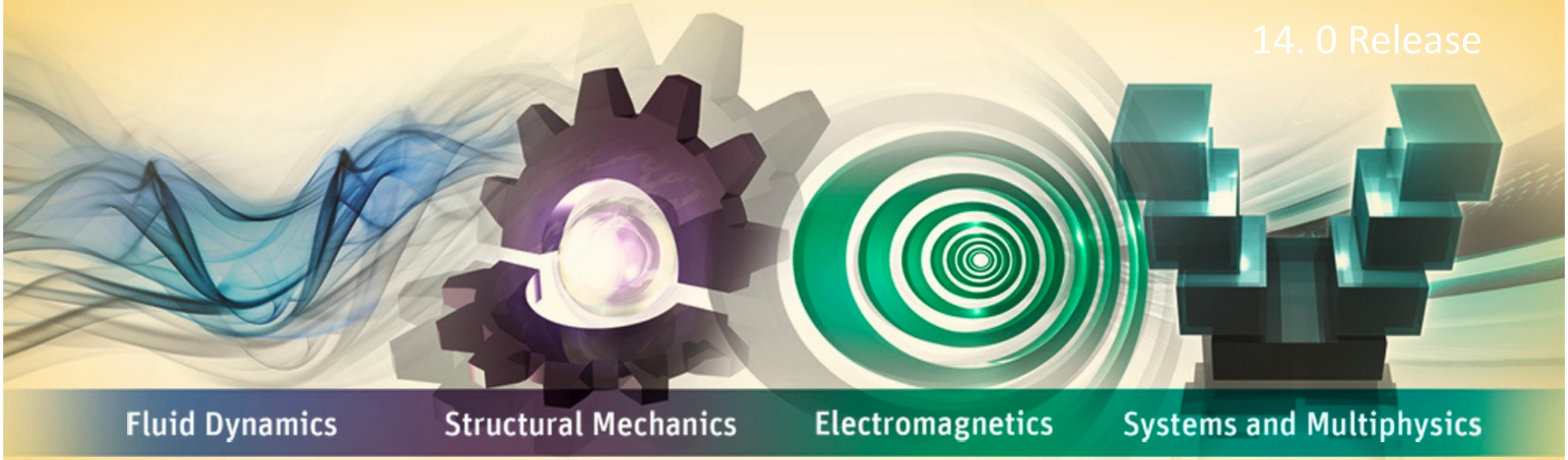
ANSYS

Realize Your Product Promise™

Lecture 3
Post-processing

14. 0 Release



Introduction to ANSYS FLUENT

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 both in CFD-Post and within FLUENT.

Learning Aims:

You will learn:

- How to perform flow field visualization and quantitative data analysis on your CFD results
- How to do this in FLUENT and in CFD-Post

Learning Objectives:

You will know what post-processing options are available and how to perform the post-processing work that your CFD simulations require

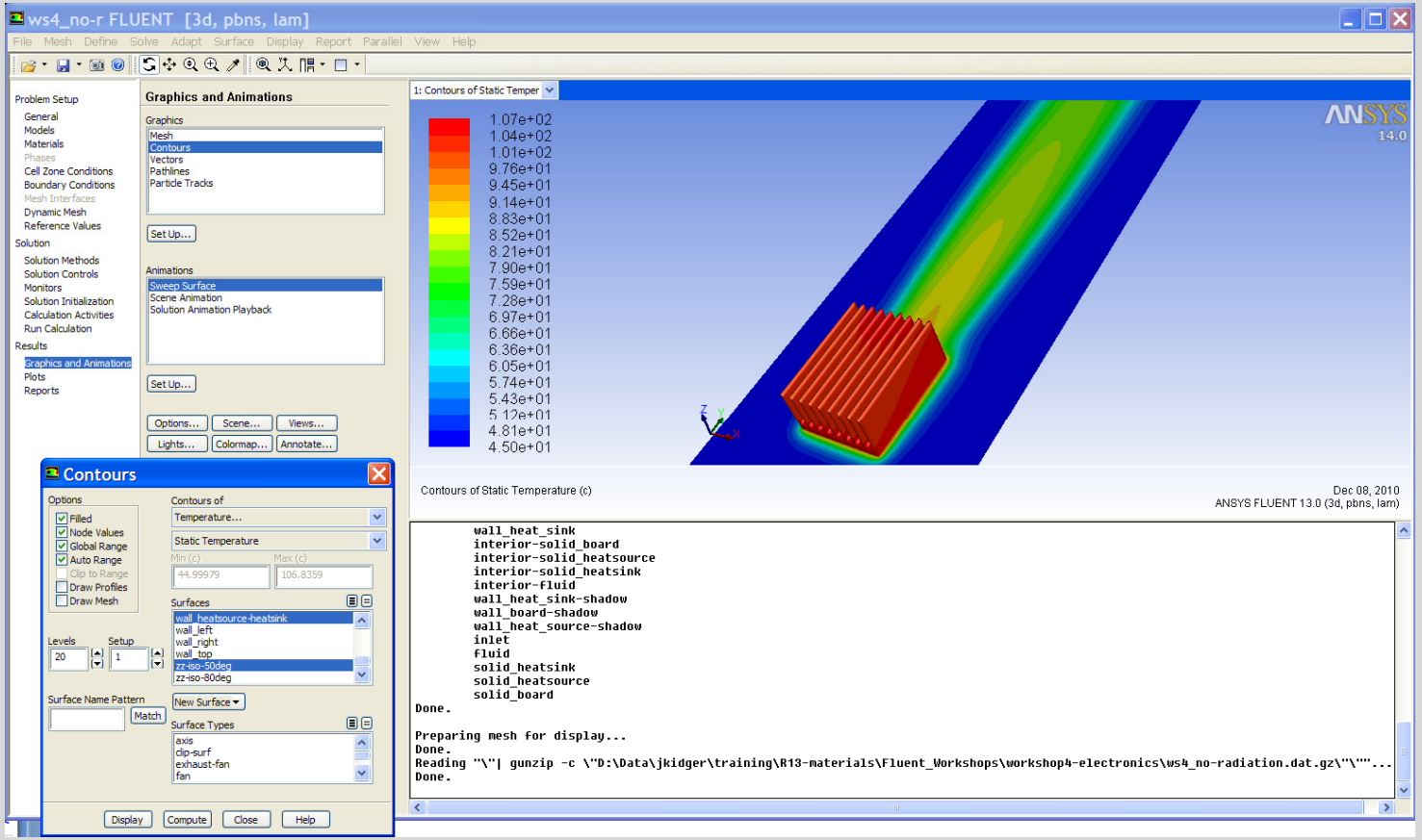
Overview

- There are two ways to post-process CFD results from FLUENT
  - **FLUENT post-processing tools** –tools integrated in the FLUENT solver.
  - **ANSYS CFD-Post application**
    - A state-of-the-art post-processor for ANSYS CFD products.
    - CFD-Post can run as a standalone post-processor, or within Workbench

- Both post-processors include many tools for analyzing CFD results
  - Isosurfaces
  - Vector plots
  - Contour plots (shaded and graded)
  - Streamlines and pathlines
  - XY plotting
  - Animation creation

Post-processing in FLUENT

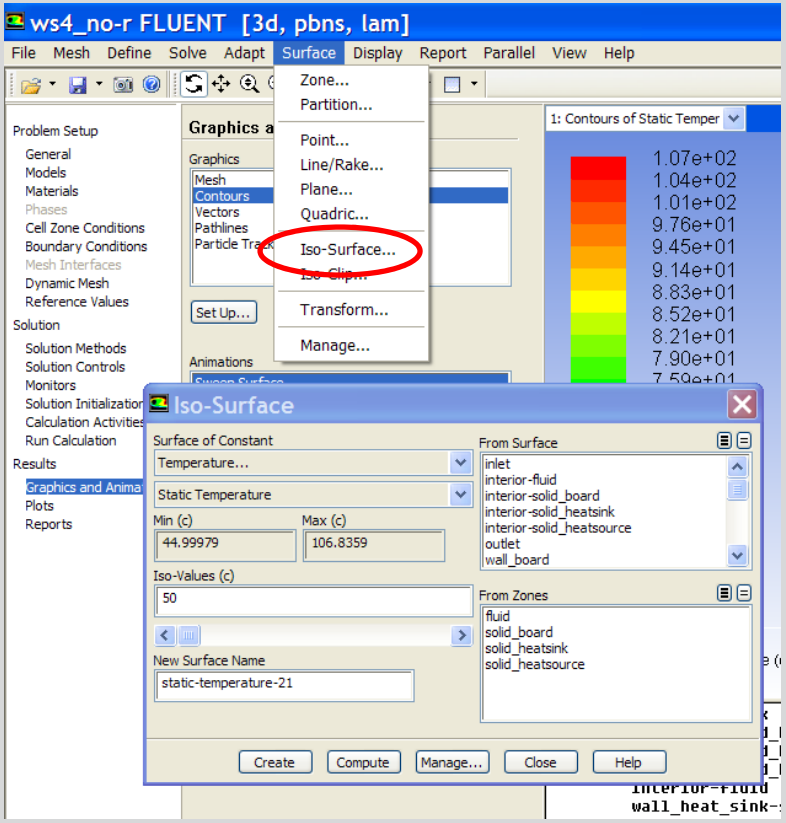
- The tools built into FLUENT have the advantage of letting you quickly review your simulation. Since all the data is still in memory for the solver, you can readily stop your simulation, interrogate the results, and then modify / continue with the calculations without waiting for the file write / read needed to use another application



Post-processing in FLUENT

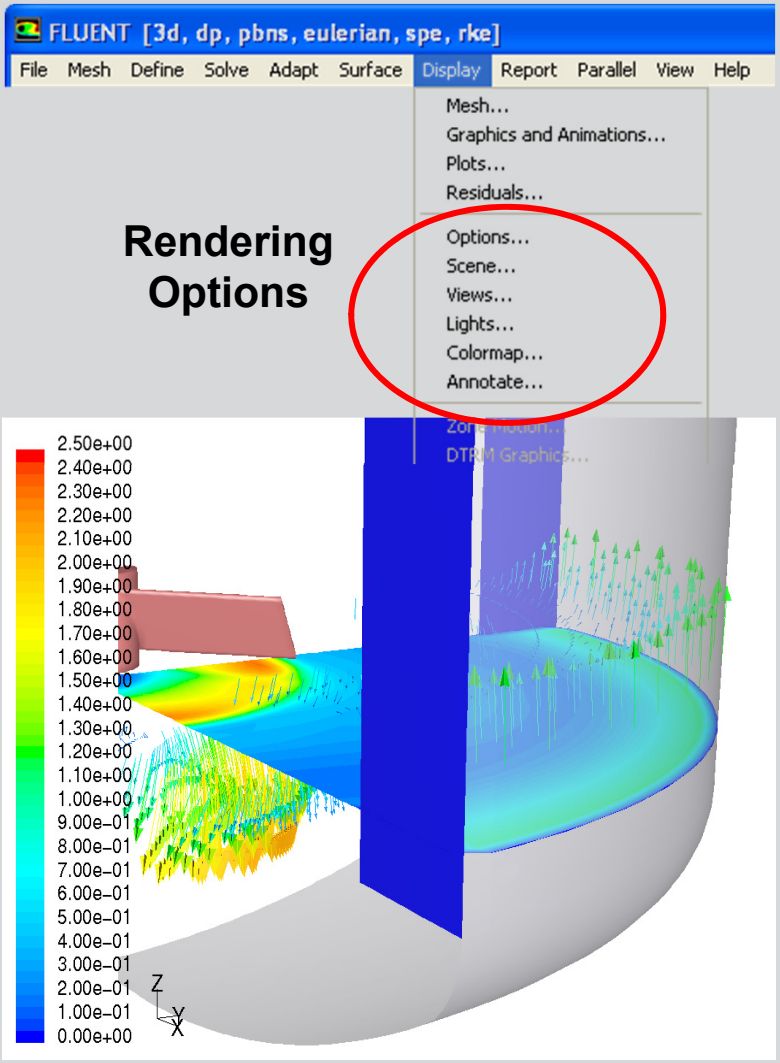
- The results can be reported / plotted either on existing surfaces present in the model, or on new surfaces.

- Commonly used options are:
  - A planar or cylindrical surface cutting through the model *(use Iso-Surface, based on constant mesh value, of x-coordinate, or radial coordinate)*
  - An Iso-surface to create a 3D surface showing where a given value exists (temperature, concentration)
  - A Line surface (used for XY plots)
  - A Point surface (used as a sampling probe to show values at a particular point)



Post-processing Rendering Options

- The rendering options in FLUENT allow control of the look-and-feel of the post-processing plots, including:
  - Views and display options
  - Colormaps for contour/vector plots
  - Shading on surfaces using Lights
  - Annotation of plots
  - Surface manipulation
  - Scene composition using plot overlays, different colors, shading, transparency
  - Scene animation (fly-throughs)

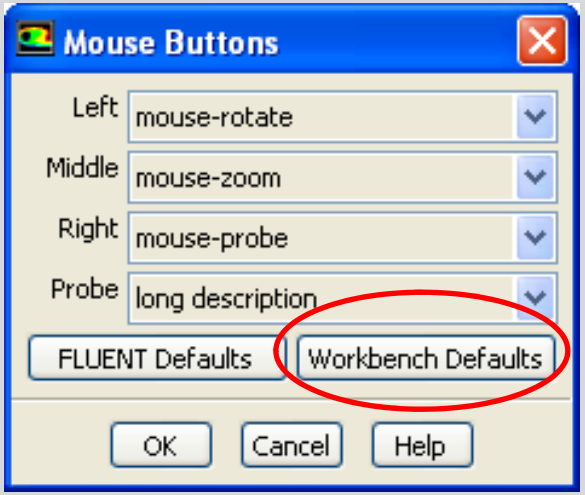


Example of scene composition: Overlay of contour and vector plot with transparent walls to show internal details.

Mouse Functionality

- Mouse button functionality depends on the chosen solver (2D / 3D) and can be configured in the solver.
- Default settings
  - 2D Solver
    - Left button translates/pans (dolly)
    - Middle button zooms
    - Right button selects/probes
  - 3D Solver
    - Left button rotates about 2 axes
    - Middle button zooms
      - Middle click on point in screen centers point in window
    - Right button selects/probes

Display → Mouse Buttons…

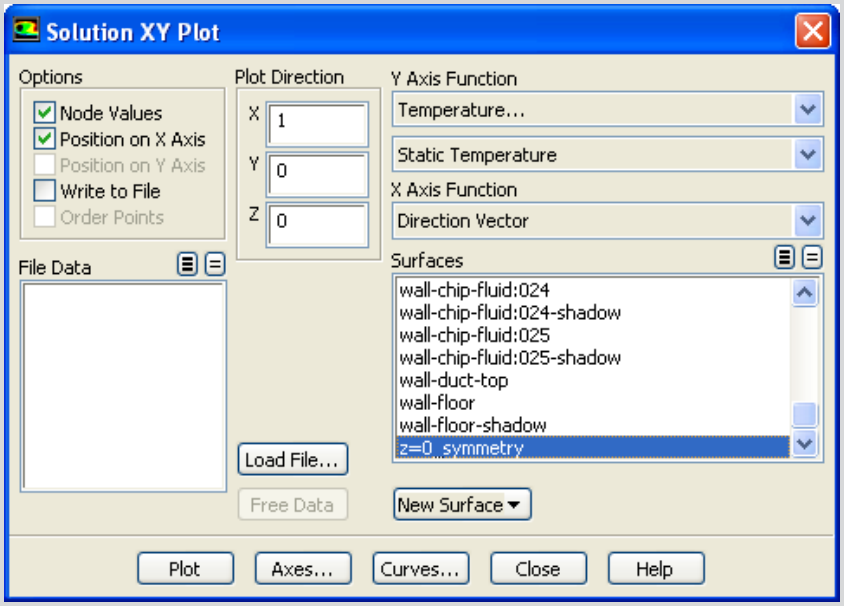


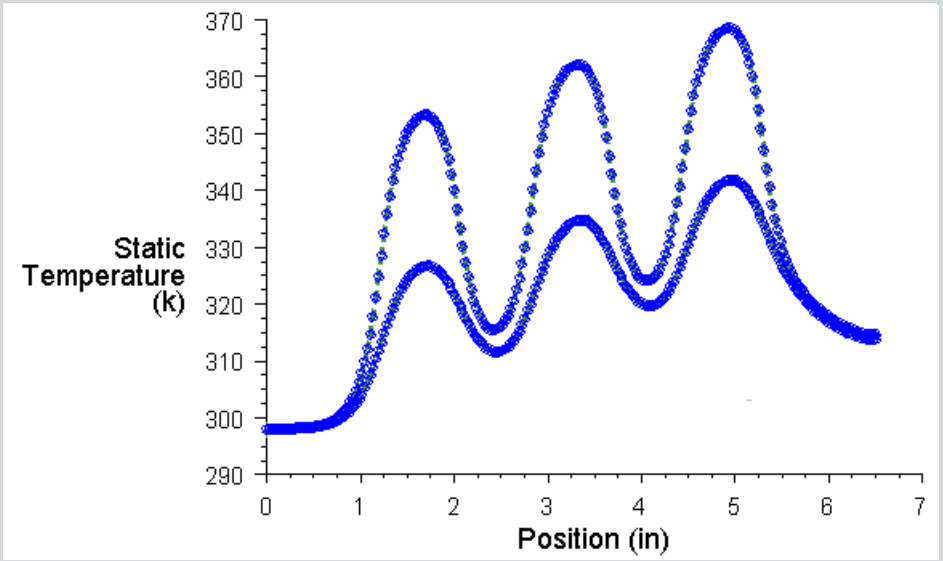
- Retrieve detailed flow field information at point with Probe enabled.
  - Right-click on the graphics display.
- User can choose between classic 'FLUENT' settings, or for mouse behavior consistent with Workbench.

