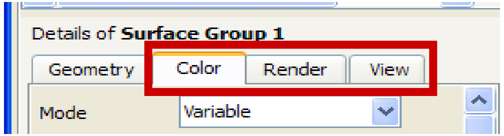




User Surface: From Contour Method
(Note: It’s generally easier to use Iso Clips instead)

Color, Render and View

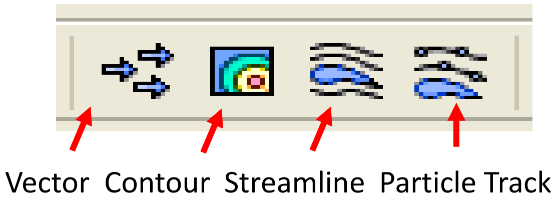
- Locations have similar Color, Render and View settings

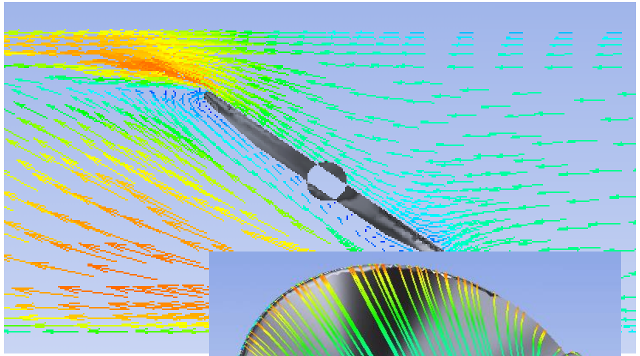


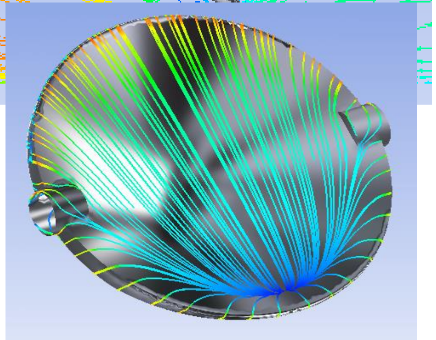
- Color
  - Select the variable with which to color the location
  - Set the Range (Global, Local, User Specified) and pick a Color Map
- Render
  - Draw Faces: shows solid surface
  - Draw Lines: shows mesh edges/intersection lines between mesh and location
  - Transparency, Lighting, Texture…
- View
  - Apply Rotation, Translation, Reflection, Scaling
  - Pick a different Instance Transform

Other Graphics Objects

- **Insert from the toolbar or the Viewer right-click menus**
- **Vectors, Contour and Streamlines use existing locations as a base**
- **Vector Plot**
  - **Can plot any vector variable**
  - **Can project Normal or Tangential to the base object**
- **Streamlines**
  - **Plot forwards and/or backwards from a seeding location**
  - **Surface Streamline to visualise velocity “on” walls**





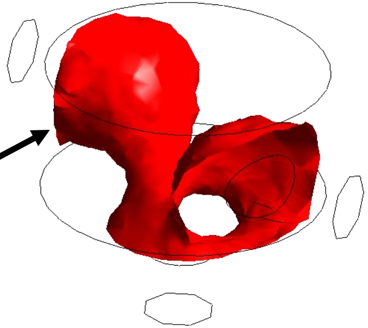


Introduction | Overview | CFD-Post | Summary

User Defined Variables Example

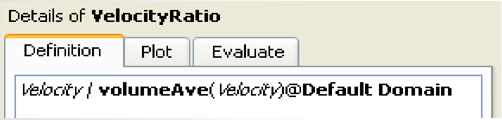
- **Goal: Plot an isosurface at VelRatio = 0.7**

where $$VelRatio = \frac{Vel_{local}}{\overline{Vel}}$$

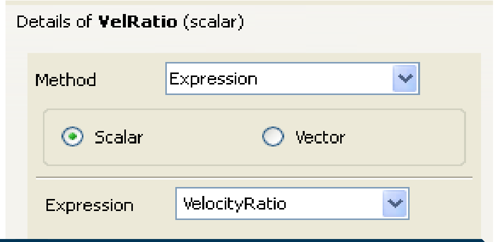


$$VelRatio = 0.7$$

1. On the Expressions tab create the expression for Velocity Ratio:

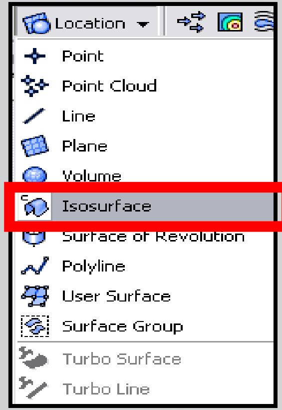


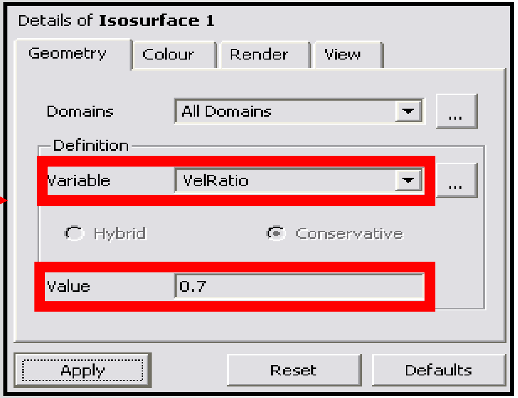
2. On the Variables tab create a new variable named VelRatio using Method = Expression



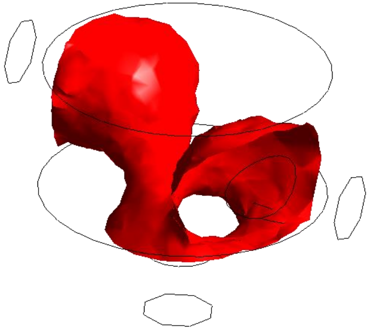
User Defined Variables Example

3. Create an Isosurface using the variable VelRatio at a value of 0.7





$$VelRatio = 0.7$$



CEL

- CEL - CFX Expression Language
  - Allows the user to create equations (can be functions of solution/system variables) that can be used in CFX-Pre and CFD-Post
- Expressions can be:
  - algebraic
    - Velocity u * X
    - tan(X/Z)
    - log(T/T0)
  - integral
    - massFlowAve(Total Pressure)@inlet or massFlow()@inlet

