

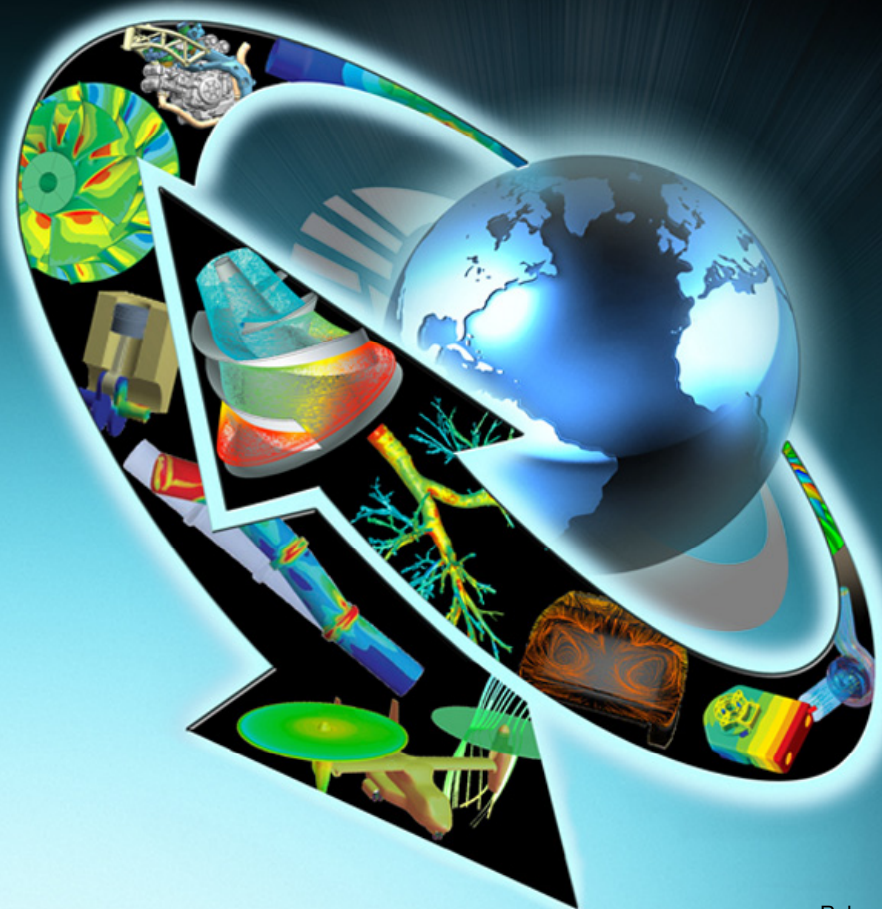


Customer Training Material

Lecture 11

Postprocessing with FLUENT and CFD-Post

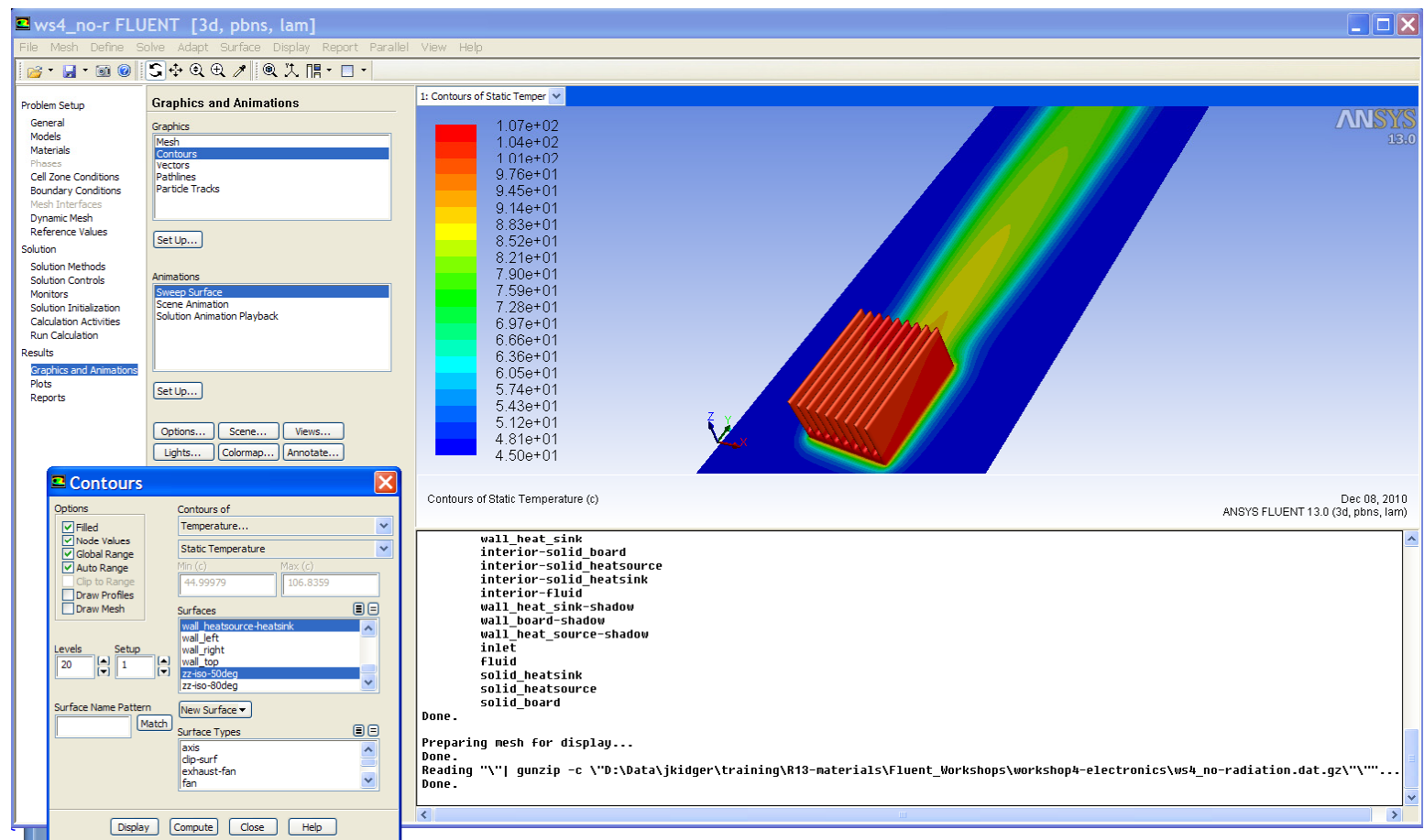
Introduction to ANSYS FLUENT



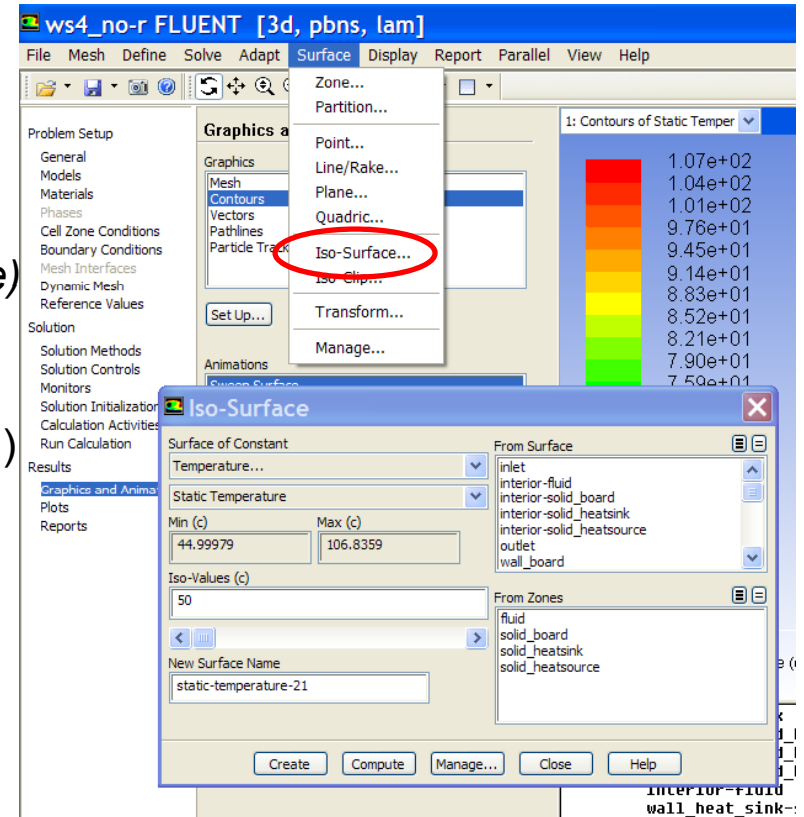
- There are two ways to postprocess CFD results from FLUENT
 - **FLUENT Postprocessing tools** –tools integrated in the FLUENT solver.
 - **ANSYS CFD-Post application**
 - A state-of-the-art postprocessor for ANSYS CFD products.
 - CFD-Post can run as a standalone postprocessor, or within Workbench
- Both postprocessors include many tools for analyzing CFD results
 - Isosurfaces
 - Vector Plots
 - Contour plots (shaded and graded)
 - Streamlines and pathlines
 - XY plotting
 - Animation creation

Postprocessing 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



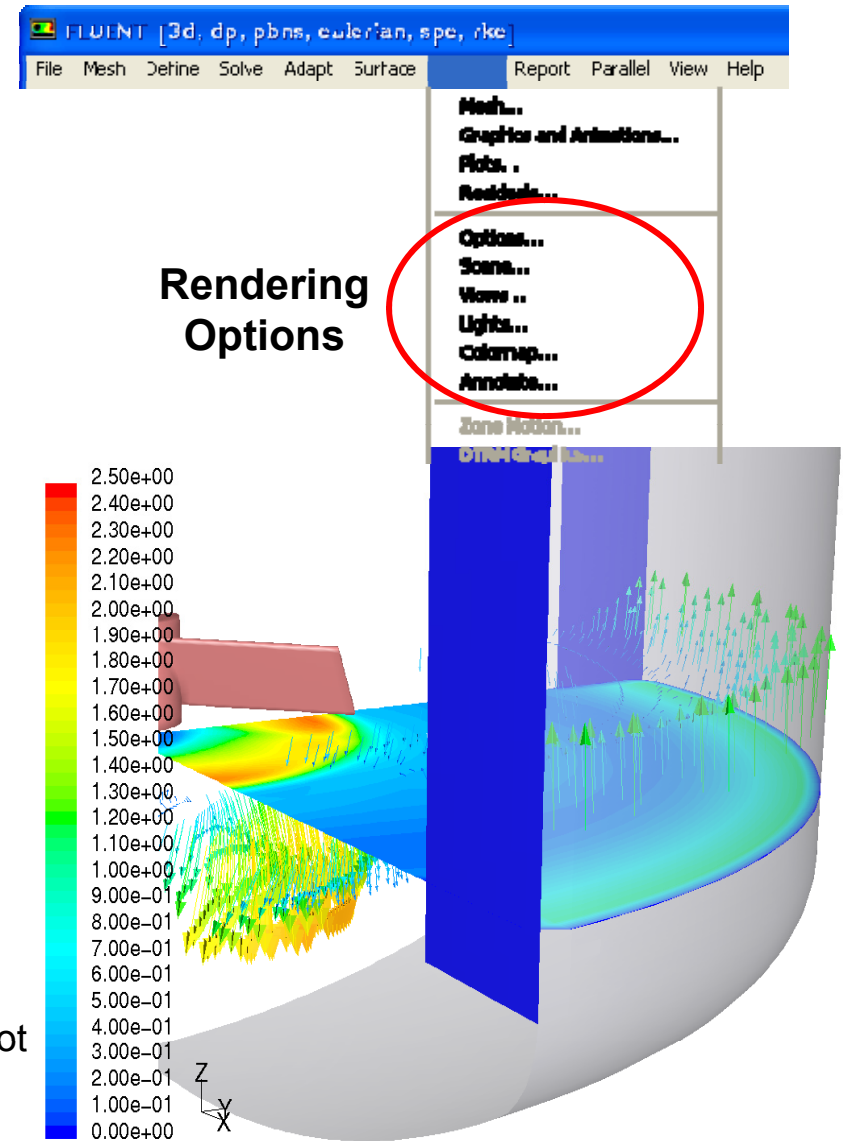
- The results can be reported / plotted either on existing surfaces present in the model, or on new surfaces.
- Commonly used options are:
 - A planar or cylindrical surface cutting through the model (*use Iso-Surface, based on constant mesh value, of x-coordinate, or radial coordinate*)
 - An Isosurface to create a 'cloud' 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)



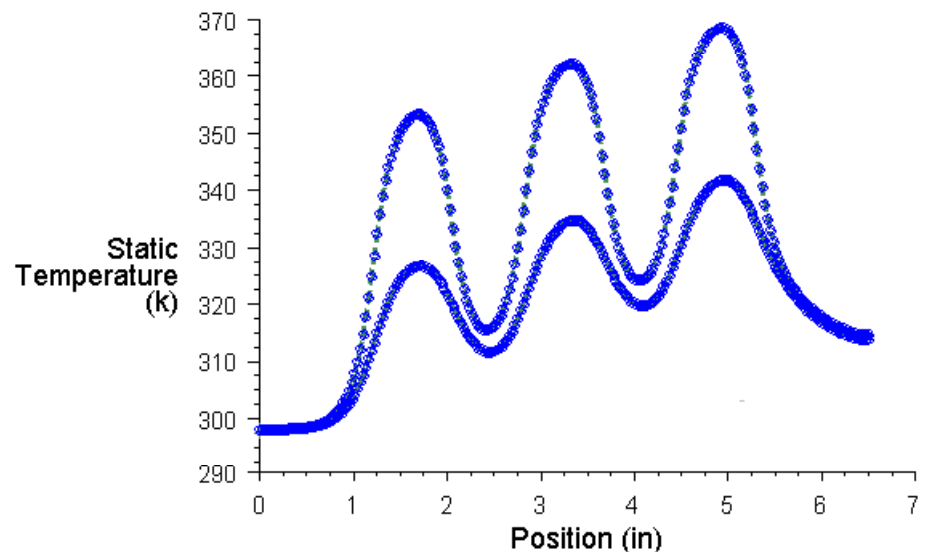
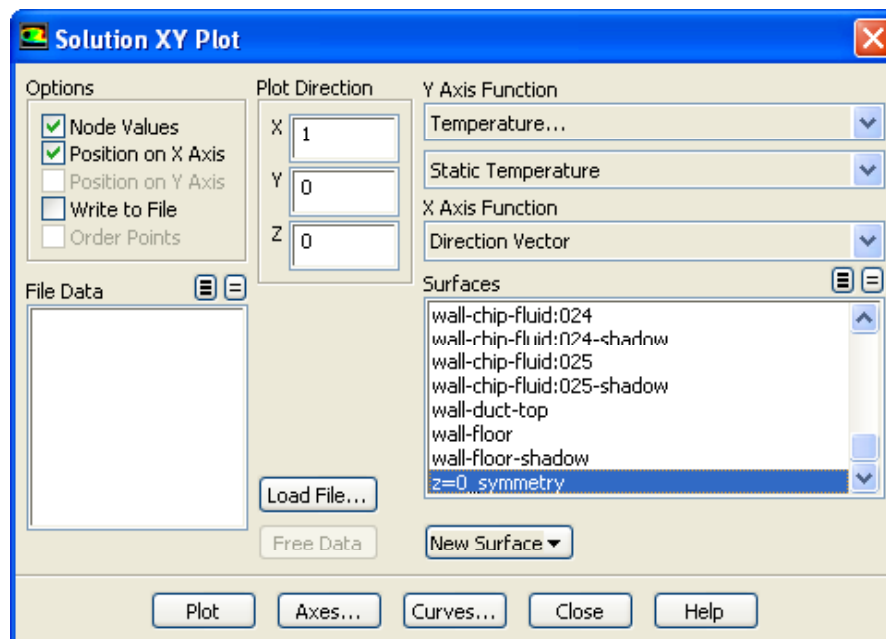
Post-Processing Rendering Options

- 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.



