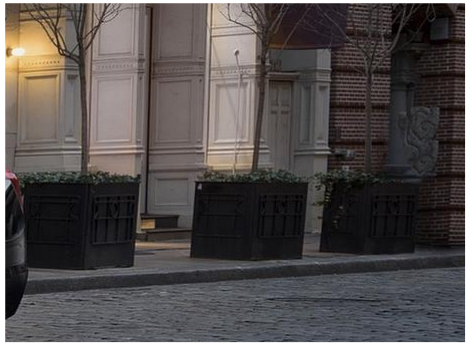


EnSight 10.2 Advanced Training

ANSYS

Martin J. Faber – Sr. Account Manager, North America


EnSight 10 Advanced Training Overview 1

More EnSight Basics

Client-Server architecture, EnSight processes, global shading and hidden lines overlay, 4 icons on the right (display information, variable legend visibility, record animation, fast display), EnSight modes

Parts and the Quick Edit Menu

Line width, visibility per viewport, element representations, displacements, visual symmetry, element labeling, parts filter elements, element blanking, shaded and hidden line mode, auxiliary clipping, fast display representation

Solution Time and Flipbook

Time and advanced tab, discrete or continuous, record an animation, flipbook load and run tab, objects and images, 4 types of data, record a flipbook

Pathlines (Transient Streamlines)

Pathlines introduction, creating pathlines, displaying pathlines, displaying pathline particle traces

Copy, Clone and Tag

Part duplicating, when to use each one, create a part copy, create a part clone, part tagging, how to

EnSight 10 Advanced Training Overview 2

Plots

Curve/graph visibility, show graph in separate window, graph attributes, axis attributes, curve attributes, select/delete graphs

More Part Creation

Contours, vector arrows, elevated surface, separation & attachment lines, vortex cores

Create/Edit Viewports

Viewport overview, linking viewports, exchange with largest viewport, background color, border and location, bounds visibility, special attributes, viewport layouts, new viewport, pop/push viewports

Texture Mapping and Skybox

Background of texture mapping, using decals, loading and Z-axis projection, repeat mode, texture mode, update the plane tool, using patterns, modulate, background of a skybox, box tool, skybox picture layout

The Keyframe Animator

Keyframe animation basics, quick animations, keyframe animation examples, run attributes, animating a clip plane, recording a KF animation, tips and tricks, examples

EnSight 10 Advanced Training Overview 3

Using Predefined Materials

Materials library overview, materials assignment example, lighting and shading per material, style feature to copy materials, notes about the materials library, using materials on the Chevy Traverse, Chevy Traverse animation

Using Multiple Light Sources

Multiple light sources overview, types of light, notes about light sources, example of using multiple light sources, default light vs 3 light sources

Ray Tracing

Ray Tracing basics, creating a Ray Traced image, creating auxiliary geometry, wheel - Whitted quality 3, wheel - Ambient Occlusion quality 3, Ray Traced images comparison, Chevy Traverse – Whitted quality 3, Chevy Traverse – Ambient Occlusion quality 3

The Transformation Editor

Global transformation, Z-clipping, tools: cursor, line, plane & quadratic, center of transform

EnSight 10 Advanced Training Overview 3

Create & Edit Attributes (Parts)

Several categories of attributes, full control of all features

Case Linking

How to, operation, viewport capabilities, toggle full view, viewport exchange, case linking off, notes

Annotations

Text strings, lines, 2D shapes, 3D arrow with text, dials, bar gauges, logos, legends, select, delete

Actions that will Slow Down your System

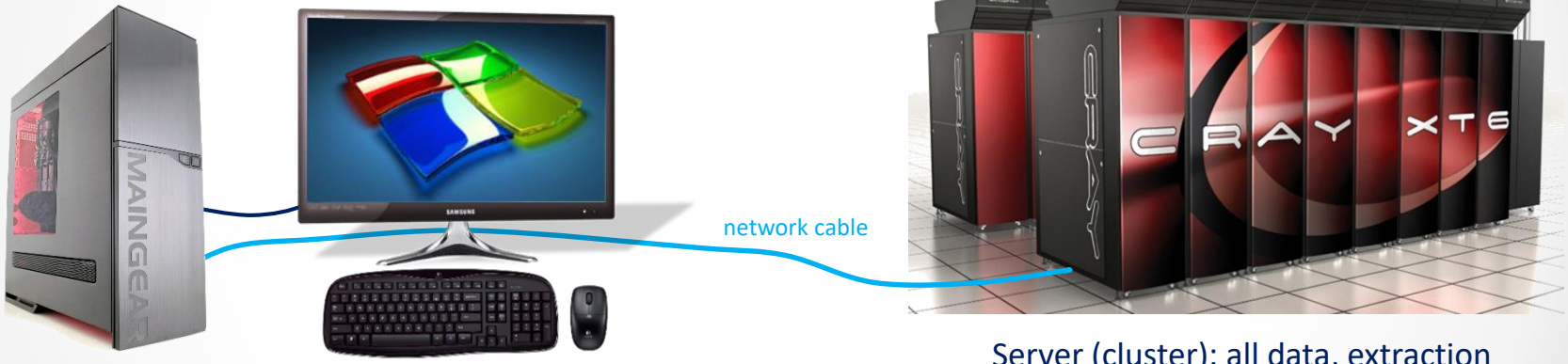
Hidden line display, opacity, element representation, keyframe animation, anti-aliasing, high resolution, label display

2017 Chevrolet Volt Cutaway Drawing

Chapter 1: More EnSight Basics

Client Server Architecture 1

- EnSight works on a client-server architecture that distributes the workloads between the client and one or more servers – this architecture leaves the data where it was computed; if EnSight is run on a single computer, both client and server are run on that machine



Client (GUI & all graphics)

Server (cluster): all data, extraction algorithms; local or remote

- The GUI (Graphical User Interface) and graphics are handled by the Client; the Server takes care of all data and the data extraction algorithms; the Server can be local or remote

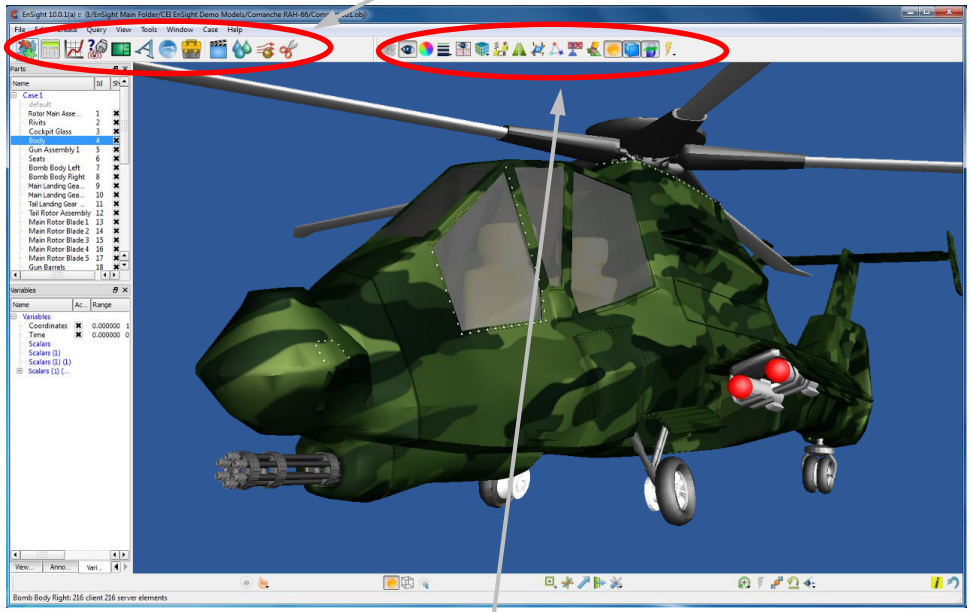
Client Server Architecture 2

- When an icon is clicked on the Feature Icon Bar, for instance to create streamlines, the Client sends a request to the Server to calculate streamlines

- The Server responds by calculating the streamlines and sends back the 3D objects; the Client then updates the display

Client Server Architecture 3

- When an icon is selected on the EnSight Feature Toolbar, an action is triggered mainly on the Server



- When an icon is selected on the Quick Edit Menu, an action is triggered on the Client

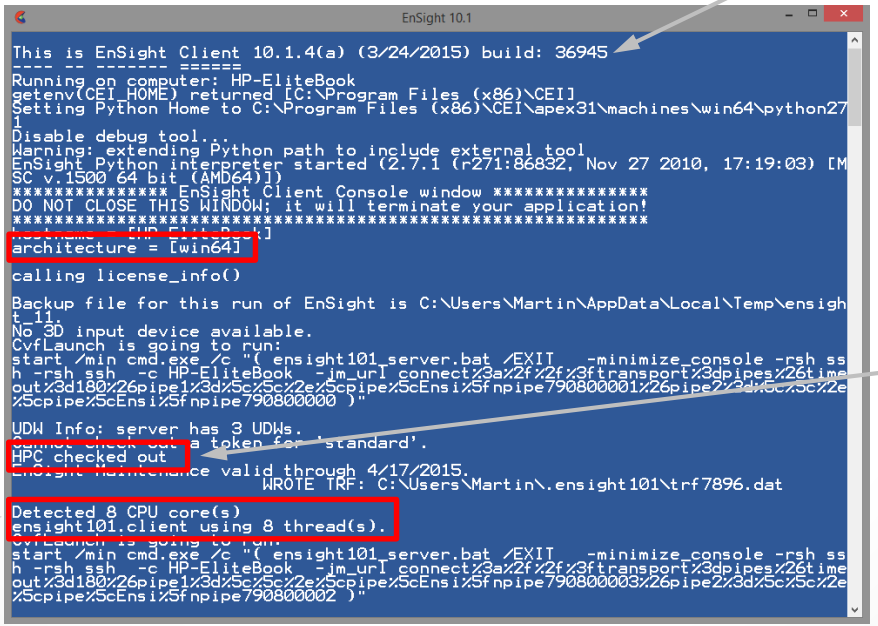
EnSight Processes 1

- When EnSight is started in Windows, an EnSight Client Console window is opened that starts the Client process; as soon as the Client process is running and the EnSight window with the GUI appears, the EnSight Client Console window is minimized (it can be opened though)

Runs in 64-bit mode

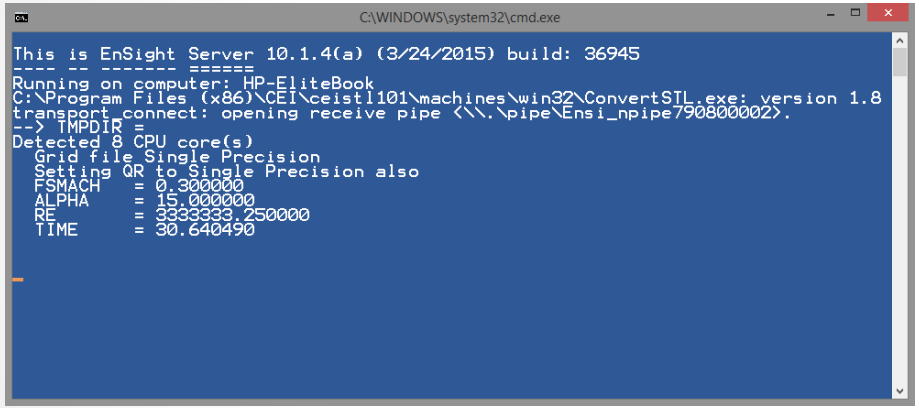
Number of CPU's detected and number of threads that are being used

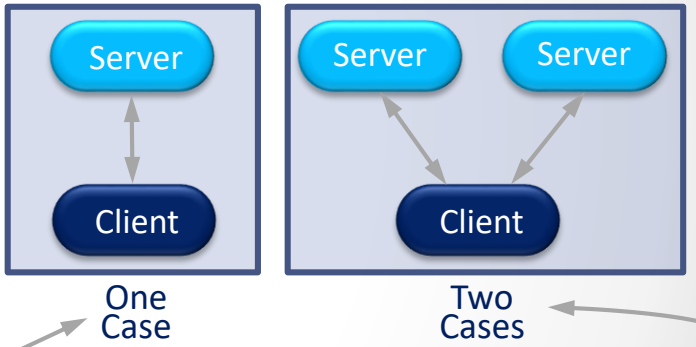
EnSight HPC license checked out



EnSight Processes 2

- As soon as the Client process is running, EnSight will start another Console window for the Server process, similar to the EnSight Client Console window; this window will also automatically be minimized (it can be opened as well)



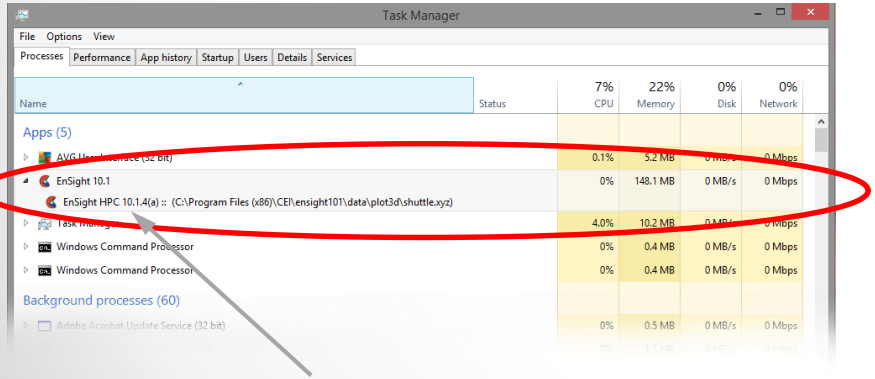


- During normal operation, EnSight will use 1 Client process and 1 Server process; when a second model (case) is loaded, EnSight will start a second Server process for the second case; this means EnSight can connect a single Client to multiple Servers at the same time, with each Server maintaining a unique dataset

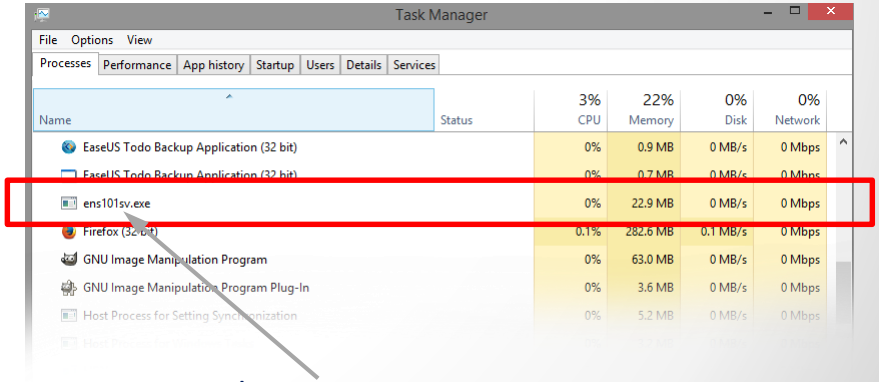
EnSight Processes 3

- Should the client cease to work (in case of a crash), one or more server processes are most likely still running; by closing the EnSight Console Server window, the EnSight Server process will also be shut down

- When EnSight is running, the Task Manager (Ctrl-Shift-Esc) in Windows (**ps** command in Linux) will show both Client and Server processes running at the same time (the Task Manager of Windows 8.1 is shown below)