CEL Rules

- The syntax rules are the same as those for conventional arithmetic
- Operators are written as:
  - + (addition)   - (subtraction)   * (multiplication)
  - / (division)   ^ (exponentiation)
- Variables and expressions are case sensitive (example: t vs. T)
- Expressions must be dimensionally consistent for addition and subtraction operations (example: 1.0 [mm] + 0.45 [yds] is OK)
- You cannot add values with inconsistent dimensions

CEL Rules

- Fractional and decimal powers are allowed (example: a^(1/2) + 1.0^0.5)
- Units of expressions are not declared – they are the result of units in the expression (example: a [kg m^-3] * b [m s^-1] has units of [kg m^-2 s^-1]
- Some constants are also available in CEL for use in expressions:
  - e Constant: 2.7182818
  - g Acceleration due to gravity: 9.806 [m s^-2]
  - pi Constant: 3.1415927
  - R Universal Gas Constant: 8314.5 [m^2 s^-2 K^-1]

Built In Functions

Numerical functions and operators are also available in CEL

- Right-click when creating expressions for a complete list
- Custom functions with User Fortran can also be created

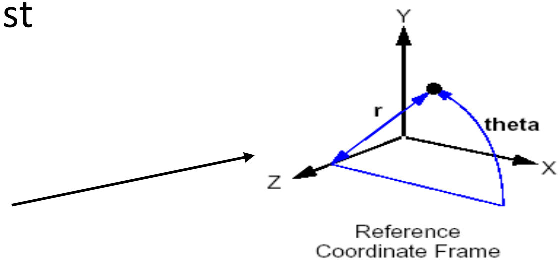
Function	Operand's Dimensions [x]	Operand's Values	Result's Dimensions
sin(x)	Angle	Any	Dimensionless
cos(x)	Angle	Any	Dimensionless
tan(x) ***	Angle	Any	Dimensionless
asin(x)	Dimensionless	$-1 \le x \le 1$	Angle
acos(x)	Dimensionless	$-1 \le x \le 1$	Angle
atan(x)	Dimensionless	Any	Angle
exp(x)	Dimensionless	Any	Dimensionless
loge(x)	Dimensionless	$0 < x$	Dimensionless
log10(x)	Dimensionless	$0 < x$	Dimensionless
abs(x)	Any	Any	[x]
sqrt(x)	Any	$0 \le x$	[x]^0.5
if(test, res1, res2)*	Any	Any	Any (res1 and res2 must have the same dimensions)
min(x,y) ****	Any	Any	[x]
max(x,y) ****	Any	Any	[x]
step(x) *	Dimensionless	Any	Dimensionless

*if functions contain a test, and two result outcomes. The first outcome, res1 will be returned if test evaluates to true. If test evaluates to false, res2 is returned. Consider the following example, where we wish to set volume fraction to 1 when X is greater than 1 [m], and 0 if X is less than 1 [m]:
if (x>1[m], 1, 0)
In this case, if the result is precisely equal to 1[m], the result is (res1+res2)/2 **step(x) is 0 for negative x, 1 for positive x and 0.5 for x=0. *** note that tan(x) is undefined for $n\pi/2$ where n=1, 3, 5 .. **** both x and y must have the same dimensions.

Solver Variables

- **Solver variables are available for use in any expression**
- **Below is a partial list of the available system variables:**
  - When creating expressions, right-click to access a full list

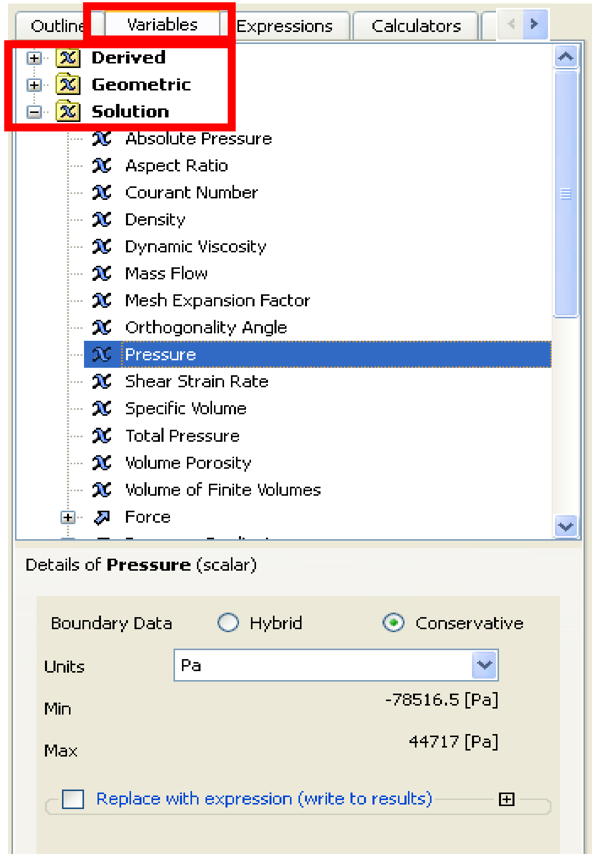
x	Direction 1 in Reference Coordinate Frame
y	Direction 2 in Reference Coordinate Frame
z	Direction 3 in Reference Coordinate Frame
r	Radial spatial location, r = (x^2+y^2)^0.5
theta	Angle, arctan(y/x)
t	Time
u	Velocity in the x coordinate direction
v	Velocity in the y coordinate direction
w	Velocity in the z coordinate direction
p	(absolute) Pressure
ke	Turbulent kinetic energy
ed	Turbulent eddy dissipation
T	Temperature
sstrnr	Shear strain rate
density	Density
rNoDim	Non-dimensional radius (rotating frame only)
viscosity	Dynamic Viscosity
Cp	Specific Heat Capacity at Constant Pressure
cond	Thermal Conductivity
AV name	Additional Variable name
mf	Mass Fraction



Depending on your physics, some variables will not be valid – e.g. you need to solve heat transfer to use T