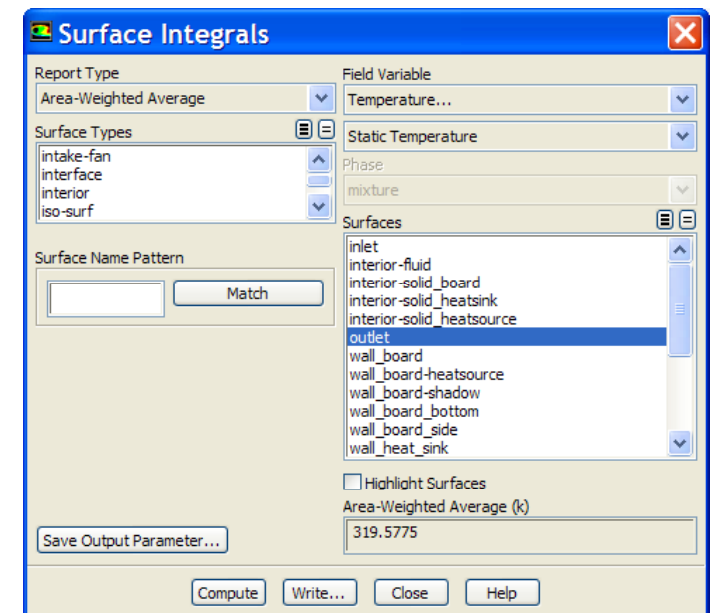
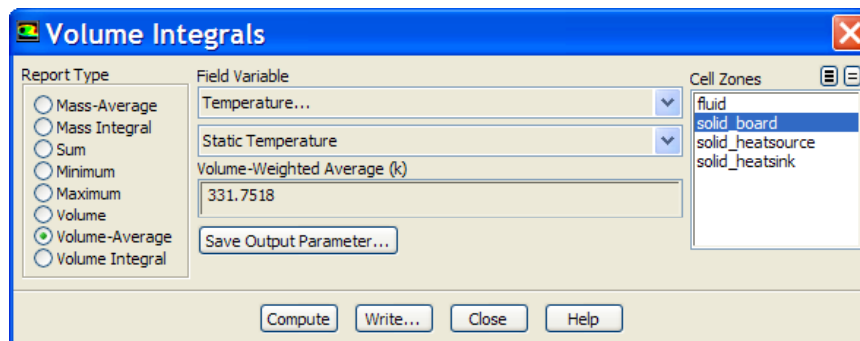
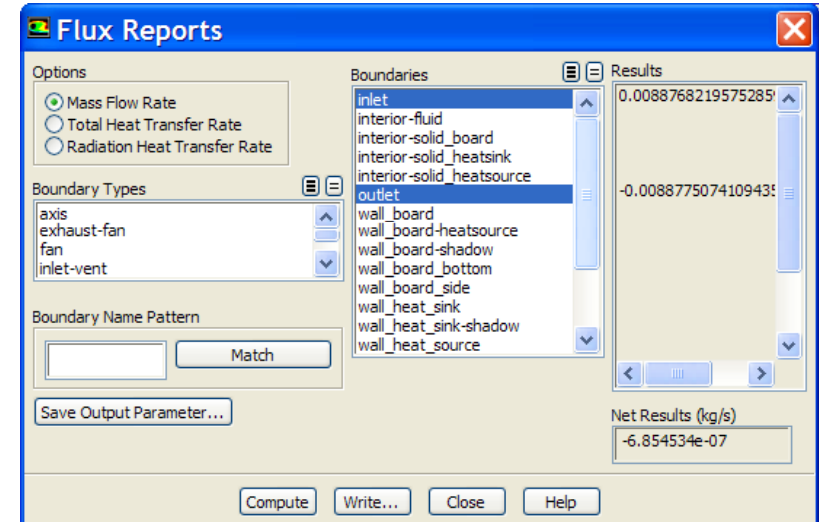
- FLUENT provides tools to generate data plots of the solution:
 - XY plots of solution variables
 - Histograms to illustrate frequency of distribution
 - Fast Fourier Transforms (FFT)
- Other data files (experimental, computational) can also be read in to compare results (as done on Workshop #2 for the airfoil)



Postprocessing 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



Postprocessing in CFD-Post

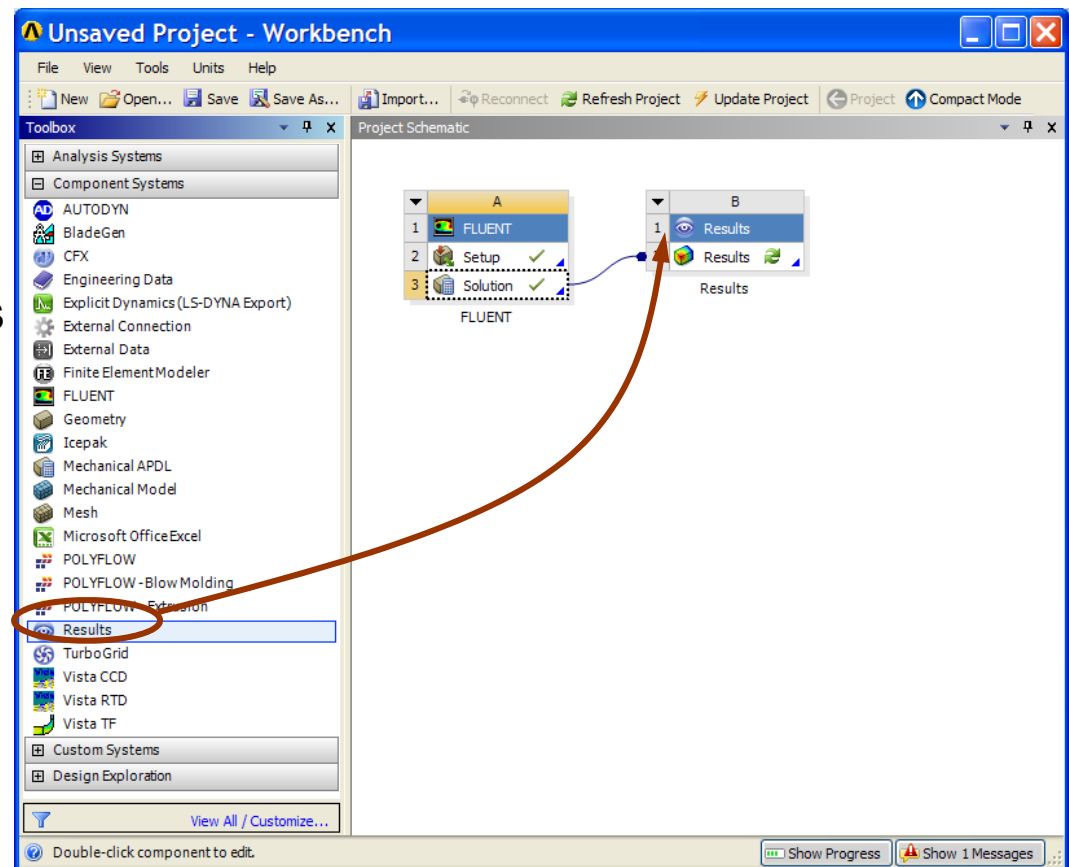
Postprocessing

Starting CFD-Post



- Within ANSYS Workbench
 - Drag the CFD-Post icon (Results) in the Component Systems list to the project tree.

- Outside of Workbench:
 - Start > Programs >
ANSYS 13.0 > Fluid Dynamics
CFD-Post



Postprocessing GUI Layout

B2 : Results - CFD-Post

File Edit Session Insert Tools Help

Outline Variables Expressions Calculators Turbo

Objects in the model

- fluid
 - inlet
 - outlet
 - wall_board
 - wall_heat_sink
 - wall_heat_source shadow
 - wall_left
 - wall_right
 - wall_top
- solid_board
 - wall_board heatsource solid_board
 - wall_board shadow
 - wall_board_bottom
 - wall_board_side
- solid_heatsink
 - wall_heat_sink shadow
 - wall_heatsource heatsink solid_heatsink
- solid_heatsource
 - wall_board heatsource solid_heatsource
 - wall_heat_source
 - wall_heatsource heatsink solid_heatsource

User-created objects

- User Locations and Plots
 - Default Transform
 - Default Legend View 1
 - Surface Group 1
 - Wireframe
- Report
 - Title Page
 - File Report
 - Mesh Report
 - Physics Report
 - Solution Report

Details of current object

Details of Surface Group 1

Geometry Color Render View

Mode: Variable

Variable: Temperature

Range: Global

Min: 318.15 [K]

Max: 379.1

Boundary Data: Linear

Color Scale: Linear

Color Map: Default (Rainbow)

Undef. Color:

Apply Reset Defaults

Various Viewers (3D, Chart, ...)

3D Viewer Table Viewer Chart Viewer Comment Viewer Report Viewer

Temperature Surface Group 1

3.800e+002

3.491e+002

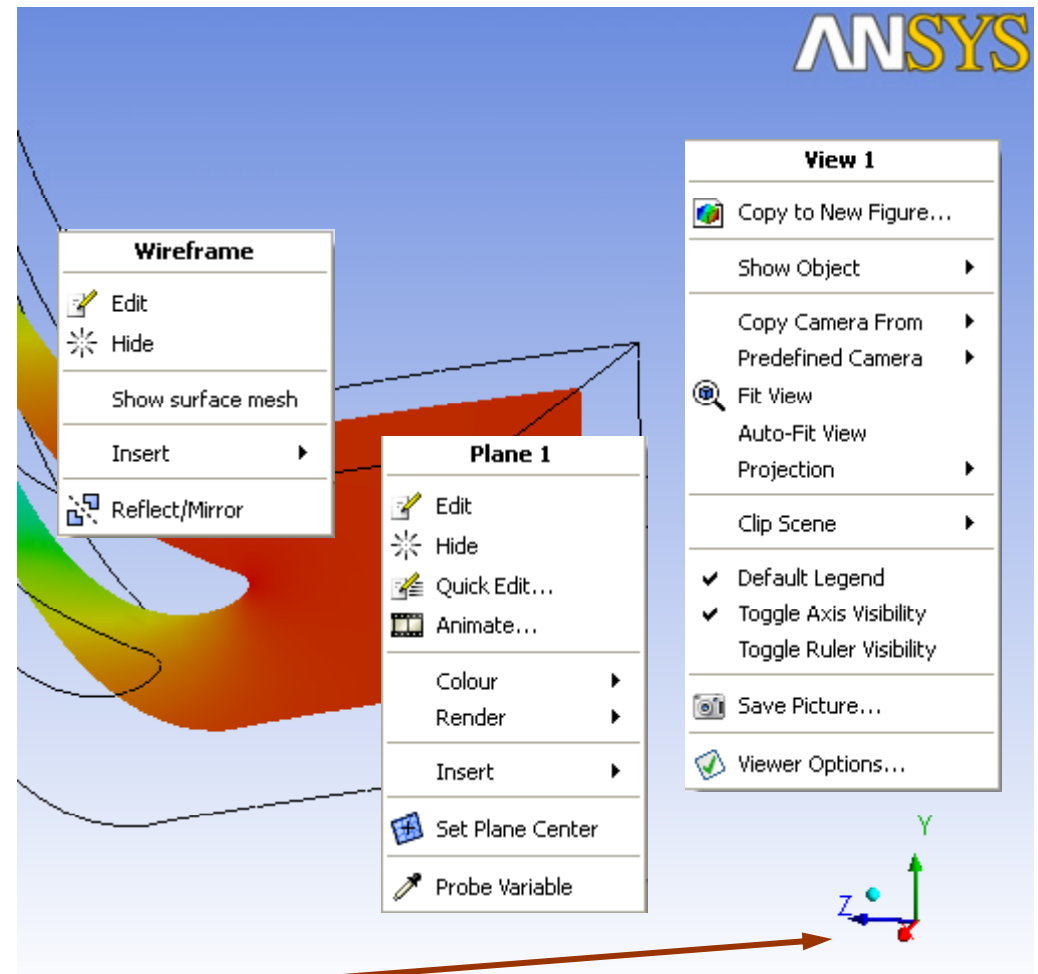
[K]

0 0.200 0.400 (m)

X Y Z

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

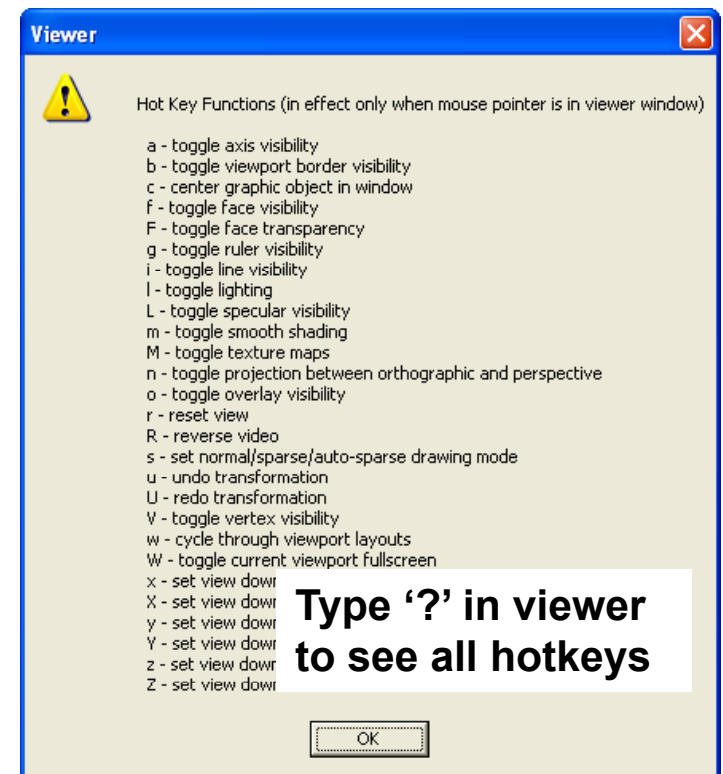
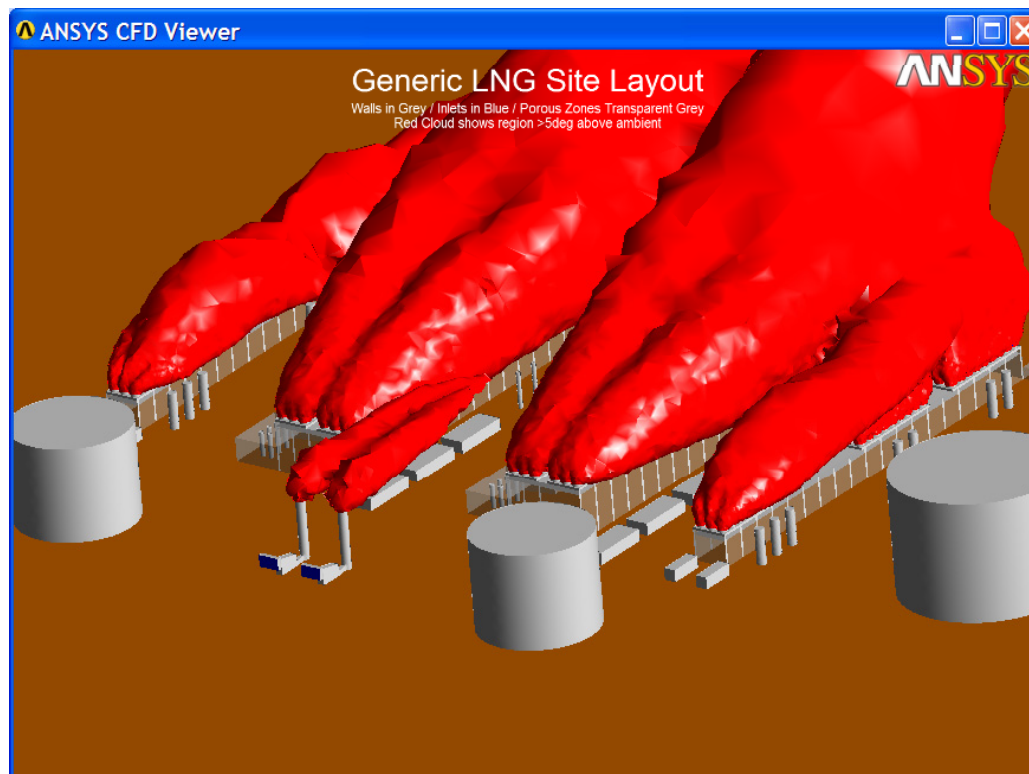