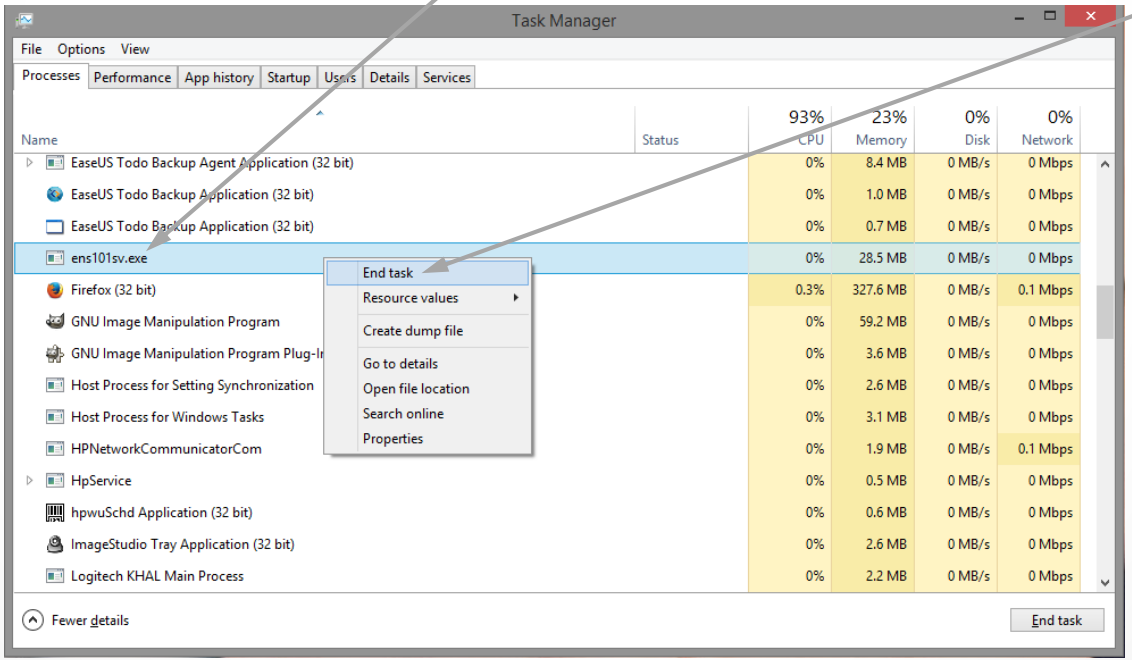
EnSight Client Process: ens101cl.exe



EnSight Server Process: ens101sv.exe

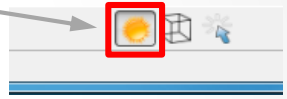
EnSight Processes 4

- The Task Manager in Windows (the **kill** command in Linux) can also be used to end a lone EnSight Server process because it can slow down the system: **Task Manager -> Processes** and left-click ens101sv.exe, then right-click and select **End Task**



Global Shading and Hidden Lines 1

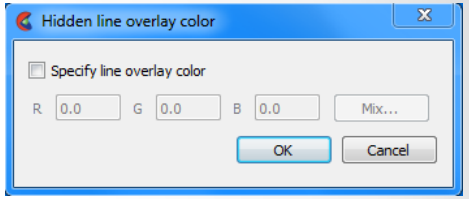
- On the Quick Settings Menu are 2 important icons:
  - The one with the icon of the sun is the global **Display Shaded Surfaces**; this button is on by default and when it is switched off there will be no shaded surfaces displayed in EnSight, just wireframe; this can also be toggled through **View** -> **Shaded**



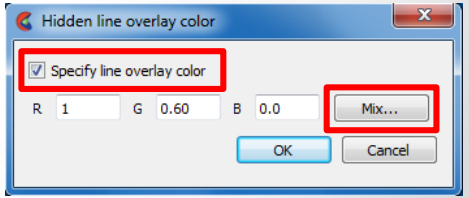
  - To the right of this icon is the global **Overlay Hidden Lines** icon that is switched off by default; this toggle enables or disables hidden line mode; this can also be toggled through **View** -> **Hidden line**

Global Shading and Hidden Lines 2

- If the current mode is Shaded when **Overlay Hidden Lines** icon is toggled on, the **Hidden Line Overlay** dialog is displayed; on this dialog a color for the overlay edges can be specified



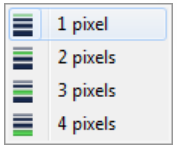
- If the **Specify Line Overlay Color** toggle is not enabled, the overlay color will be set to the native color of each part; if it is enabled, the color can be specified either by entering red, green or blue color values or by clicking the **Mix...** button and selecting a color from the palette



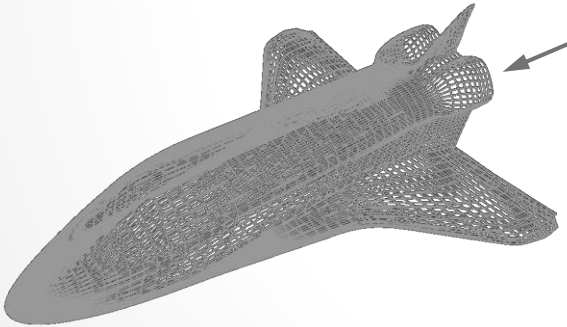
Chapter 2: Parts and the Quick Edit Menu

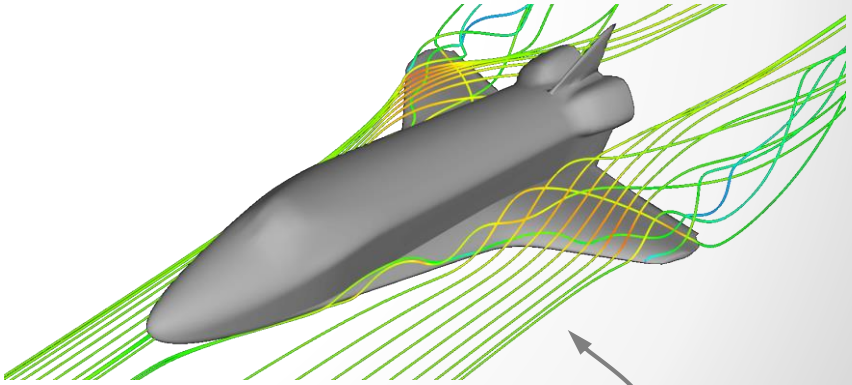
Line Width

- The **Line Width** icon on the Quick Edit Menu controls the line width (in pixels) of any part; by default it is set to a width of 1 pixel; this icon will update to reflect attribute changes



- When displaying for instance the Shuttle model in wireframe, the width of the wireframe can be adjusted; in this example from 1 to 3 pixels

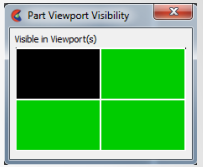




- In this example the streamlines are adjusted from 1 to 4 pixels

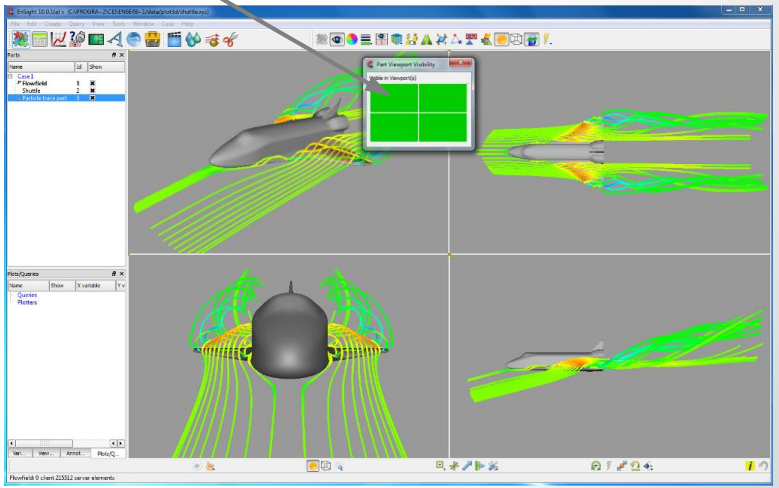
Visibility per Viewport

- The **Visibility per Viewport** icon controls whether a part is visible in a certain viewport or not; black means the selected part is not visible



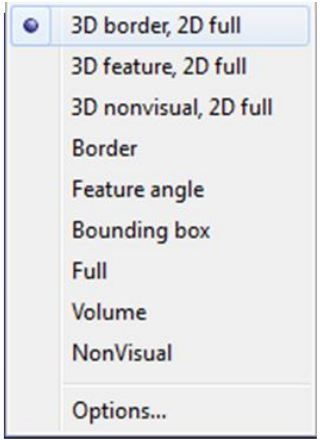
- When multiple viewports are used this feature will display a thumbnail of the viewports; click on the thumbnails to make a part visible or invisible in a particular viewport

- In this example the streamlines in the top left viewport are not displayed; by selecting the thumbnail on the menu the streamlines can be displayed



Element Representations 1

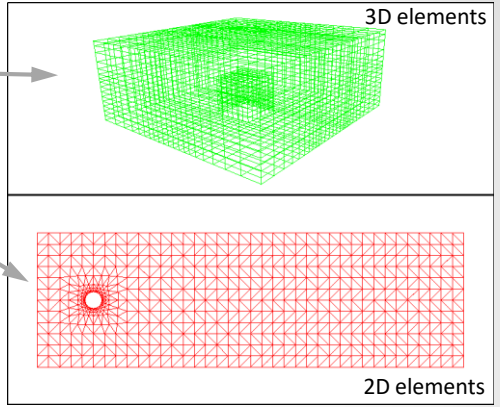
- Click the **Part Element Settings** icon to change the way a part is displayed; the following menu appears:



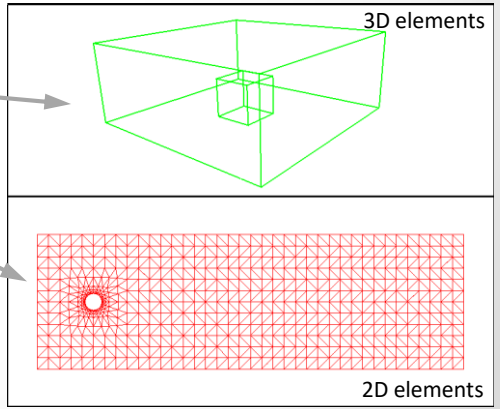
- The default setting is **3D border, 2D full**; this means a 3D element will only be displayed by its border lines (no internal element lines will be shown) and a 2D element will be fully displayed
- This does not modify a part's geometry, just how it is displayed on the screen

Element Representations 2

- **3D Border, 2D Full** (default)
  - 3D elements are displayed in border mode
  - 2D elements are displayed in full representation mode

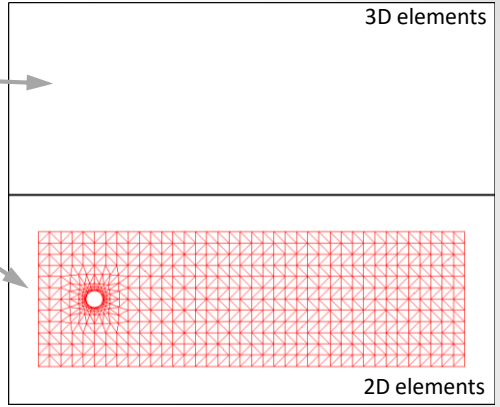


- **3D Feature, 2D Full**
  - 3D elements are displayed in feature mode
  - 2D elements are displayed in full representation mode

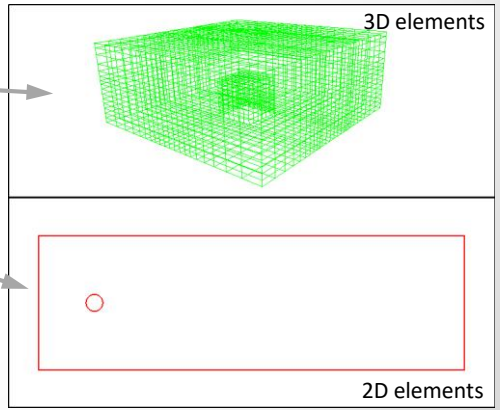


Element Representations 3

- **3D NonVisual, 2D**
  - Full 3D elements are not loaded
  - 2D elements are displayed in full representation mode

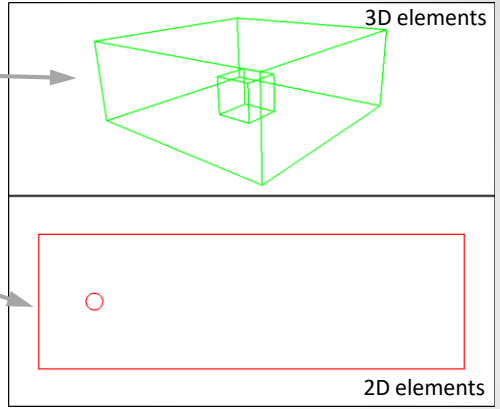


- **Border**
  - Shared faces (3D elements) are removed
  - Shared edges (2D elements) are removed

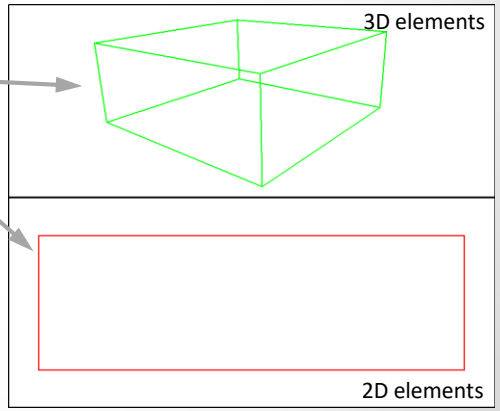


Element Representations 4

- **Feature Angle**
  - For 3D, operates off of Border elements
  - Looks at 2D element normals – the edge between 2D elements are removed if the angle is less than what the user has specified

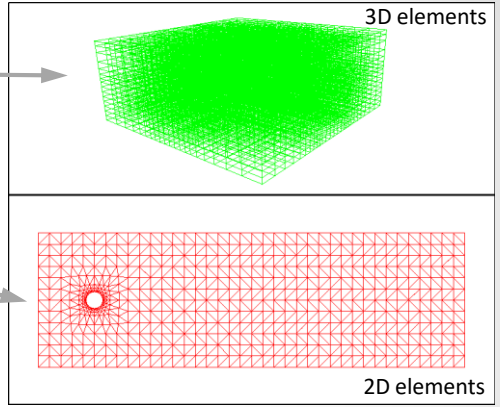


- **Bounding Box**
  - Displays a bounding box around the geometry

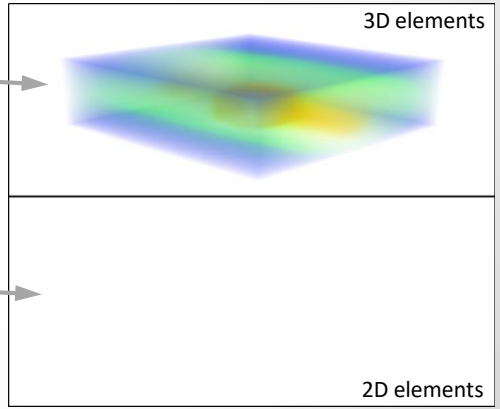


Element Representations 5

- **Full**
  - Every face (3D elements) is visible
  - Every edge (2D elements) is visible

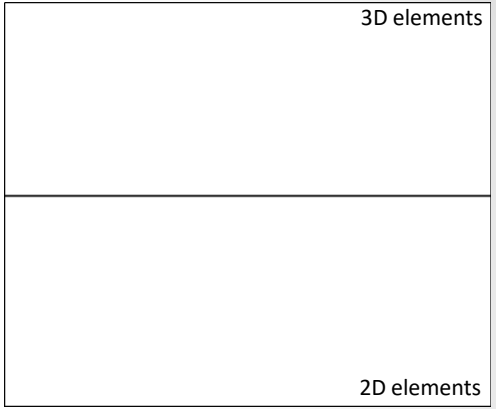


- **Volume**
  - 3D elements are rendered as volumes, including alpha values (transparencies)
  - 2D elements become nonvisual

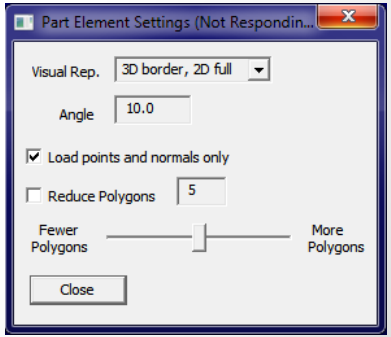


Element Representations 6

- **NonVisual**
  - The part is kept only on the EnSight server; it's geometry is not displayed on the EnSight client - useful for external flowfields

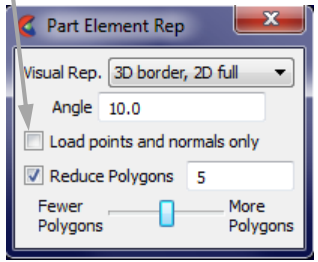


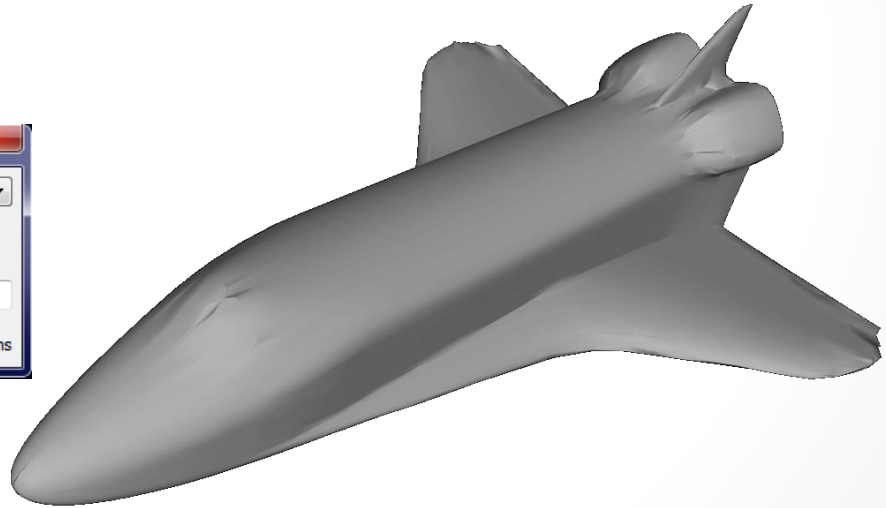
- Click the toggle **Load Points and Normals Only** to display only the element centers of the model