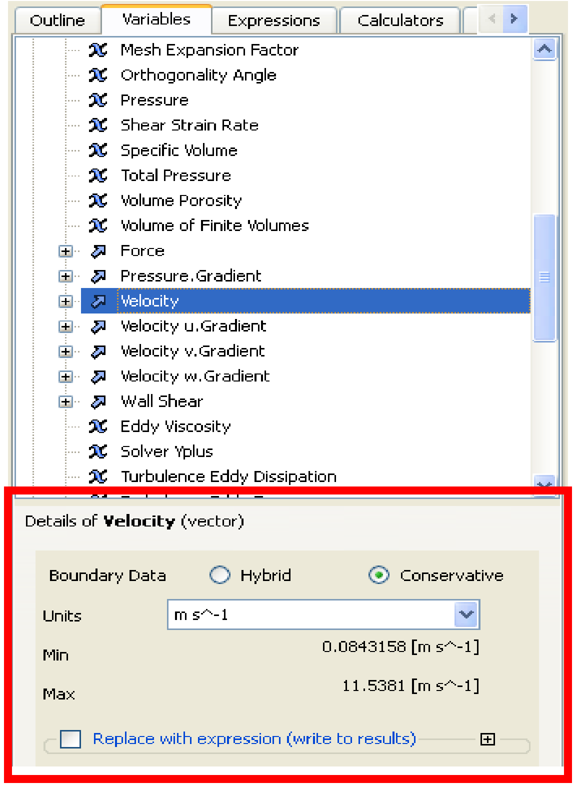
Variables Tab

- **The Variables Tab shows information about all available variables**
- **Derived variables**
  - **Calculated by CFD-Post – not contained in results file**
- **Geometric variables**
  - **X, Y, Z, Normals , mesh quality data**
- **Solution variable → from the results file**
- **User Defined variables → create new derived variables**
- **Turbo variables → additional variables automatically created for turbomachinery cases**



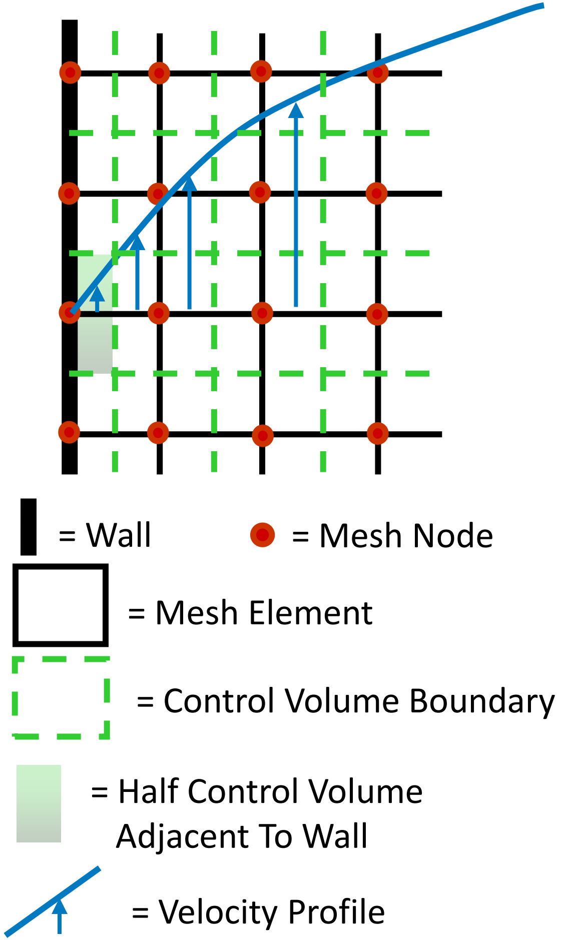
Variables Tab

- **Details pane shows information for selected variable**
  - Different options for *User Defined* variables
- **You can replace any variable with an expression**
  - New values are stored in the results file, so you can close CFD-Post and the data is retained
  - Old values can be restored at any time
  - Example: modifying results for an initial guess
- **Switch between *Hybrid* and *Conservative* variables**
  - Only applicable to CFX results
  - By default CFD-Post uses Conservative values for all calculations and Hybrid values for all graphics
  - Can switch between Hybrid and Conservative on the *Colour* tab



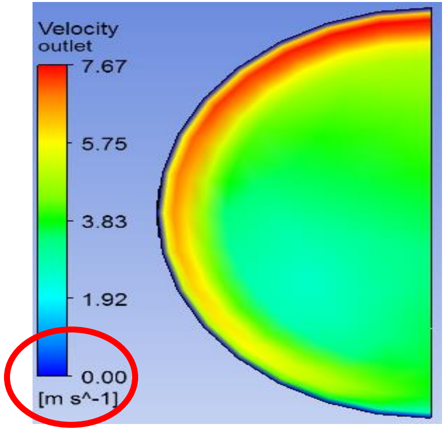
Hybrid vs. Conservative

- **The finite volumes used by the CFX-Solver are constructed from the mesh, but are not equal to the mesh elements**
  - Mesh nodes lie at the centre of control volumes
- **Values are stored in the results file at nodes and represent “average” control volume values**
- **Next to wall boundaries you have a half control volume with some representative non-zero velocity**
  - This non-zero velocity is stored at the wall node
  - But we *know* that the velocity on a wall is zero
- **Conservative values = control volume values**
- **Hybrid values = specified boundary condition values**

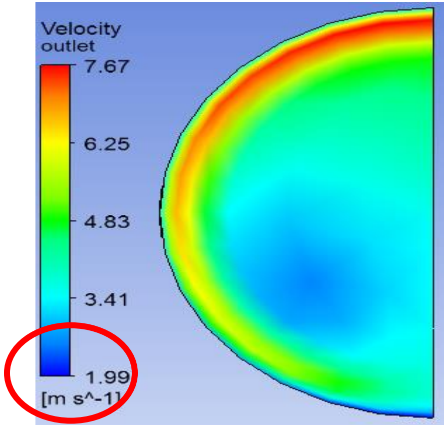


Hybrid vs. Conservative

- **For visualization, ANSYS CFD-Post uses hybrid values by default, because you usually don't want to see non-zero wall velocities**
- **For calculations conservative values are used by default**
  - This is good! For example mass flow is calculated correctly — a velocity of zero would produce zero mass flow through the wall adjacent control volume which is clearly wrong
- **So in most cases you don't need to worry about Hybrid vs Conservative since CFD-Post does the right thing**
  - User Defined variables will be derived from conservative values by default
  - Take care when interpreting plots! The range will be different for hybrid and conservative values