Postprocessing

3D Viewer Files



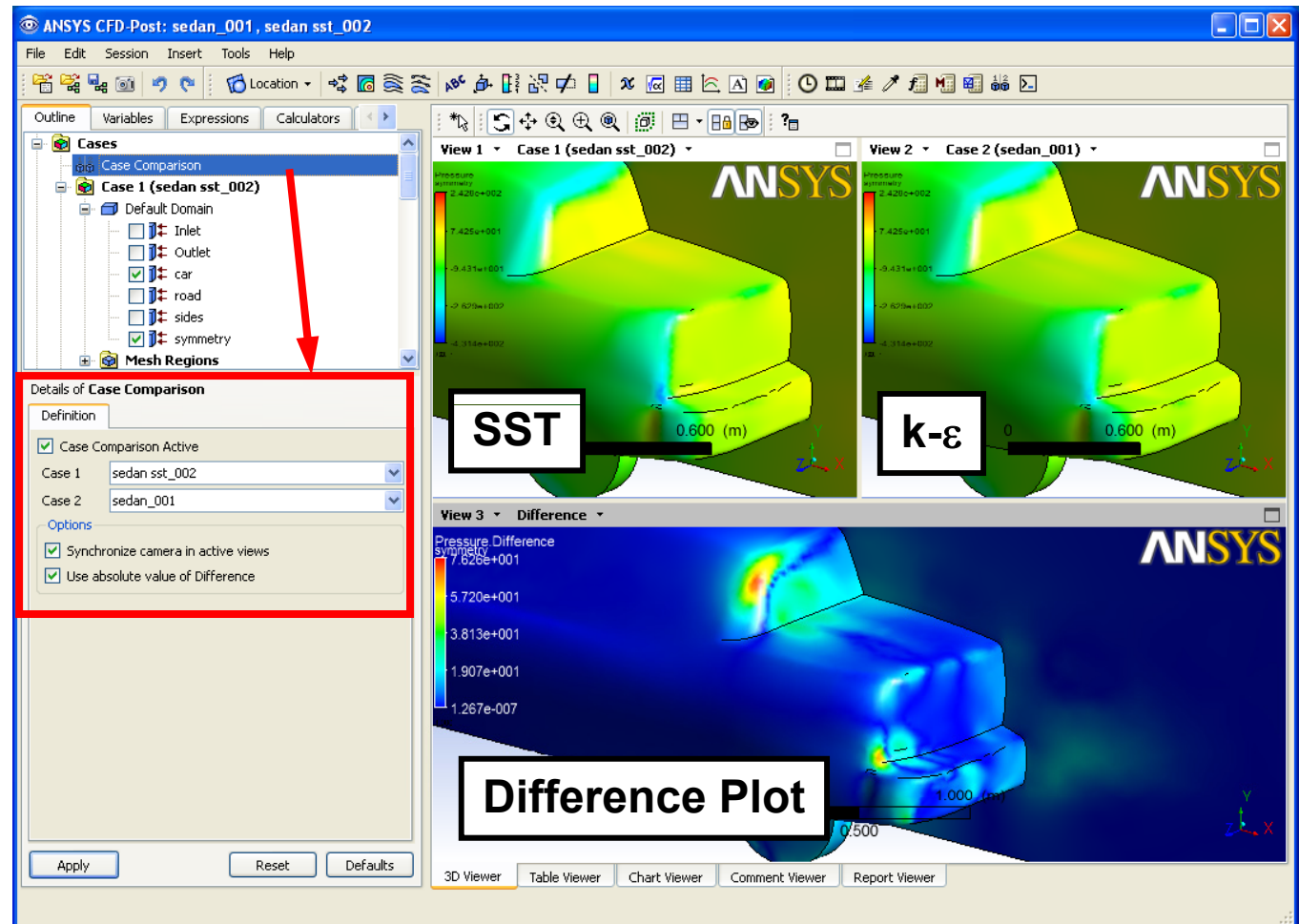
- Save *Picture* in the *CFX Viewer State (3D)* file format (.cvf file)
- Can then use the stand-alone Viewer to view the file, rotate, pan, zoom, etc
 - Unlicensed and free to distribute to your customers
 - Can embed 3D Viewer files in PowerPoints and HTML files
 - Download from the ANSYS Website (search for “CFD Viewer”)



Postprocessing

Case Comparison

- When multiple files are loaded you can select **Case Comparison** from the **Outline** tree
 - Automatically generates difference variables and plots



Postprocessing

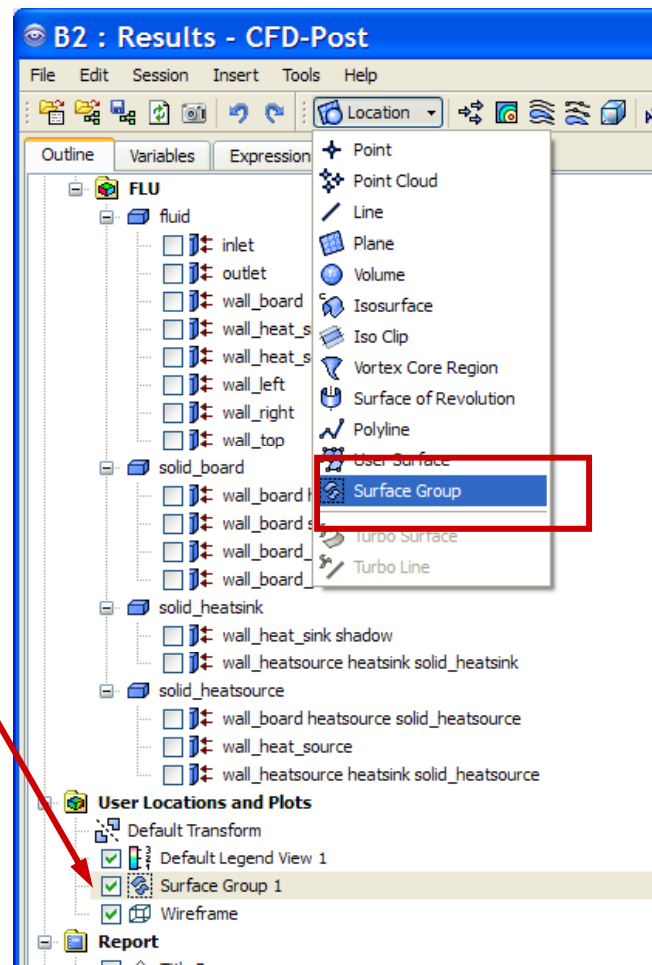
Creating Locations

- Locations are created from the *Insert* menu or from the toolbar
- 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*



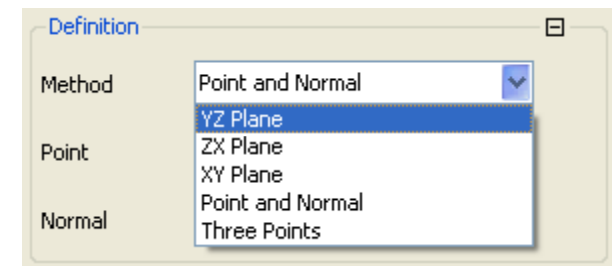
Postprocessing

Location Types



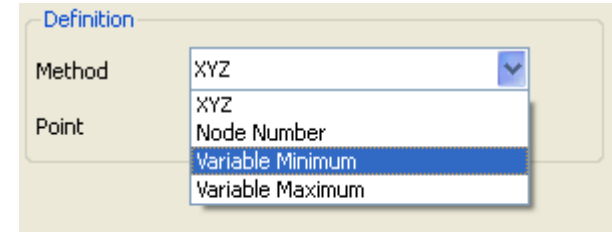
- **Planes**

- *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)



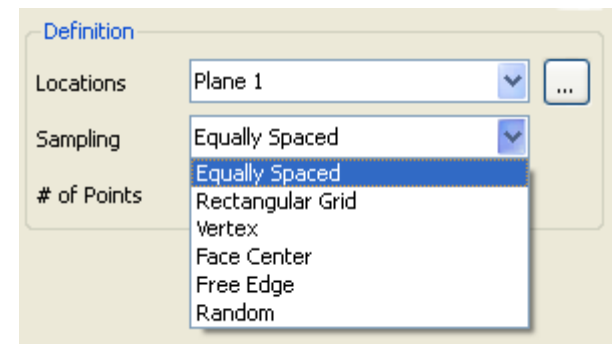
- **Point**

- *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

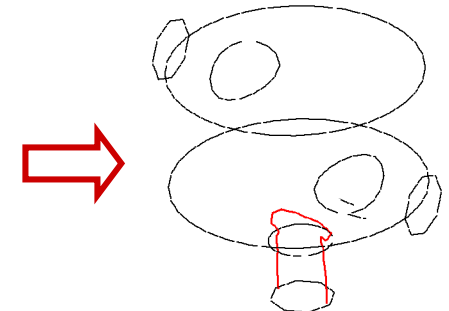
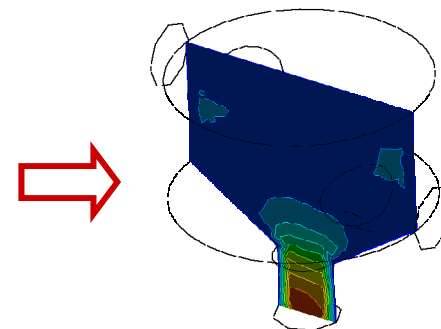
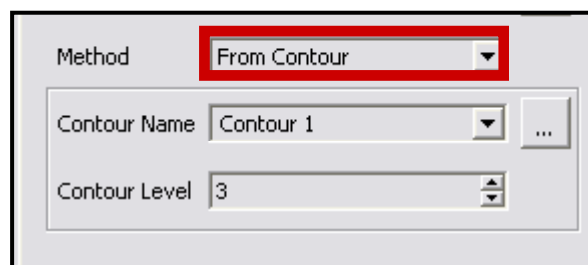
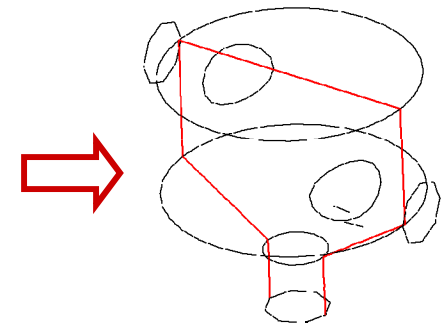
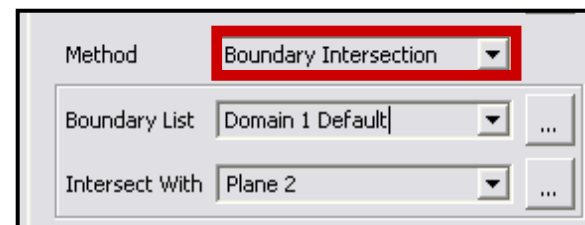
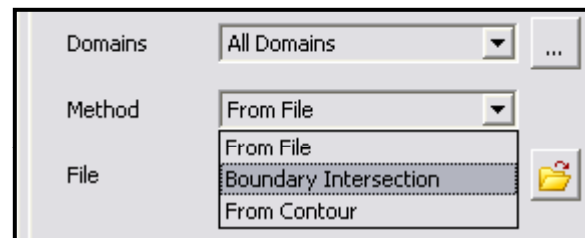


- **Lines**

- Straight line between two points
- Usually used as the basis for an XY Chart

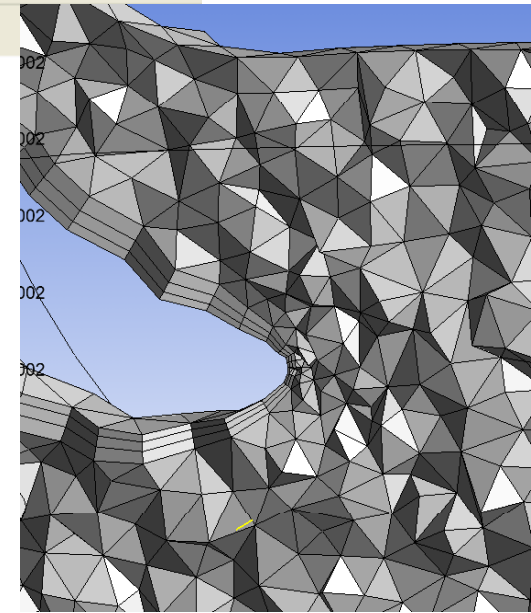
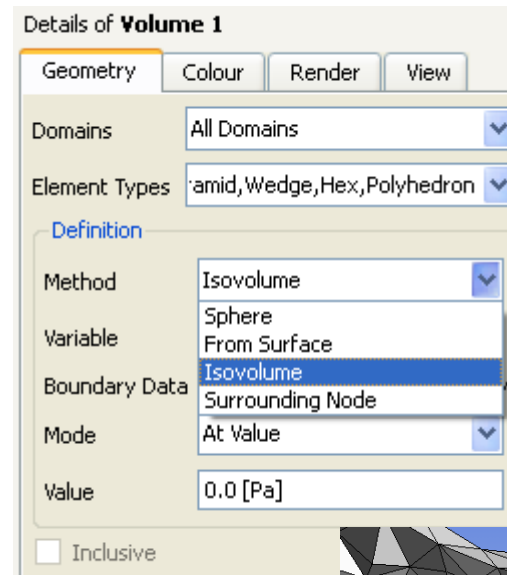
- **Polylines**