Element Representations 7

- **Reduce Polygons** speeds up visualization processing by thinning out the number of polygons that are rendered; the trade off is in image quality and speed; the model is not changed though, just the visual representation of it on the display
- Toggle **Reduce Polygons** to activate and use the slider for more or fewer polygons



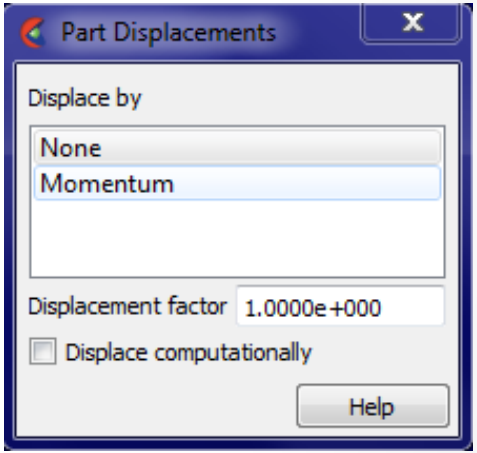


Part Displacement

- For structural mechanics simulations, EnSight can display and animate displacements to visualize the relative motion of the geometry

- Select the nodal variable to use

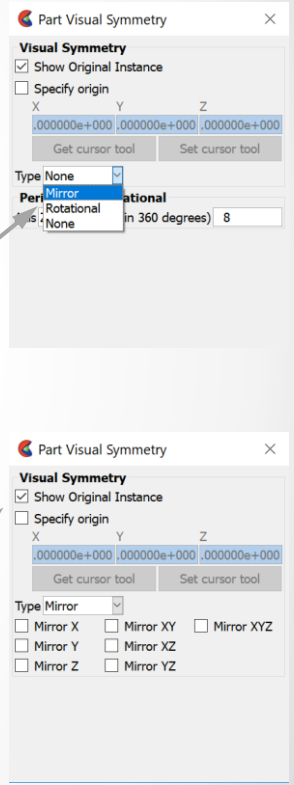
  Please note the following restriction:
  displacements can only use nodal vector variables



- The **Displacement Factor** is a scalar that scales the vectors and exaggerates the displacements; a value of 1.0 will give ‘true’ displacements

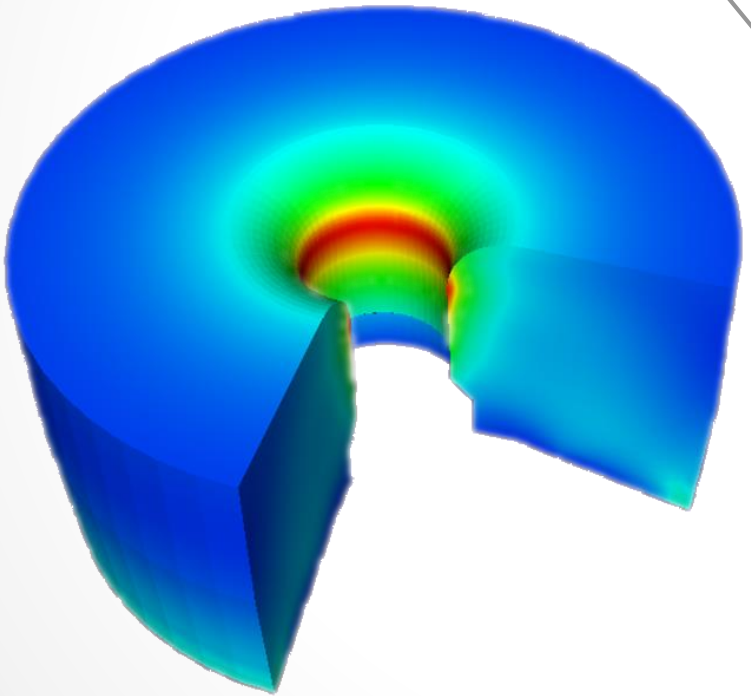
Part Visual Symmetry 1

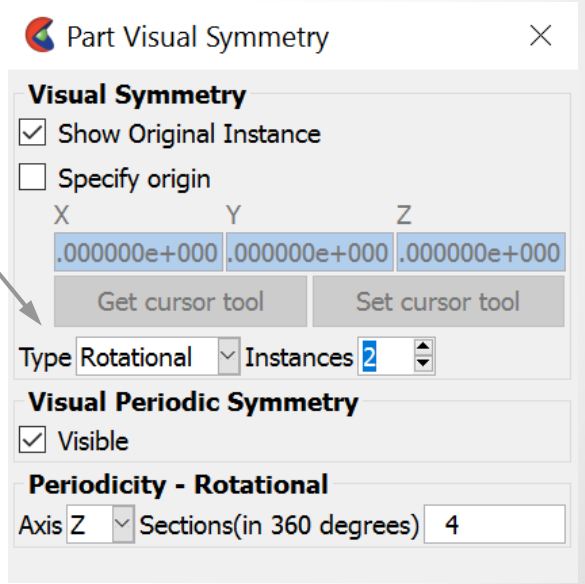
- Visual symmetry is done on the client so it's a copy of the original; the symmetry is performed with respect to the reference frame of the part (see the Frame chapter)
- This feature is useful if there is symmetry in a model and only part of the model has been analyzed
- There are 2 options: **Mirror** and **Rotational** symmetry
- Click on **Mirror** and select the mirror plane or axis
- There's a toggle to show or hide the original instance



Part Visual Symmetry 2

- For rotational symmetry, click on **Rotational**, select the **Axis**, the number of **Sections in 360 degrees** and the number of **Instances**





Exercise 1 – EnSight Processes & Element Rep

- See the EnSight 10 Advanced Training Exercises handout and do Exercise 1

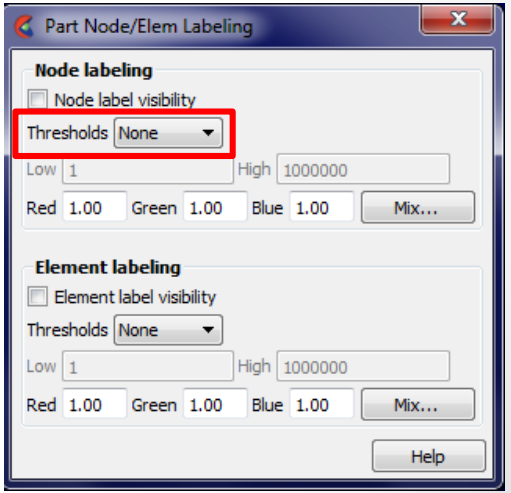
Element Labeling 1

Be careful with this feature because it can cause your system to slow down significantly if used improperly

- Sometimes it is useful to identify specific nodes or elements within a model; EnSight can display node and element labels in the Graphics Window

- Select the desired part and click the **Element Labeling** icon and the labeling menu is displayed; start by selecting a filter in the **Thresholds** box; this has 5 filters to display only selected ranges of labels:

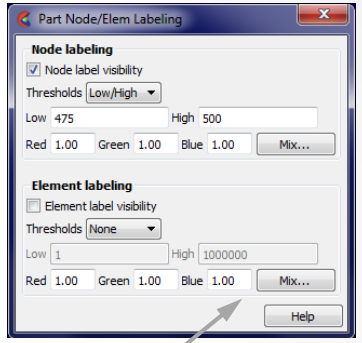
None - show all labels
Low - remove all labels < the **Low** value
Band - remove all labels >= **Low** and <= **High**

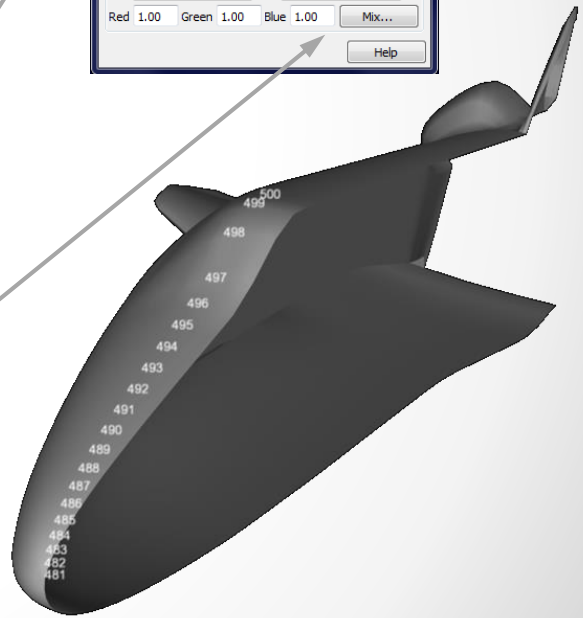


Element Labeling 2

High - remove all labels > the **High** value
Low/High - remove all labels < the **Low** value as well as those > the **High** value

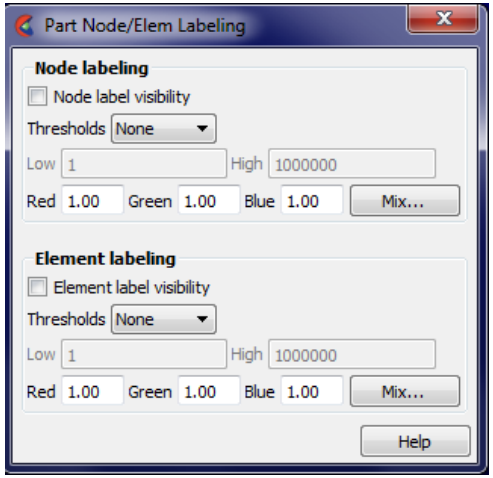
- Click on the **Node Label Visibility** toggle to display the labels
- The example shows the Space Shuttle model with a **Low/High** filter on the nodes of < 475 and > 500
- The **Mix** button controls the color of the labels





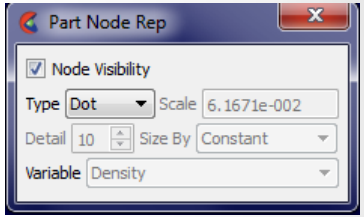
Element Labeling 3

- The menus for **Element Label Visibility** are identical to the **Node Label Visibility** menus

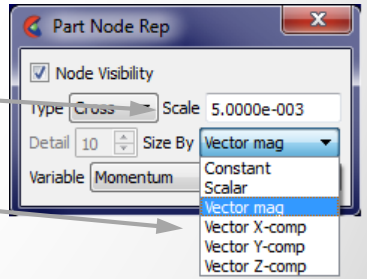


Node Representation

- EnSight can display the nodes of a model in the Graphics Window
- There are 3 display types: **Dot**, **Cross** and **Sphere**
  - **Dot**: nodes are displayed as points
  - **Cross**: nodes are displayed as crosses and can be fixed size or sized based on a variable
  - **Sphere**: nodes are displayed as spheres and can be fixed size or sized based on a variable



- **Scale** changes the display size of the nodes



- **Crosses** and **Spheres** can be scaled by a variable (scalar or vector)

Part Filter Elements 1

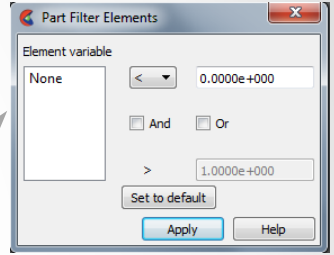
- In EnSight filters can be used to remove certain elements; the filter criteria can be one or two variables or it can be a variable (such as a Von Mises stress/strain) threshold for which limiting values and conditions are entered as a filter

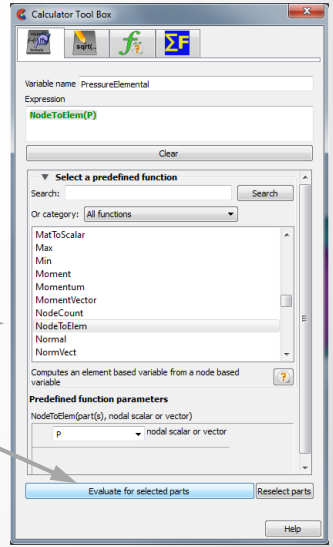
- Elements are not just visually removed of the client, but are also removed for all calculations of the server; if elements need to be removed for visual purposes only, consider **Element Blanking**

- The variable used must be an Elemental Variable, not a nodal one; use the **NodeToElem** function in the **Variable Calculator** to convert variables if needed

Part Filter Elements 2

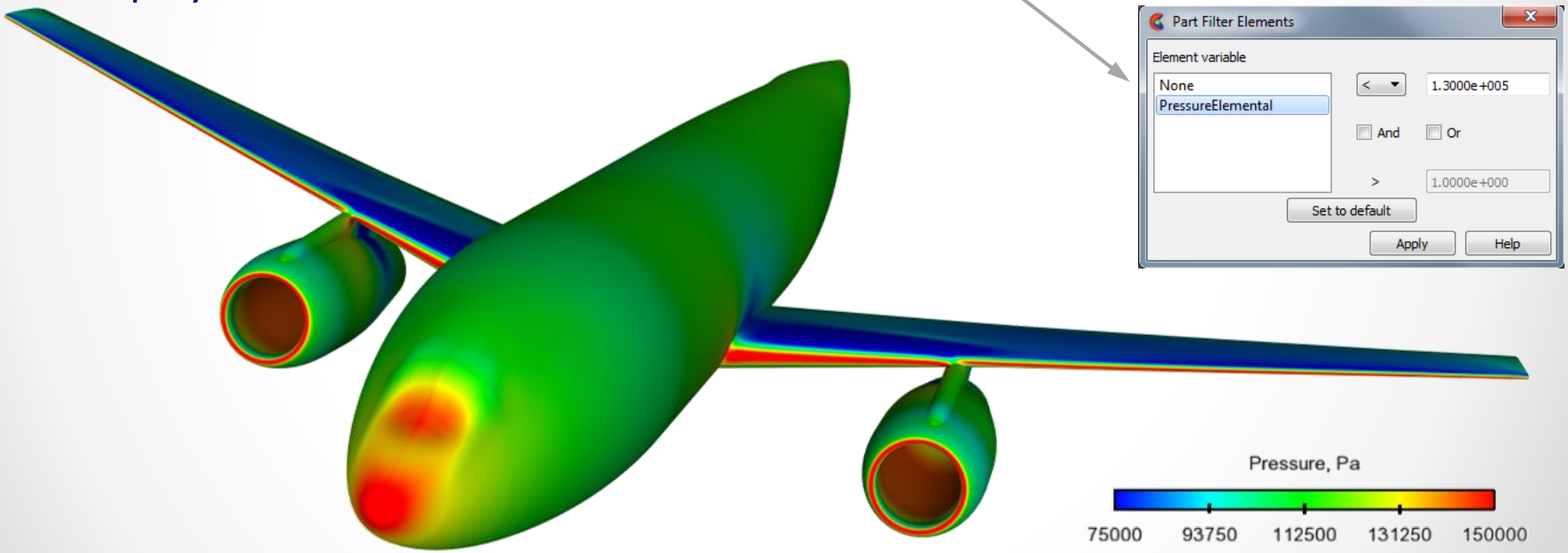
- To use **Part Filter Elements**, do the following:
  - Select the part(s) to use
  - Click the **Part Filter Elements** icon; the is displayed and the available Element Variable(s) are listed (they may have to be calculated from Nodal Values)
  - To calculate Elemental Variables from Nodal Values, open the **Variable Calculator** and select the **NodeToElem** function; then type in a new variable name and select the variable, in this case P; click the **Evaluate for Selected Parts** button to create the new variable





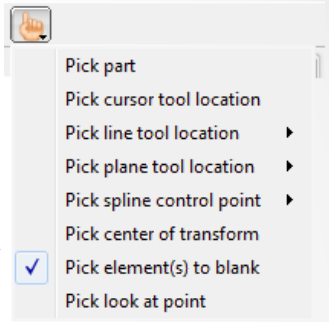
Part Filter Elements 3

- The Part Filter Element Settings menu now displays the new PressureElemental variable that was just calculated; type for example < 130,000 (Pa) as the filter criteria, click **Apply** and only elements with a pressure higher than that number are displayed