Plots

- FLUENT provides tools to generate data plots of the solution:
  - XY plots of solution variables
  - Histograms to illustrate frequency of distribution
  - Fast Fourier Transforms (FFT)

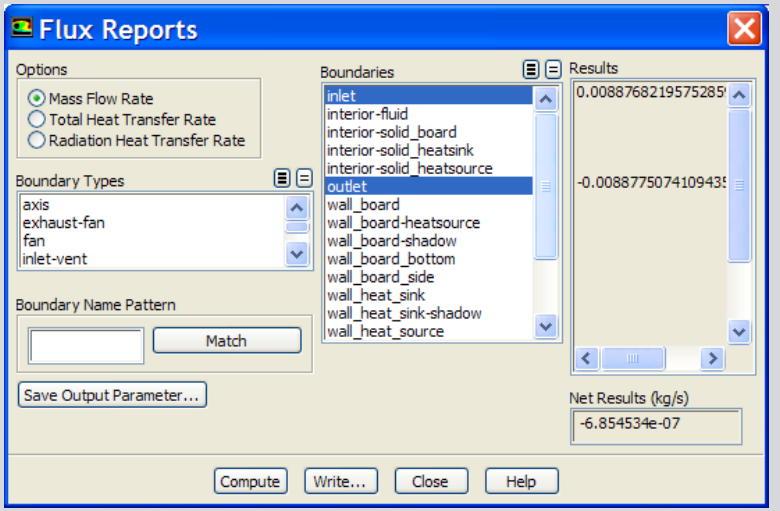
- Other data files (experimental, computational) can also be read in to compare results

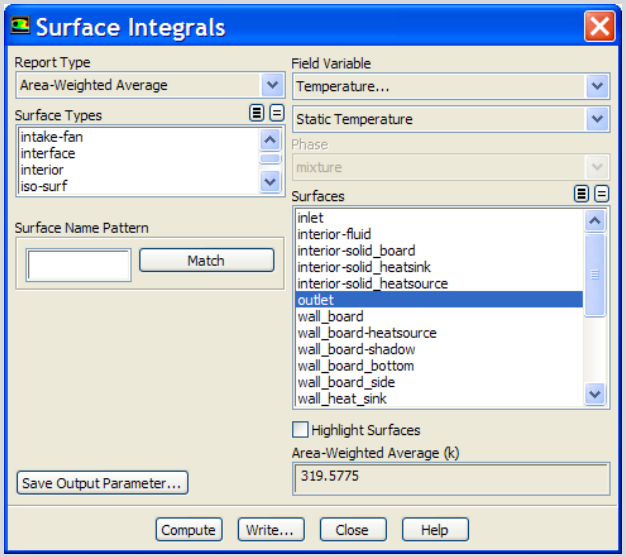


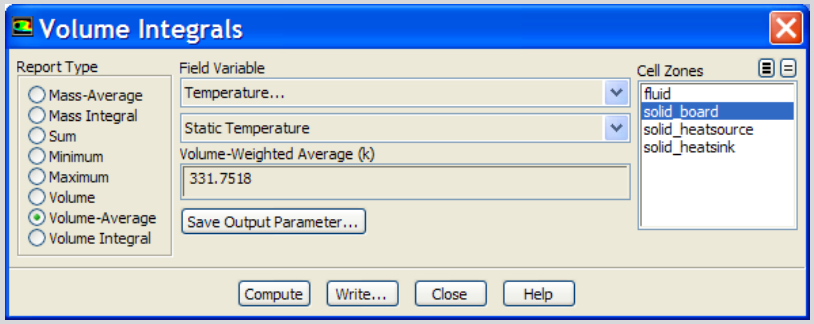


Reports

- The tools in FLUENT can also provide numerical data for analysis. For example:
  - **Flux Reports -** these show the total mass or energy flow through a boundary (a useful assessment of convergence)
  - **Surface Integrals** – these can show the total, average or max/min values of any quantity on any surface
  - **Volume Integrals** – show the sum, max/min or average values in a cell zone

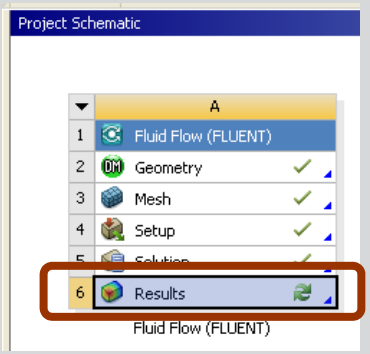


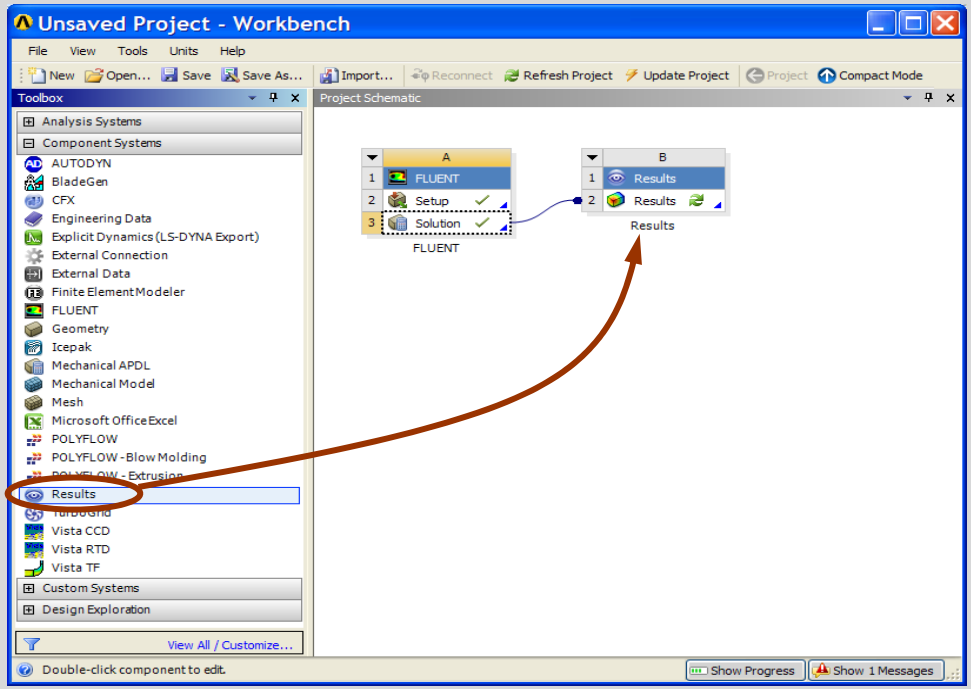




Starting CFD-Post

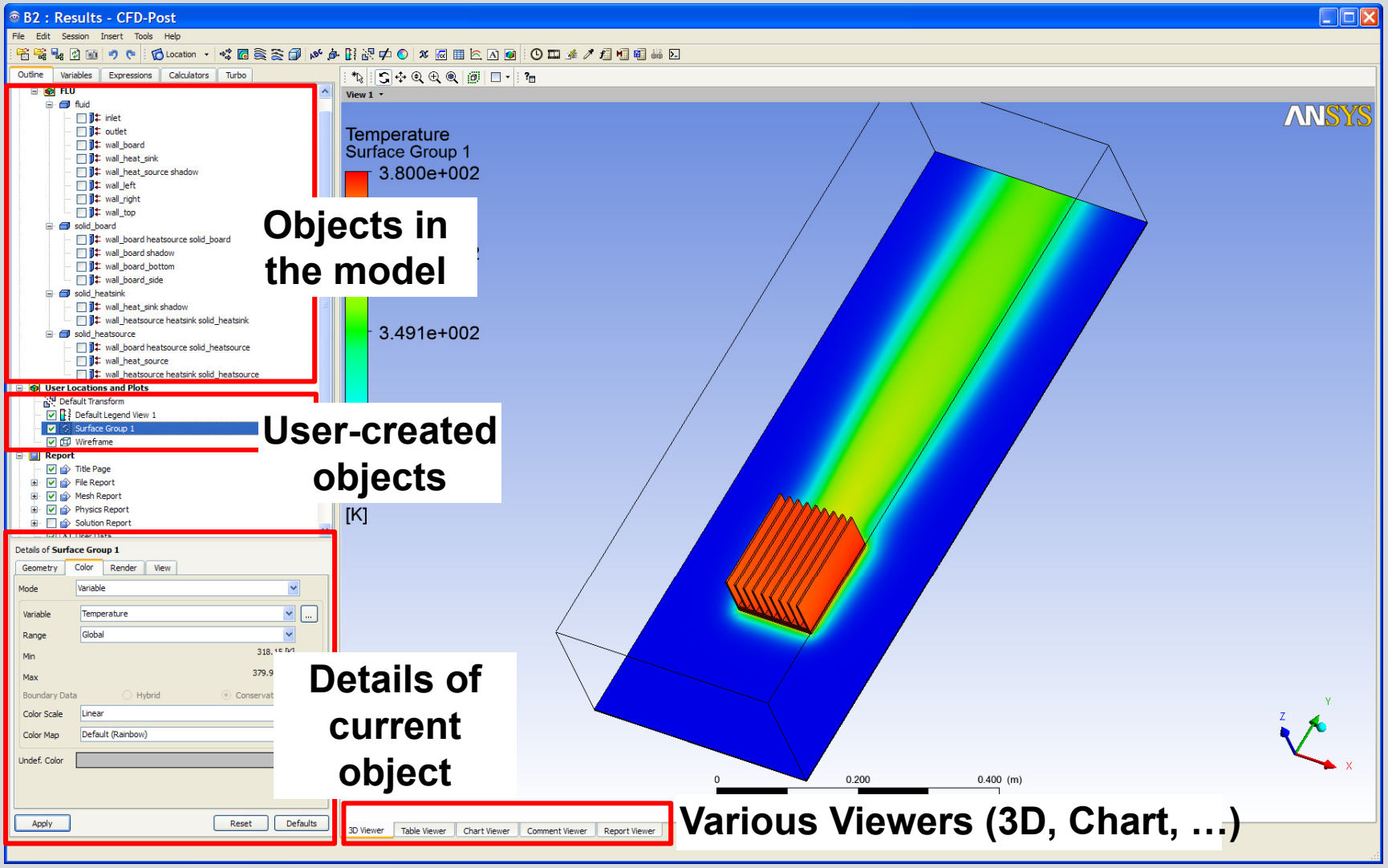
- Within ANSYS Workbench
  - If a FLUENT Analysis System was used, double click in the Results cell in the Project Schematic after the solution has been calculated
  - If a FLUENT Component System was used, Drag the CFD-Post icon (Results) in the Component Systems list to the project schematic.





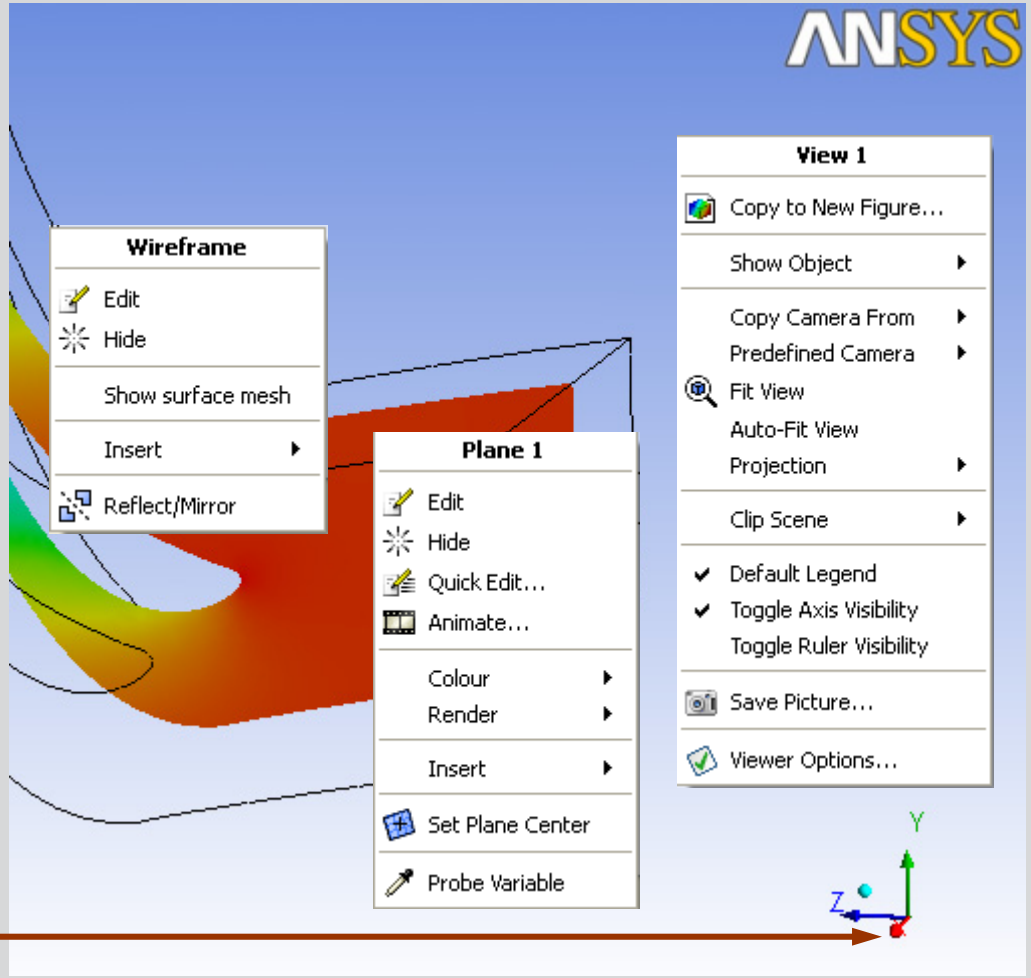
- Outside of Workbench:
  - **Start > Programs > ANSYS 14.0 > Fluid Dynamics CFD-Post**

GUI Layout


B2 : Results - CFD-Post
File Edit Session Insert Tools Help
Outline Variables Expressions Calculators Turbo
Objects in the model
User-created objects
Details of current object
Temperature
Surface Group 1
3.800e+002
3.491e+002
[K]
ANSYS
0 0.200 0.400 (m)
3D Viewer Table Viewer Chart Viewer Comment Viewer Report Viewer
Various Viewers (3D, Chart, …)

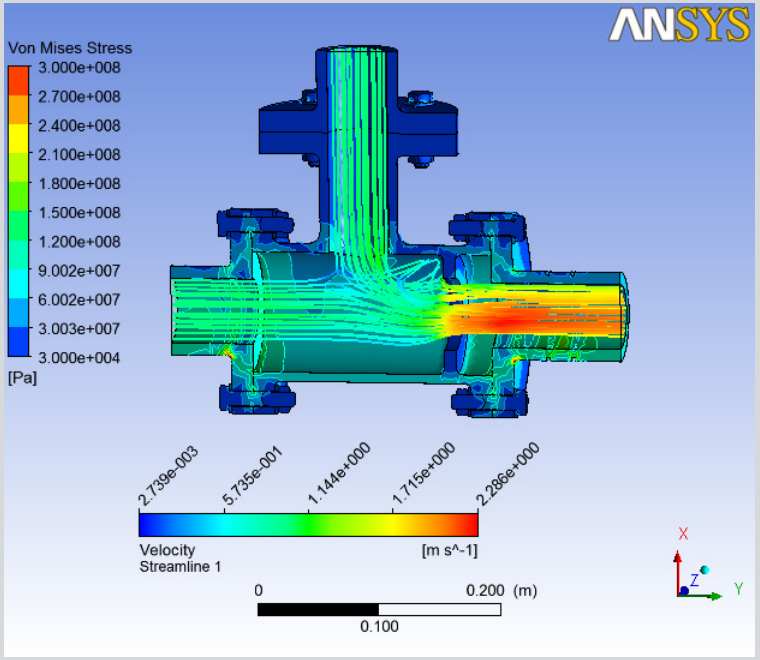
Viewer Right-click Menus

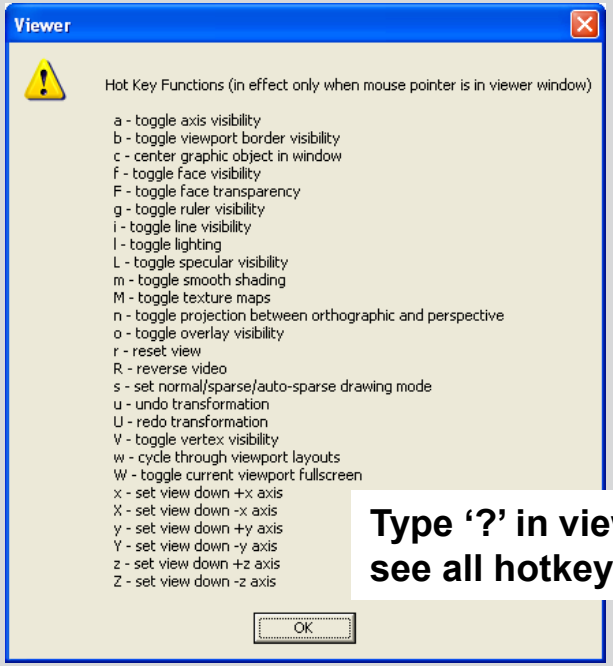
- Right-clicking in the Viewer provides a context-sensitive menu.
  - Right-clicking on an object (e.g. Wireframe, Plane) shows options for that object
  - Can also insert new objects based on the current location (such as a vector plot on a plane, contour plot on a surface, etc.)
  - Right-clicking in empty space shows options for the current View
- Click on the axes to orient the view



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 PowerPoints and HTML files
  - Download from the ANSYS Website (search for “CFD Viewer”)



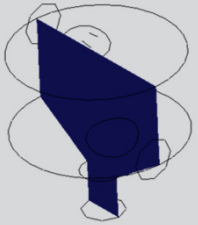


Type ‘?’ in viewer to see all hotkeys

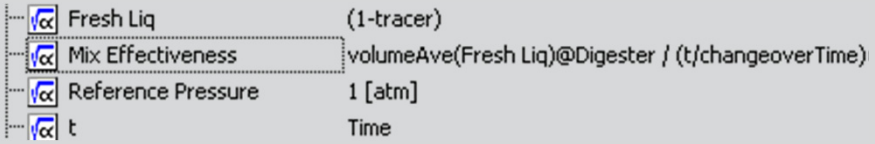
3D FSI Results (Courtesy of Cadfem Gmbh)

CFD-Post General Workflow

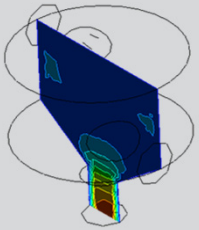
1. **Prepare Locations where data will be extracted from or plots generated**

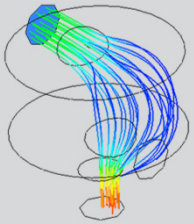


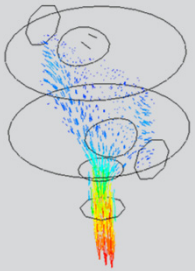
2. **Create variables/expressions which will be used to extract data (if necessary)**



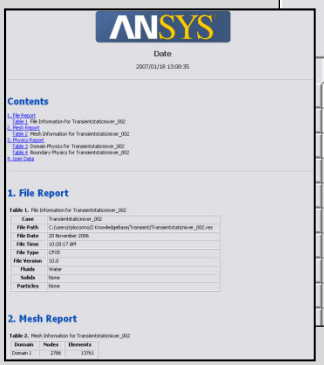
3. **i) Generate qualitative data at Locations**

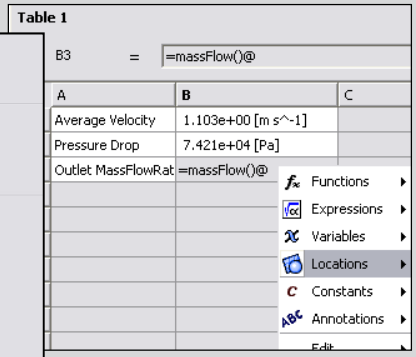


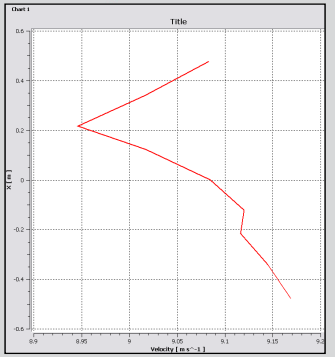




ii) Generate quantitative data at Locations







4. **Generate Reports**

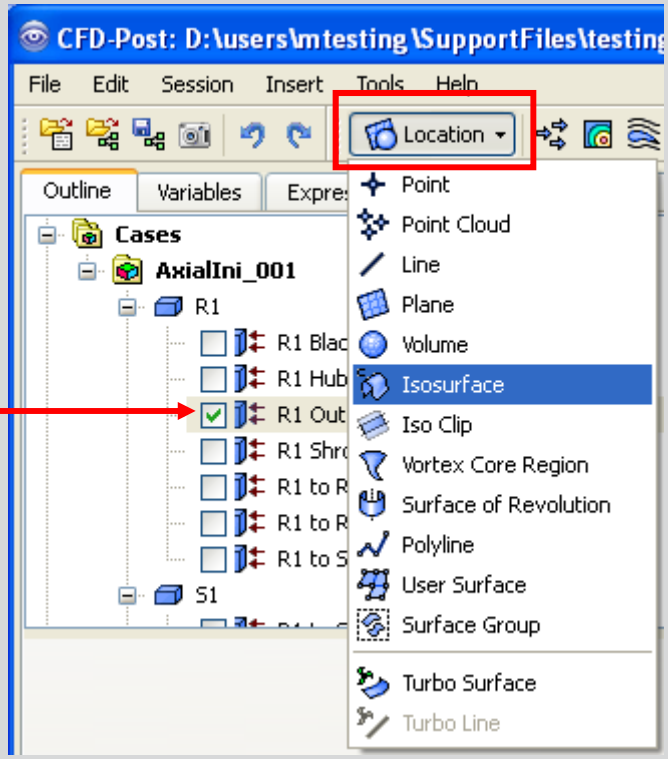
Creating Locations

- Locations are created from the *Insert* menu or from the toolbar
  - Many different types available (list appears in figure) and details of each type in the appendix
- Once created, all Locations appear as entries in the *Outline* tree