- 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



- **Volumes**

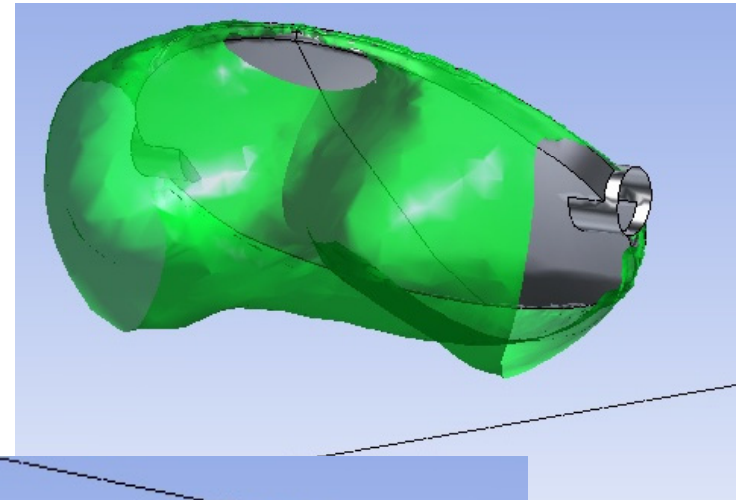
- Elements are either in or out
 - No cut volumes
- From Surface
 - A volume is formed from all elements touching (or above / below) the selected location
 - Can be useful for mesh checking
- Isovolume
 - Based on a variable at, above or below a given value, or between two values



- **Isosurfaces**

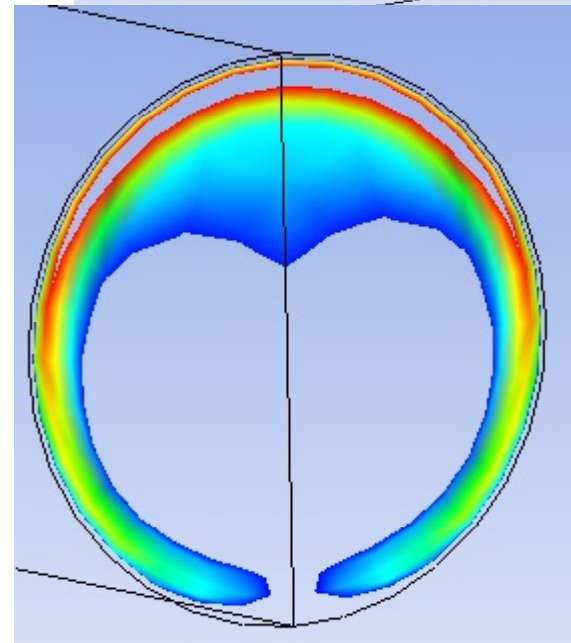
- Surface of a variable at a specified value

Isosurface of pressure behind a flap valve



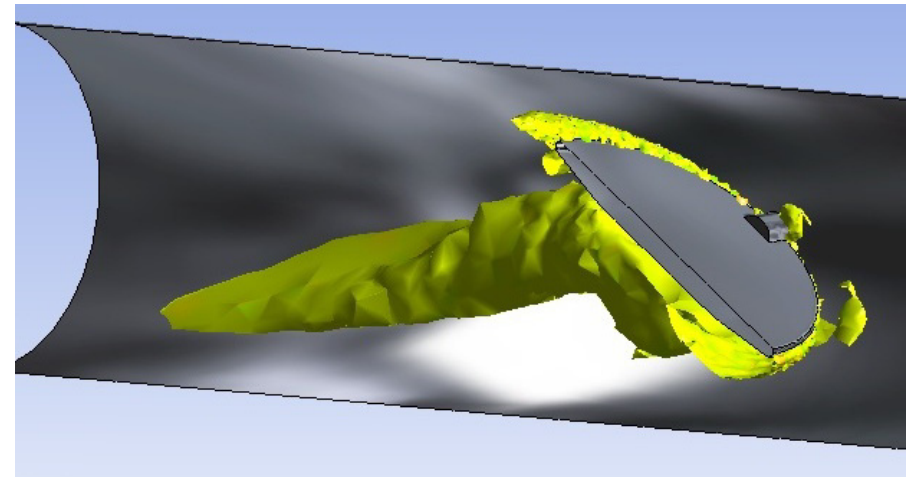
- **Iso Clip**

- An Iso Clip takes a copy of any existing location and then clips it using one or more criteria
 - E.g. a outlet boundary plot which is then clipped by Velocity ≥ 10 [m/s] and Velocity ≤ 20 [m/s]
- Can clip using any variable, including geometric variables



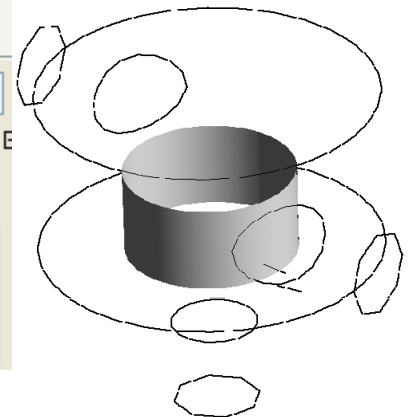
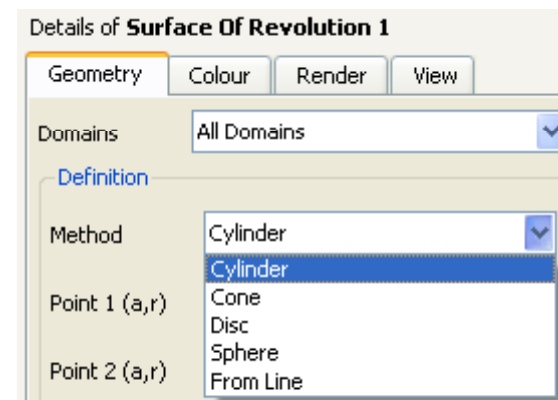
- **Vortex Core Region**

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



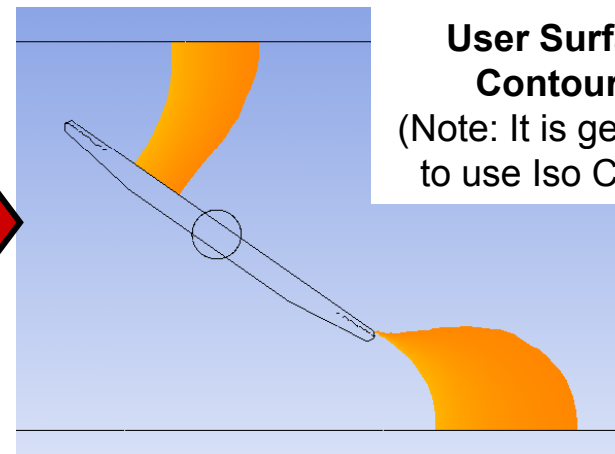
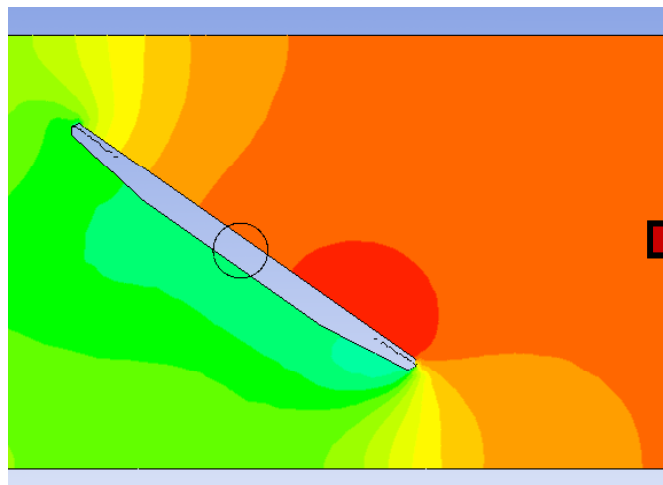
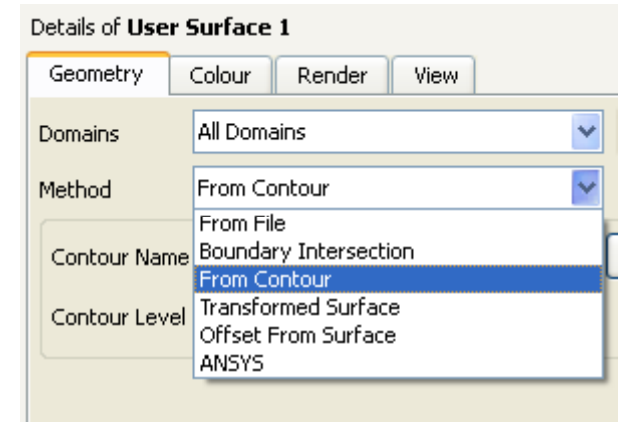
- **Surface of Revolution**

- 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



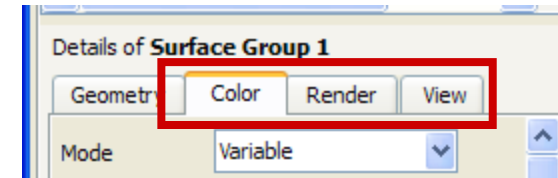
- **User Surface**

- 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 and existing surface
- **Offset From Surface**: offset an existing surface in either the Normal direction or by Translating



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

- 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*



- **Vector Plot**

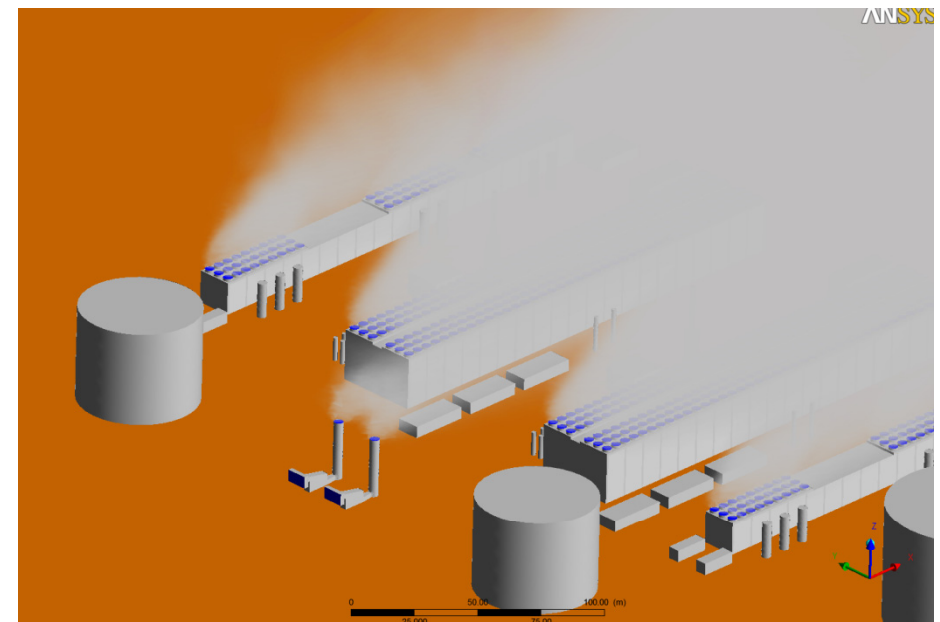
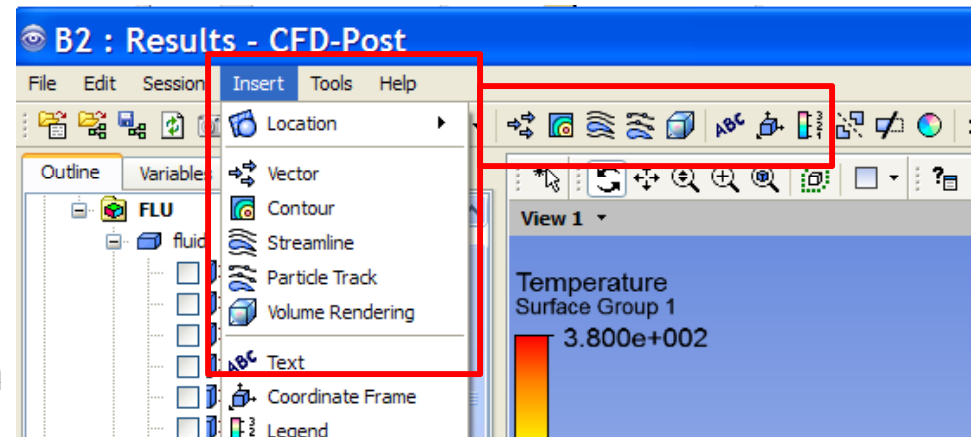
- Can plot any vector variable; usually velocity

- **Streamlines**

- Can proceed forwards and/or backwards from a seeding location

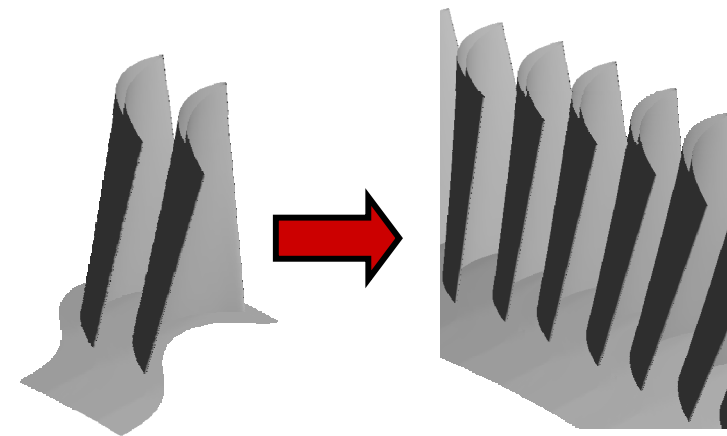
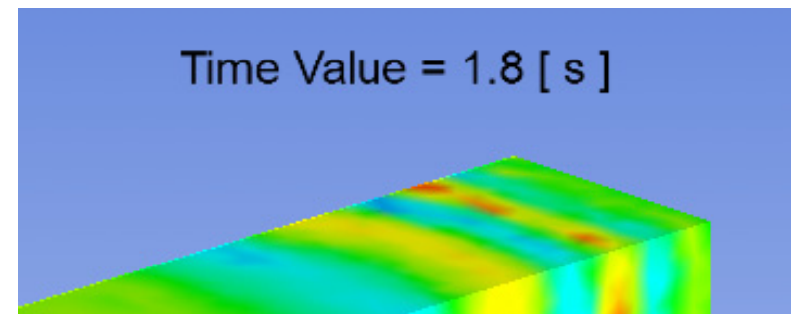
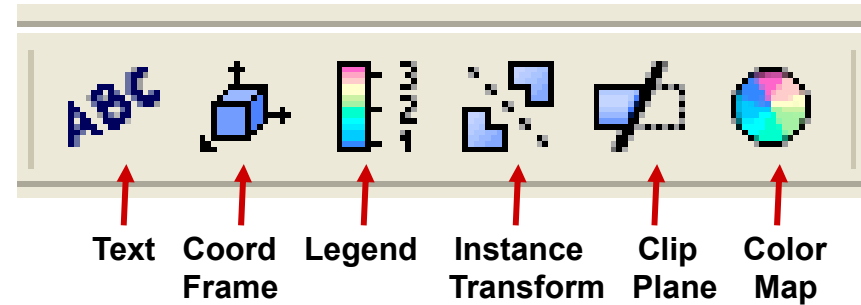
- **Volume Rendering**

- Shades every grid cell with a transparency based on a variable
- So for example, transparency could be based on smoke concentration. The result will give an idea of visibility through the smoke,