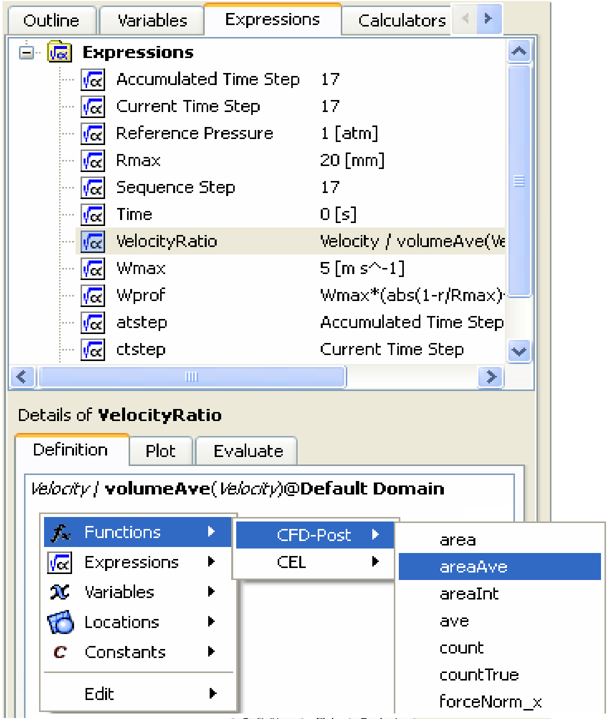
Hybrid

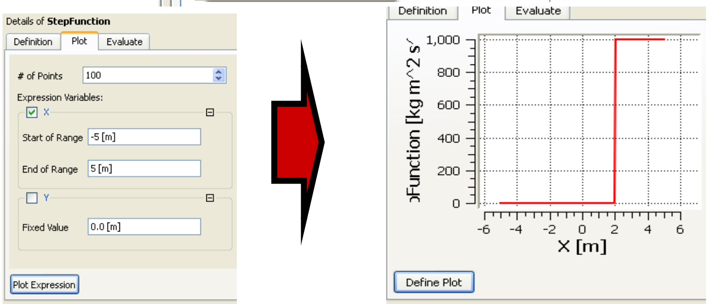


Conservative

Expressions Tab

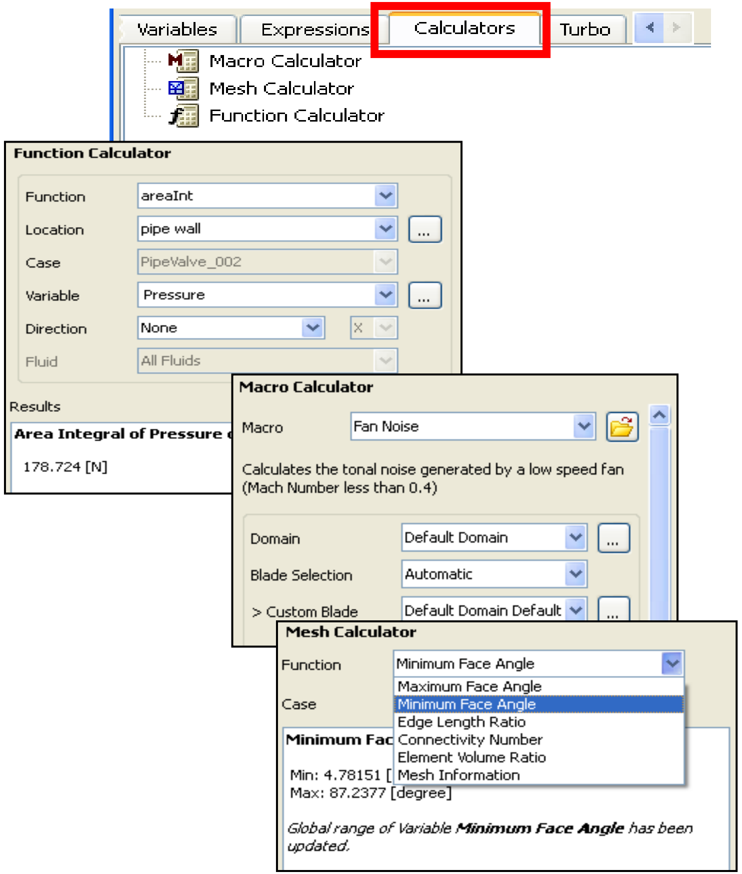
- **The *Expressions* tab shows all existing expressions and allows you to create new expressions**
  – Right-click in the top area > *New*
- **Enter expressions on the *Definition* tab in *Details* view**
  – Right-click to select *Functions*, *Variables* etc. for building your expression
- **Use the *Plot* tab to view an XY plot of the expression**
  – Enter a range for one variable and fixed values for the others





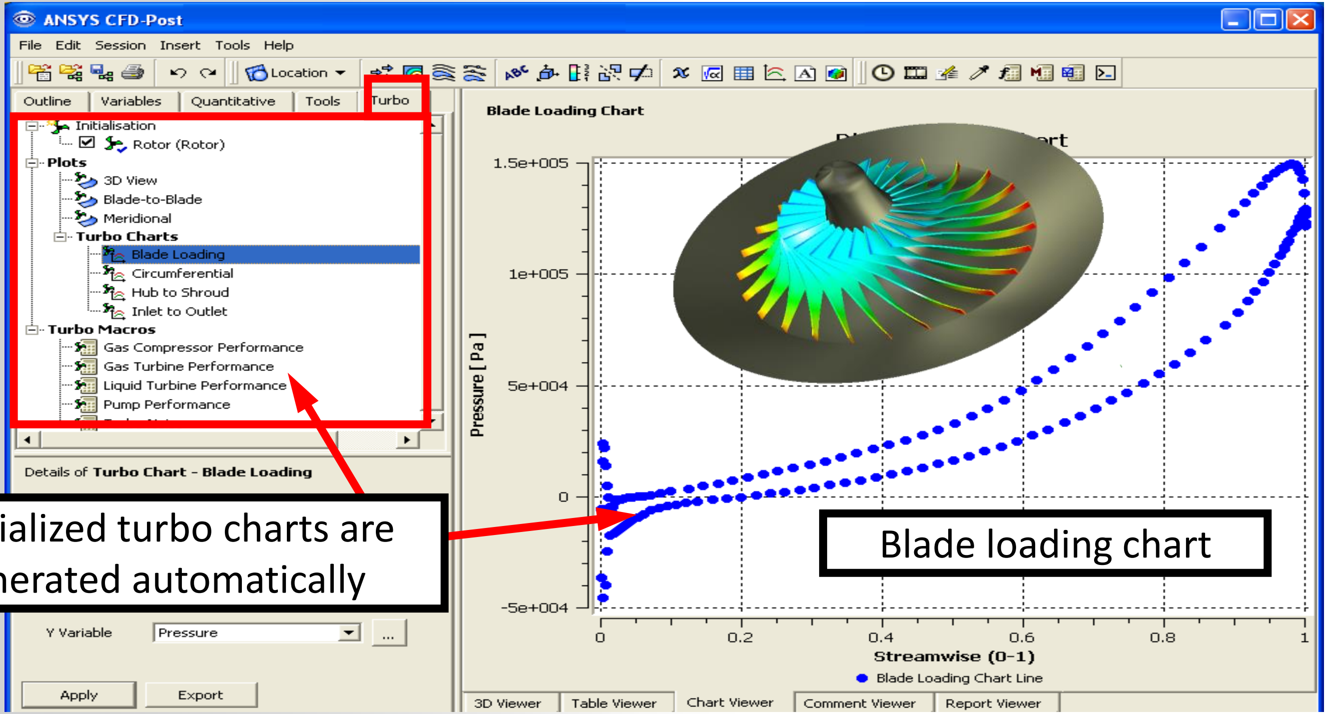
Calculators Tab

- **Function Calculator**
  - Extract engineering data from the results
  - Many functions, see doc to see how they operate
  - Same function used as when creating expressions
- **Macro Calculator**
  - Run predefined Macros
  - Write your own Macros and have them appear here
  - More in Scripting lecture
- **Mesh Calculator**
  - Mesh quality metrics and stats
  - Field variables exist for all the metric and can be plotted



Turbo Post Processing

- The *Turbo* tab contains tools for post-processing turbomachinery cases.