Use the check boxes next to each object in the Outline tree to quickly control visibility

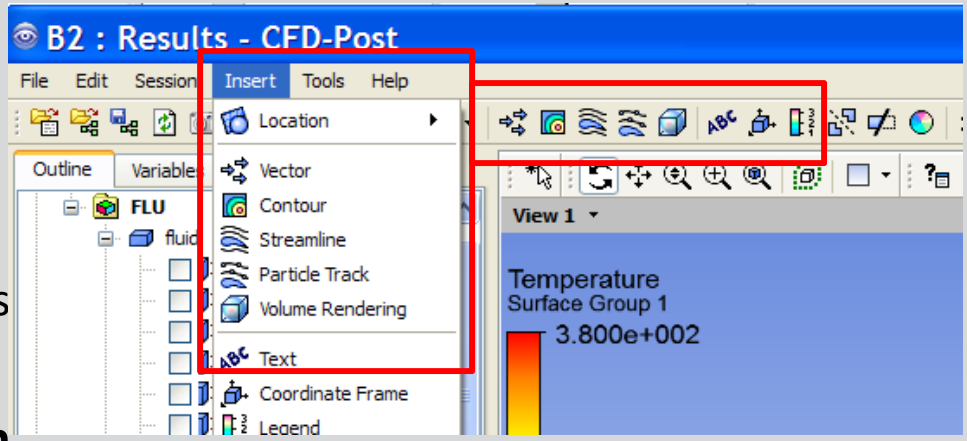
Double-click objects in the Outline tree to edit

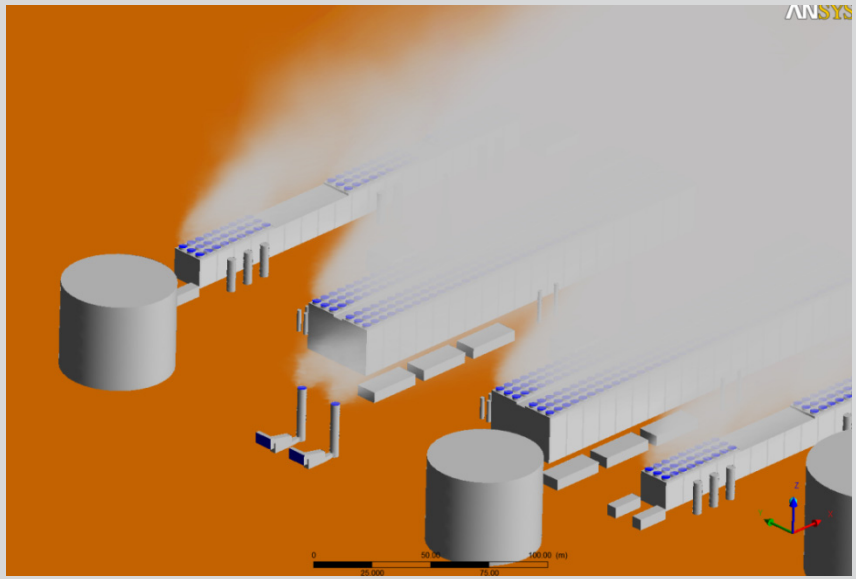
Right-click objects in the Outline tree to Duplicate or Delete



Other Graphics Objects

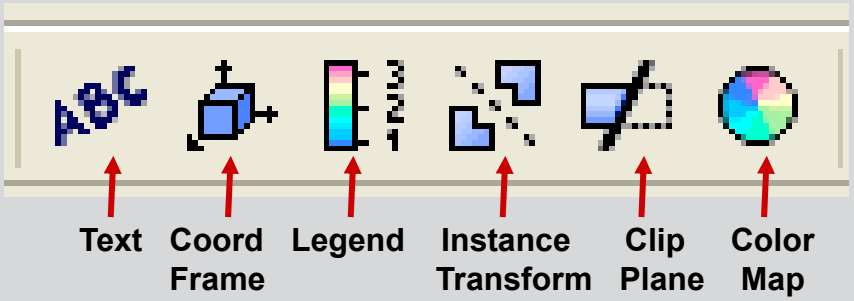
- **Vector Plot**
  - Can plot any vector variable; usually velocity
- **Streamlines**
  - Can proceed forwards and/or backwards from a seeding location
- **Vectors, streamlines and contours can use any existing object as a base**
- **Volume Rendering**
  - Shades every grid cell with an opacity based on the magnitude of a variable
  - So for example, opacity could be based on smoke concentration. The result will give an idea of visibility though the smoke.

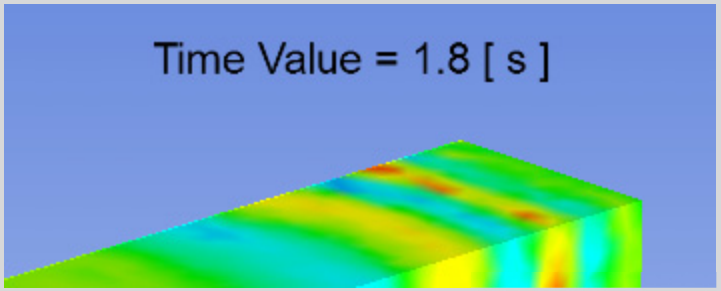


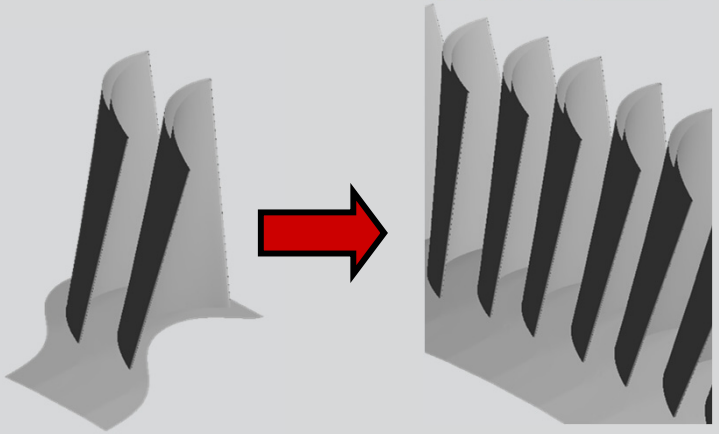


Other Graphics Objects

- **Text:** add your own labels to the Viewer
  - Auto-text allows you to show time step/values, expressions, filenames and dates that change automatically
- **Coord Frame**
  - Insert a new local coordinate frame, useful for generating plots relative to features in the model
- **Legend**
  - Create additional legends that are tied to a specific plot (the default legend changes automatically with the active plot)
- **Instance Transform**
  - Create patterns and duplicates of the geometry
  - Usually used to re-create full plots from symmetric/periodic solution data

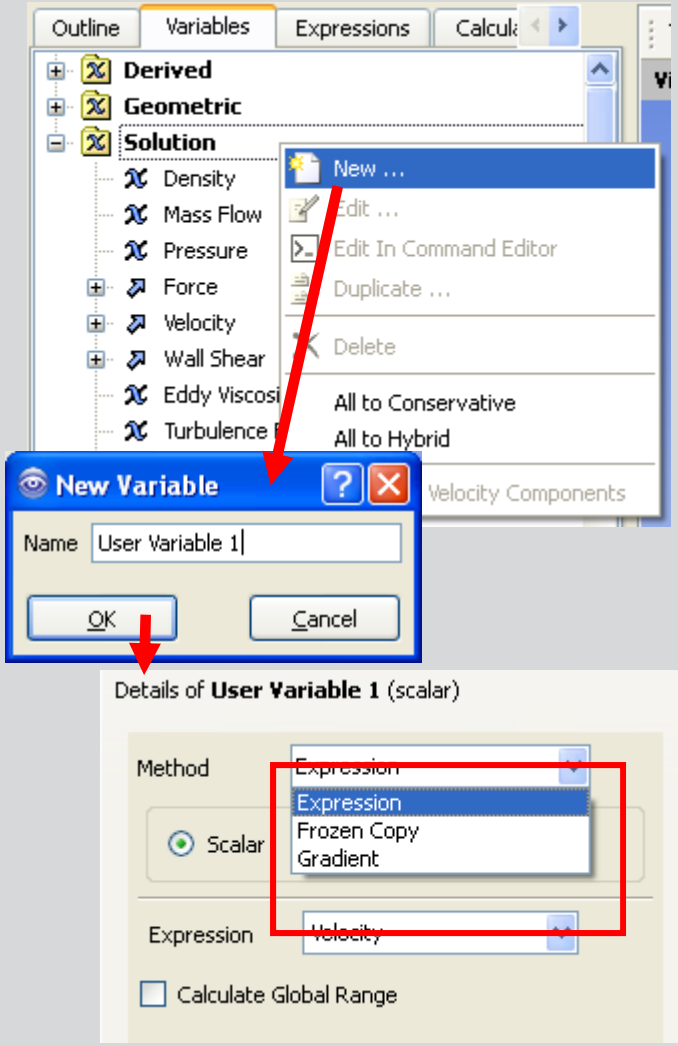






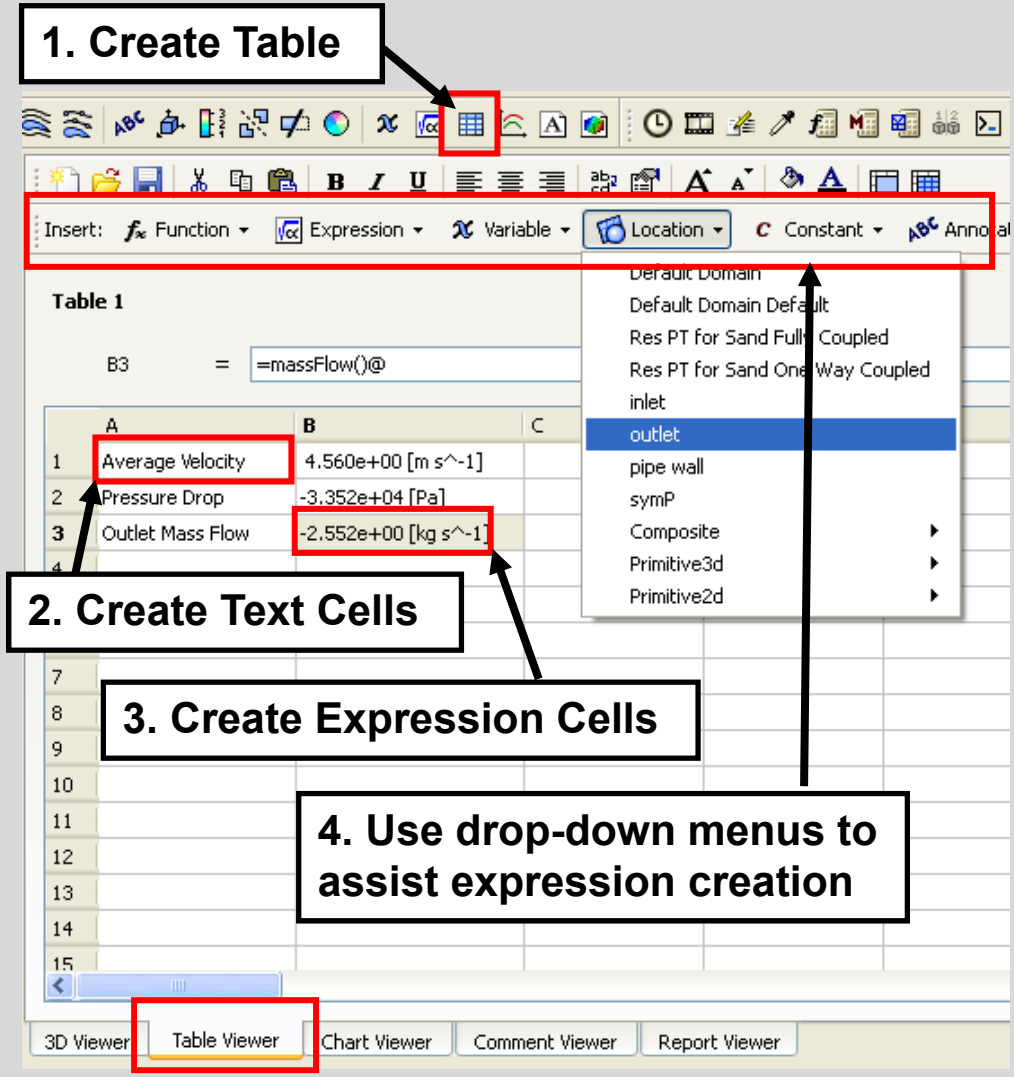
Variables Tab: User Defined Variables

- **User defined variables can be used to form new variables which are functions of the basic solver variables such as pressure, temperature, velocity, mass flow rate, wall shear, ...**
- **Create new variables by *Right-click > New…* in the top half of the *Variables* tab**
- **There are 3 methods for User Defined variables:**
  - The *Expression* method defines a variable via an expression, which can be a function of any other variable
    - Usually create the expression first on the *Expressions* tab ➔ *Example in Appendix*
  - *Frozen Copy* has been superseded by *Case Comparison*
  - The *Gradient* method calculates the gradient of any existing scalar variable
    - Produces a new vector variable

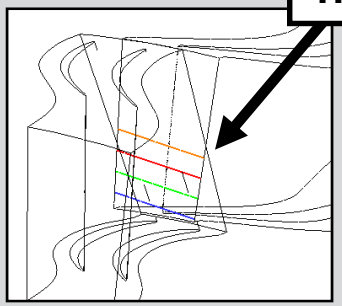


Tables

- **Select *Insert > Table* or use the toolbar icon to create a new table**
  - *3D Viewer* will switch over to the *Table Viewer*
- **Tables allow you to display data and expressions in a tabular view**
- **Tables are 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
- **This is not a spreadsheet**
  - Cannot reference other cells



Charts



1. Create Lines

2. Create Chart

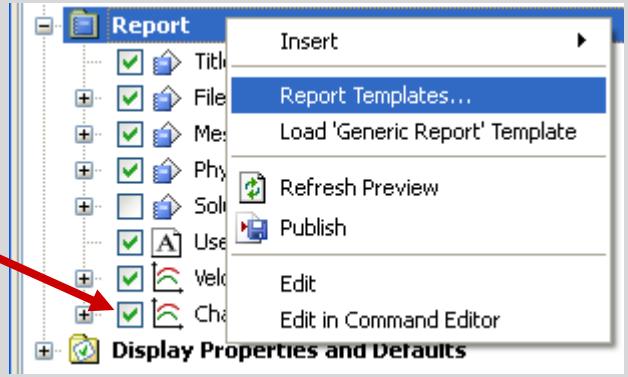
3. Select Chart Type

4. Create Data Series (Lines)

5. Select X and Y Axis variables

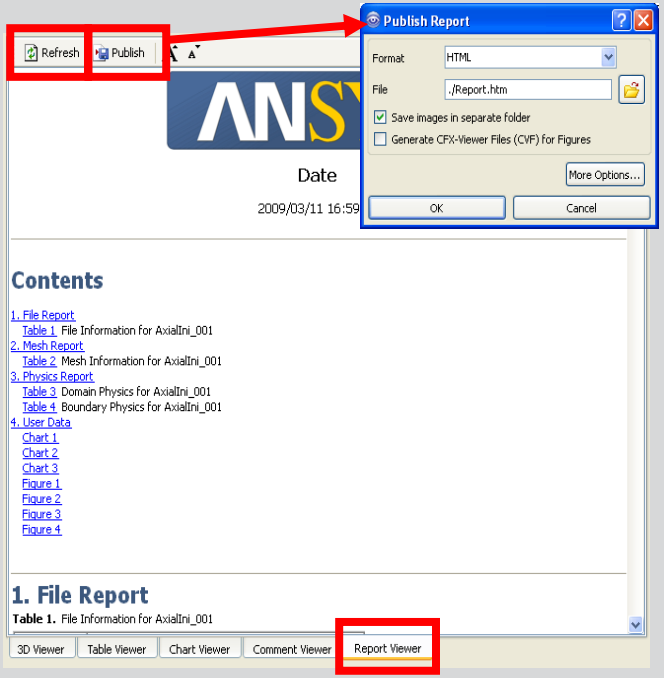
Reports

- **CFD-Post has report generation tools which allow 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



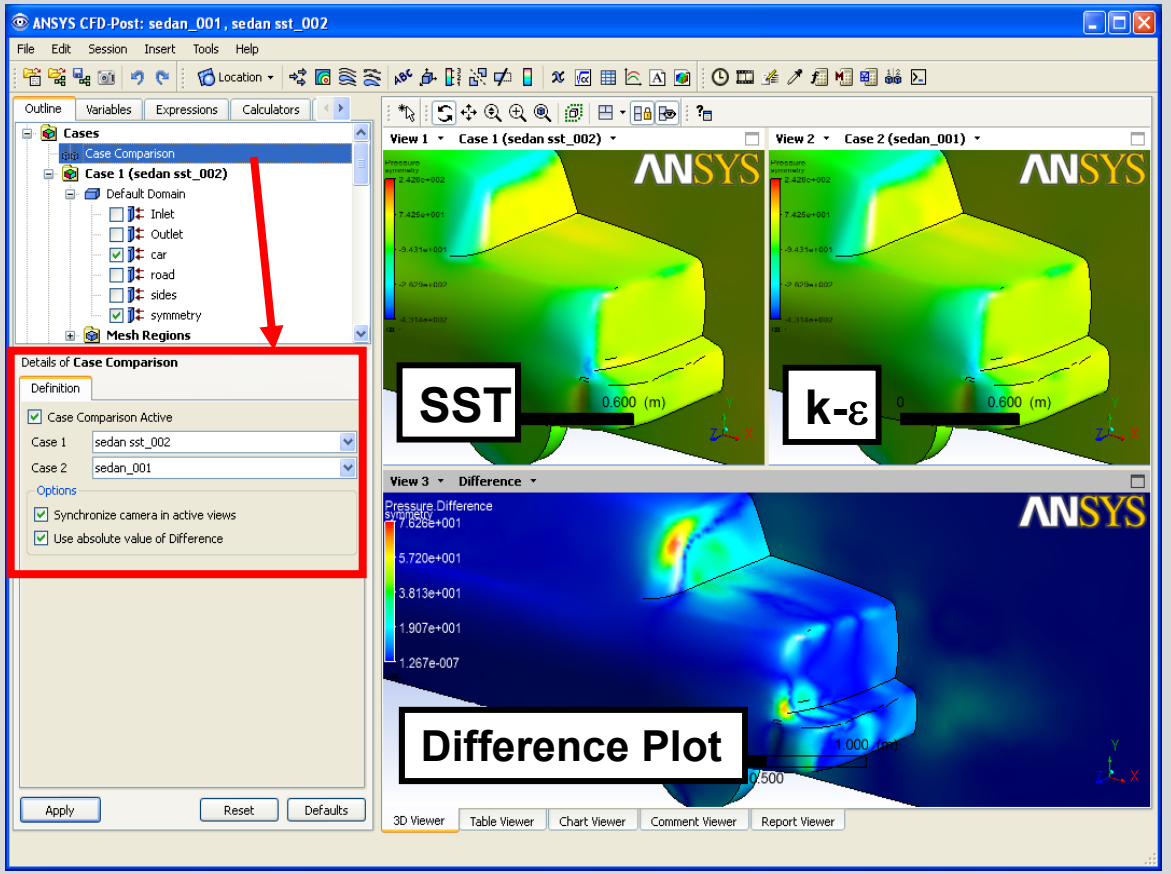
- **Reports are template based**

- ***Publish* writes out an HTML or Text copy of the report**



Case Comparison

- **CFD-Post allows multiple cases to be loaded simultaneously**
- **When multiple files are loaded you can select *Case Comparison* from the *Outline* tree**
  - Automatically generates contour plots of the difference between the two cases for any variable
  - The displayed views can be synchronized between the two cases