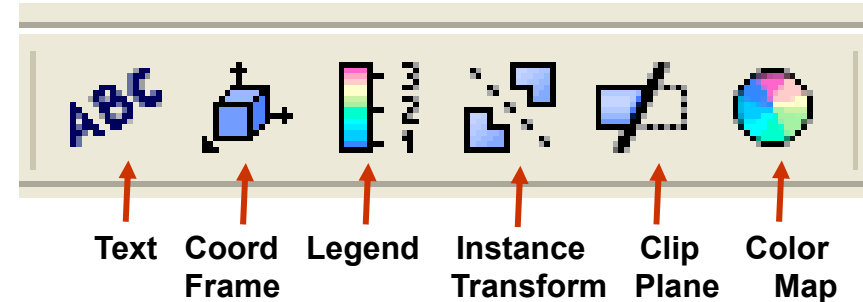
Other Graphics Objects

- **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 new coordinate frame, useful for generating plots relative to features in the model
- **Legend**
 - Create addition legend that are tied to a specific plot (the default legend changes automatically with the active plot)
- **Instance Transform**
 - Usually used to re-create full plots from symmetric/periodic solution data



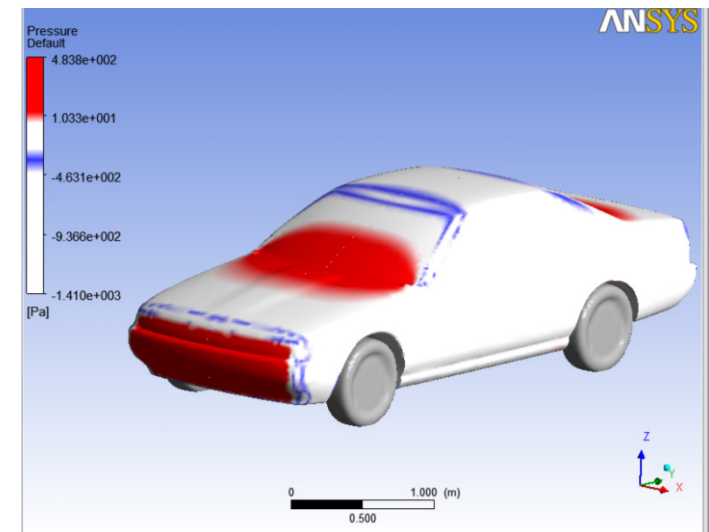
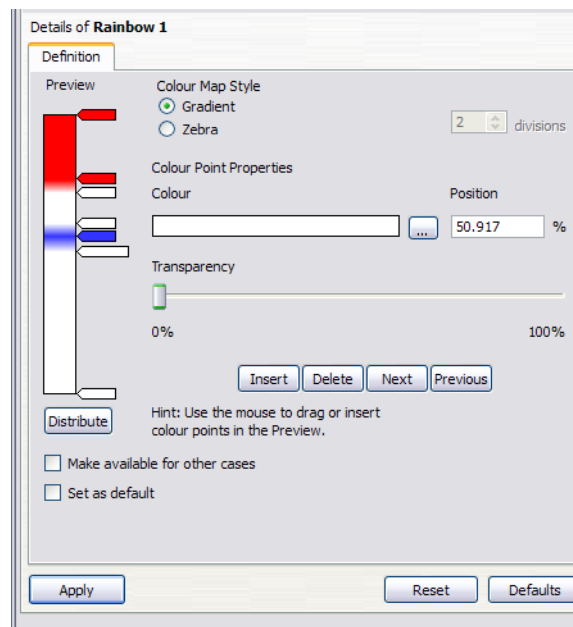
• Clip Plane

- Define a plane; when active all viewer object in front / behind this plane are hidden



• Color Map

- Create custom color maps

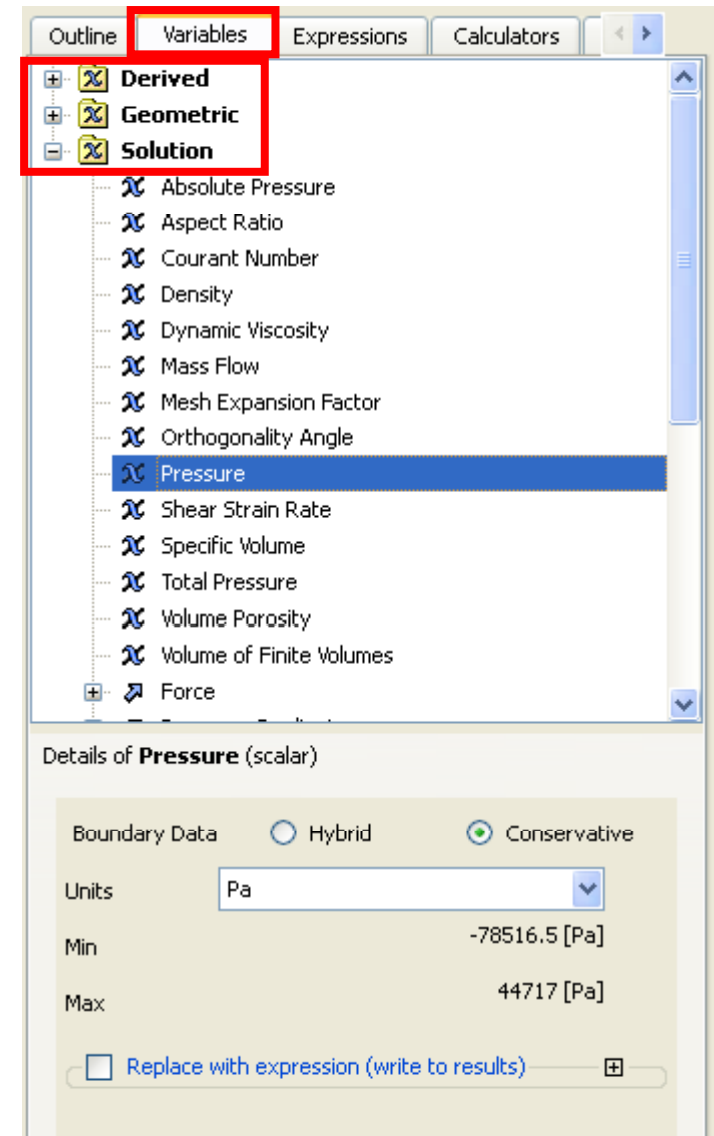


Postprocessing

Variables Tab

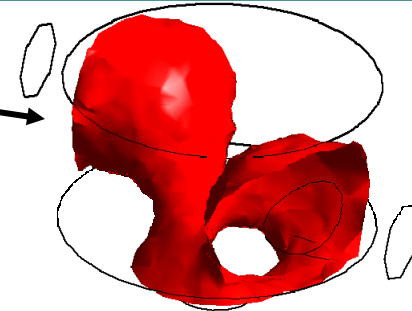


- The *Variables* Tab shows information about all available variables
- **Derived variables**
 - Calculated by CFD-Post – they are not contained in the results file
- **Geometric variables**
 - X, Y, Z, Normals , mesh quality data
- **Solution variables**
 - From the results file
- **User Defined variables**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

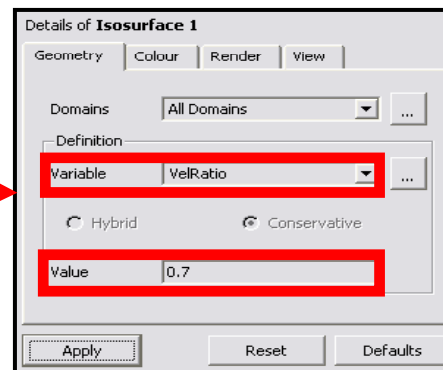
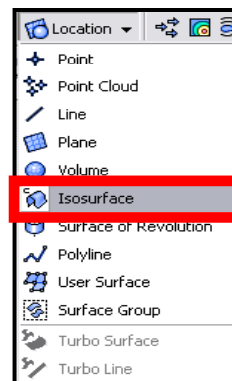
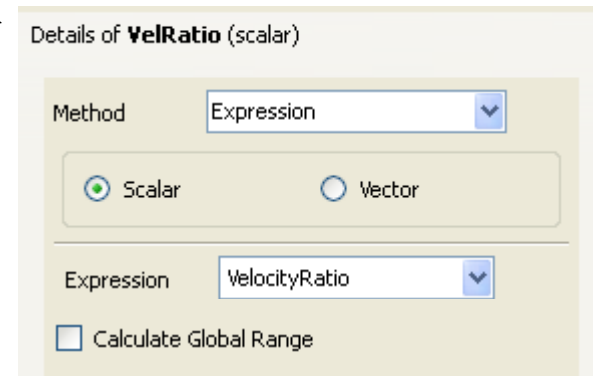
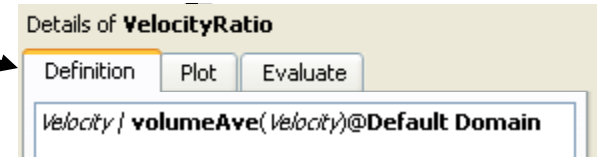


User Defined Variables Example

- Goal: Plot an isosurface at VelRatio = 0.7
where $VelRatio = \frac{V_{local}}{V}$



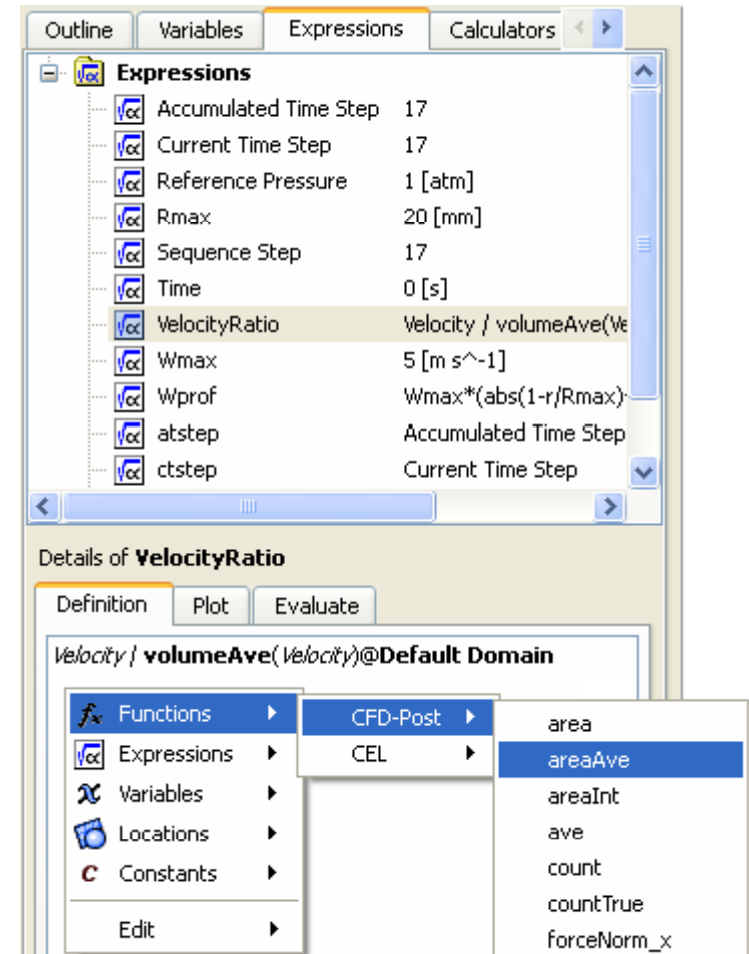
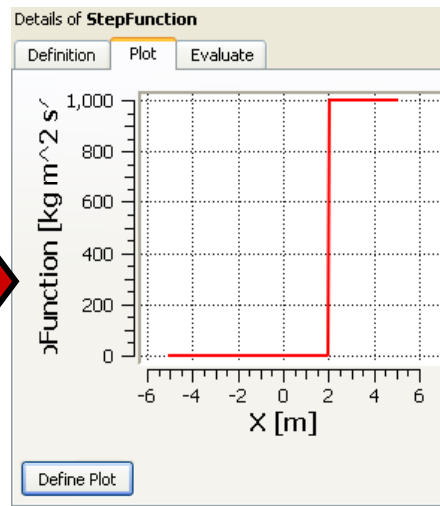
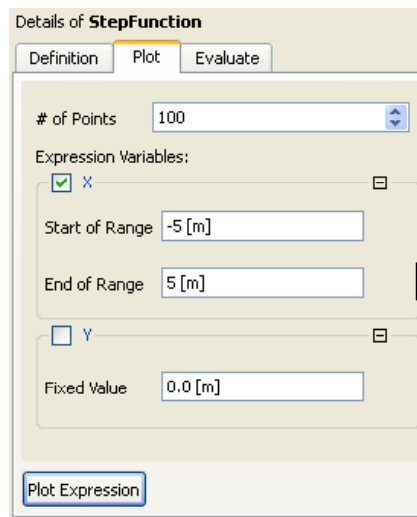
1. On the *Expressions* tab create the expression for Velocity Ratio:
2. On the *Variables* tab create a new variable named *VelRatio* using *Method = Expression*
3. Create an Isosurface using the variable *VelRatio* at a value of 0.7