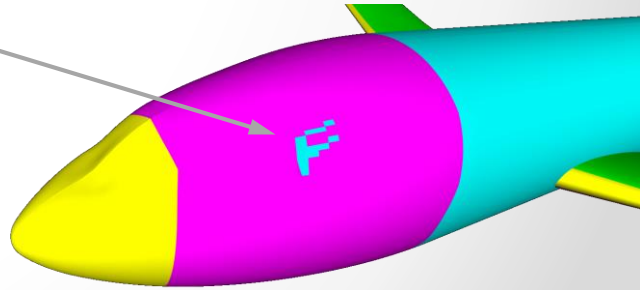


Element Blanking/Visibility 1

- Elements can be blanked; this can be useful to peek inside of certain parts or remove only portions of a part; element blanking is just a visual removal on the client

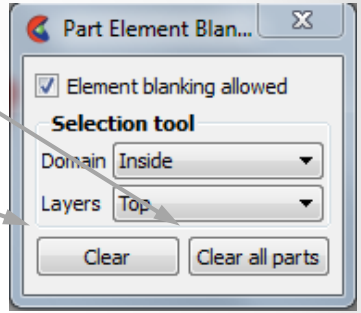


- Do the following to blank elements:
  - Set the pick action to **Pick Element(s) to Blank**
  - Select one or more parts in the part list that needs elements blanked
  - Place the mouse pointer over an element that needs to be blanked and press the **P**-key P on the keyboard or click the middle mouse button to blank the element



Element Blanking/Visibility 2

- To make the element(s) visible again, click the **Element Blanking/Visibility** icon and click the **Clear** button or the **Clear All Parts** button

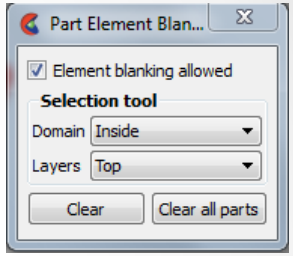


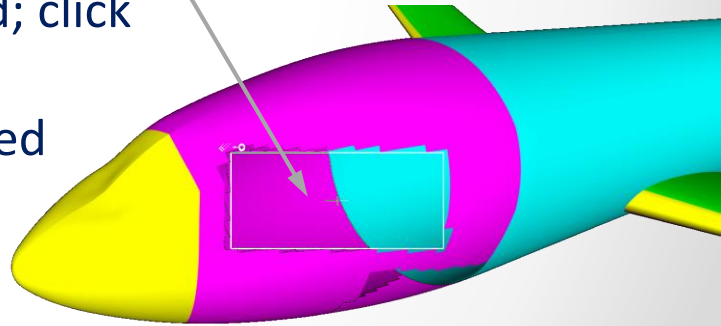
- The **Selection Tool** can also be used to blank elements on a larger scale

- Do the following to blank elements using the **Selection Tool**:
  - Select the part(s) on which to blank elements
  - Toggle the **Selection Tool** icon on
  - Position the tool and adjust its size if needed

Element Blanking/Visibility 3

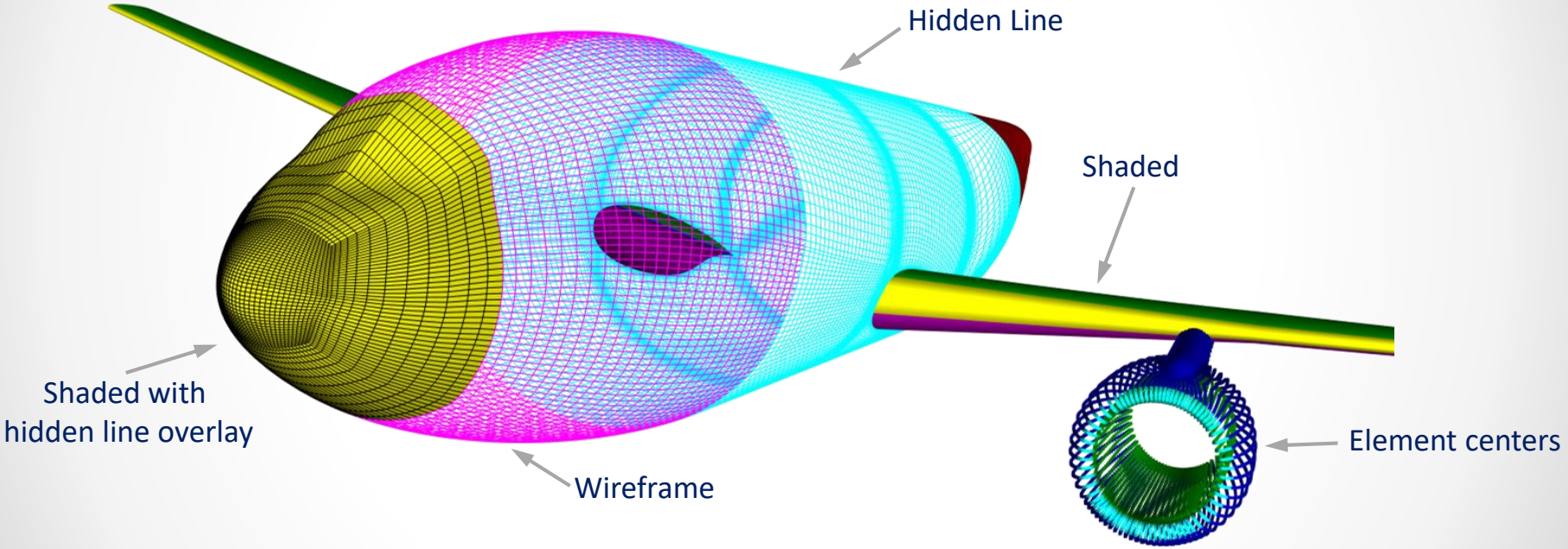
- Click the **Element Blanking/Visibility** icon and select the **Domain** (**Inside** or **Outside**) and which **Layers** to blank (**Top** or **All**); **Inside** blanks the elements inside of the **Selection Tool** - **Outside** does the opposite; **Top** only blanks the top visible layer while **All** blanks all layers with a single click
- Click the **Eraser Symbol** on the top left corner of the tool and the first layer of elements is removed; click it again and the next layer is removed
- Toggle the **Selection Tool** icon off when finished





Part Shading and Part Hidden Line

- **Part Shading** and **Hidden Line** display can both be switched on and off per part; this means that wireframe, shaded and hidden line views can be mixed in a model



- Remember to switch on the global **Display Shaded Surfaces** and **Overlay Hidden Lines** to see these features

Part Auxiliary Clipping

- **Auxiliary Clipping** can be set per part; by default it is set to on but it can be switched on or off per part
- In the example below, the body parts of the car are the only parts that have **Auxiliary Clipping** set to on; when the Plane Tool is moved, it seems like the outer skin of the car is peeled away
- Auxiliary Clipping can be toggled on and off by clicking on **View** -> **Auxiliary Clipping**

Part Fast Display

- **Fast Display** can be set per part; by default it is set to display a **Box**

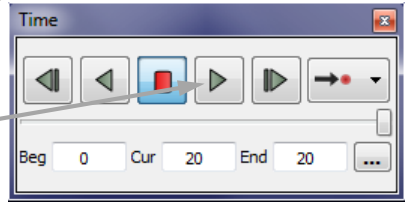
- In this example the main rotor and the cockpit glass are set to a **Box** display while the other parts are set to **Off**
- To make the lines more visible, the **Line Width** is set to **2**
- To activate global **Fast Display**, click on **View** -> **Fast Display**

Procter & Gamble Pantene Products

Chapter 3: Solution Time and Flipbook

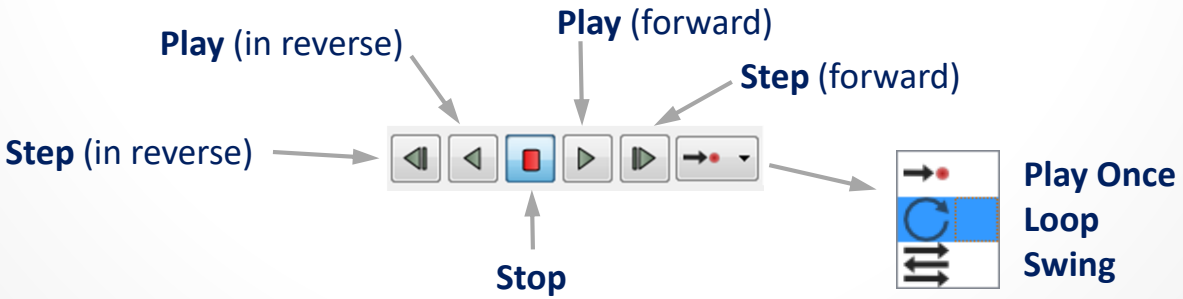
Time Menu 1

- EnSight can handle both steady state and transient data; when a transient model is loaded the Time menu will appear on the screen (by default above the Parts List)



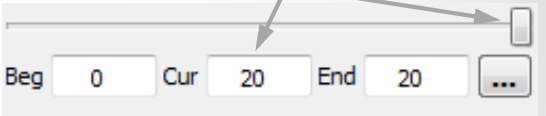
- Click the Play button to run the model through the time steps

- The DVR like buttons have the following functions:



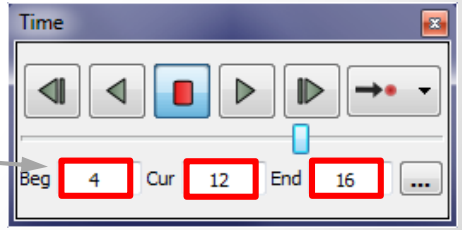
Time Menu 2

- When a transient data set is loaded, by default the last time step is displayed; this can be seen by the slider that's at the end of the time range or by the **Cur**(rent) field that has the last time step in it (20 in this example)



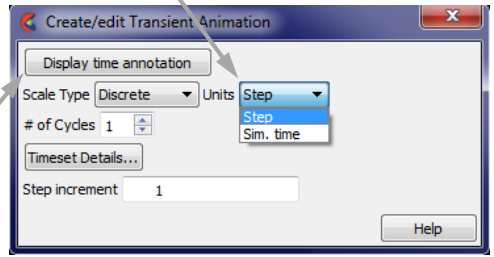
  Note: The user can select whether the last or first step is displayed by default by clicking on **Edit** -> **Preferences** -> **Data** -> **If Starting Time Step is Not Specified, Load:** select **First Step** for instance

- The slider can be dragged to any specific time step; in this example the slider has been dragged to time step 12, the **Beg**(inning) has been changed to step 4 and the **End** has been modified to 16 by just typing in these values



Create/Edit Transient Animation menu 1 + ...

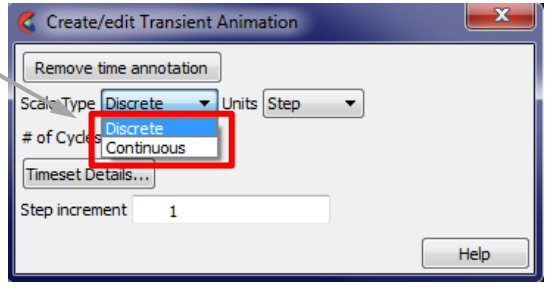
- Click the ... icon for some advanced features; the time **Units** can be set as either discrete **Step**(s), running from zero to the total number of steps minus one or the actual simulation time values found in the results data set



- Click the **Display Time Annotation** to display the time in the Graphics Window; this is a toggle and the time display can be moved and resized by dragging the touch-n-go handles
- The time annotation can of course also be displayed by dragging and dropping the Time variable in the Variable List onto the background of the graphics window

Create/Edit Transient Animation menu 2 + ...

- The time **Scale Type** can either be **Discrete** or **Continuous**:

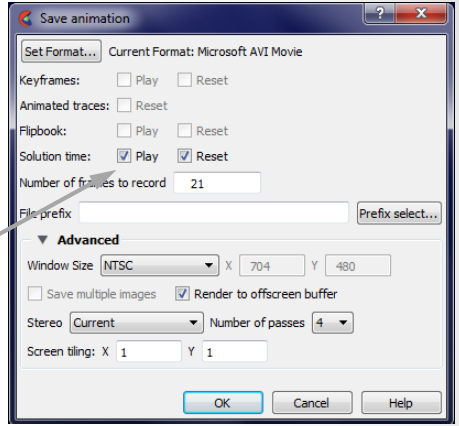


  - If the **Scale Type** is set to **Discrete,** only actual time steps that are listed in the results data set can be displayed
  - If the **Scale Type** is **Continuous**, results can be displayed between actual output time steps; all variable values are linearly interpolated between the two time steps

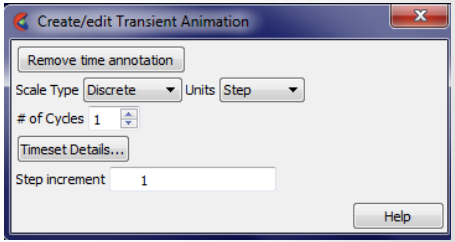
    Note: if the mesh is changing over time, results can't be displayed continuously
- The **Timeset Details** button controls how multiple timelines of several datasets can be synchronized and played simultaneously

Record Current Graphics Window Animation

- Record a transient animation for instance by clicking on the **Record Current Graphics Window Animation** icon on the Quick Settings menu and select the options on the Save Animation menu such as movie format, window size, number of passes etc (some of these options are under the **Advanced** tab); select the **Play** and **Reset** toggles



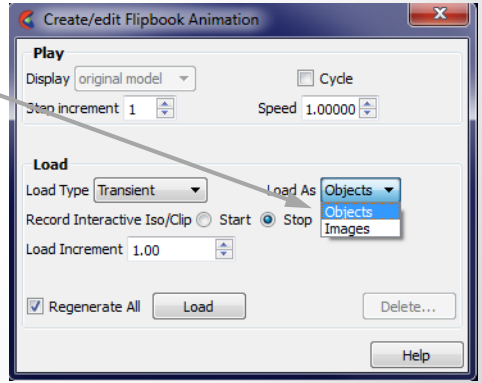
- The **# of Cycles** field determines the number of times the solution time is played when an animation is recorded



Flipbook 1

- A flipbook is a quick and easy way to make an animation by playing back a series of images; for each step, a graphical ‘page’ is created and stored in RAM memory; when the flipbook is played, the pages are sequentially displayed as fast as the graphics card allows; click the flipbook icon

- A flipbook can be created as **Objects** or **Images**; if the flipbook is created using **Objects**, during playback of the animation the model can still be rotated or zoomed but for bigger models the RAM on the client that is needed can be substantial; in that case **Images** (pixel data only) is a better choice due to lower RAM requirements but the animation can not be rotated or zoomed during playback

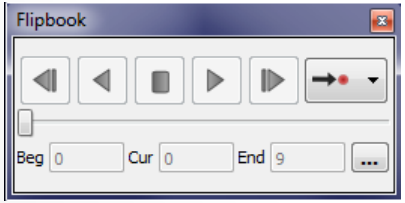


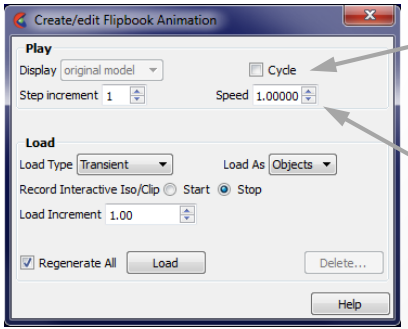
Flipbook 2

- Most EnSight users use a flipbook to animate and record transient data but a flipbook can animate 4 types of data:

Transient	Pages are constructed by stepping from the current beginning to ending time range and rebuilding all time-dependent parts based on each time step in sequence.
Mode Shapes	Pages are constructed by applying a cosine-driven scaling factor to a displacement variable.
Create Data	Pages are constructed by applying a delta to either a clip part or an isosurface.
Linear Load	Pages are constructed by applying linear interpolation ranging from zero to the maximum (displacement) vector field value.