- **Expression syntax:**
  - `function()@CASE:#.Location`
  - E.g: `areaAve(Pressure)@CASE:1.Inlet`
  - `See Appendix for details of Expression usage`

Comparison of FLUENT and CFD-Post

- In CFD-Post
  - Automate post-processing through Session files, State files and Report templates
  - Make use of Expressions and User Variables to extract engineering data
  - Compare solutions using the Multi-file mode and the Case Comparison tools
  - Save images in the 3D CFX Viewer format to provide management or your customers with a better understanding of the flow

- In FLUENT
  - Post-processing can be performed from within the solver, which is often more convenient for most basic post-processing operations

Summary and Conclusions

Summary:

Post-processing can be performed using either FLUENT or CFD-Post

Many basic operations such as contour plots, vector plots and streamlines can be performed in both

FLUENT may be more convenient when only basic operations are required because the post-processing is performed in the solver – not necessary to write files and start a separate program

CFD-Post contains many powerful, sophisticated post-processing capabilities including 3D-viewer files, user variables, automatic html report generation and case comparison

What Next:

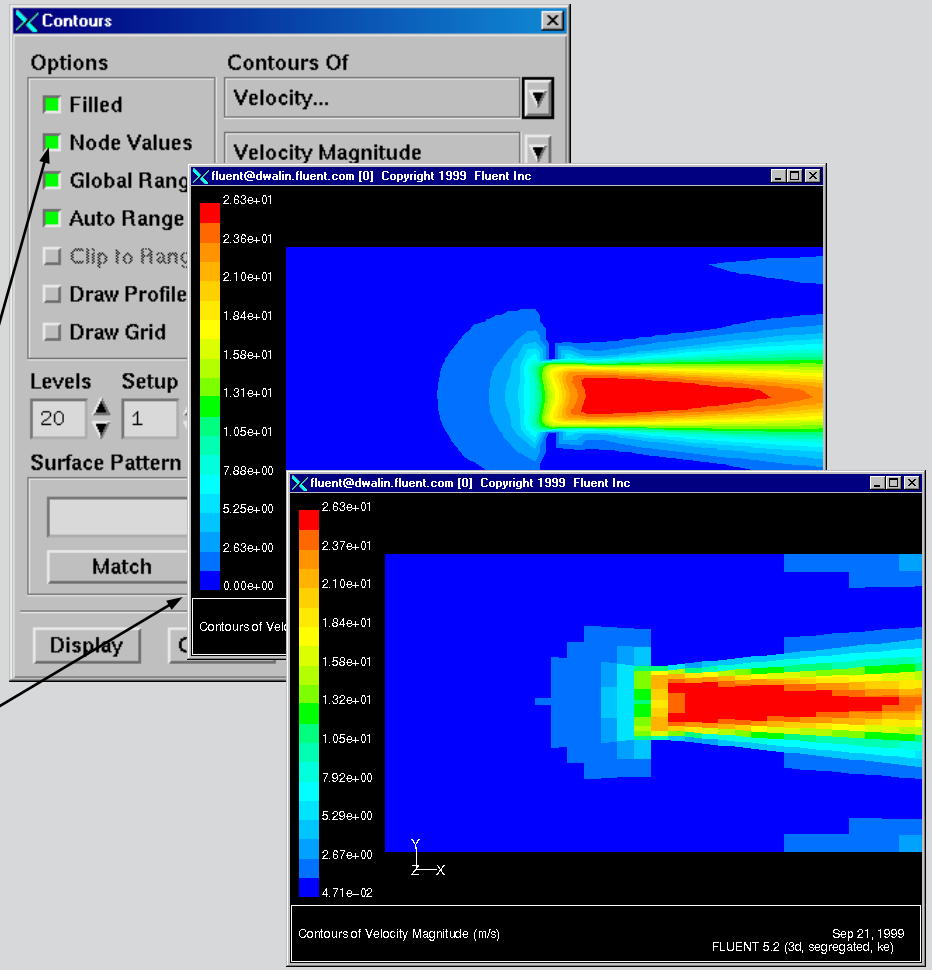
Post-processing is best learned in a hands-on manner. Details of the operations described in the lecture will be covered during the completion of Workshop 3

ANSYS

Appendix

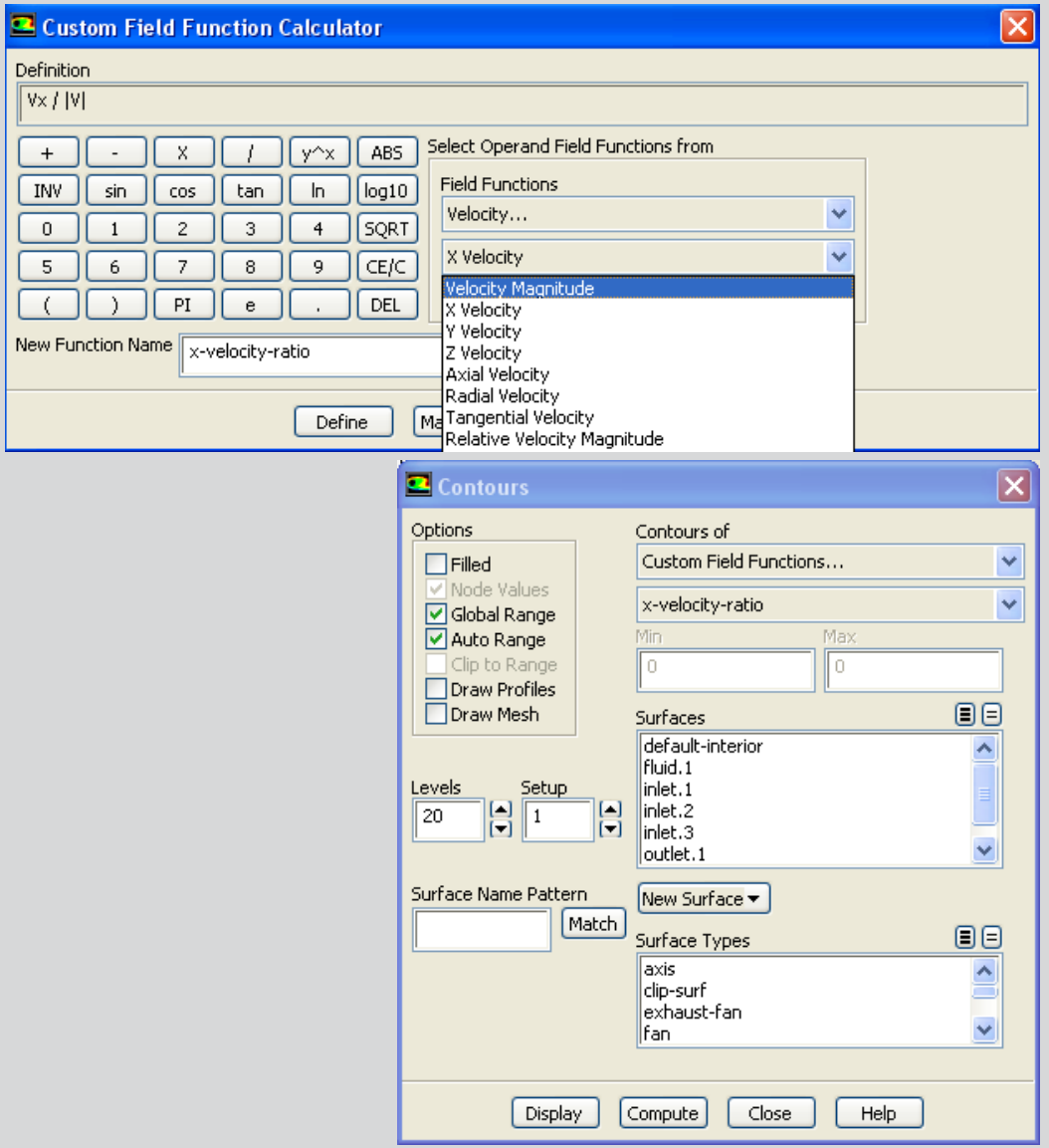
Post-Processing in FLUENT: Node Values

- **Fluent calculates field variable data at cell centers**
- **Node values are either:**
  - calculated as the average of neighboring cell data away from boundaries
  - defined explicitly on boundaries with boundary condition data (when available)
- **Node values on surfaces are interpolated from grid node data**
- **Data files store:**
  - data at cell centers
  - node value data for primitive variables at boundary nodes
- **Enable Node Values to interpolate field data to nodes**



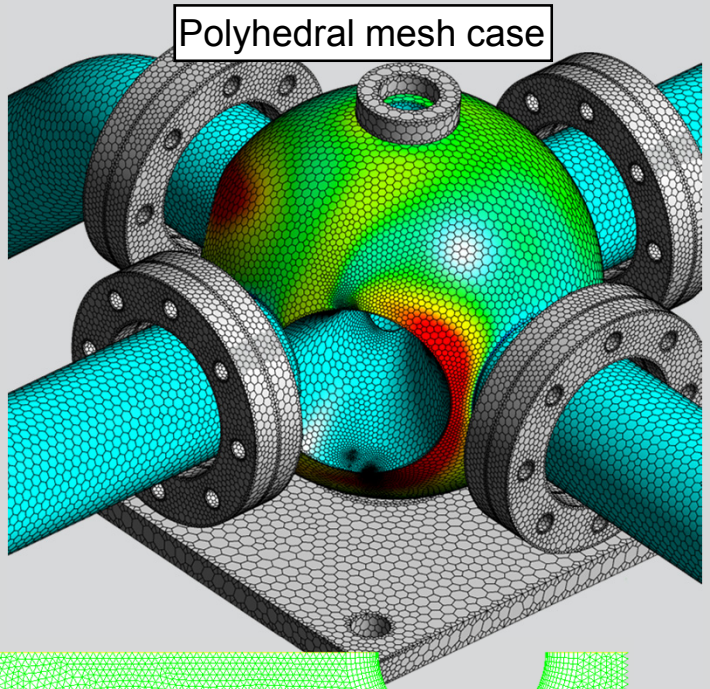
Custom Field Functions in FLUENT

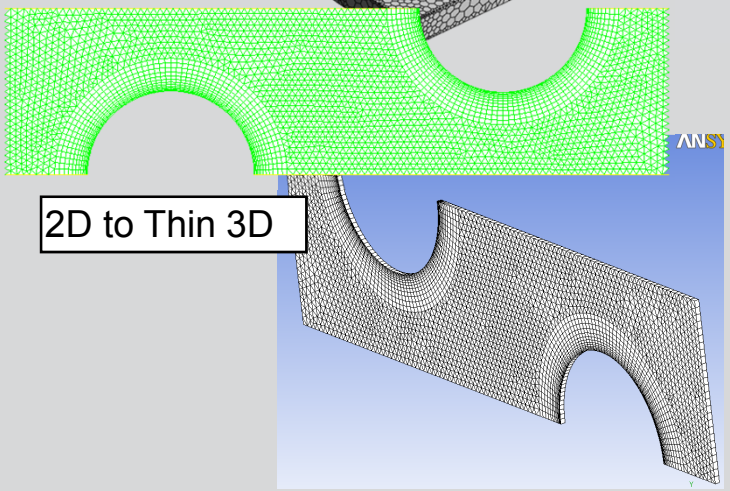
- In addition to the basic field variables provided by FLUENT, it is possible to define custom field functions for use with the post-processing tools
  - In the menu bar Define > Custom Field Functions, to open the Custom Field Function Calculator panel
- Use the menus and calculator buttons to form expressions from the basic field variables
- Custom Field Function variables will appear in post-processing menus
- Custom field function definitions are saved in the case file and can be transferred to other case files



Limitations of CFD Post

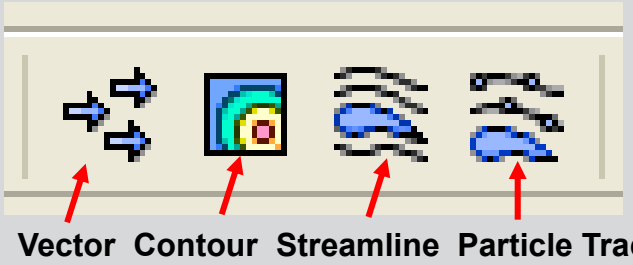
- **All mesh types supported**
  - Polyhedral, non-conformal, adapted, ...
  - 2D FLUENT meshes are extruded to thin 3D domains
  - 2D axisymmetric meshes are converted to 3D wedges
- **Some data may not be in the standard .dat file**
  - Export through the Data File Quantities or the Export to CFD-Post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

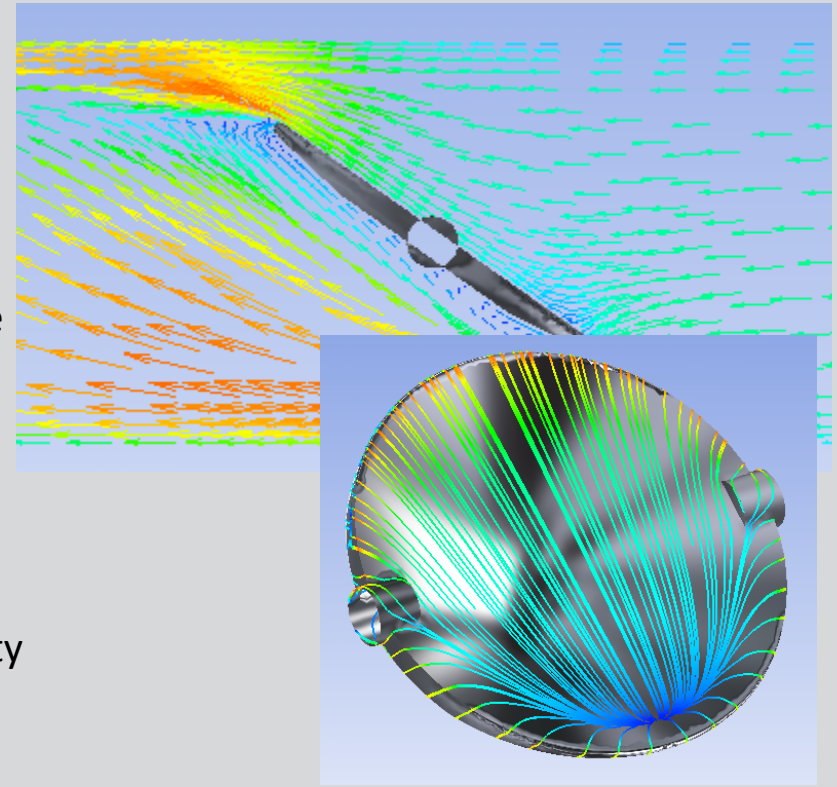




Other Graphics Objects

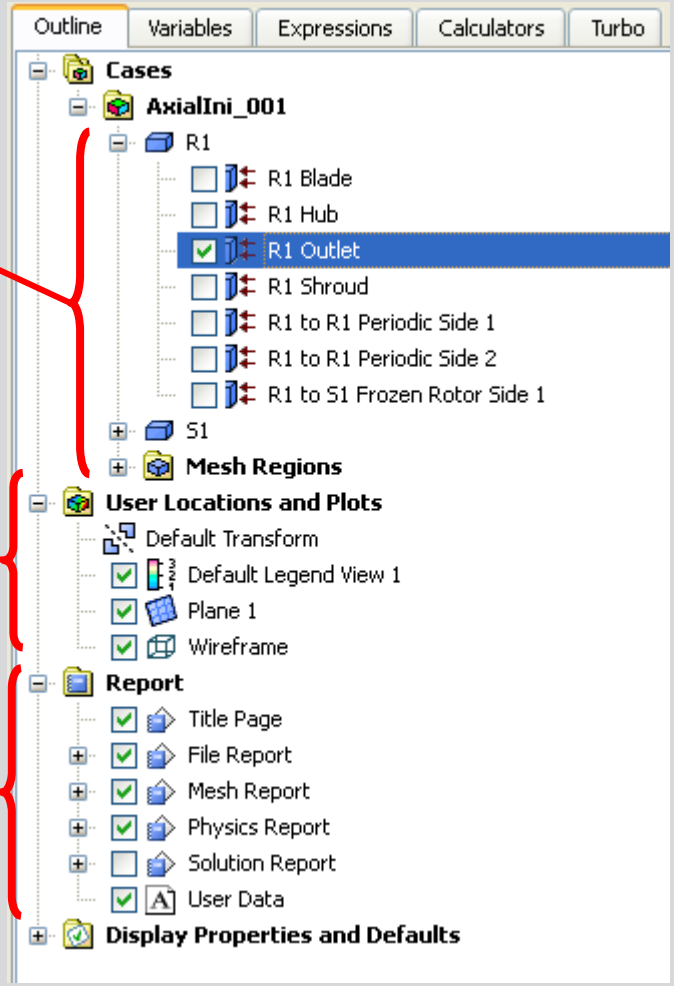
- **Insert from the toolbar or the Viewer right-click menus**
- **Vectors, Contour and Streamlines use existing Locations as a base**
- **Vector Plot**
  - Can plot any vector variable
    - usually velocity
  - Can project vectors Normal or Tangential to the base object
- **Streamlines**
  - Can proceed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





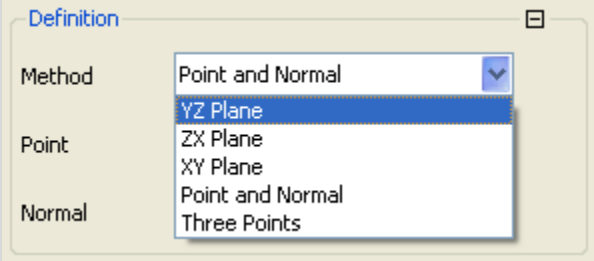
Creating Locations

- **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**

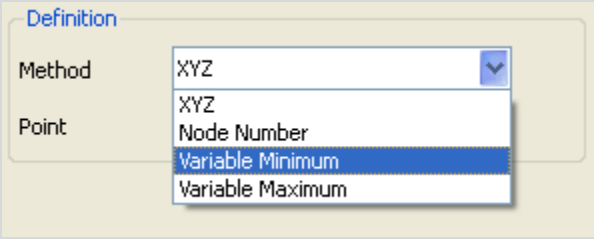


Location Types

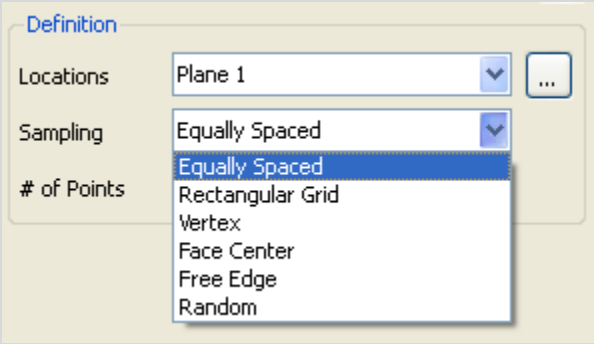
- **Planes**
  - *XY Plane*, *Point and Normal*, etc.
  - Can define a circle or rectangle to bound the plane, otherwise it's bounded only by the solution domain(s)