Postprocessing

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

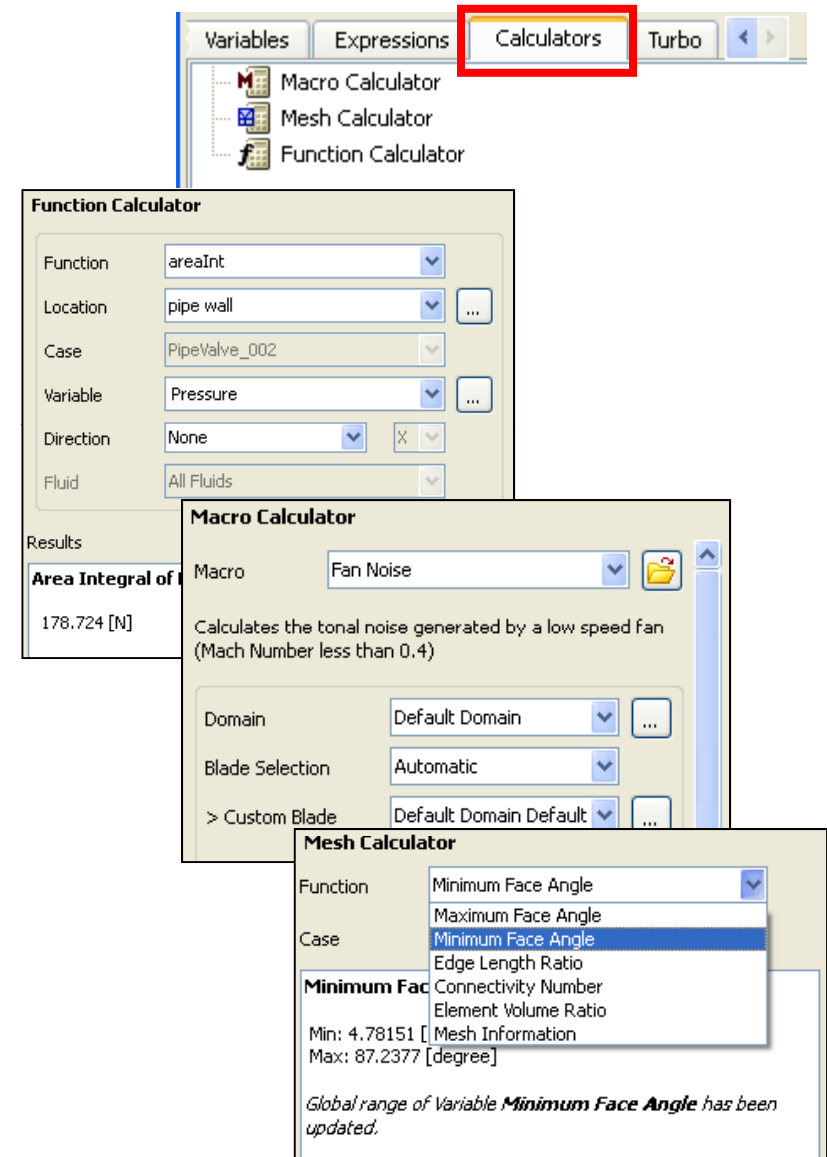


Postprocessing

Calculators Tab



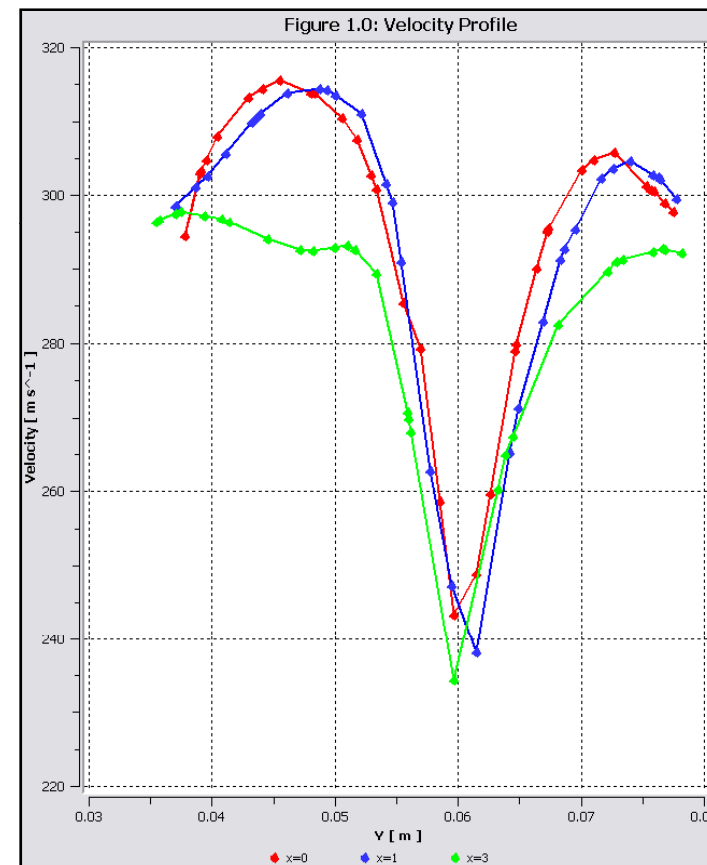
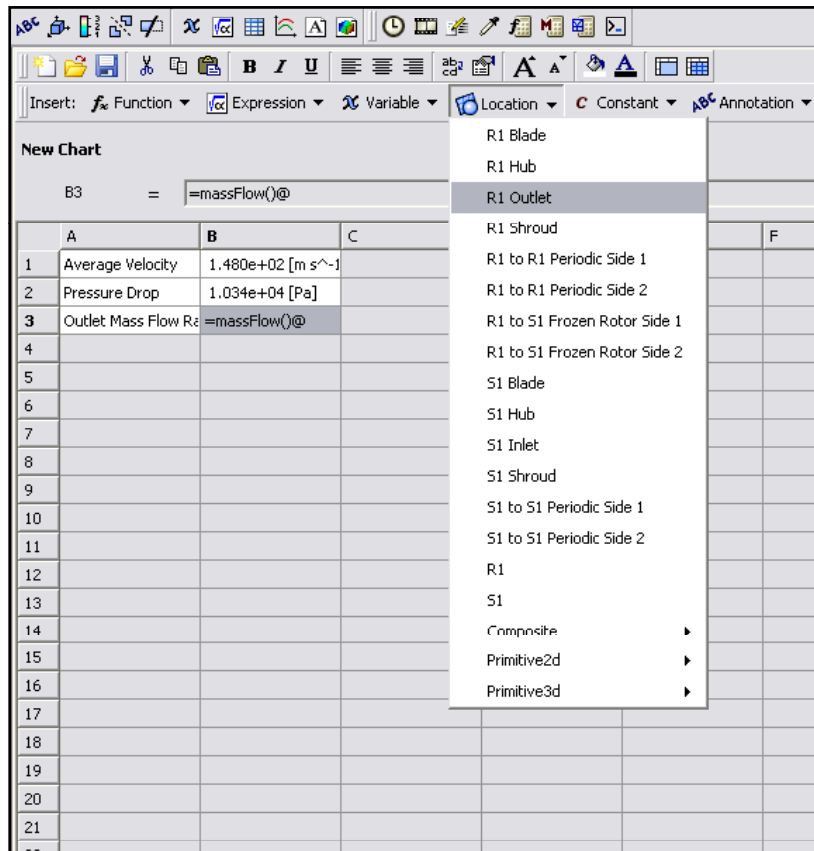
- **Function Calculator**
 - Extract engineering data from the results
 - Same function used as when creating expressions
- **Macro Calculator**
 - Run predefined Macros
 - Write your own Macros and have them appear here
- **Mesh Calculator**
 - Mesh quality metrics and stats
 - Field variables exist for all the metric and can be plotted



Postprocessing

Generating Tables and Charts

- Tables and Charts can be created to format and present results



- 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

The screenshot shows the ANSYS Table Viewer interface. The top toolbar contains various icons, with the 'Table' icon highlighted by a red box and labeled '1. Create Table'. Below the toolbar, the 'Insert' menu is open, showing options like 'Function', 'Expression', 'Variable', 'Location', 'Constant', and 'Annotation'. A red box highlights the 'Location' dropdown menu, which is labeled '4. Use drop-down menus to assist expression creation'. The main table area shows a table with columns A, B, and C. The first row is labeled 'Average Velocity' and the second row is labeled 'Pressure Drop'. The third row is labeled 'Outlet Mass Flow' and the fourth row is labeled 'Outlet Mass Flow'. The values in the B column are 4.560e+00 [m s^-1], -3.352e+04 [Pa], and -2.552e+00 [kg s^-1]. A red box highlights the 'Outlet Mass Flow' row, and a callout labeled '2. Create Text Cells' points to the first column. Another callout labeled '3. Create Expression Cells' points to the B column. The bottom of the interface shows the 'Table Viewer' tab selected, with other tabs like '3D Viewer', 'Chart Viewer', 'Comment Viewer', and 'Report Viewer' visible.

1. Create Table

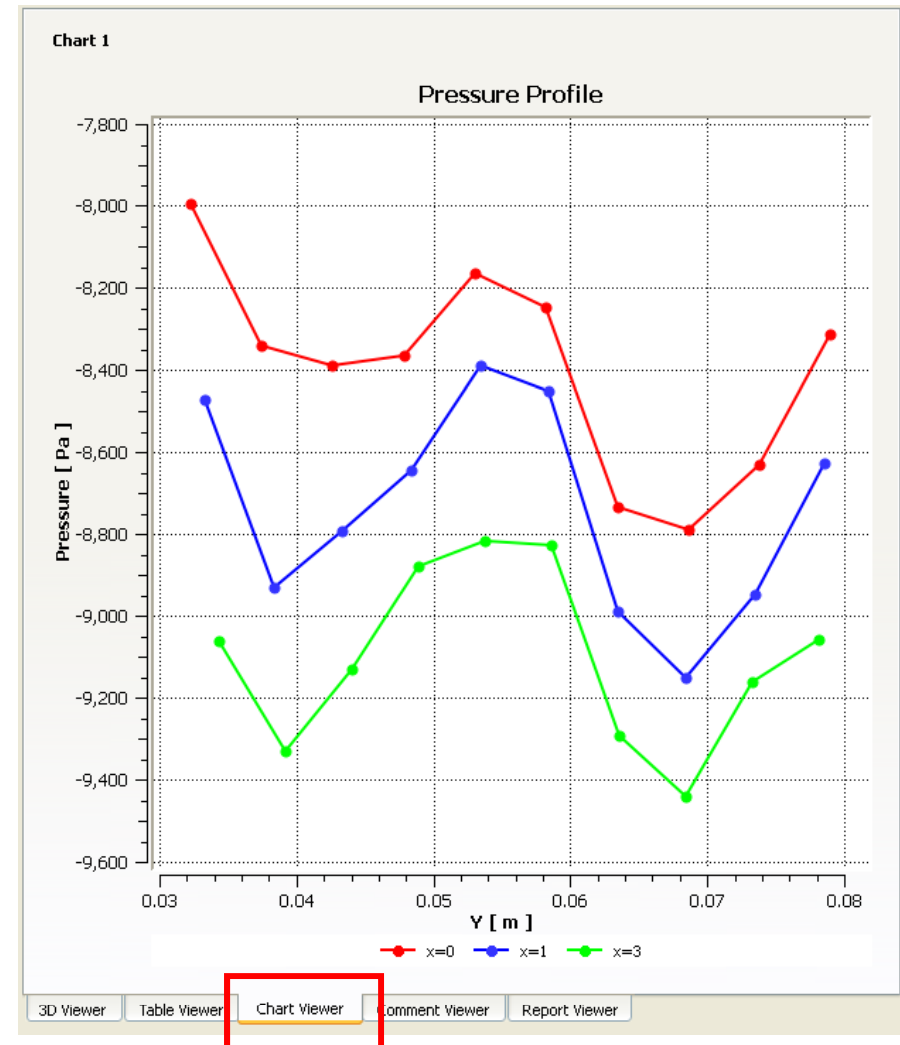
2. Create Text Cells

3. Create Expression Cells

4. Use drop-down menus to assist expression creation

	A	B	C
1	Average Velocity	4.560e+00 [m s ⁻¹]	
2	Pressure Drop	-3.352e+04 [Pa]	
3	Outlet Mass Flow	-2.552e+00 [kg s ⁻¹]	
4			

- Plot a relationship between two variables along a line/curve
 - Need to create the line first
 - Polyline, Boundary Intersection curve, Contour line, etc.
- Charts are automatically added to the Report
- Chart Points are not necessarily evenly spaced
 - Data points usually correspond to where the line/curve intersects a mesh face
- Multiple lines can be plotted on a single chart

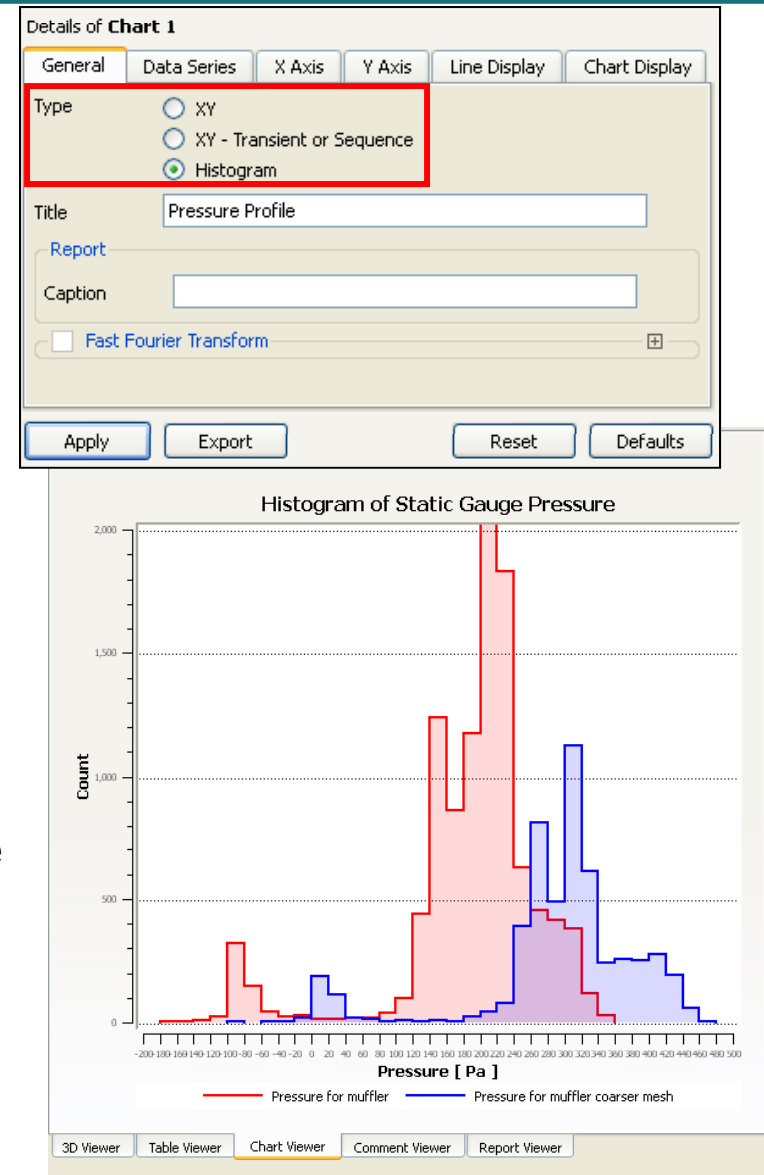


Postprocessing

Charts: Type

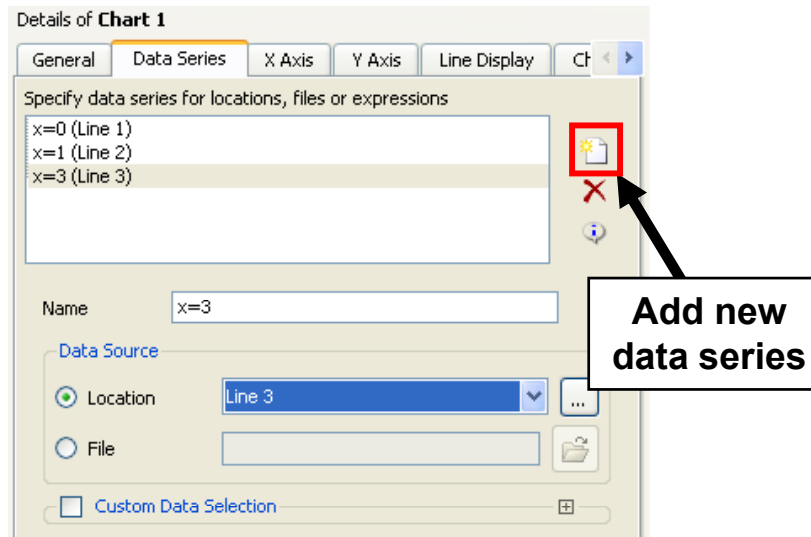


- Charts can be one of three types:
 - **XY**
 - Standard XY plots based on line locators
 - **XY – Transient or Sequence**
 - Plots an expression (usually Time) versus a variable at a point locator
 - Typically used to show the transient variation of a variable at a point
 - **Histogram**
 - Can be 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