- Once a flipbook is loaded, the **Flipbook** menu uses DVR like buttons just like the **Solution Time** panel to play an animation



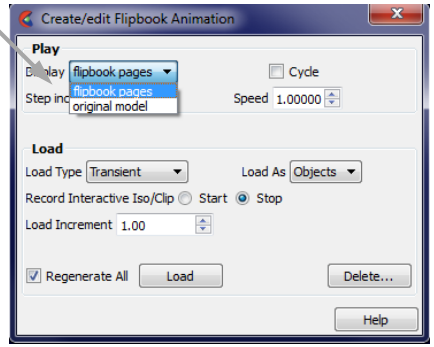


Cycle can play the flipbook forward and in reverse

Display Speed can slow down the animation

Flipbook 3

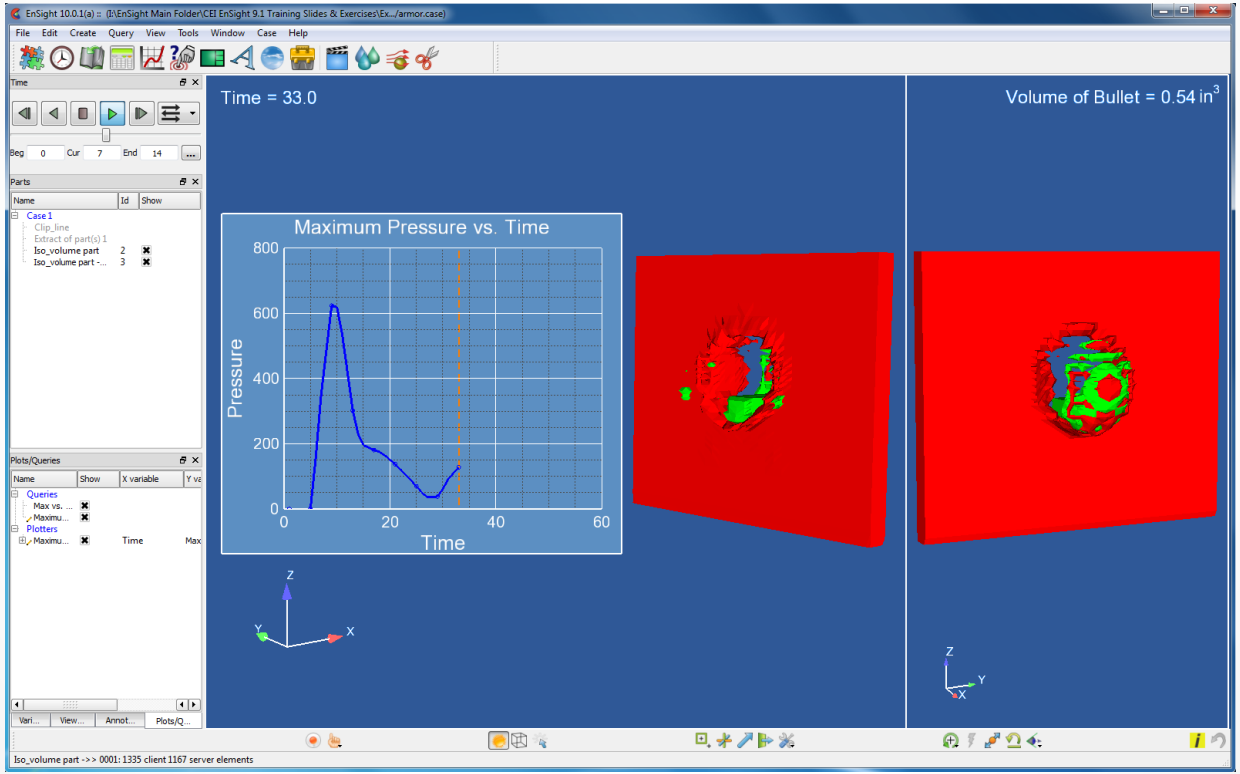
- If the flipbook pages were created using **Images**, don't forget to set the **Display** back to **Original Model** otherwise rotate, pan and zoom will not work (because the display is still set to **Flipbook Pages**)



- When a flipbook has been created it can be recorded to a movie file using the **Record Current Graphics Window Animation** icon on the Quick Settings menu
- For more complex and sophisticated animations, use the Keyframe Animator and not a flipbook

Solution Time and Flipbook Exercise

- See the EnSight 10 Advanced Training Exercises handout and do Exercise 2



Chapter 4: Pathlines (Transient Streamlines)

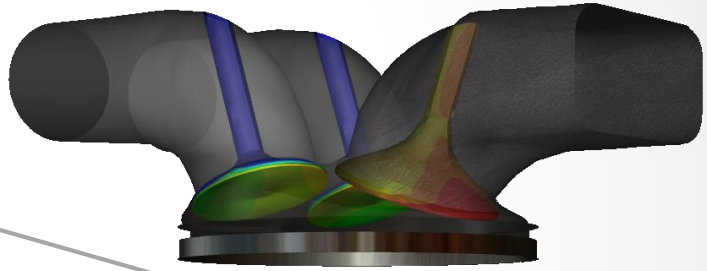
Pathlines Introduction

- The path calculated using a transient flowfield that is updated as the calculation proceeds is known as a pathline
- By default, pathlines will start at the first time step of the simulation and terminate at the last step (unless stopped earlier)
- A delta value can be specified for an emitter that will cause additional particles to be emitted into the flow at regular intervals (Multiple Pulses); this type of pathline is also called a streakline or smoke trace
- Pathlines can be animated just like streamlines with the same options on the **Particle Trace** -> **Animate** menu

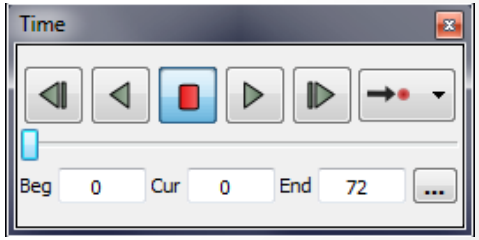
Creating Pathlines 1

- A pathline is created with the same icon as a streamline, so click the **Particle Trace** icon and in the **Type** selection box select **Pathline**

- Select the flowfield and click the **Create** button and the pathlines part will be created; however most likely no pathlines will be displayed on the screen

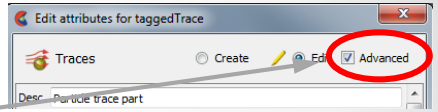


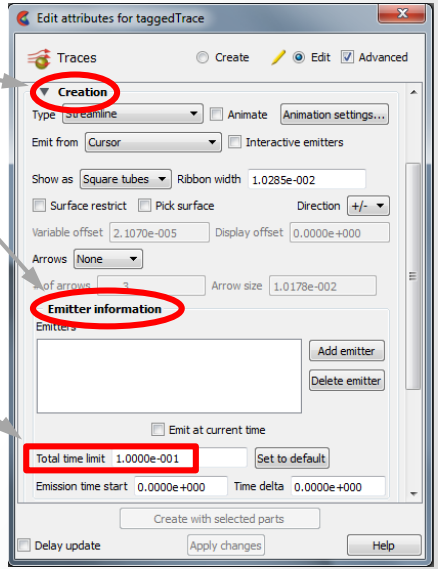
- This is caused by the settings of 2 menus:
  1. The **Solution Time** menu should be set to the first time step (usually **Start** = **0**)



Creating Pathlines 2

2. Double click the Particle Trace Part in the Parts List; this brings up the Edit Attributes menu for that part; click the **Advanced** toggle on; this opens up many more options under the **Creation** label

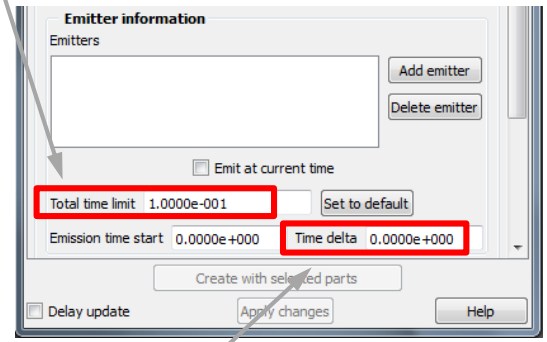




The **Emitter Information** fields must have values that are appropriate for the data set; these values can be found in the results data set: First the **Total Time Limit**, (the end time minus the start time) - for this model it is .1

Creating Pathlines 3

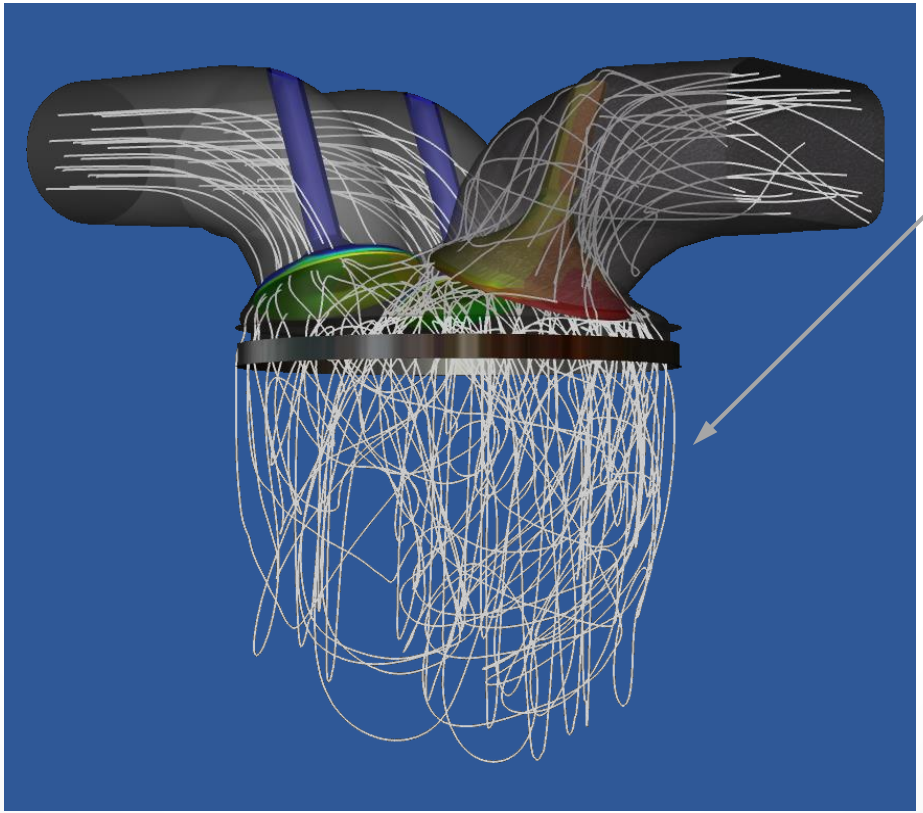
Then the **Emission Time Start** (the time when the analysis starts); for many models this is at time = 0)



3. The **Time Delta** is set by default to 0 and this value doesn't need to be changed unless more particle traces are needed; if a value is typed in, a new set of traces will be emitted at S, S+D, S+2D etc into the flow field (S is the start time and D is the delta value)

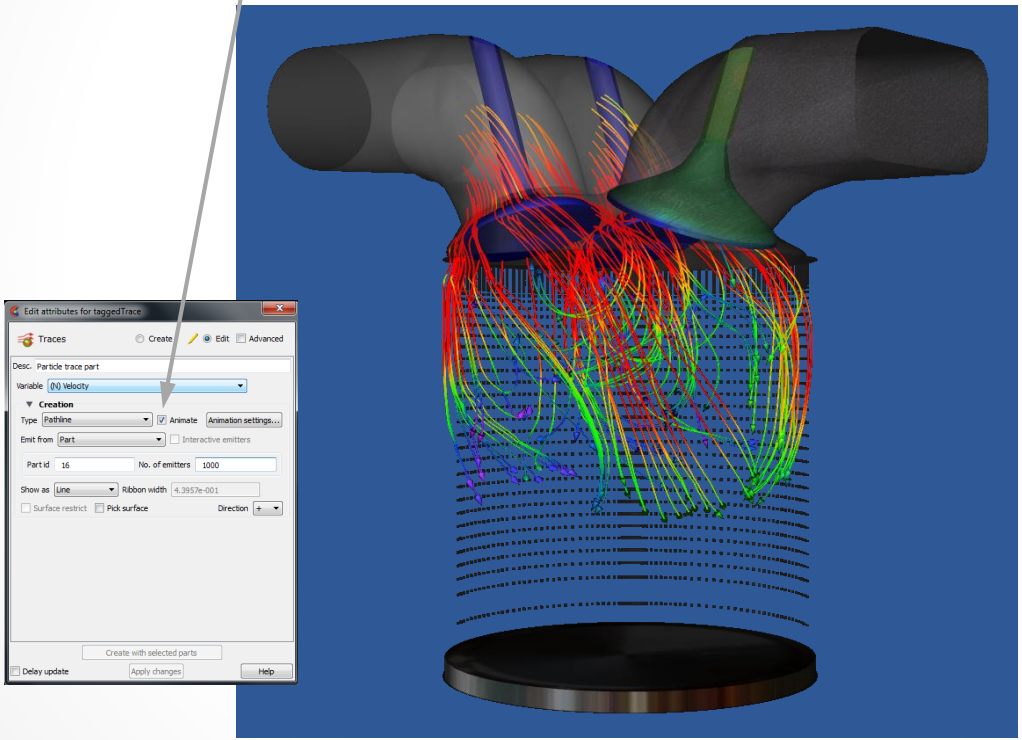
Displaying Pathlines

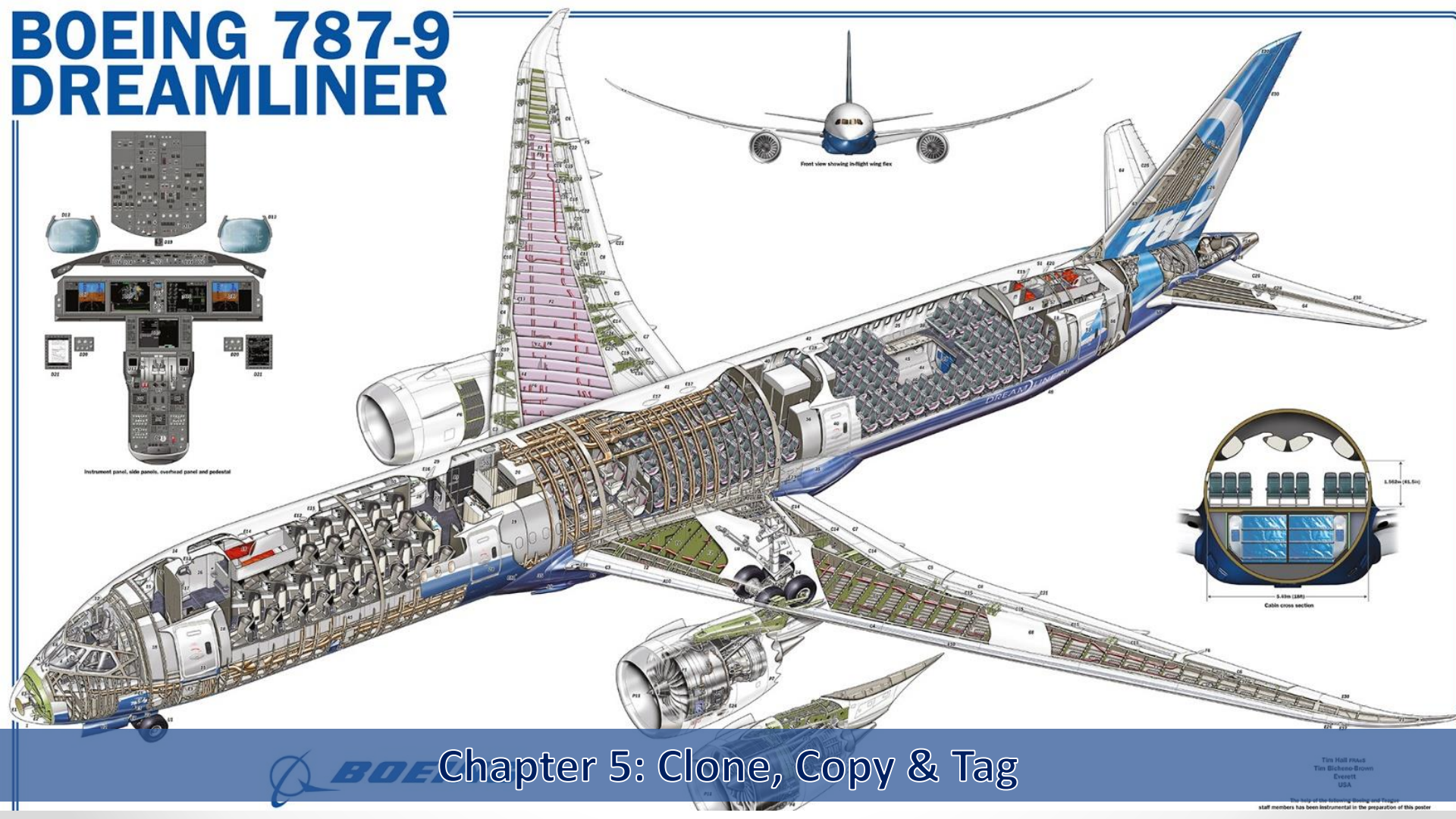
- After the adjustments using the **Emitter** settings, the pathlines are displayed