- **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

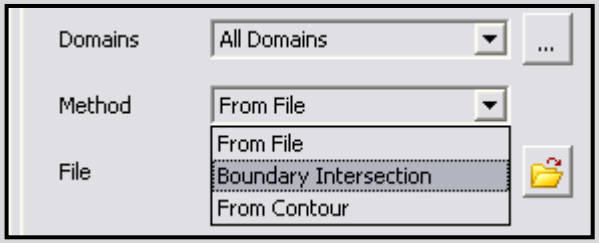


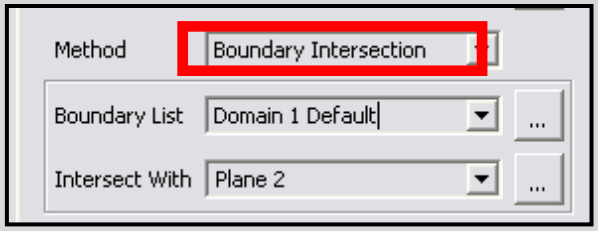
- **Point Cloud**
  - Create multiple points
  - Usually used as seeds to streamlines, vectors

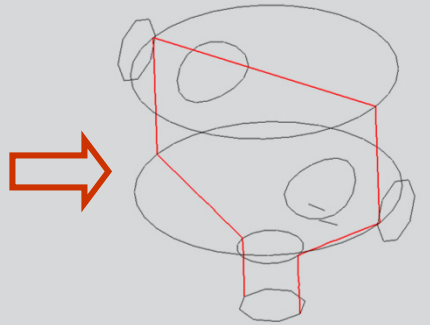


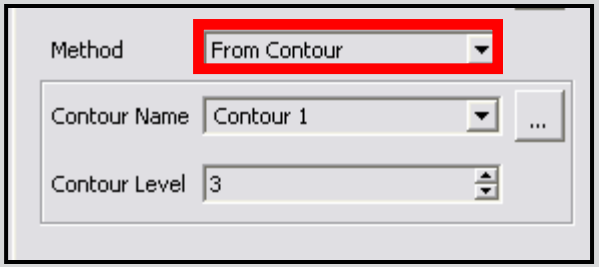
Location Types

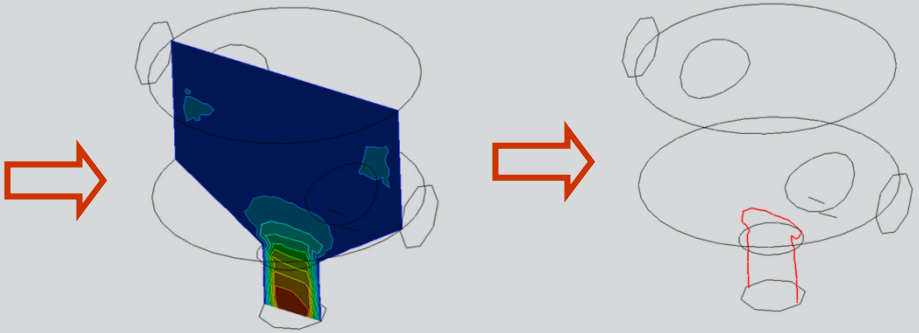
- **Lines**
  - Straight line between two points
  - Usually used as the basis for an XY Chart
- **Polylines**
  - Also used for Charts
  - Read points from a file
  - Use the line of intersection between a boundary and another Location
  - Extract a line from a contour plot





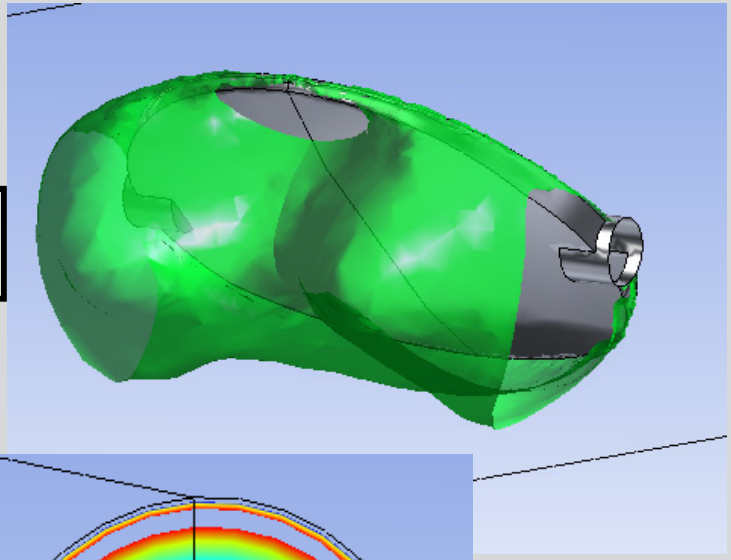






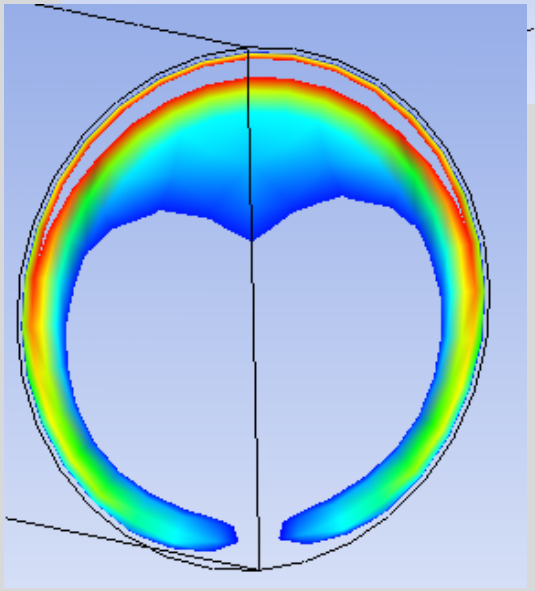
Location Types

- **Isosurfaces**
  - Surface of a variable at a specified value



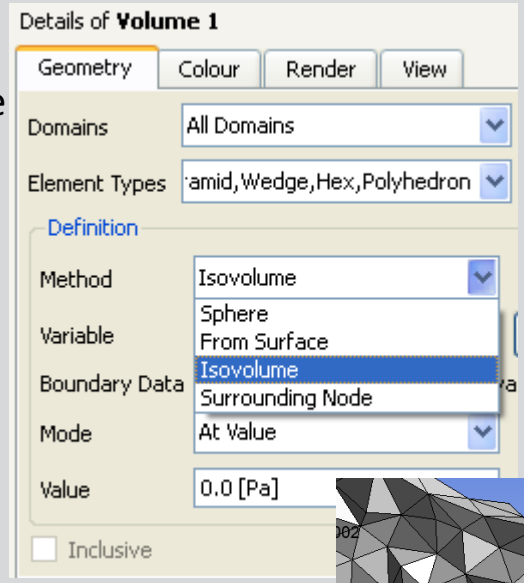
Isosurface of pressure behind a flap valve

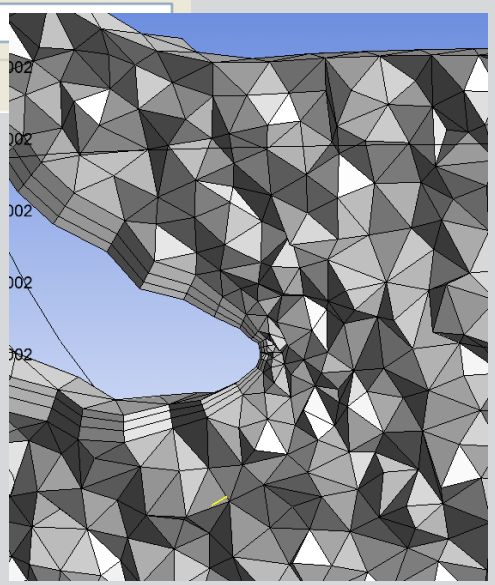
- **Iso Clip**
  - An Iso Clip takes a copy of any existing Location and then clips it using one or more criteria
    - E.g. the velocity contour plot on the outlet boundary is then clipped by Velocity >= 10 [m/s] and Velocity <= 20 [m/s]
  - Can clip using any variable, including geometric variables



Location Types

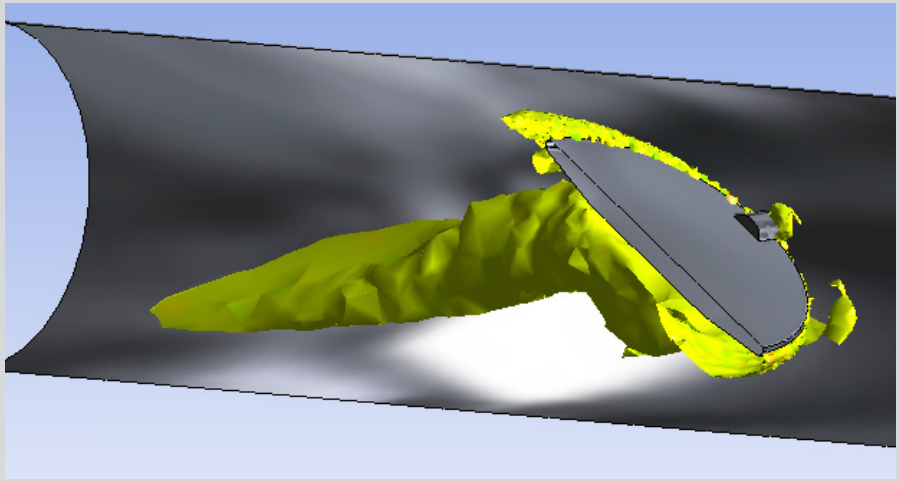
- **Volumes**
  - Mesh elements are either in or out of the volume
    - Volumes will not be displayed as perfect shapes (for example a perfect sphere) because the volume cannot cut through mesh elements
  - From Surface Location
    - A volume is formed from all elements touching (or above / below) the selected Location
    - Can be useful for mesh checking
  - Isovolume
    - Base on a variable at, above or below a given value, or between two values



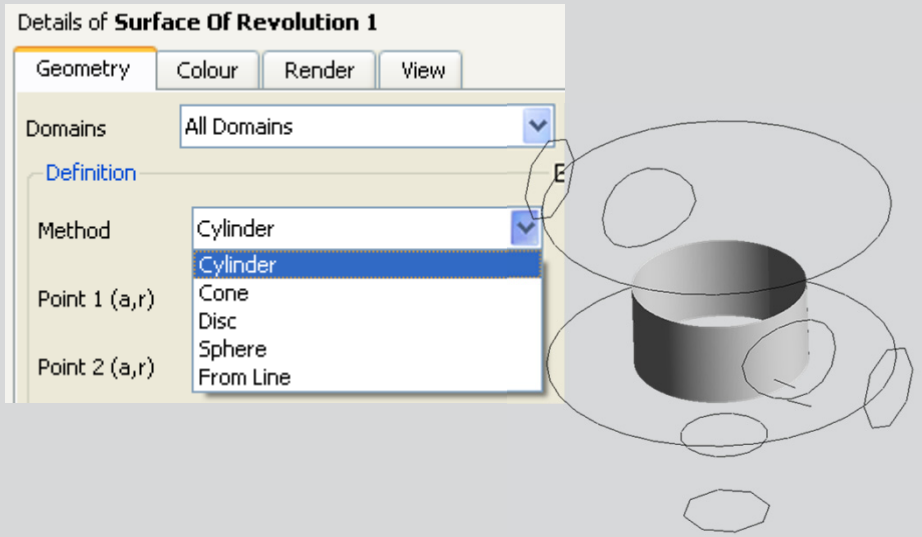


Location Types

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

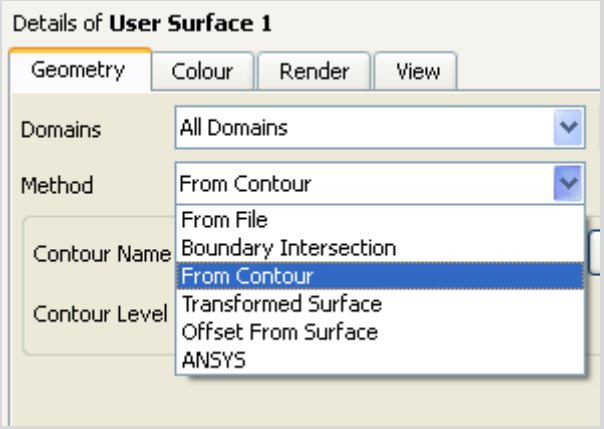


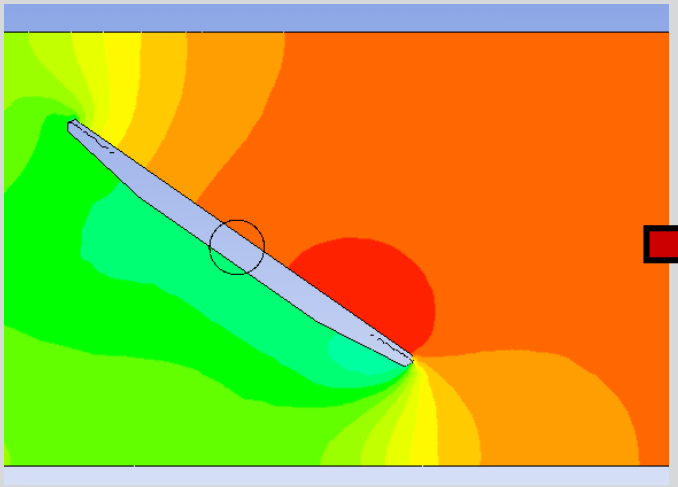
- **Surface of Revolution**
  - Predefined options for *Cylinder*, *Cone*, *Disc* and *Sphere*
  - *From Line* is much more general
    - Any line (existing Line, Polyline, Streamline, Particle Track) is rotated about an axis

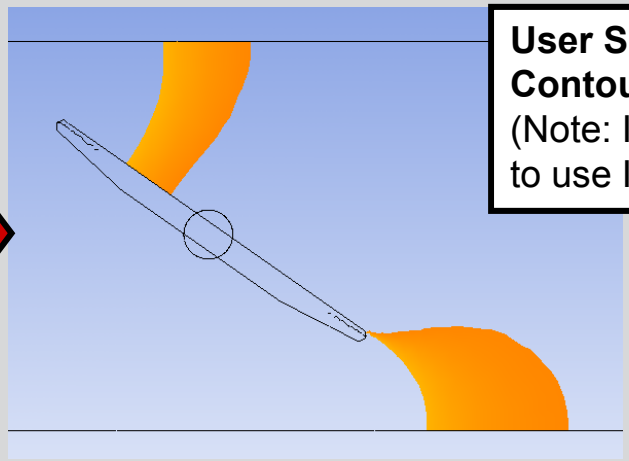


Location Types

- **User Surface**
  - Provides a number of 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 or scale an existing surface
    - *Offset From Surface*: offset an existing surface in either the Normal direction or by Translating



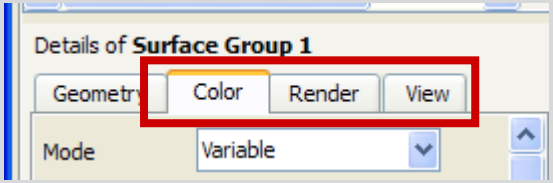




User Surface: From Contour Method
(Note: It's generally easier to use Iso Clips instead)

Color, Render and View

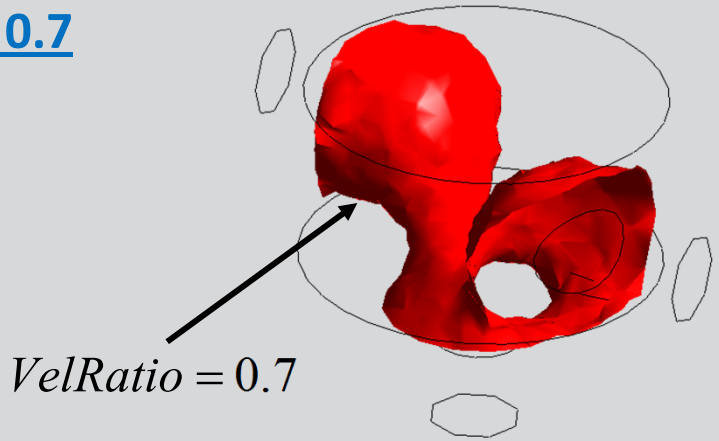
- All Locations have similar *Color*, *Render* and *View* settings
- **Color**
  - Select the variable with which to color the Location
  - Set the *Range* (*Global*, *Local*, *User Specified*)
  - Pick a *Color Map*
- **Render**
  - *Draw Faces*: shows solid surface
  - *Draw Lines*: shows mesh edges or intersecting lines between mesh edges and the plot
  - *Transparency*, *Lighting*, *Texture*…
- **View**
  - Apply Rotation, Translations, Reflection, Scaling
  - Pick a different *Instance Transform*



User Defined Variables Example

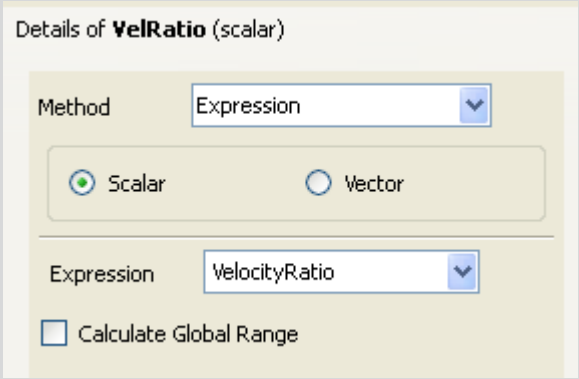
- **Goal: Plot an isosurface at VelRatio = 0.7**

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



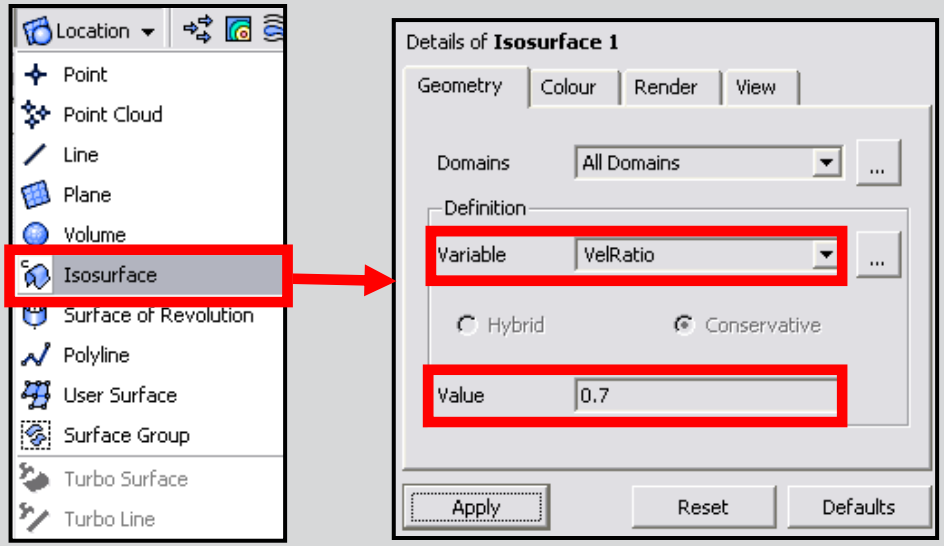
1. **On the Expressions tab create the expression for Velocity Ratio:**