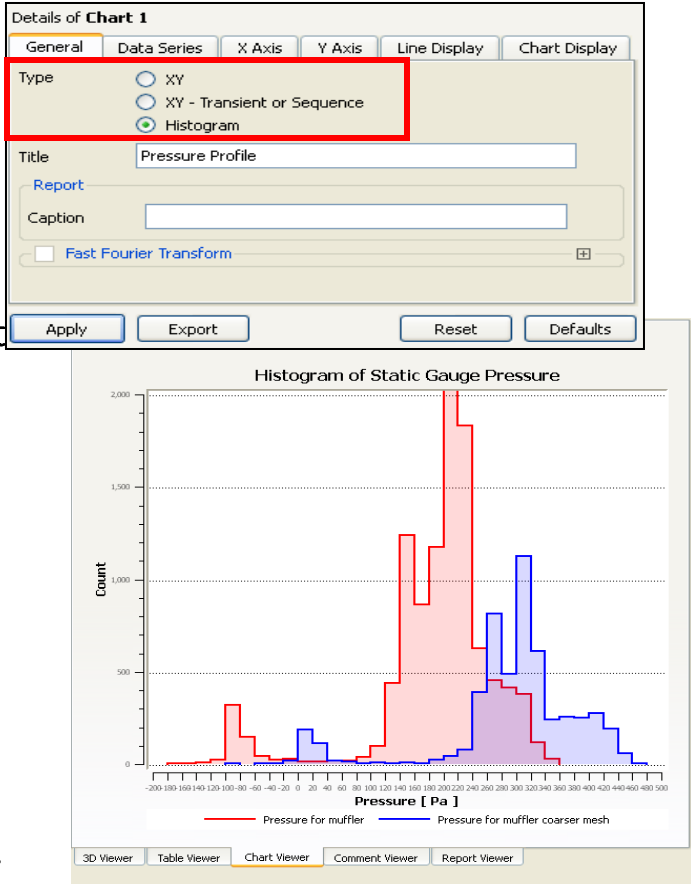


Specialized turbo charts are generated automatically

Blade loading chart

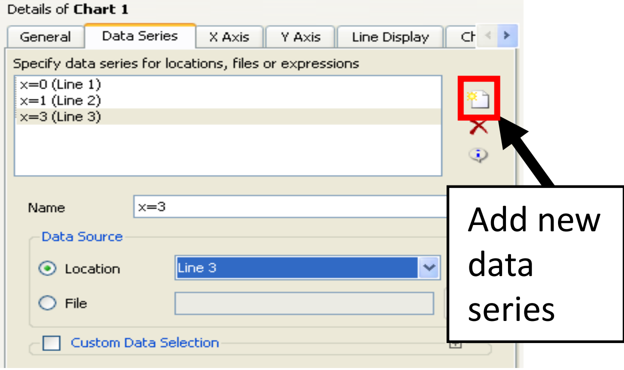
Charts: Type

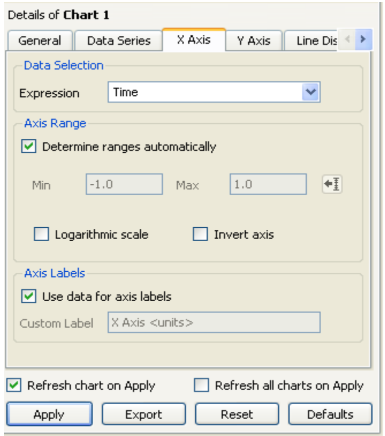
- **Charts can be one of three types:**
  - **XY:** Standard XY plots based on <u>line</u> locators
  - **XY – Transient or Sequence**
    - Plots an expression (usually Time) versus a variable at a <u>point</u> locator
    - Typically used to show the transient variation of a variable at a point
      - Data must be present in the trn files
  - **Histogram**
    - Based on a locator that contains multiple data locations – lines, surfaces, planes, domains
    - Plots a variable divided into discrete bands on the X Axis versus the frequency of occurrence on the Y Axis

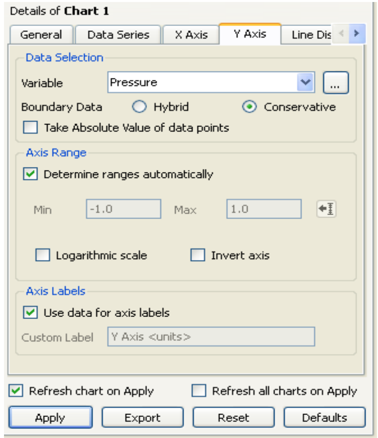


Charts: Data Series and Axes

- **Each data series corresponds to a location (line, point, etc.) which corresponds to a curve on the chart**
- **Use the *X* and *Y Axis* tabs to set the variables on the axes**
- **The remaining tabs are for various display options**