Animating Pathline Particle Traces

- Pathline particle traces are displayed by double clicking the Particle Trace Part in the Parts List and selecting the **Animate** toggle




BOEING 787-9
DREAMLINER
Front view showing inflight wing flex
Instrument panel, side panels, overhead panel and pedestal
Cabin cross section
Chapter 5: Clone, Copy & Tag

Part Duplicating

- There are two main methods to duplicate parts in EnSight*
  - (1) Part Copy
    - Creates a ‘linked’ copy of the parent part
      - Users may change color of the copy, display attributes
      - Updates to parent are reflected in the copy
        - ❑ Move the parent, and the copy moves as well
  - (2) Part Clone
    - Creates a ‘separate’ clone of the parent
      - Initially the clone will have all of the same attributes (color, position, display, etc)
      - All attributes of the clone are independent from the parent
      - The Clone is a completely separate from the parent

*(You can re-load model parts from dataset to duplicate model parts only)

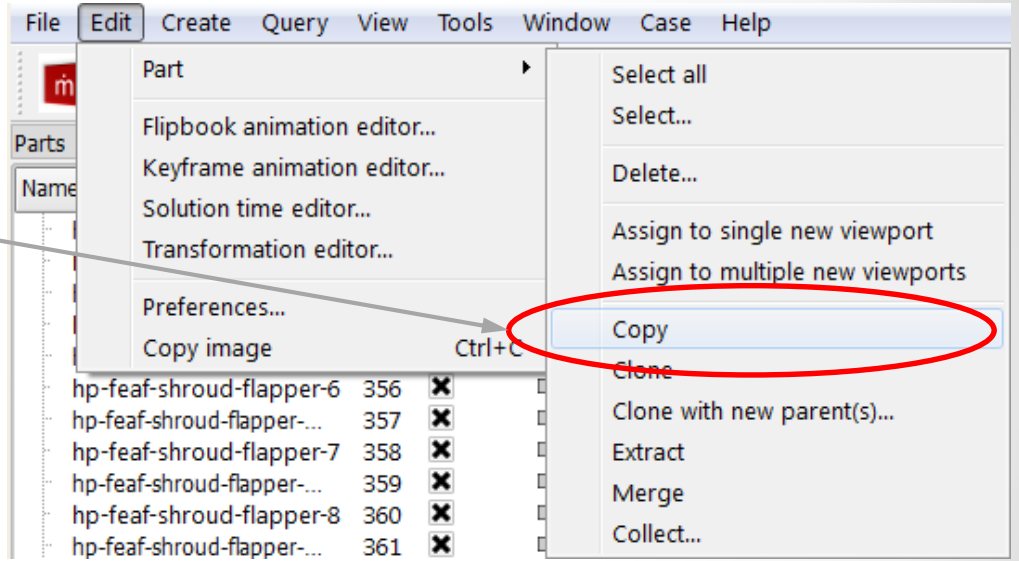
When to use each one

- (1) Part Copy
  - When you want to color the SAME part by a different variable
  - The Location of the Parent and Copy remain the same
  - It is really a ‘linked’ copy
  - Not a independent part

- (2) Part Clone
  - You have created/setup a part (location, color, style), but now want another one just like it (to start with)
  - Initially it has all the same attributes, but you want the ability to change anything
  - Completely Independent

How to Create a Part Copy

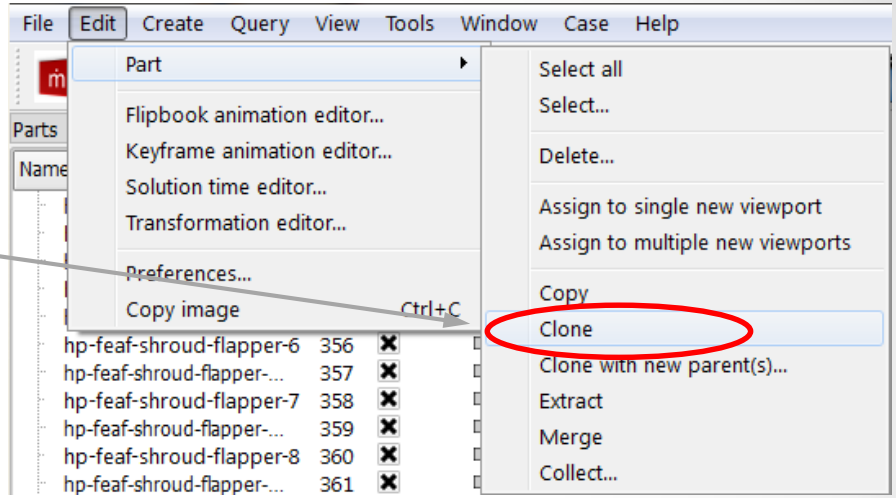
- (1) Part Copy
  - Select the part(s)
  - Top Pull Down Menu
    - **Edit** -> **Part** -> **Copy**



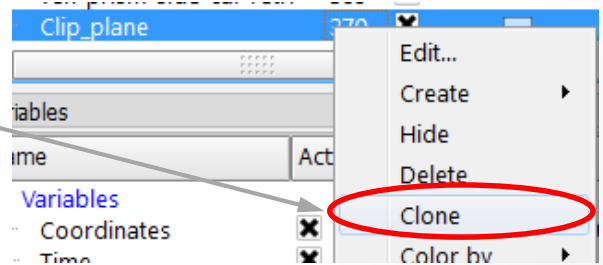
- The new Part can have only its visual properties changed (color, reflection, viewport assignment)

How to Create a Part Clone

- (2) Part Clone
  - Select the part(s)
  - Top Pull Down Menu
    - **Edit -> Part -> Clone**

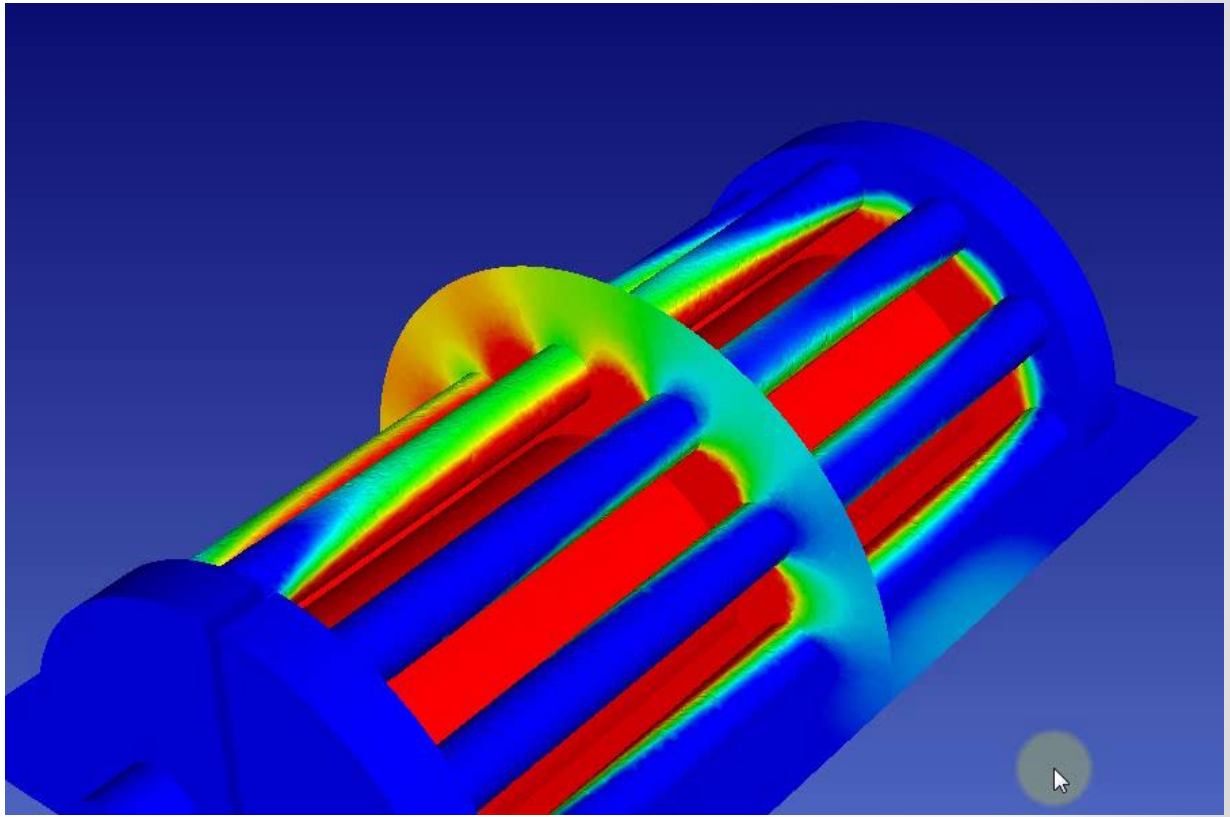


- Alternatively, you can use the Right Click:
  - Select the part(s)
  - Right Click -> **Clone**



Creating and Using Part Clone

- Example of quickly creating similar part via ‘Clone’ Method



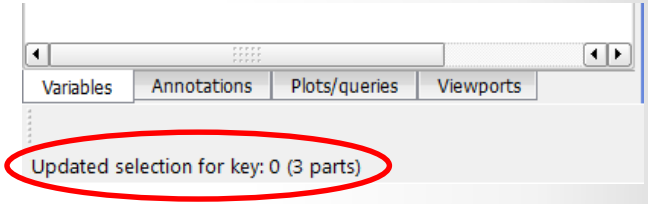
Part Tagging

- Part Selection can be tagged (new for EnSight 10.1)
  - Ability to Save and Recall Part Selection - up to 10 saved/recalled selections
- Part Selection Tagging is handled via:
  - Saved using Alt + [0-9]
  - Recalled using Ctrl + [0-9]
- ‘Groups’ are not created - simply an ability to save & recall part selection
- Meant for quick save/recall of complex part selections
- Parts can exist in multiple selections (Part #4 can be in both Alt-0 and Alt-4)

Part Tagging – How To

- Select the Part(s) you wish to save the selection
  - Press Alt + [0-9] to save the selection to one of 10 selections
  - Press Ctrl + [0-9] to recall that selection

- Each Selection Creation is noted in the output area



- Part Tagging selections are saved and restorable from Context, Session, and Archive files

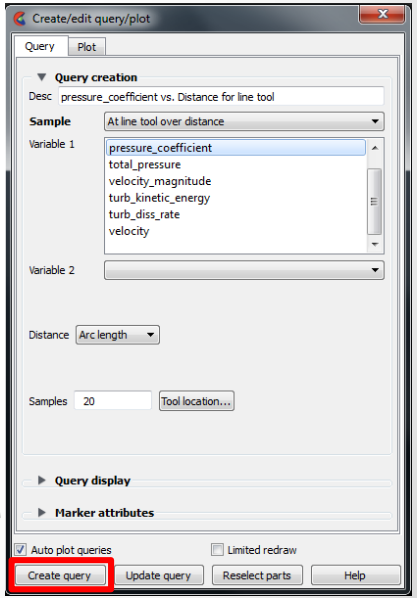
- Part Tagging is not saved or restored in Journal files (no command equivalent)

Mercury Marine Verado
350 SCi outboard engine

Chapter 6: Query and Plot

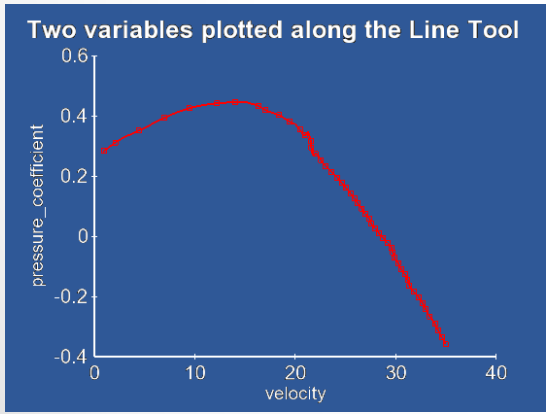
Query/Plot 1

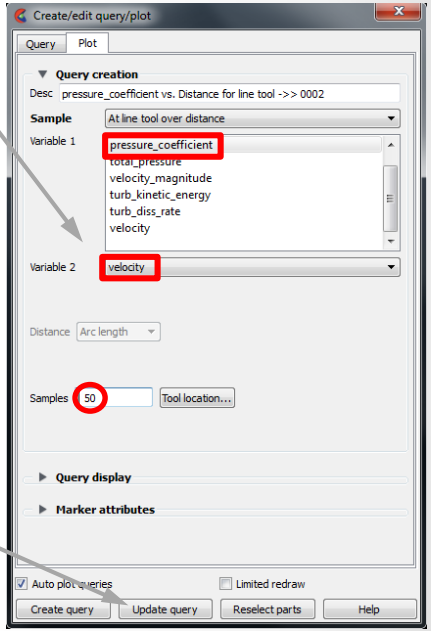
- Queries and plots can be generated by right clicking the Line Tool or right clicking a clip part and selecting **Query (Variable)** as described in the EnSight Basic Training
- A query and plot can also be created by clicking the Query icon on the Feature Toolbar; the Create/Edit Query/Plot menu is displayed
- On this menu select the **Sample**, in this example the Line Tool, **Variable 1**, for instance the `pressure_coefficient` then select the appropriate part(s) and click on the **Create Query** button
- Since the **Auto Plot Queries** is toggled on, the query will be immediately plotted as well



Query/Plot 2

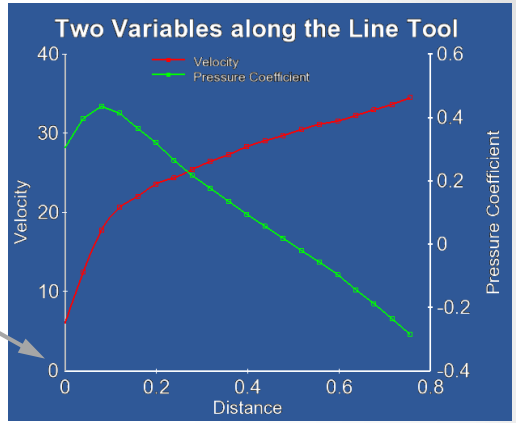
- If **Variable 2** is clicked and a second variable is selected, the result is a scatter query and plot of two different variables along the line tool
- The number of **Samples** controls the number of points along the sample tool (the Line Tool in this example)
- Click the **Update Query** button if the number of **Samples** is changed for an existing plot



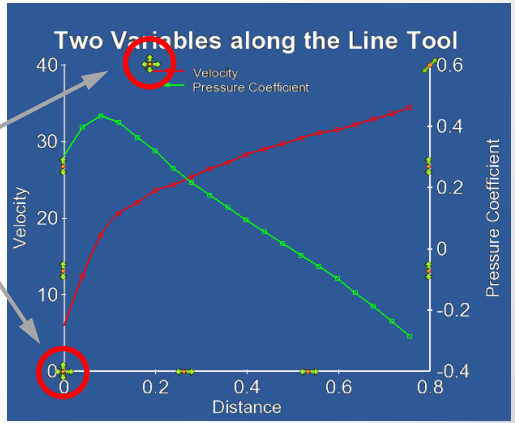


Query/Plot 3

- Of course it's also possible to create a plot with the distance along the Line Tool on the X-axis and the 2 variables on the Y and Y2 (right) axes; to do this, create the first query and plot, then create the second query and drag-and-drop it onto the origin or one of the axes of the existing plot

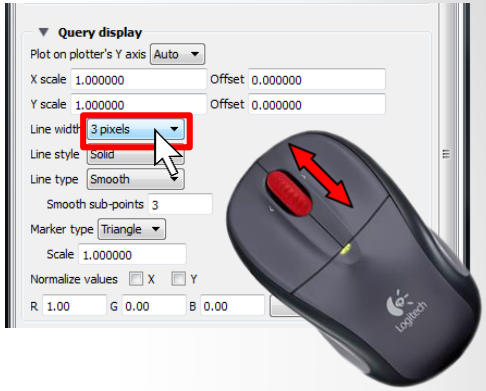


- Move the Query Description by double clicking the origin of the plot; new handles will appear to move the description

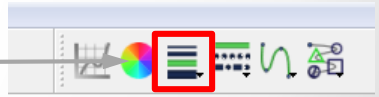


Query/Plot 4

- Toggle the **Query Display** to change the **Line Width** of the graph; as mentioned in the EnSight Basic Training, just put the cursor over the drop down list and roll the mouse wheel; this will change the **Line Width** without actually having to go into the drop down list