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

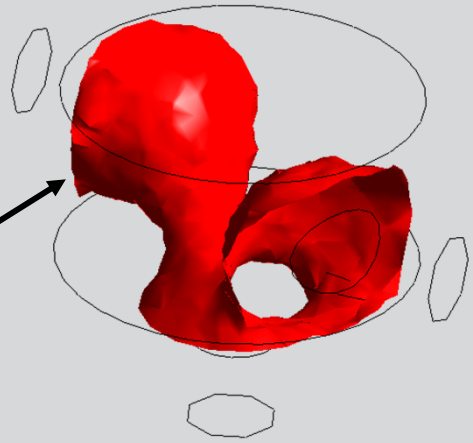


User Defined Variables Example

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)

- or integral

- massFlowAve(Total Pressure)@inlet
- 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)**

- **Mixed units are allowed in expressions as long as the dimension makes sense**
  - Cannot add 1.0 [mm] + 3.6 [kg] in an Expression
- **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

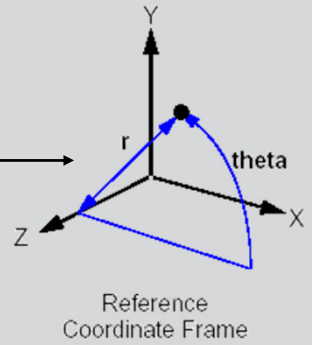
***if** functions contain a test, and two result outcomes. The first outcome, **res1** will be returned if test evaluates to true. If test evaluates to false, **res2** is returned. Consider the following example, where we wish to set volume fraction to 1 when X is greater than 1 [m], and 0 if X is less 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

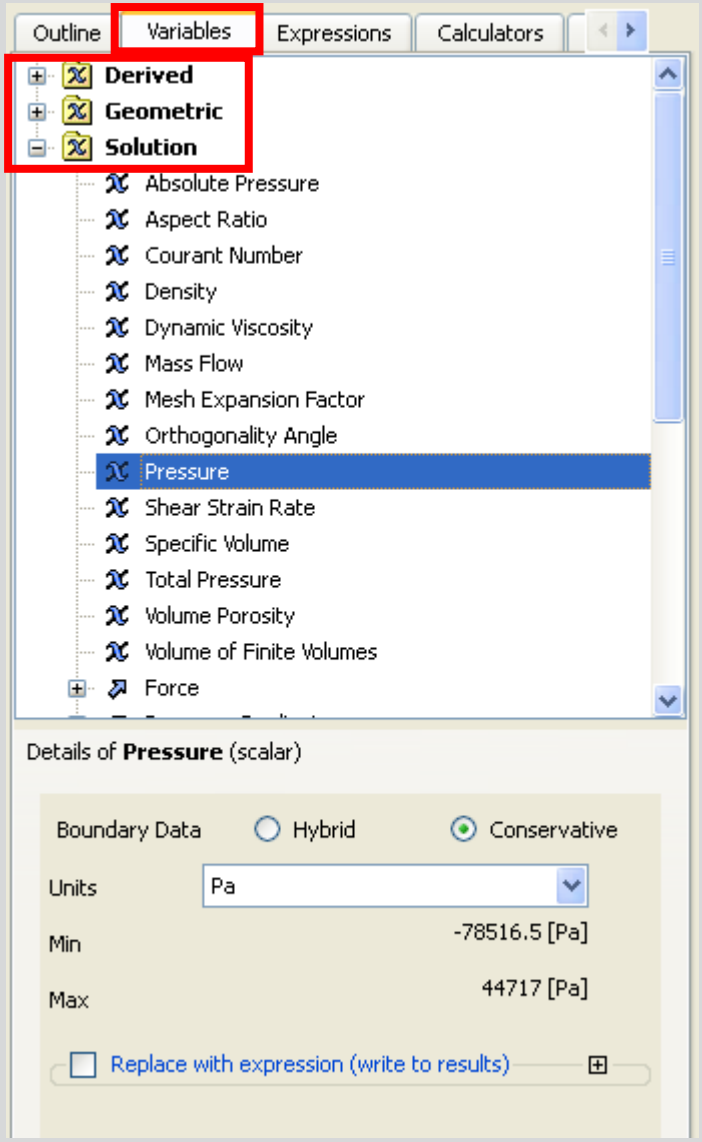
x	Direction 1 in Reference Coordinate Frame
y	Direction 2 in Reference Coordinate Frame
z	Direction 3 in Reference Coordinate Frame
r	Radial spatial location, r = (x^2+y^2)^0.5
theta	Angle, arctan(y/x)
t	Time
u	Velocity in the x coordinate direction
v	Velocity in the y coordinate direction
w	Velocity 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

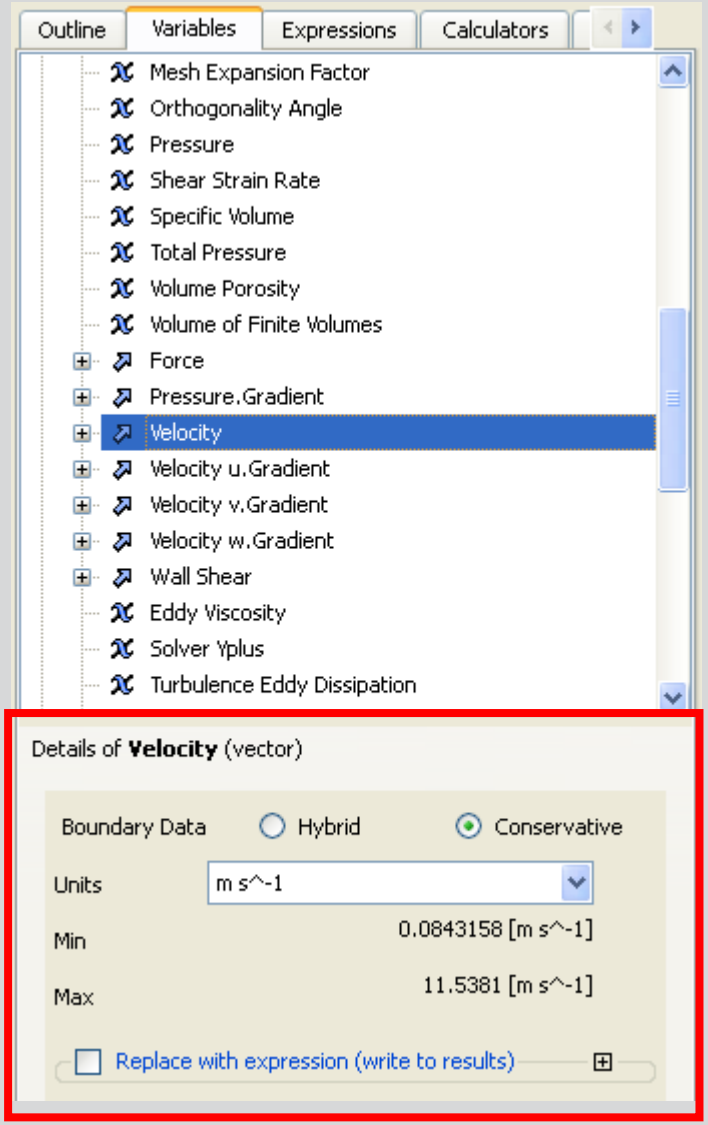
Variables Tab

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



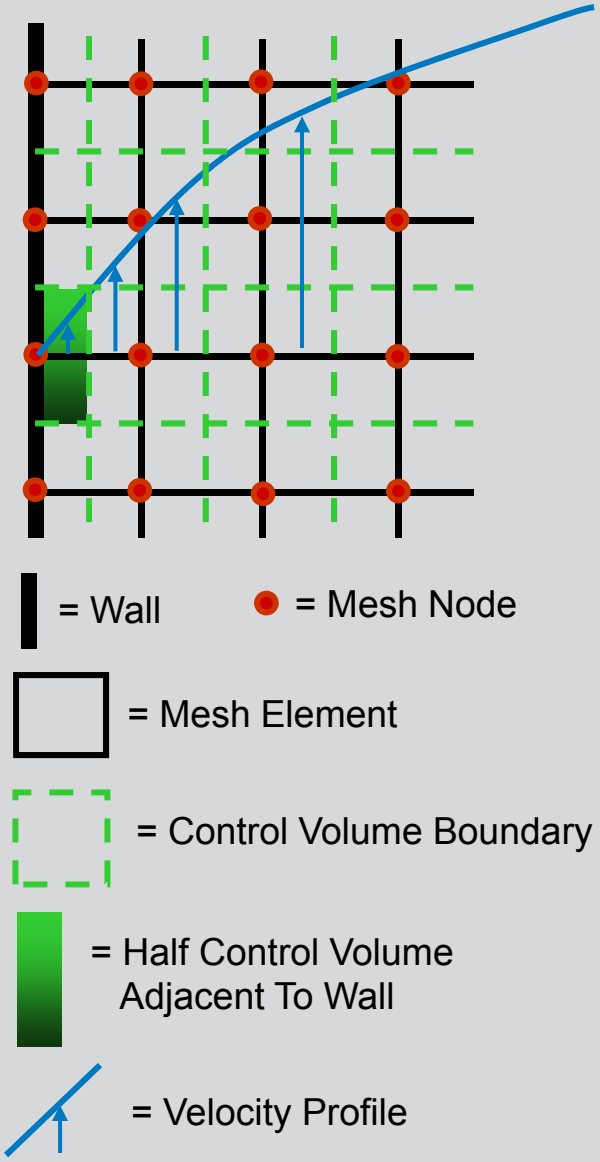
Variables Tab

- **The *Details* pane shows information for the 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* variable definitions**
  - Only applicable to CFX results
  - By default CFD-Post uses Conservative values for all calculations and Hybrid values for all graphics; more details next …
  - Can also switch between Hybrid and Conservative on the *Colour* tab for each plot



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**

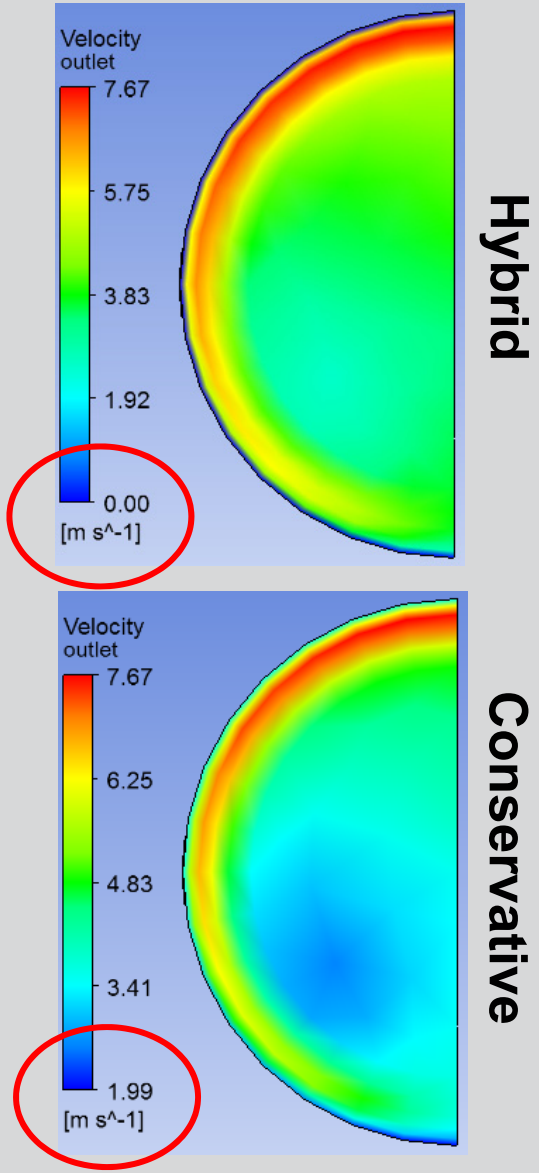


Hybrid vs. Conservative

- **For visualization purposes, ANSYS CFD-Post uses hybrid values by default, because you usually don't want to see non-zero wall velocities**

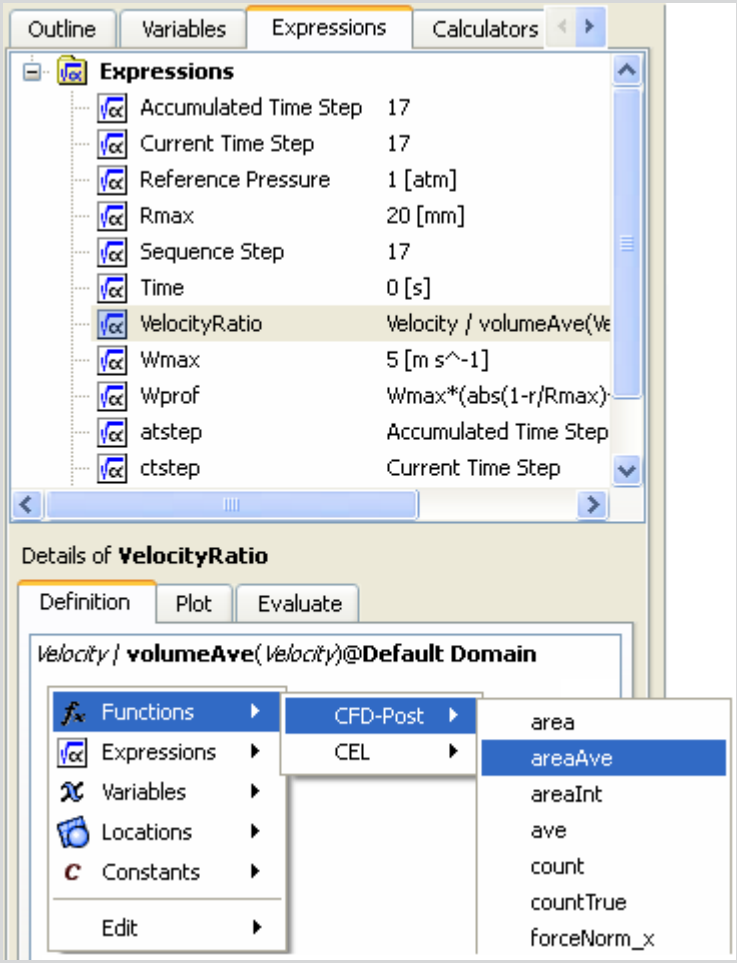
- **For calculation purposes conservative values are used by default**
  - This is good! For example mass flow is calculated correctly — a velocity of zero would produce zero mass flow through the wall adjacent control volume which is clearly wrong

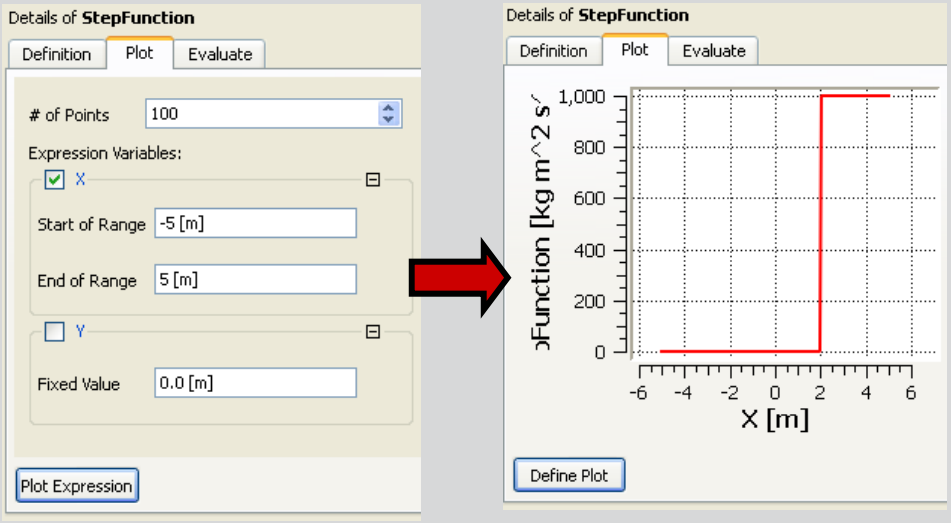
- **So in most cases you don't need to worry about Hybrid vs Conservative since CFD-Post does the right thing**
  - User Defined variables will be derived from conservative values by default
  - Take care when interpreting plots! The range will be different for hybrid and conservative values



Expressions Tab

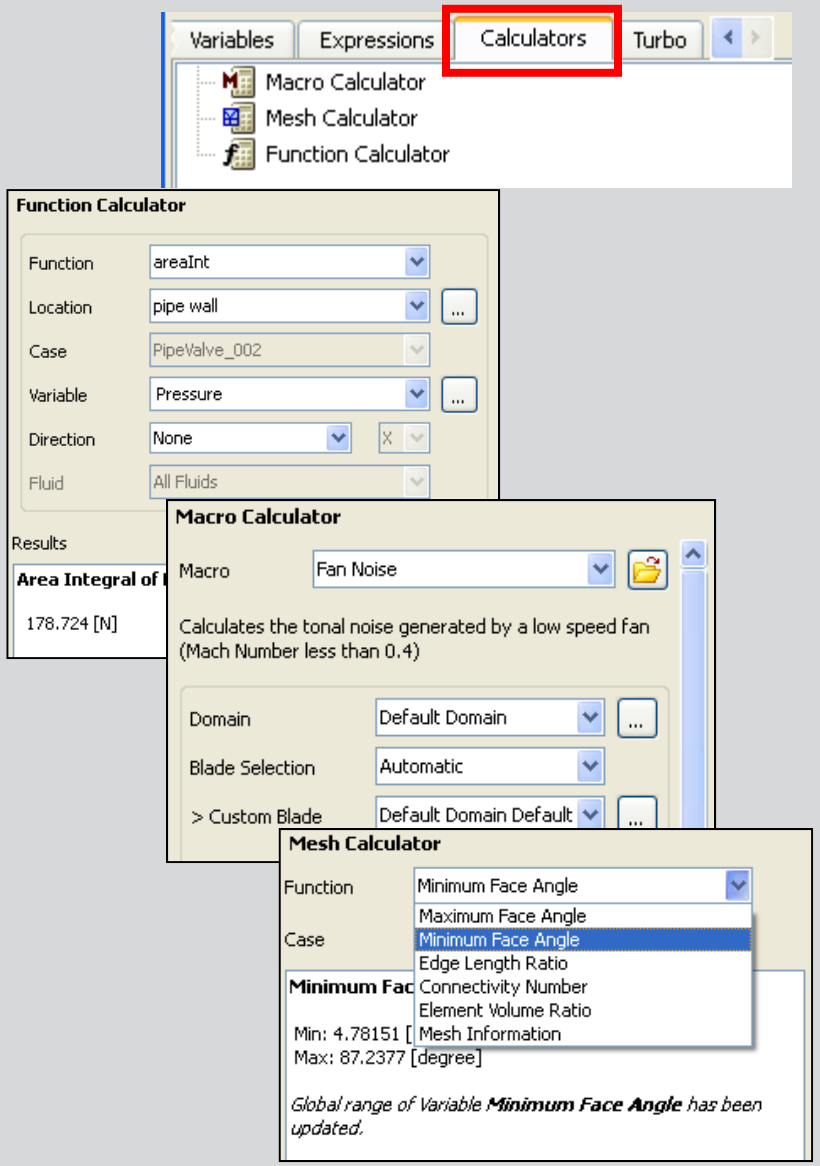
- **The *Expressions* tab shows all existing expressions and allows you to create new expressions**
  - Right-click in the top area > *New*
- **Enter the new expressions on the *Definition* tab in the *Details* view**
  - Right-click to view *Functions*, *Variables* etc. that can be used to build your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