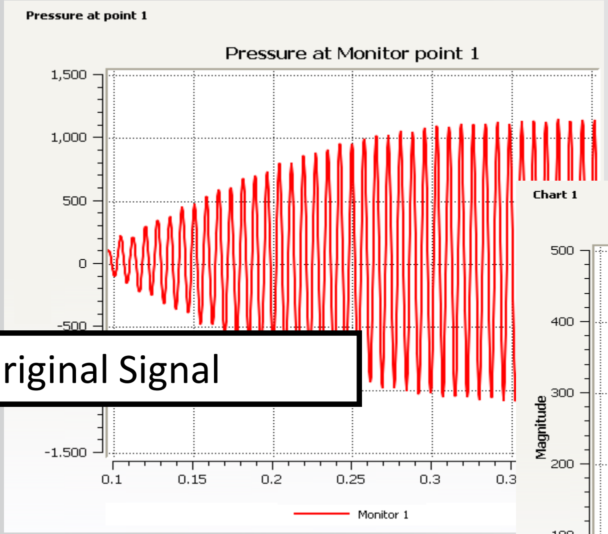




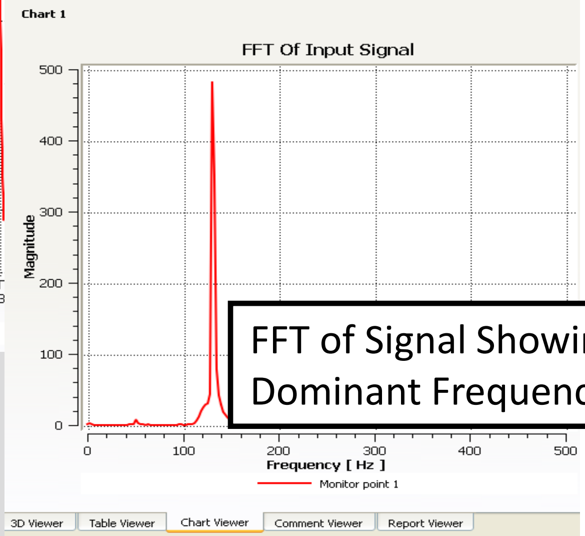


Fast Fourier Transform

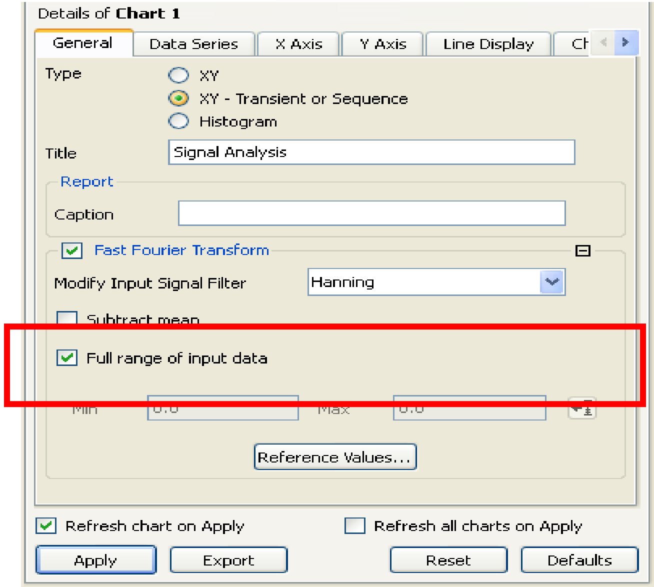
FFT can be applied to signals to extract frequency data



Original Signal

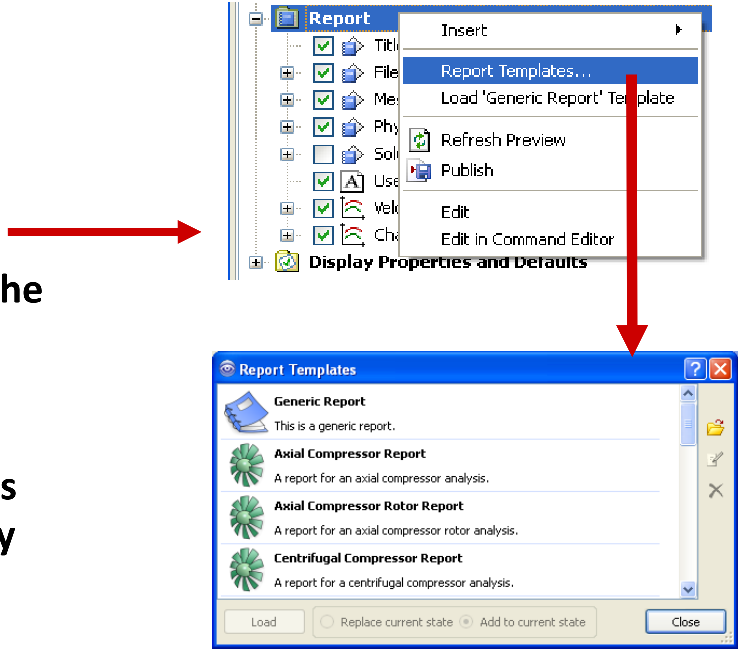


FFT of Signal Showing Dominant Frequency



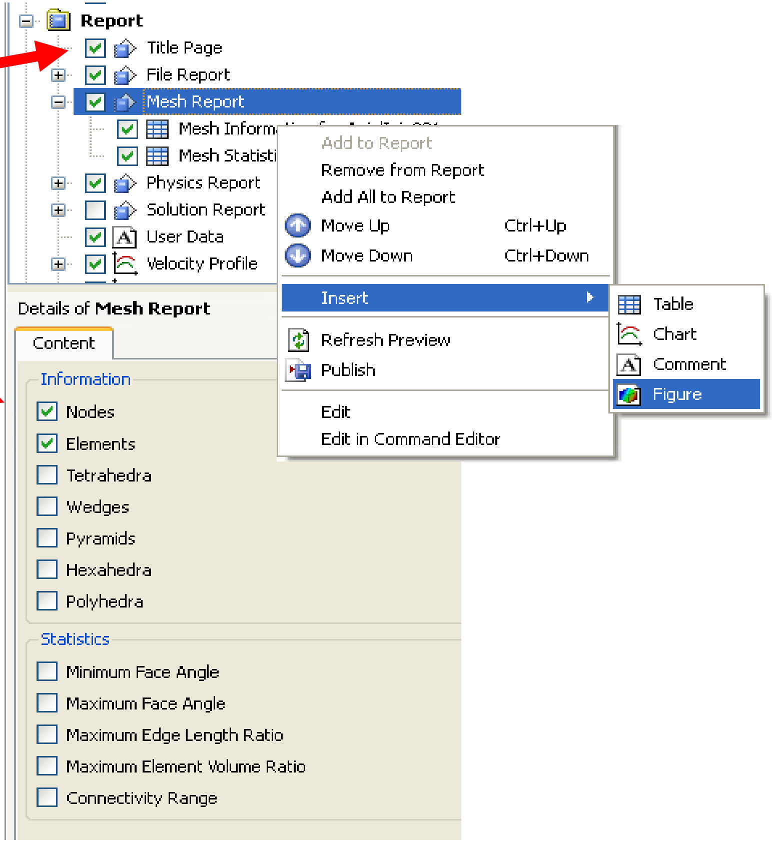
Reports

- CFD-Post has report generation tools for rapid creation of customized reports
  - To view the report, click the Report Viewer tab
  - Use the check boxes to control what is included in the report
- Reports are template based
  - Depending on the information contained in a results file, a report template will be selected automatically
  - Right-click on Report to select a different template
    - You can create your own custom templates or modify existing templates
    - E.g. add you company logo, add Charts, Tables, Plots etc