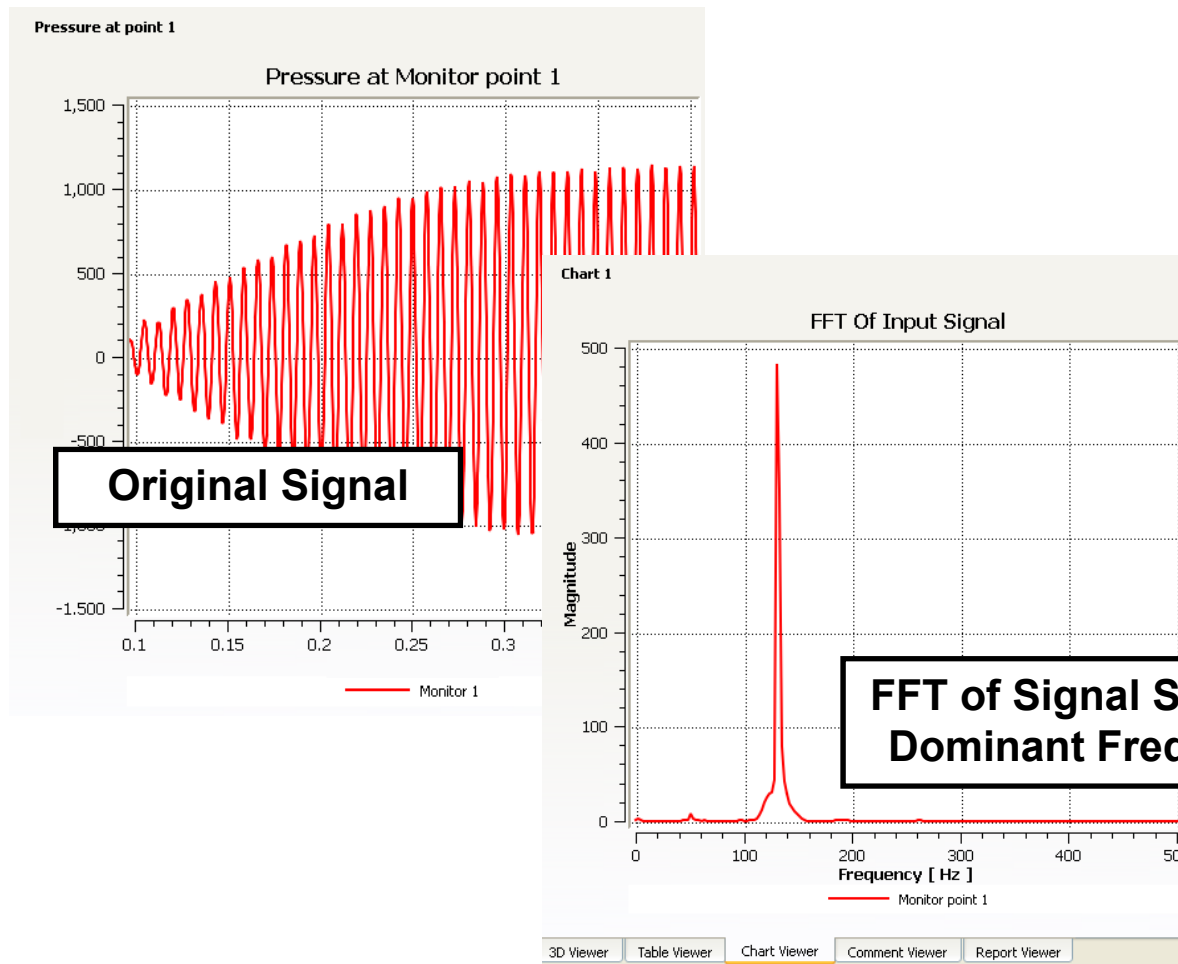


Charts: Data Series and Axes

- Each data series corresponds to a location (line, point, etc.) which corresponds to a curve on the chart



- FFT can be applied to signals to extract frequency data



Details of **Chart 1**

General | Data Series | X Axis | Y Axis | Line Display | Ch < >

Type: ☐ XY ☒ XY - Transient or Sequence ☐ Histogram

Title:

Report:

☒ **Fast Fourier Transform**

Modify Input Signal Filter:

☐ Subtract mean

☒ Full range of input data

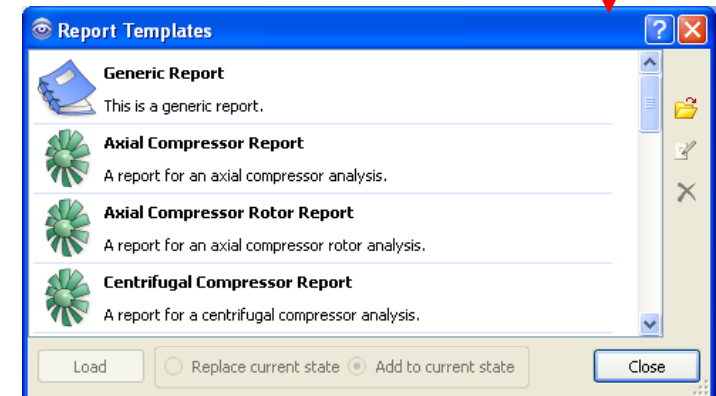
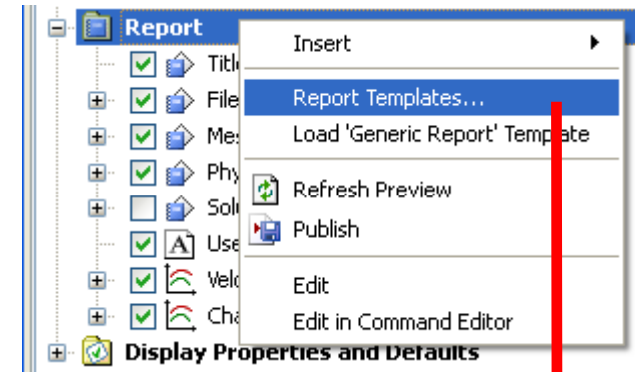
Min: Max:

☒ Refresh chart on Apply ☐ Refresh all charts on Apply

Postprocessing Reports

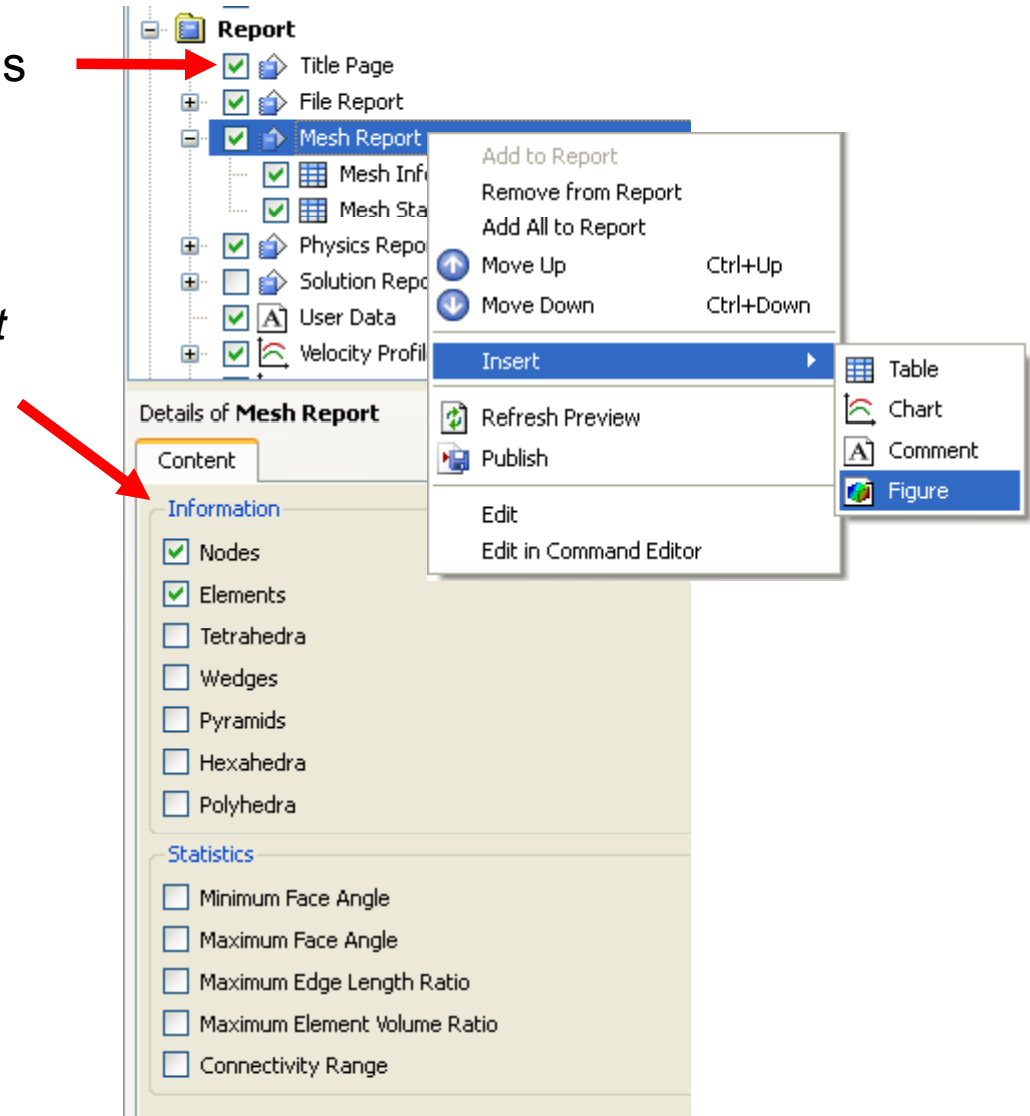


- CFD-Post has **report generation tools** which allow for rapid creation of customized reports
- Reports are template based
 - Depending on the information contained in a results file, a report template will be selected automatically
 - Right-click on *Report* to select a different template
 - You can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

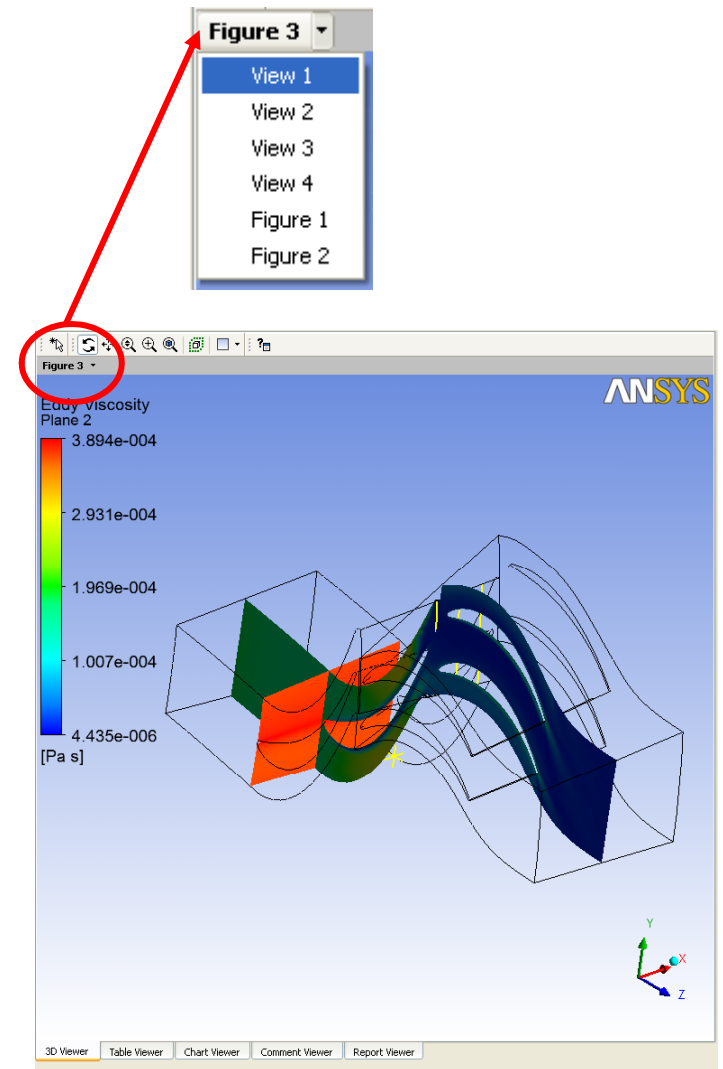


Postprocessing Reports

- 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



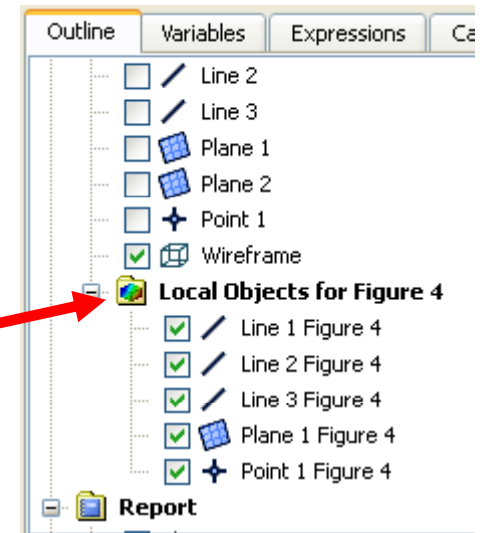
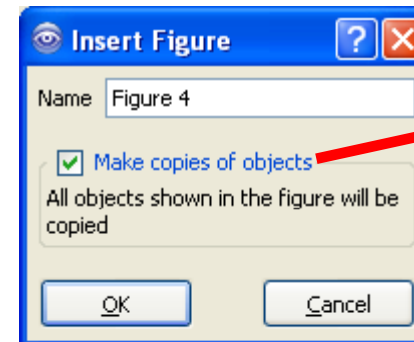
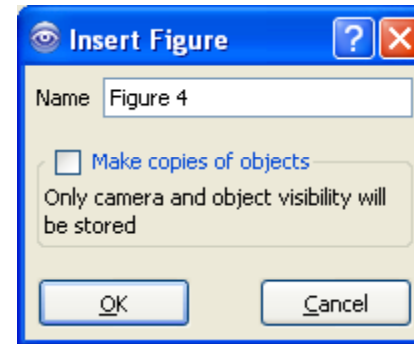
- When you add a new *Figure* it will be listed in the drop-down menu in the top corner of the Viewer
- Figures are not static, you can change them after they have been created
 - If you do not want to change a Figure, make sure one of *View 1* – *View 4* is selected from the drop down menu
- To change 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