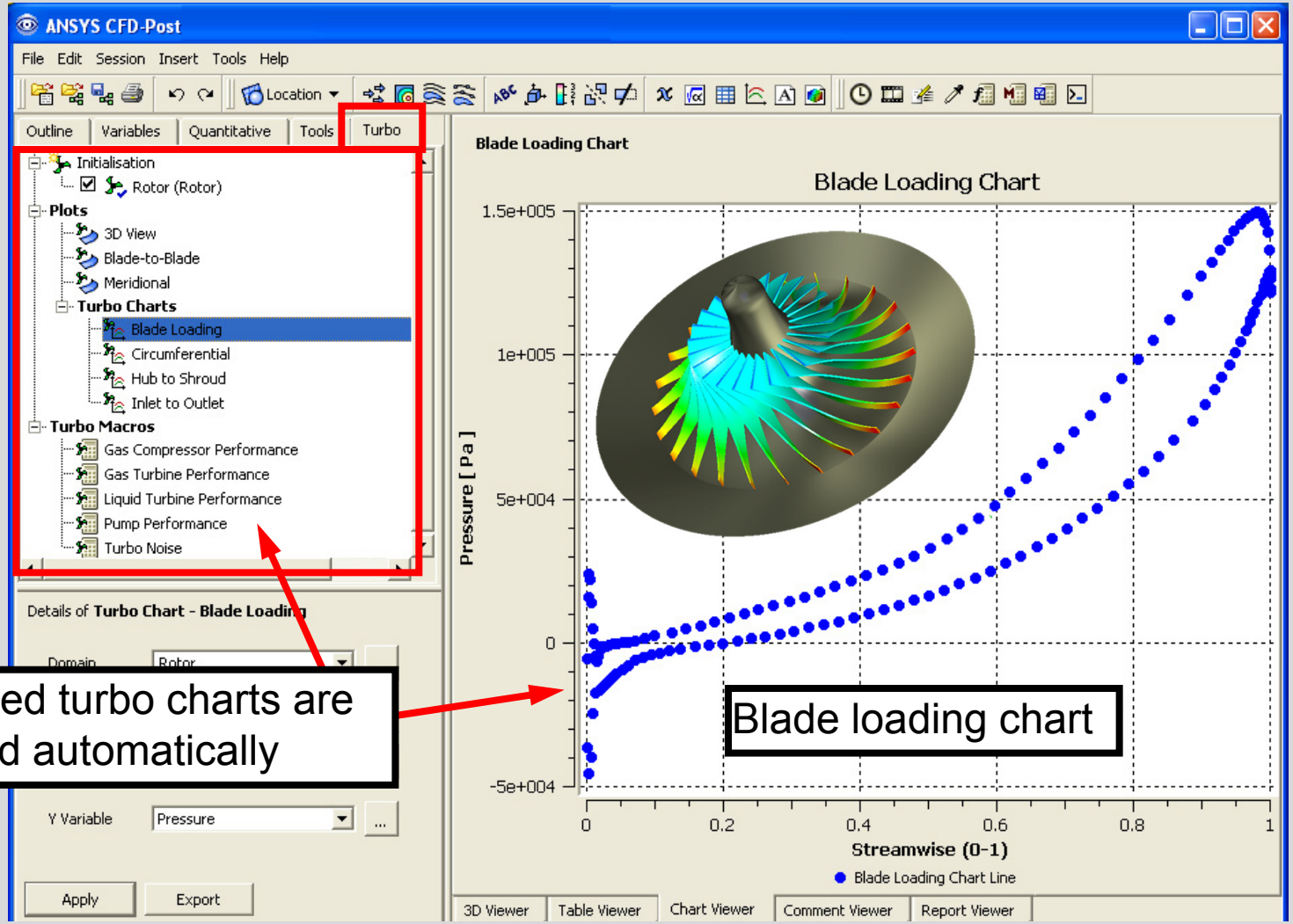
Calculators Tab

- **Function Calculator**
  - Extract engineering data from the results
  - Many functions, see doc to understand 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



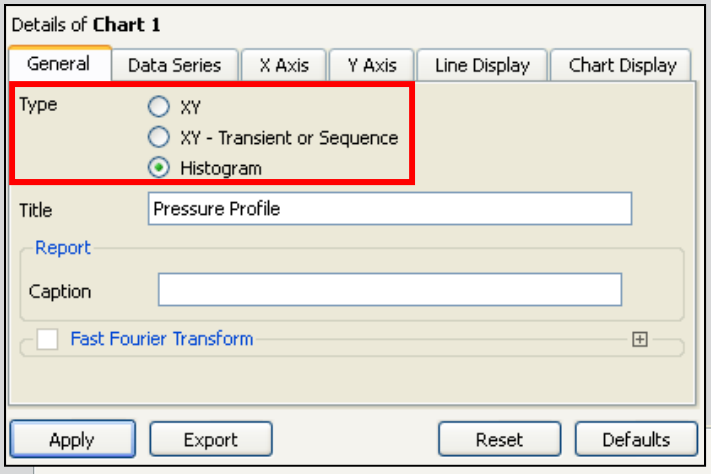
Turbo Post Processing

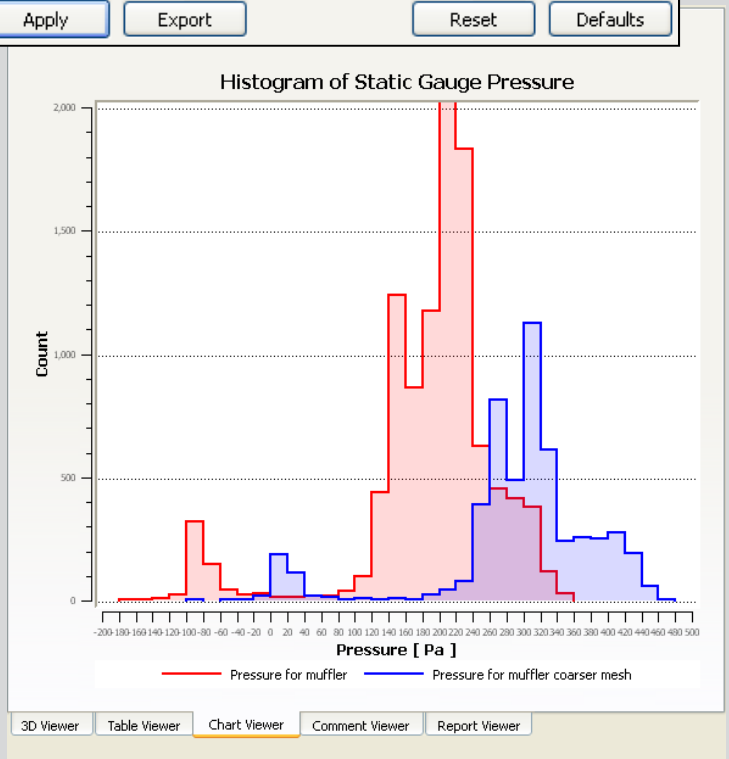
- **The *Turbo* tab contains tools for post-processing turbomachinery cases. See Appendix B for details**



Charts: Type

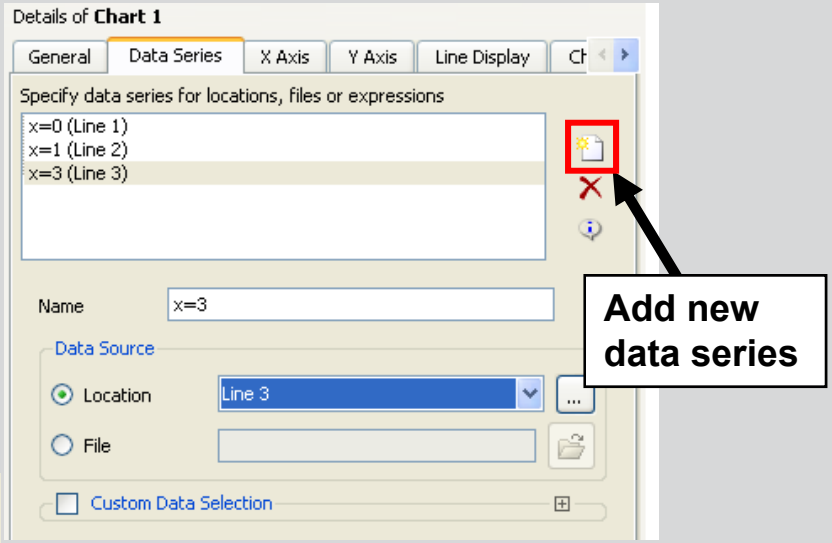
- **Charts can be one of three types:**
  - **XY**
    - Standard XY plots based on line locators
  - **XY – Transient or Sequence**
    - Plots an expression (usually Time) versus a variable at a point locator
    - Typically used to show the transient variation of a variable at a point
      - Data must be present in the trn files
  - **Histogram**
    - Can be based on any locator that contains multiple data locations – lines, surfaces, planes, domains (but not points)
    - Plots a variable divided into discrete bands on the X Axis versus the frequency of occurrence on the Y Axis

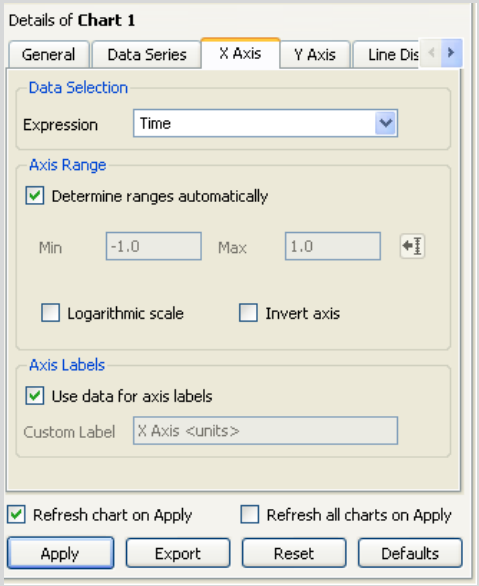


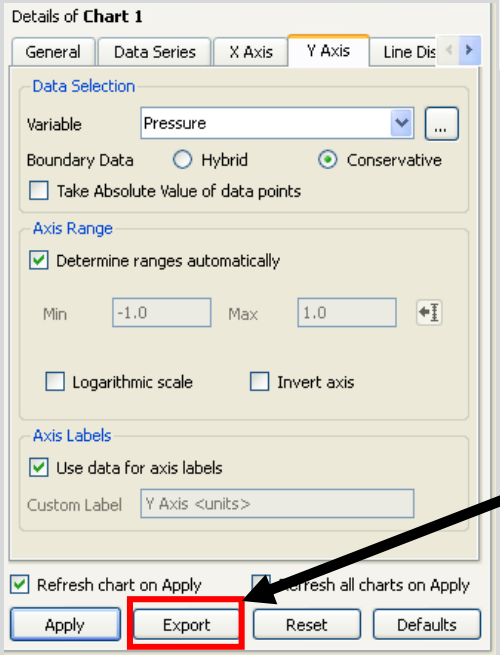


Charts: Data Series and Axes

- 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



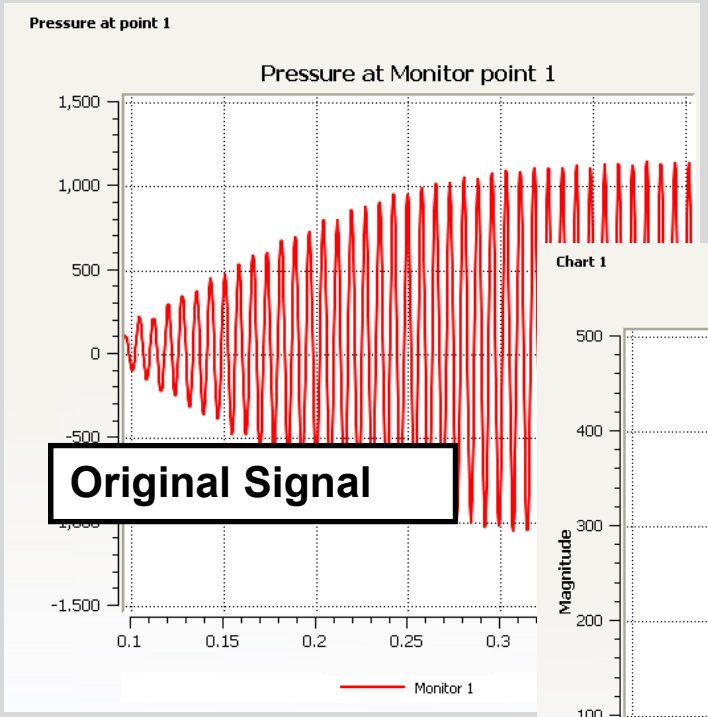




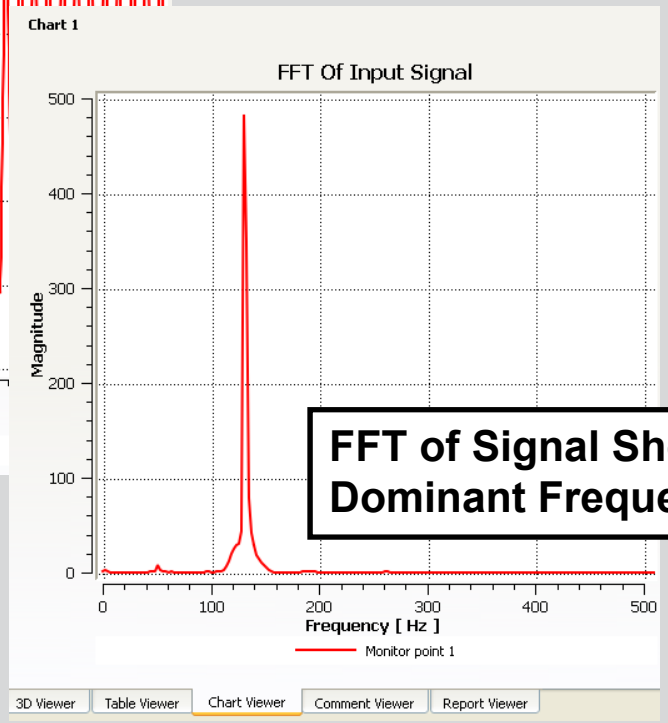
The Export button allows the generated plot data to be written to a .csv or .txt file for use in Excel, Matlab or other plotting programs

Fast Fourier Transform

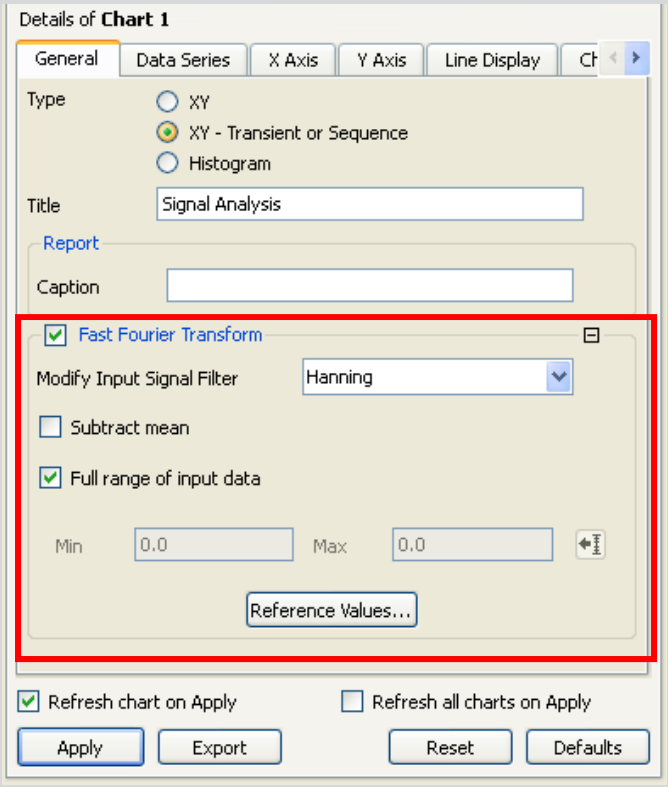
- FFT can be applied to signals to extract frequency data



Original Signal

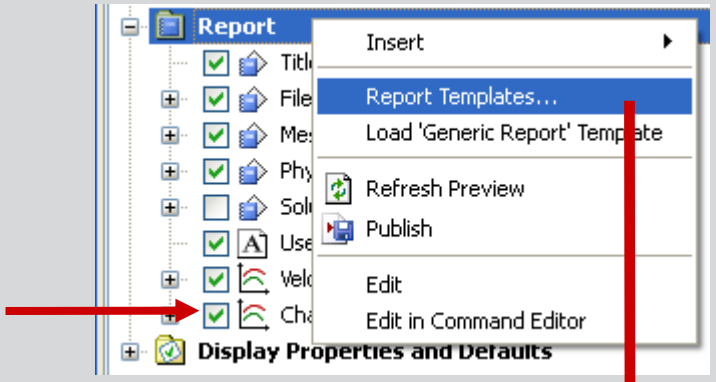


FFT of Signal Showing Dominant Frequency

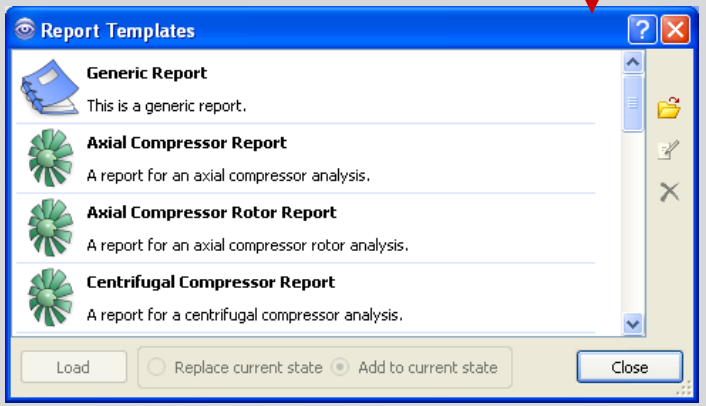


Reports

- **CFD-Post has report generation tools which allow 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