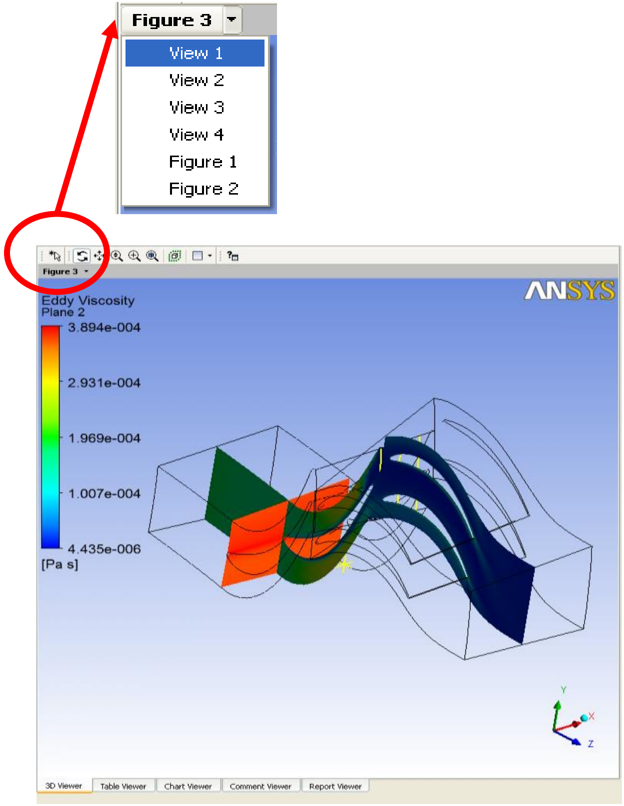
Reports

- **Use the check boxes to control what is included in the report**
- **Double-click items to edit**
  - For example, editing the *Mesh Report* shows that additional items can be included
- ***Tables* and *Charts* are automatically added to the report. Other items that can be added are *Comments* and *Figures*.**
  - *Right-click > Insert* to add new items
- **Can also right-click on each item to move it up or down in the report**



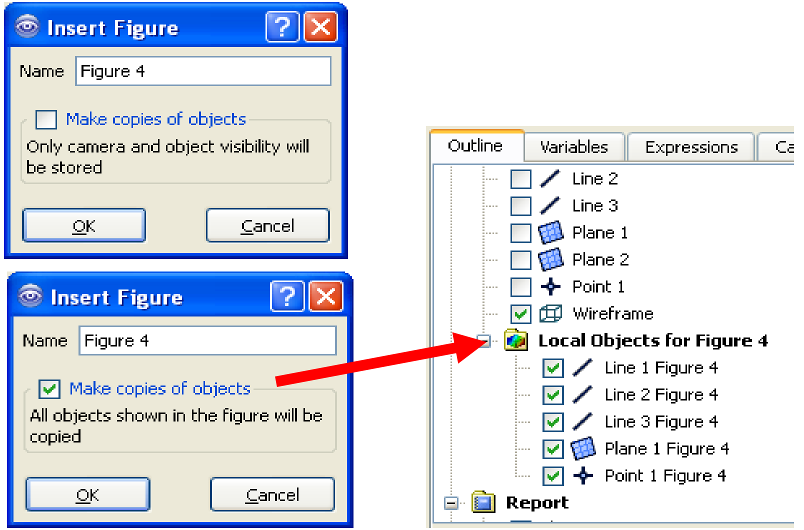
Reports: Figures

- When you add a new *Figure* it will be listed in the drop-down menu in the top corner of the Viewer
- Figures are not static, you can change them after they have been created
  – <u>If you do not want to change a Figure</u>, make sure one of *View 1 – View 4* is selected from the drop down menu
- To change the camera position for a figure (i.e. rotate / pan / zoom) select the figure from the Viewer drop down menu and move as necessary
  – All changes are automatically saved to the Figure



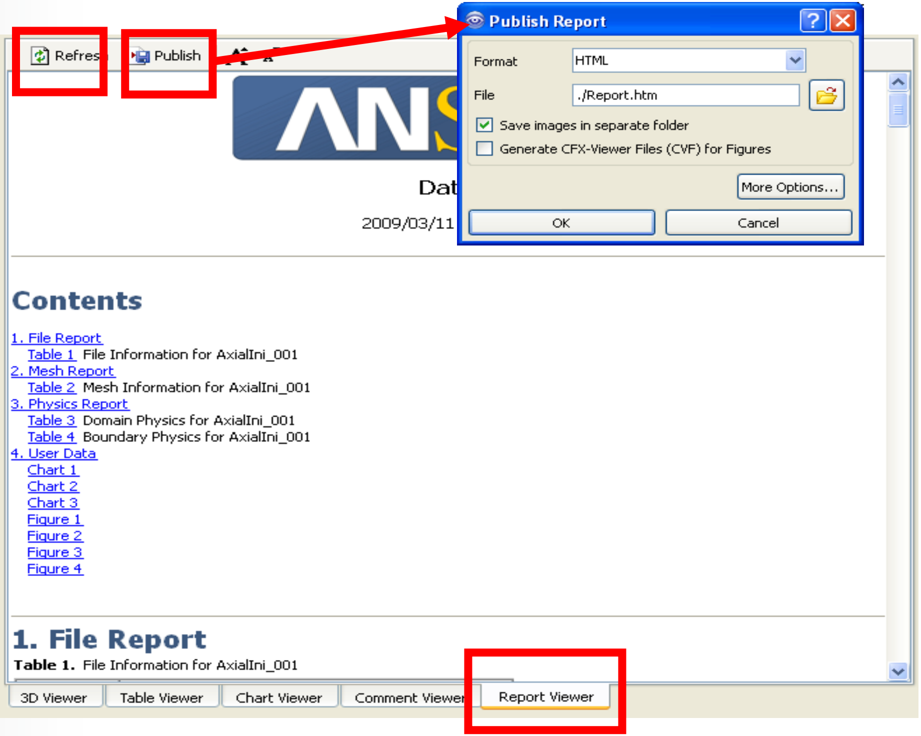
Reports: Figures

- **When you create a Figure, you have the option to *Make copies of objects***
  - **If you disable this only the camera and object visibility is stored with the figure**
    - So changing global objects will *always* cause the Figure to change
    - Good if you want the Figure to update automatically
  - **If you enable this a local copy of all the current objects is created and shown in the *Outline* tree**
    - Changing global object will not change the Figure, you must edit the local objects
  - **In both cases the camera position and object visibility can only be changed when the Figure is active**