Postprocessing

Reports: Figures

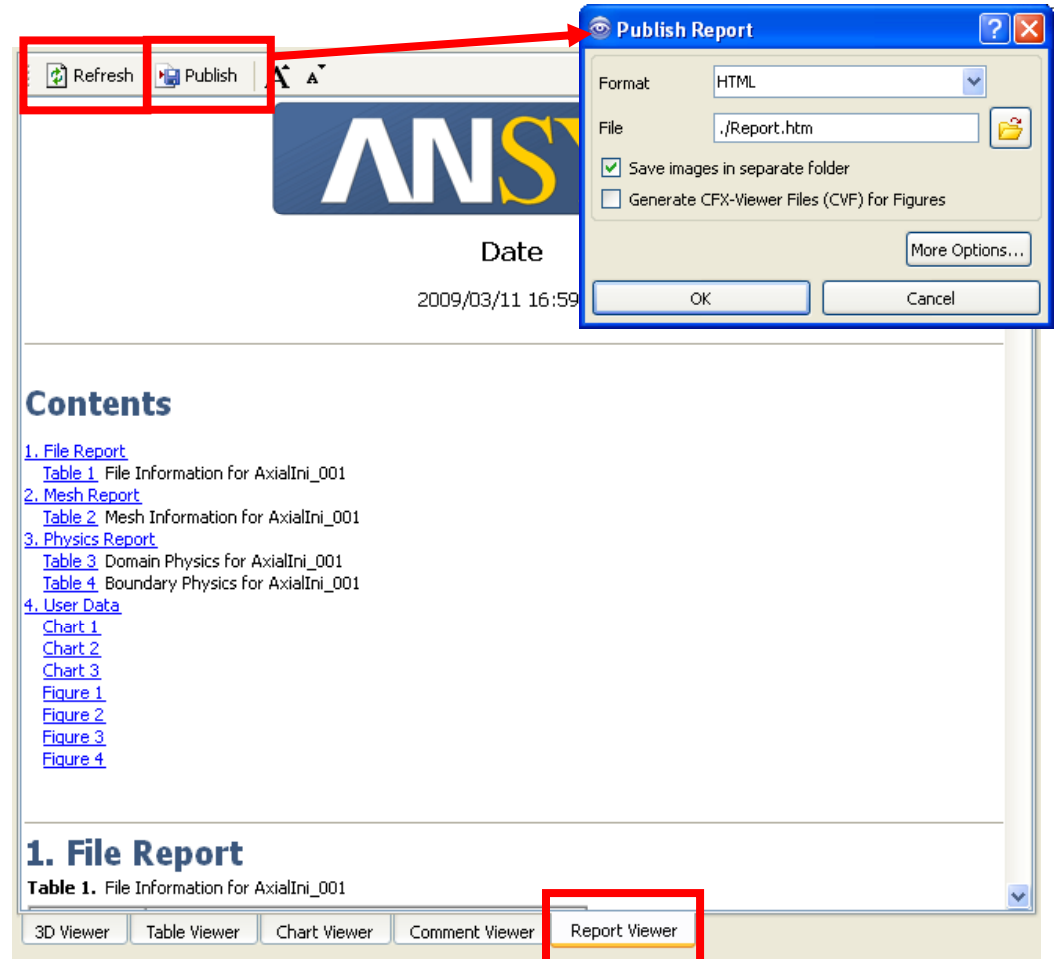
- When you create a Figure, you have the option to *Make copies of objects*
 - If you disable this only the camera and object visibility is stored with the figure
 - This is used if your figures only vary in terms of the objects visible.
 - If you enable this a local copy of all the current objects is created and shown in the *Outline* tree
 - Suppose you have a Plane, and in the different figures this Plane will show different variables like velocity, temperature etc, then take this option
 - In both cases the camera position and object visibility can only be changed when the Figure is active



Postprocessing Reports



- To view the report 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



- **Timestep Selector**

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

- **Animation**

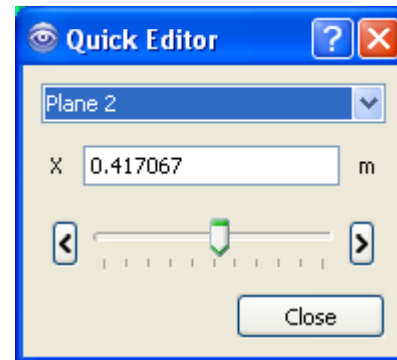
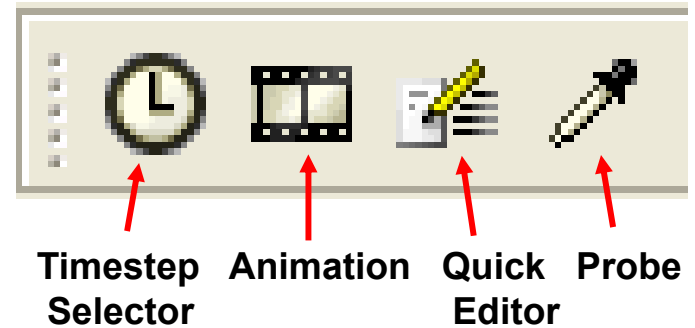
- Animate objects, create MPEGs
- More on next slide

- **Quick Editor**

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

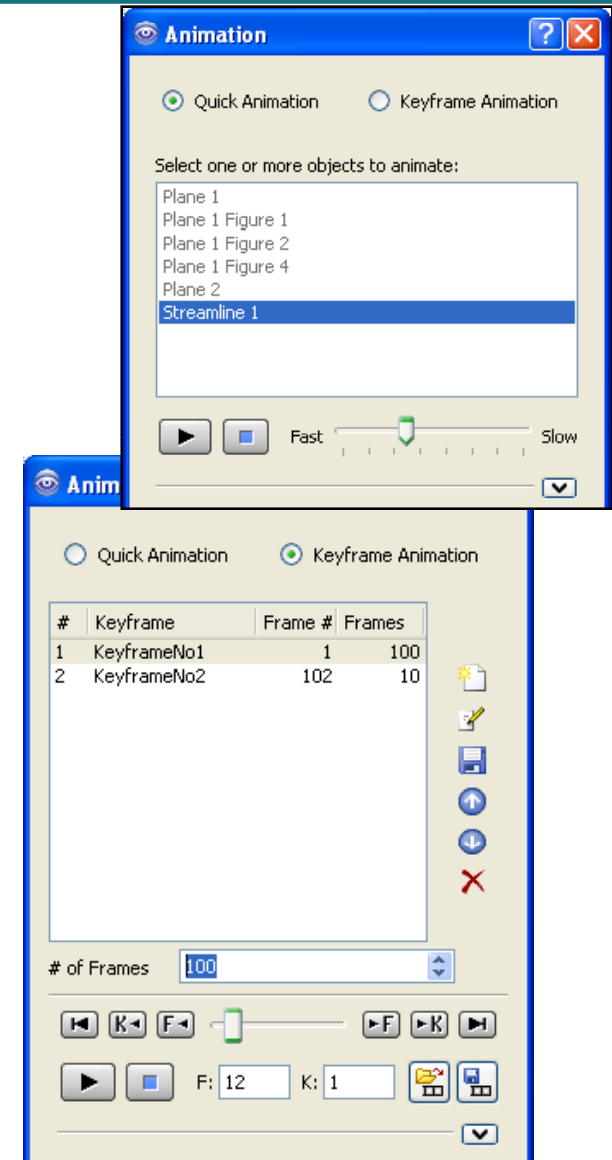
- **Probe**

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



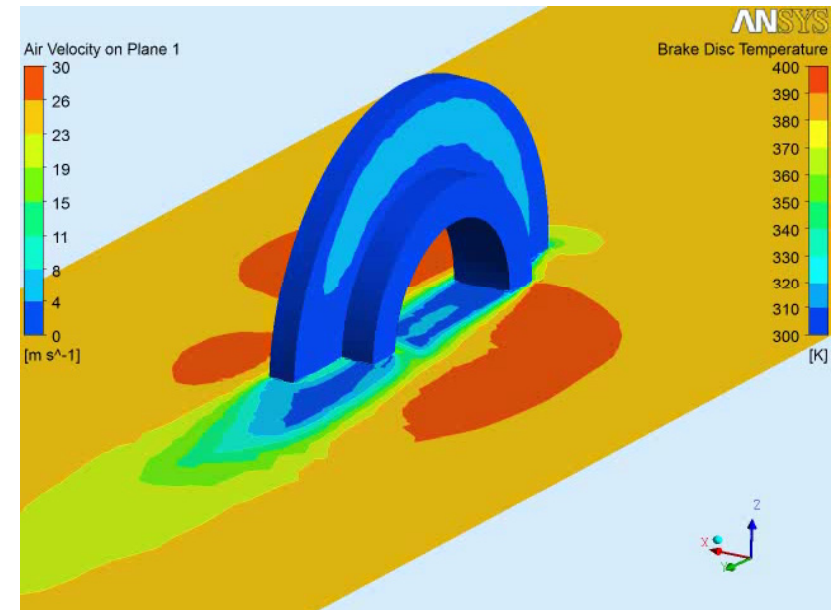
Postprocessing Animations

- Animations have two modes, *Quick* and *Keyframe*
- In *Quick Animation* mode you simply pick an object and click the *Play* button
 - The “primary” variable for that object is animated
 - Limited control
- *Keyframe* mode gives you much more control
 - When you create a Keyframe a snapshot of the current state is stored with that Keyframe
 - A series of Keyframes represent a series of different states
 - Camera position, object visibility, selected timestep, or anything else can be different between Keyframes
 - An animation needs at least two Keyframes (one for the start and one for the end)
 - Enter the *# of Frames* between each Keyframe
 - Everything that is different between the Keyframes gets interpolated over the *# of Frame*



Typical Keyframe Animation Procedure

1. Using the *Timestep Selector* load the first timestep
2. Create necessary plots and position the view
3. Create the first *Keyframe*
4. Load the last timestep
5. If necessary change the plots and the view
6. Create the second *Keyframe*
7. Select the first *Keyframe* and set the *number of Frames*
 - This is the number of Frames between the first and second Keyframes
 - If we have a total of 100 timesteps, then setting # of Frames to 98 will produce a total of 100 frames (98 plus first, plus last) and therefore 1 frame per timestep
8. Set the Movie options
9. Rewind to the first *Keyframe* and click *Play*

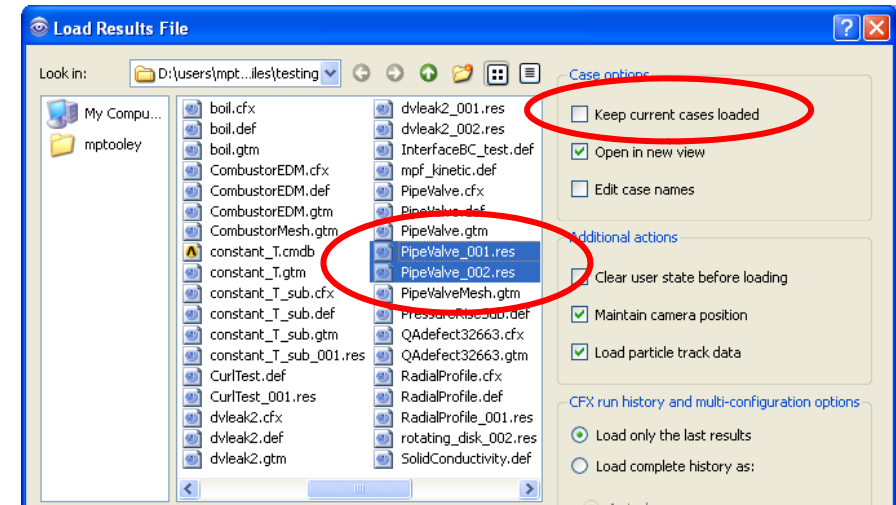


In this example the first and second Keyframes used a different view position and the Transparency of the Plane was changed from 0 to 1. The changes between Keyframes are interpolated gradually over each animation frame

Postprocessing

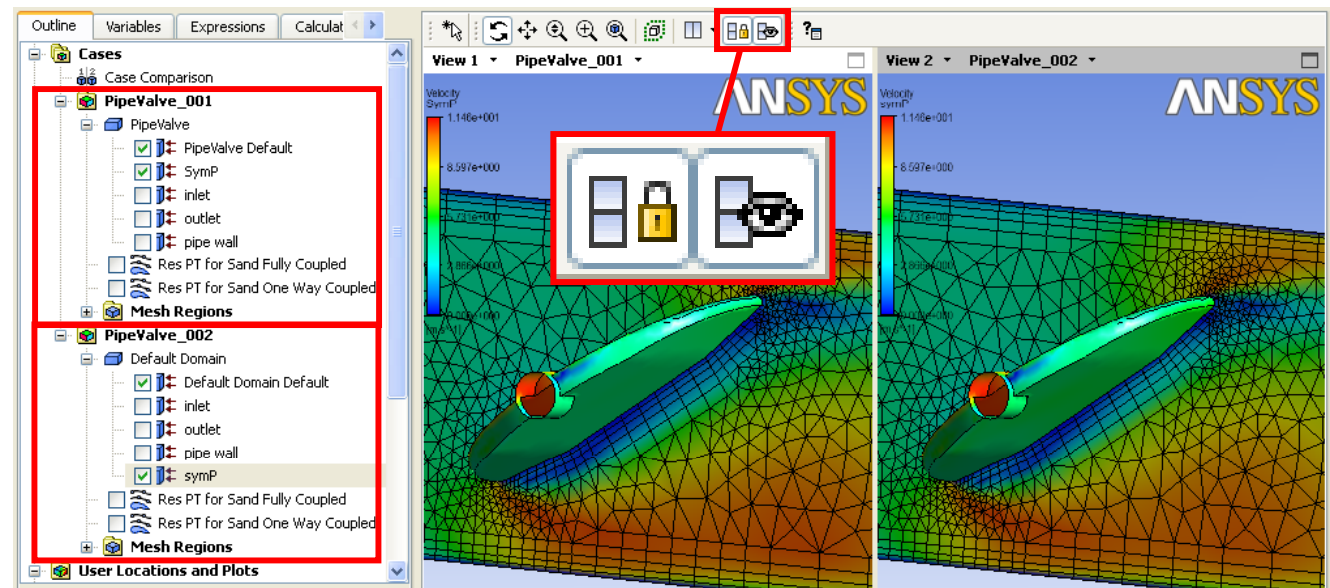
Multi File Mode

- To post-process multiple files simultaneously you can:
 - Multi-select files when loading
 - Or 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



- **Results**

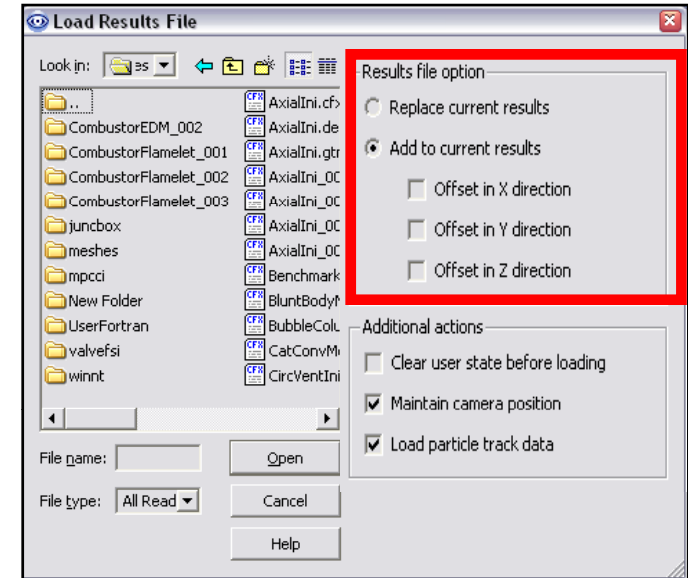
- ANSYS – CFD Post can read ANSYS results for temperature, velocity, acceleration, magnetic forces, stress, strain, and mesh deformation

- **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

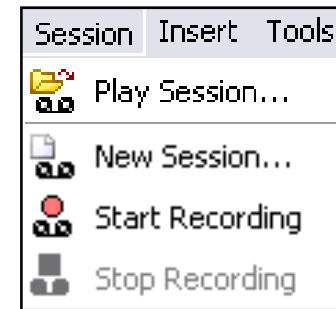
- **Export**

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



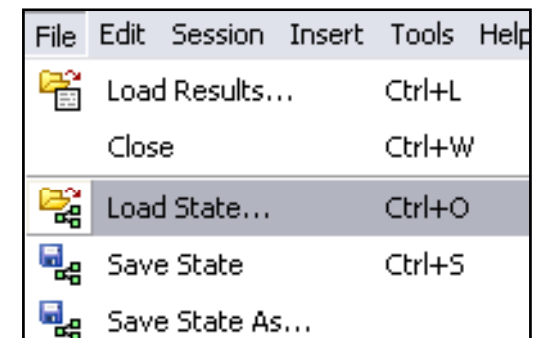
- **Session**

- 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 snapshot of all objects
- Excludes actions (e.g. file output)



- **Macros**

- Useful for automating tasks.