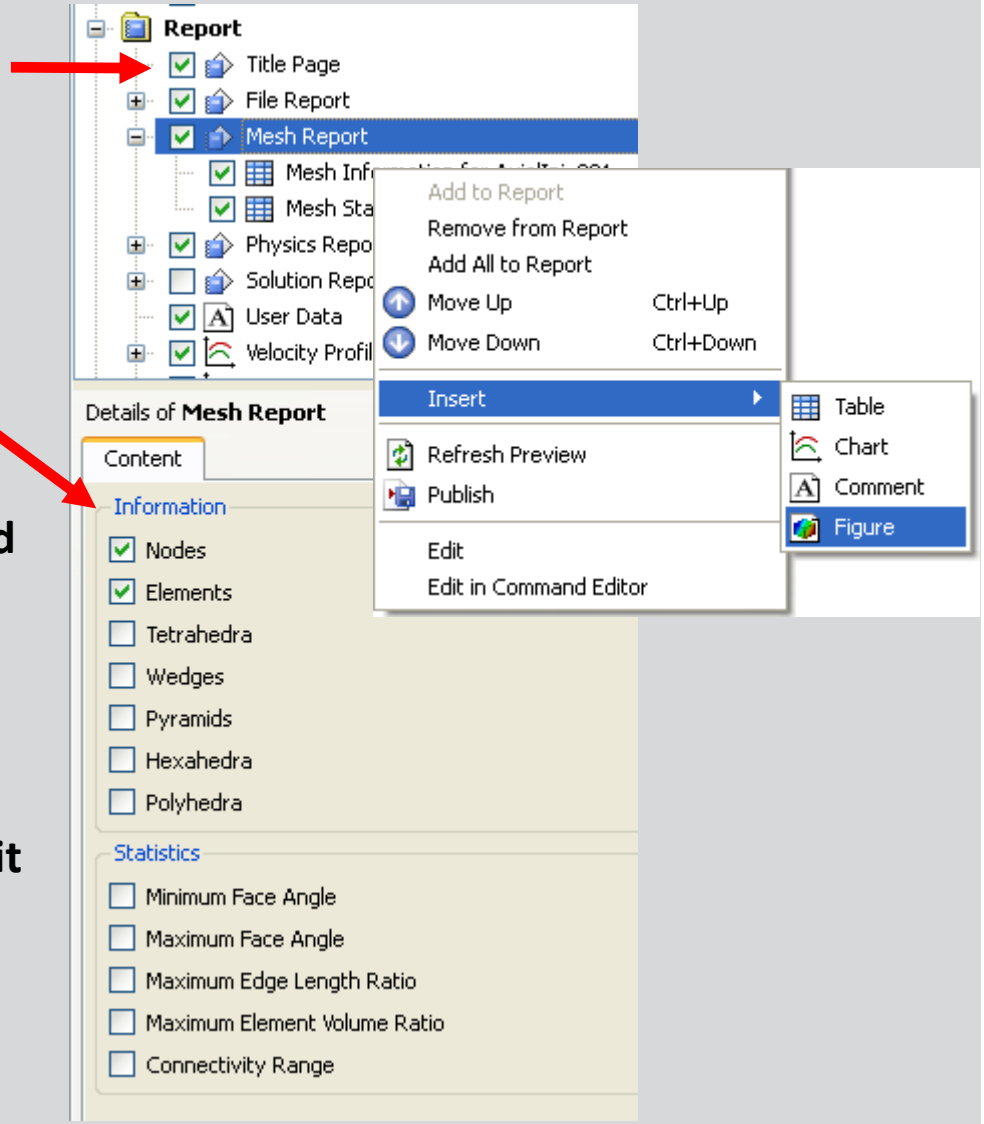


- **Reports are template based**
  - Depending on the information contained in a results file, a report template will be selected automatically
  - Right-click on *Report* to select a different template
    - You can create your own custom templates or modify existing templates
    - E.g. add you company 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**

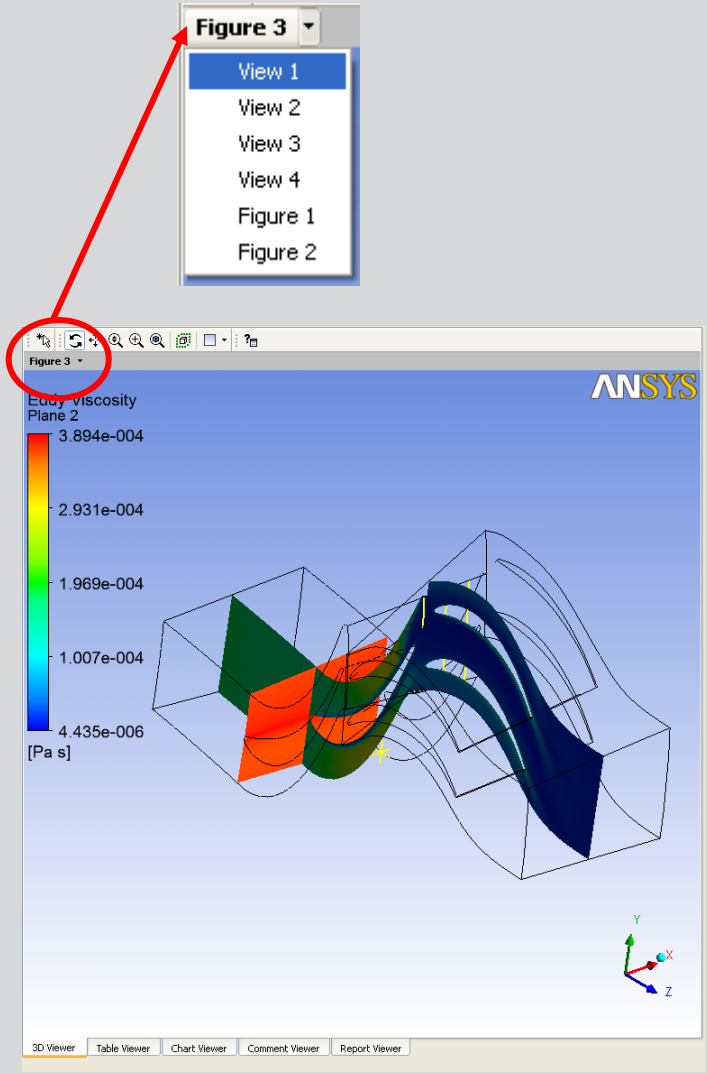


Reports: Figures

- **When you add a new *Figure* it will be listed in the drop-down menu in the top corner of the Viewer**

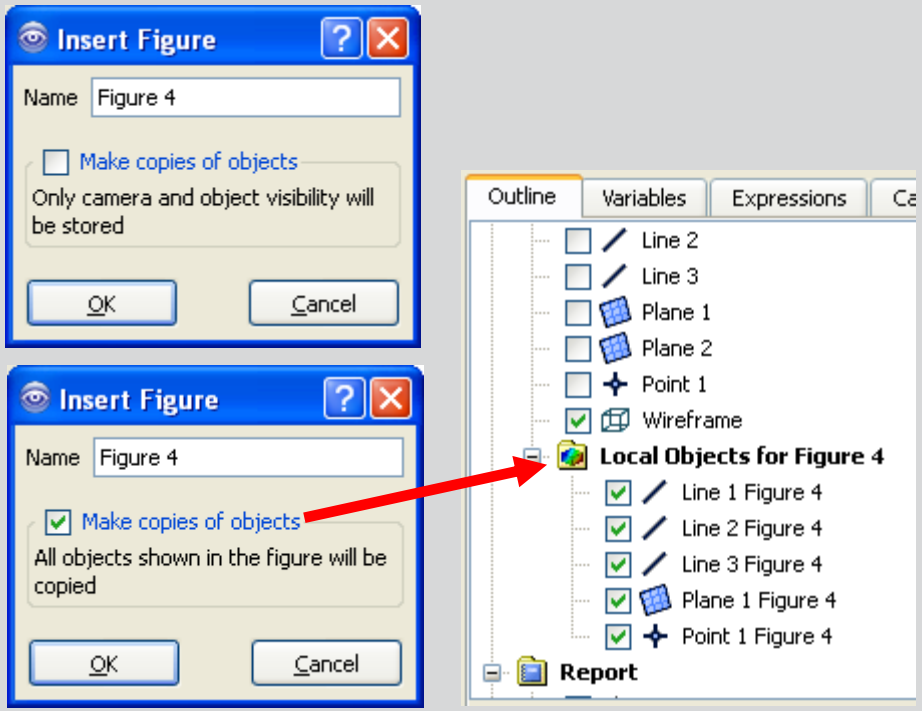
- **Figures are not static, you can change them after they have been created**
  - If you do not want to change a Figure, make sure one of *View 1 – View 4* is selected from the drop down menu

- **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



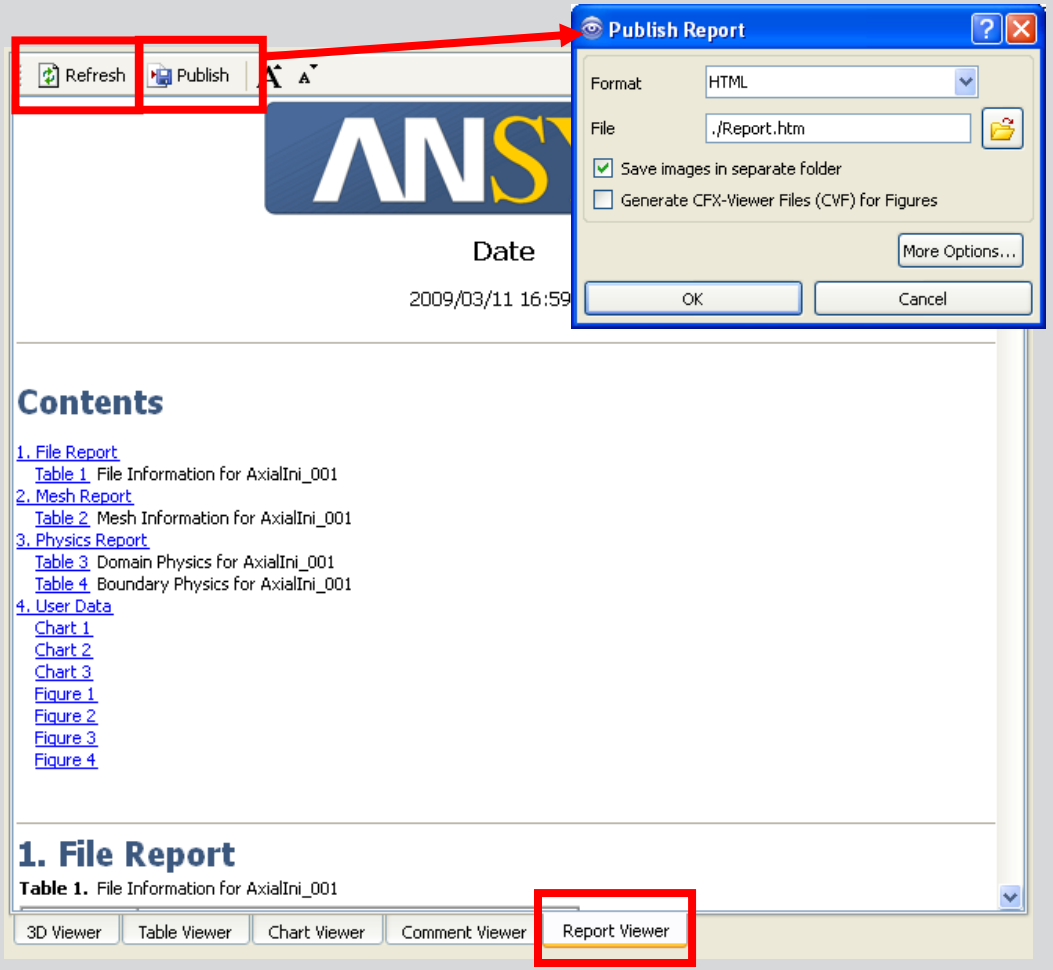
Reports: Figures

- **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**



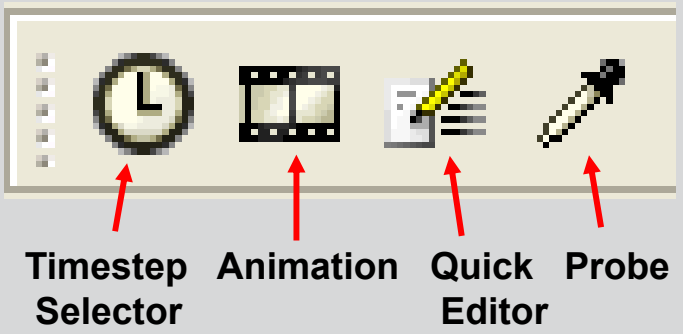
Reports

- **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



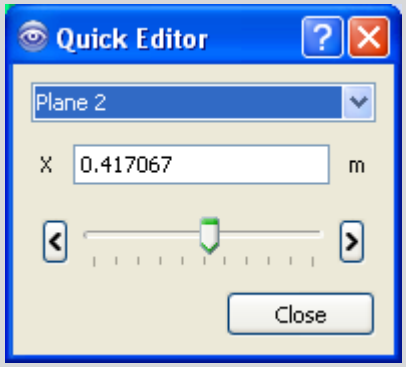
Other Tools

- **Timestep Selector**
  - Transient results are post-processed by loading in the final results (.res) file, then selecting different timesteps from the *Timestep Selector*



- **Animation**
  - Animate objects, create MPEGs
  - More on next slide

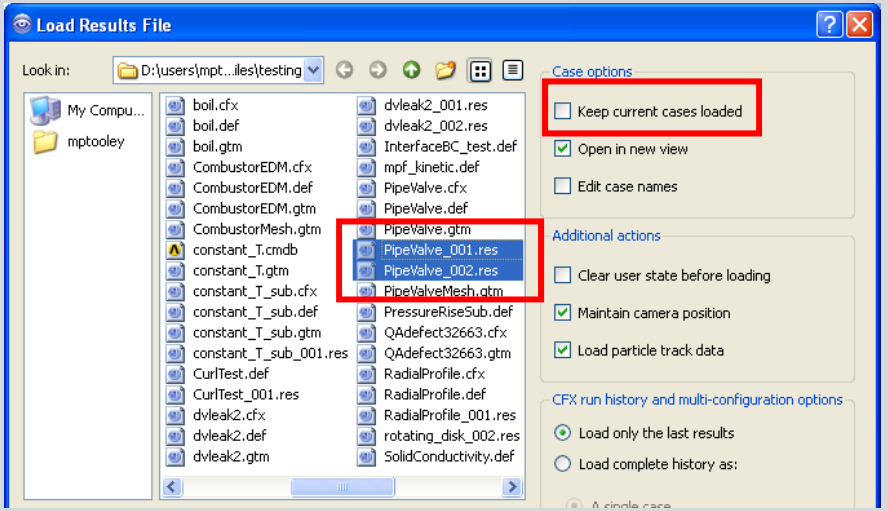
- **Quick Editor**
  - Provides a very quick way to change the “primary” value associated with each object



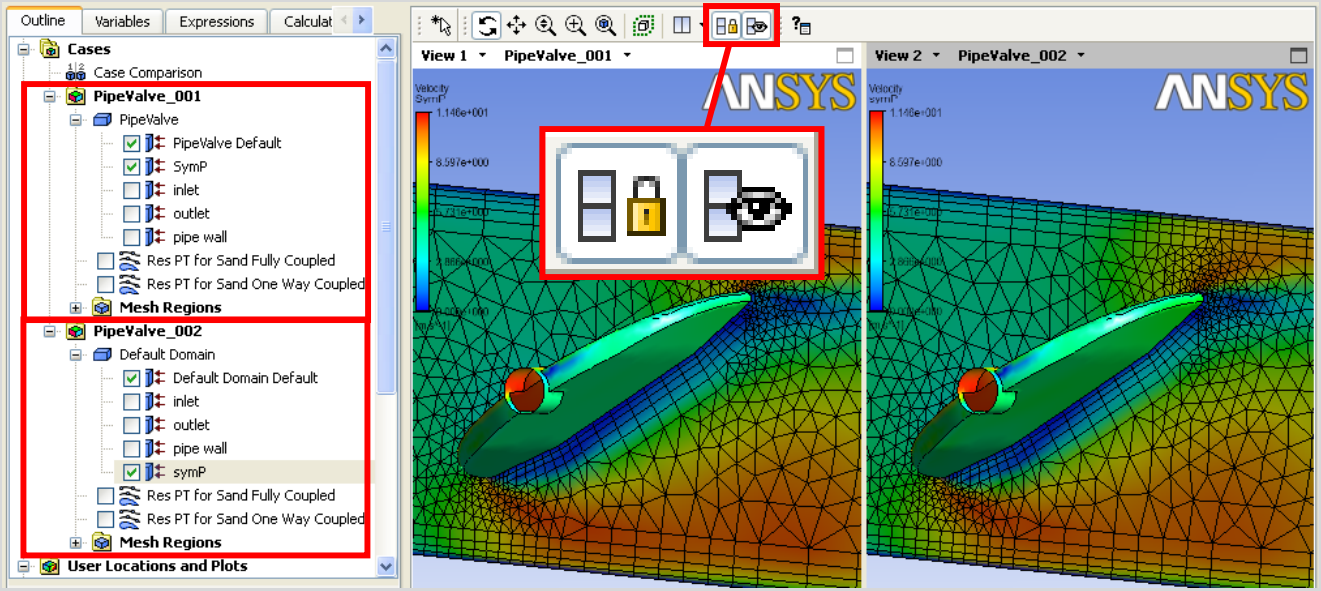
- **Probe**
  - Pick a point from the Viewer and probe a variable value at that point

Multi File Mode

- **To post-process multiple files simultaneously you can:**
  - Multi-select files when loading
  - Load a multi-configuration results file (.mres) using *Load complete history as> Separate Cases*
  - Or 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**
  - **All Views move the same**
- **Sync objects**
  - **The visibility of all User Locations and Plots is the same**

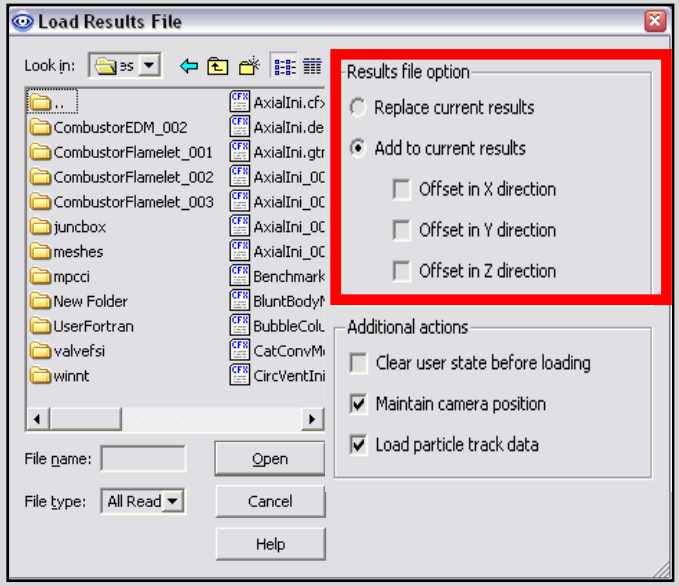


Files

- **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**
    - CFD-Post is able to read ANSYS results for temperature, velocity, acceleration, magnetic forces, stress, strain, and mesh deformation

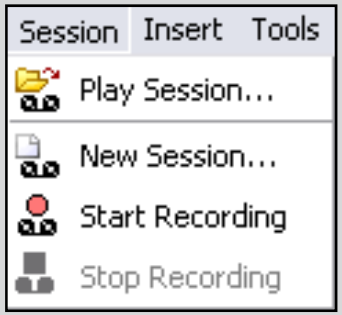


- **Import**
  - Locations: .csv files which contain point data which defines a polyline or surface
  - ANSYS Surface Mesh (.cdb): To allow for export of data on a surface for use as a boundary condition in ANSYS

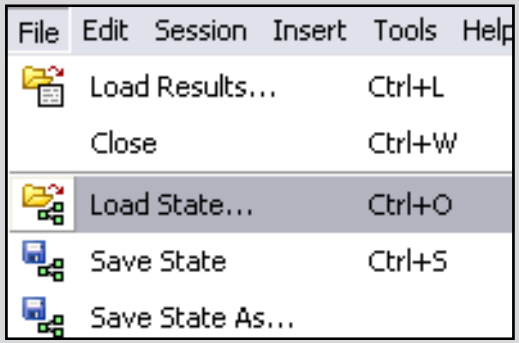
- **Export**
  - Profile Boundary Data: for use in CFX-Pre
  - General formatted results data
  - ANSYS Load Data: Written onto an imported ANSYS .cdb file

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**
  - More later in Scripting and Automation lecture