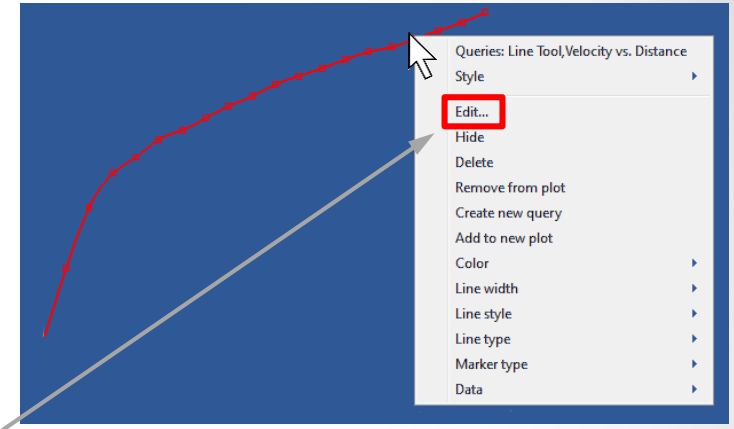


- Please note that the Curve Line Width icon on the Quick Edit Menu is dynamically updated as the mouse wheel is rolled

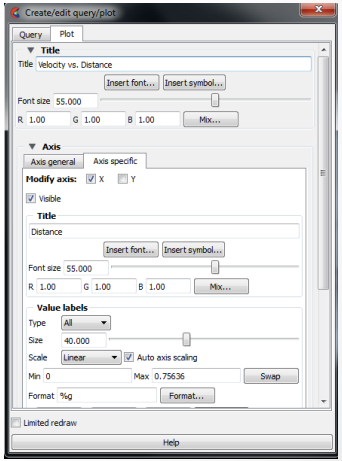


Query/Plot 5

- Right click on a curve to get instant access to many parameters of the curve

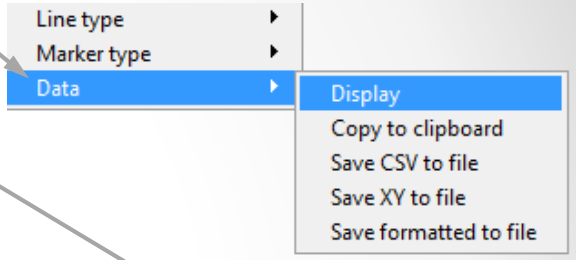


- Select **Edit** to display the Create/Edit Query/Plot menu which enables complete control over both the query and the plot

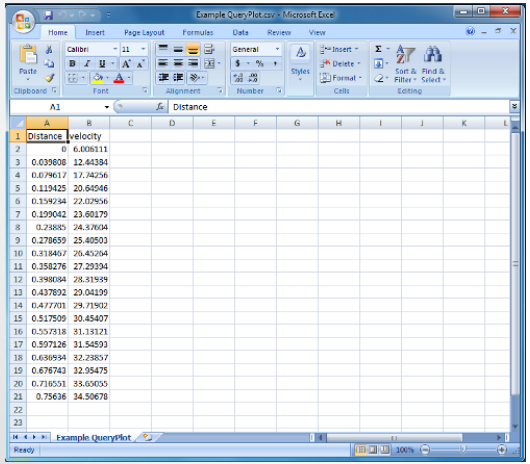


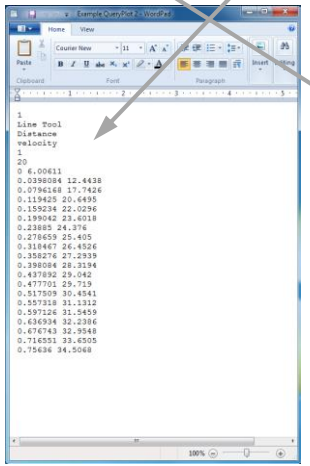
Query/Plot 6

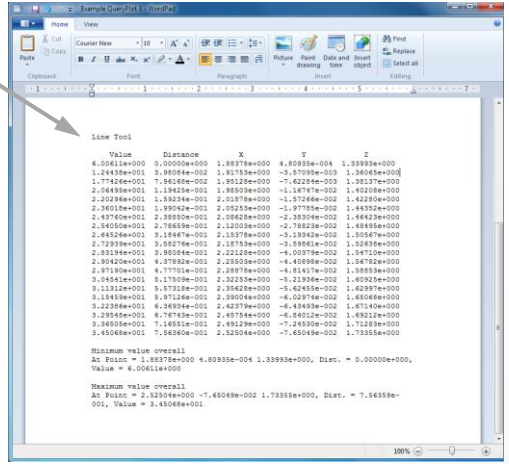
- The last option on this menu is **Data** and it provides a way to **Display** the data for the current Query/Plot

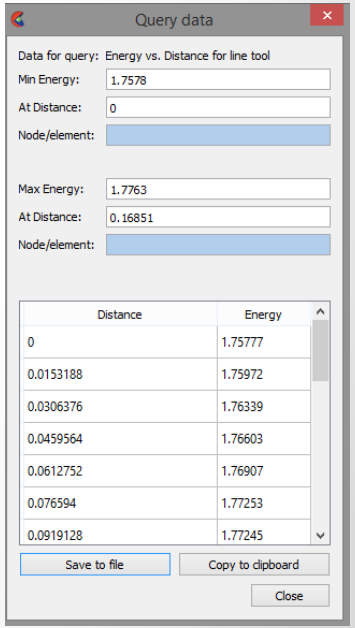


- The information can also be saved to the clipboard, a **CSV** file which can be read by Excel, an **XY** plain text file or a **formatted** plain text file



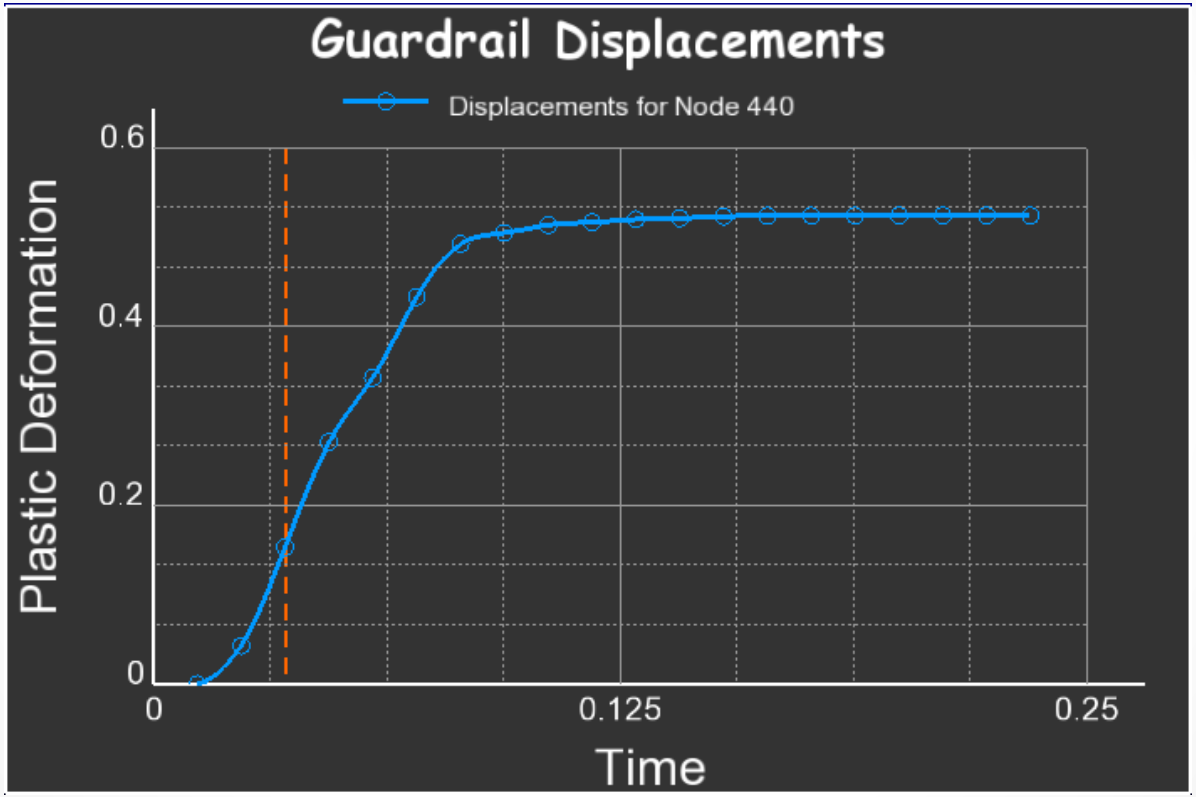






Query/Plot 7

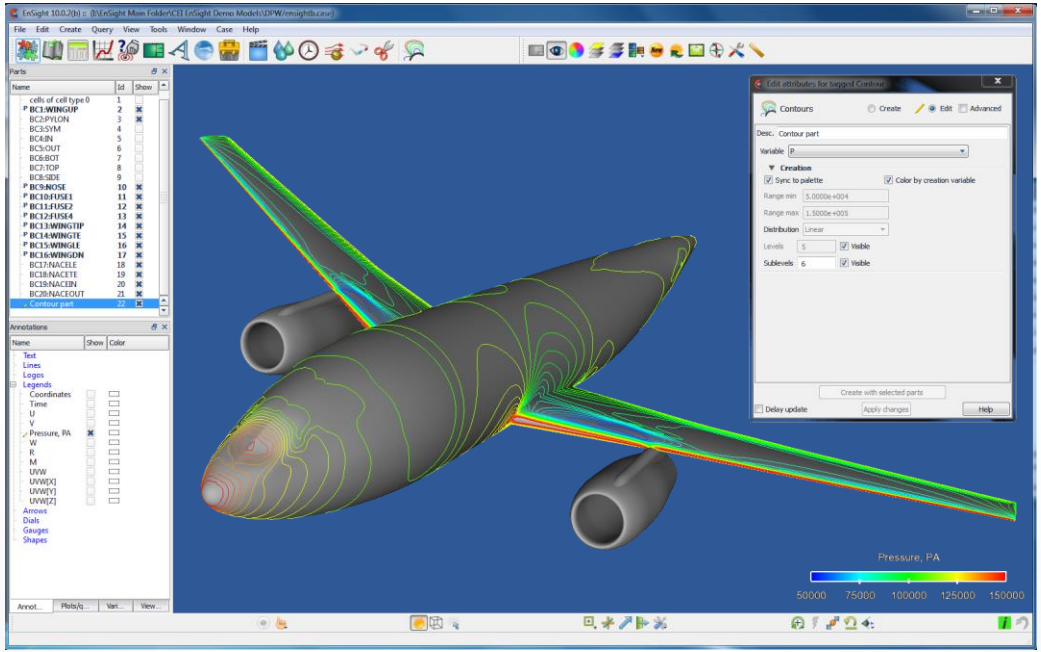
- This is an example of a finished graph



Chapter 7: More Part Creation

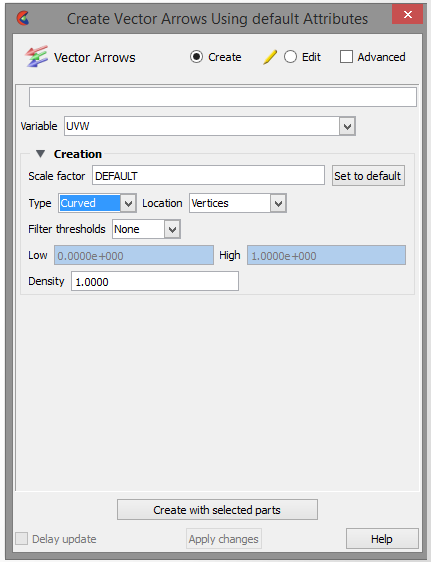
Contours

- A contour is a line of constant value on a 2D surface
- Select the parts the contours should be displayed on
- Click the Contours icon , select a variable and click the **Create with Selected Parts** button

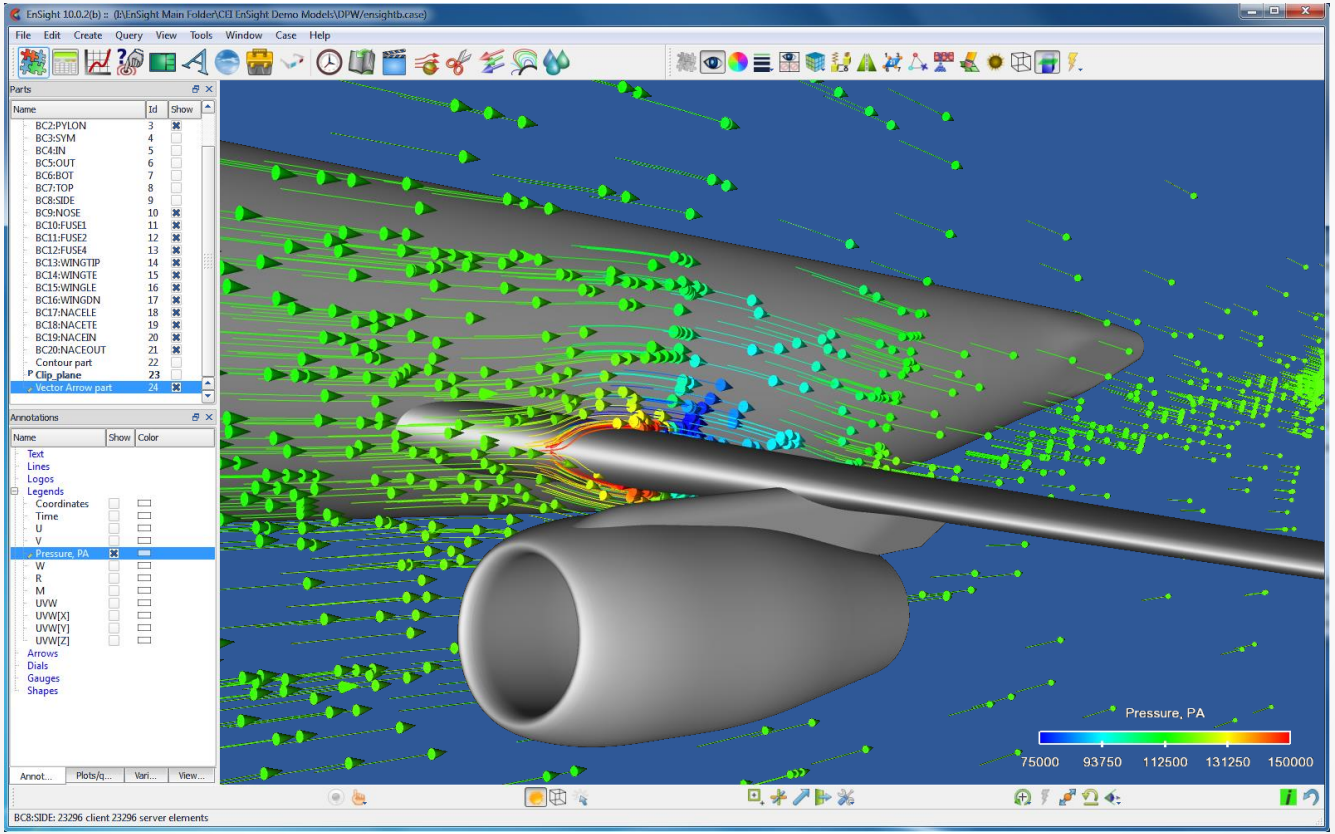


Vector Arrows 1

- Represents the direction and magnitude of a vector field
- Create a Clip plane in a fluid domain for instance and select the Clip Part
- Click the Vector Arrows icon, select a variable and click the **Create with Selected Parts** button
- The Scale Factor controls the size of the vector arrows
- Vector arrows can be **Straight** or **Curved**
- The **Density** field controls the number of arrows