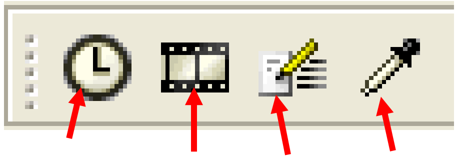
Reports

- To view the report, click the Report Viewer tab
- After making changes to objects contained in the report you will need to Refresh
- Publish writes out an HTML or Text copy of the report
  – You have the option can generate 3D Viewer files (see below) for all Figures

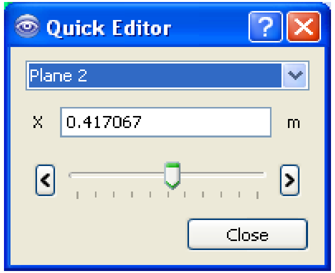


Other Tools

- **Timestep Selector**
  - Transient results are post-processed by loading in the end results file, then selecting different timesteps from the Timestep Selector



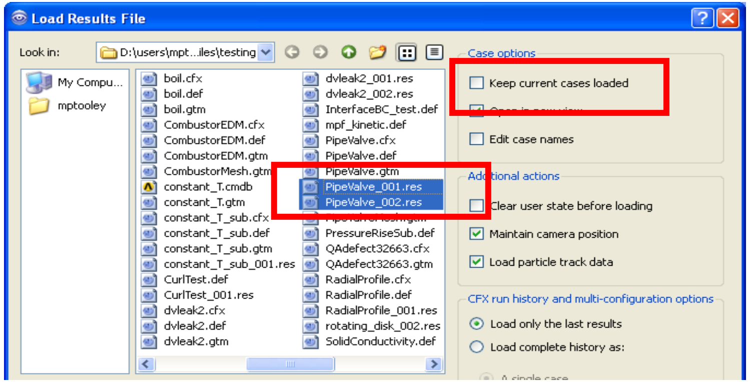
- **Animation**
  - Animate objects, create MPEGs
- **Quick Editor**
  - Provides a very quick way to change the “primary” value associated with each object

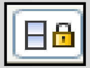


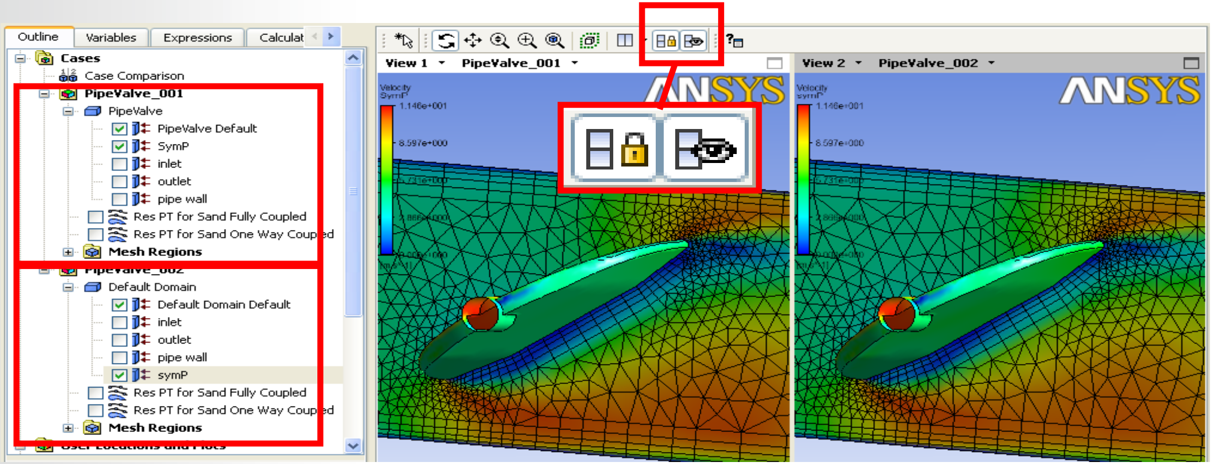
- **Probe**
  - Pick a point from the Viewer and probe a variable value at that point

Multi File Mode

- **Post-process multiple files simultaneously by:**
  - Multi-select files when loading
  - Load additional results and enable the *Keep current cases loaded* toggle
  - Each file is shown separately in the *Outline* tree and the *Viewer*



- **Sync cameras** 
  - Views move together
- **Sync objects** 
  - Visibility of locations and plots is the same

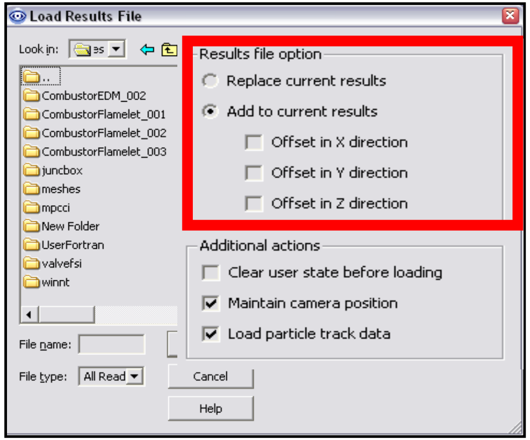


Files

- CFD-Post can interact with a number of different files including:
  - Results Files: CFX .res, ANSYS .rst, FLUENT.dat
  - Mesh Files: CFX .def., ANSYS .cmdb, FLUENT .cas,
  - Import: Polyline .csv, User Surface .csv, ANSYS surface .cdb
  - Export:Profile Data .csv, General Formatted Results .csv, ANSYS load file .csv
  - Recorded Session Files (.cse)
  - State Files (.cst)
  - Macros (.cse)

Files

- **Results**
  - **ANSYS**
    - Read ANSYS results for temperature, velocity, acceleration, magnetic forces, stress, strain, and mesh deformation
- **Import**
  - Locations: .csv files which contain point data which defines a polyline or surface
  - ANSYS Surface Mesh (.cdb): To allow for export of data on a surface for use as a boundary condition in ANSYS
- **Export**
  - Profile Boundary Data: for use in CFX-Pre
  - General formatted results data
  - ANSYS Load Data: Written onto an imported ANSYS .cdb file



Files

- **Session**
  - Session files can be used to quickly reproduce all the actions performed in a previous CFD-Post session
  - Session recording in CFX Command Language (CCL)
- **State**
  - Saves a snap-shot of all objects
  - Excludes actions (e.g. file output)
- **Macro**