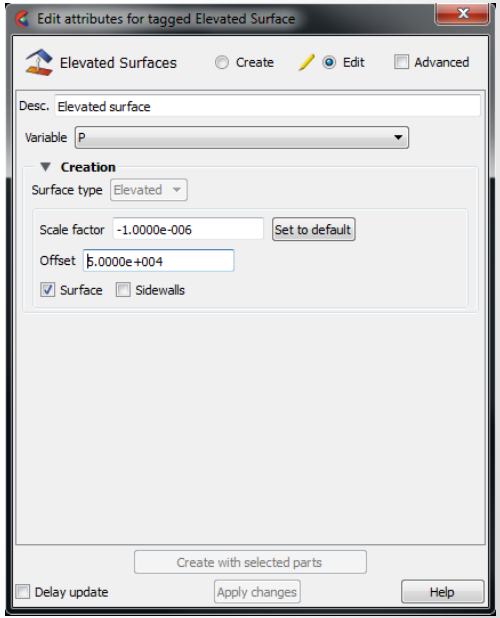


Vector Arrows 2

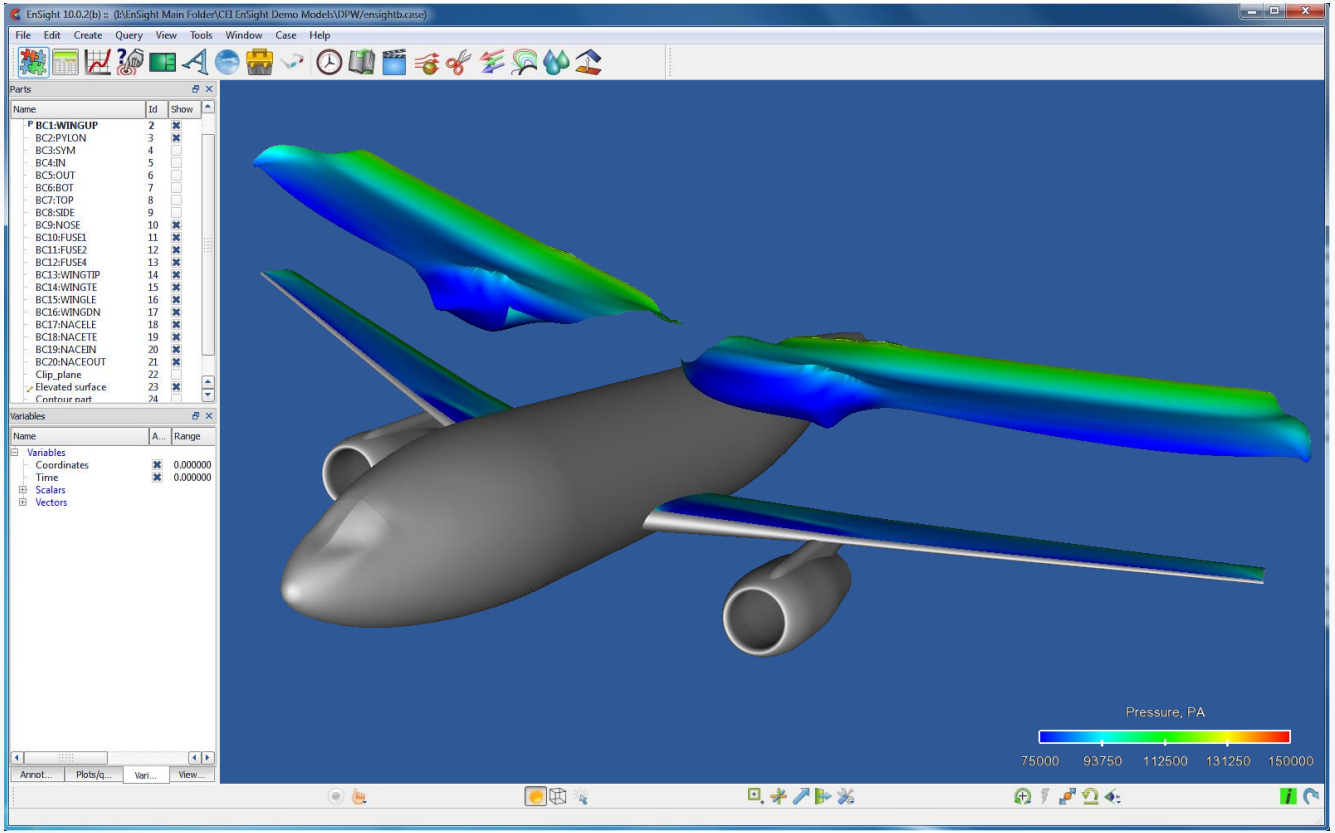


Elevated Surfaces 1

- Represents an offset surface of a scalar variable
- Select a part then click the Elevated Surfaces icon, select a variable and click the **Create with Selected Parts** button
- The **Surface Type** can be **Elevated** or **Offset**
- The **Scale Factor** controls the size of the elevated surface
- The **Offset** field allows to shift the elevated surface away from the parent, but does not affect the shape
- An Elevated Surface can have **Sidewalls**



Elevated Surfaces 1



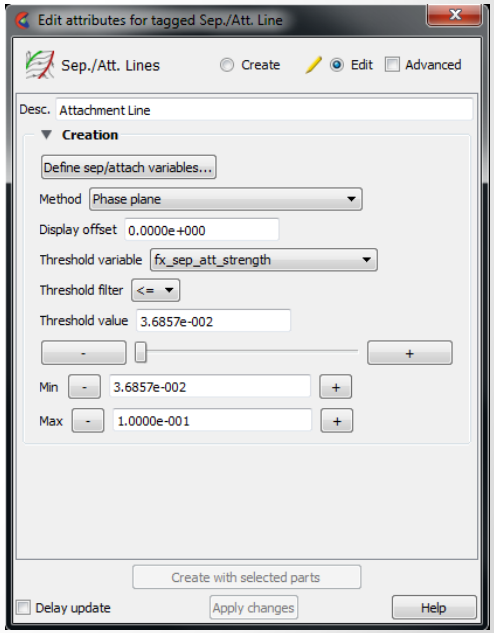
Separation & Attachment Lines 1

- Separation and attachment lines are created on any 2D surface and show interfaces where flow abruptly leaves (separates) or returns (attaches) to the surface

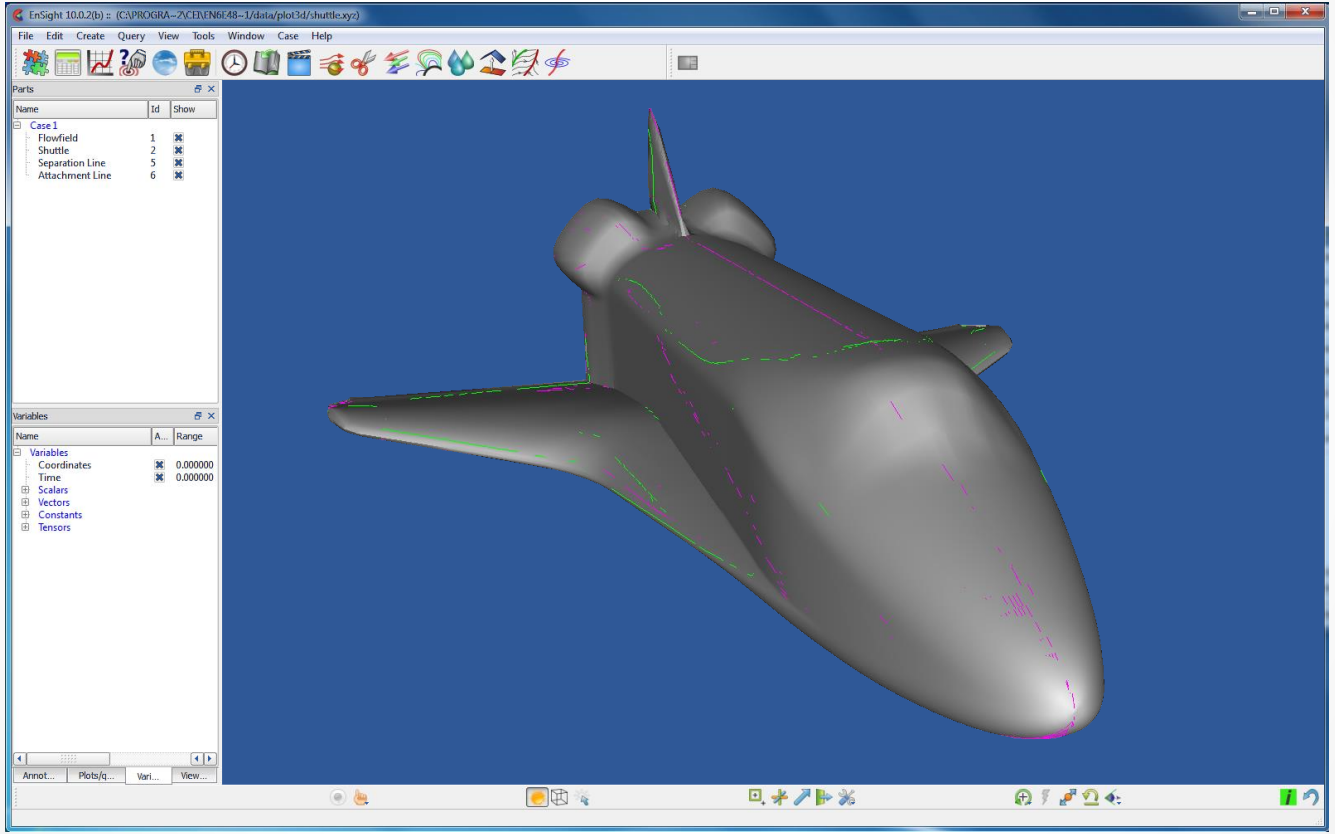
- Select a part then click the Separation & Attachment Lines icon, select a variable and click the **Create with Selected Parts** button

- The separation lines are green and the attachment lines are pink

- There are various menus to define the **Method** that is used and tweak the separation & attachment lines

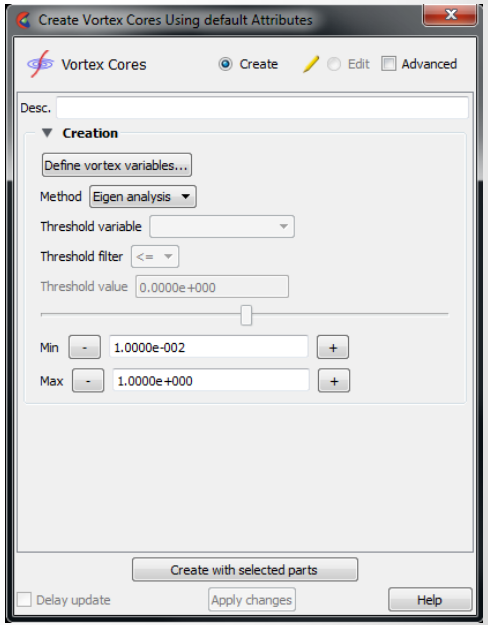


Separation & Attachment Lines 2

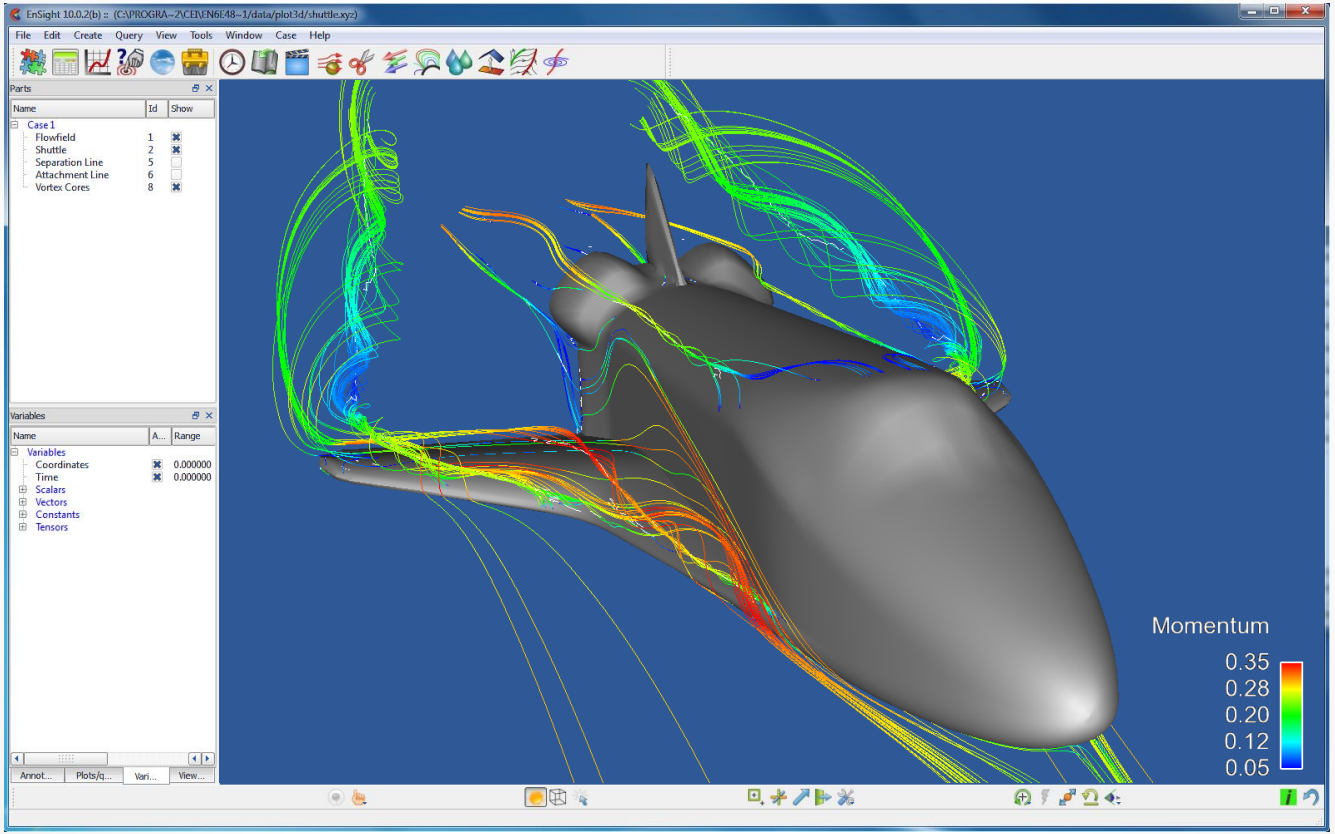


Vortex Cores 1

- Vortex cores are centers of swirling flow where the velocity is parallel to the vorticity
- Select the fluid domain then click the Vortex Cores icon
- There are various menus to define The vortex variables and the **Method** that is used
- Click the **Create with Selected Parts** Button
- Some CFD users like to use vortex cores to generate streamlines from



Vortex Cores 2



More Part Creation Exercise

- See the EnSight 10 Advanced Training Exercises handout and do Exercise 3

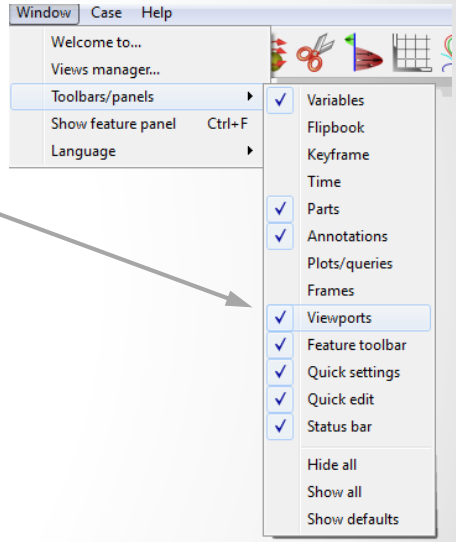
Polaris Snowmobile

Chapter 8: Create/Edit Viewports

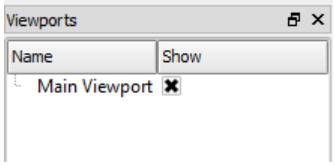
Viewport Overview 1

- Advanced control over one or multiple viewports can be achieved through the **Viewport** tab in the Objects List

- If the Viewports tab is not displayed in the Objects List, click on **Window** -> **Toolbars/Panels** -> **Viewports** and it will be toggled on; this is also how it can be toggled off

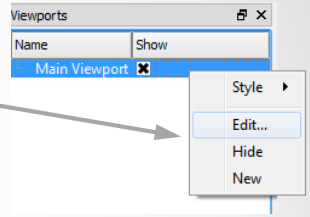


- Click the **Viewports** tab in the Objects List and the Main Viewport will be listed

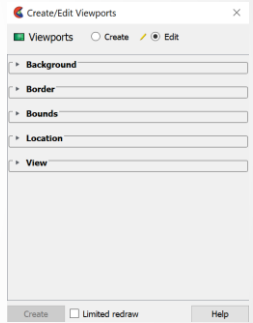


Viewport Overview 2

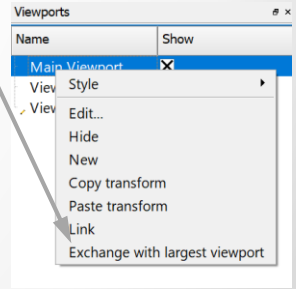
- Right click in the background of the the Main Viewport - since there is only 1 viewport in this example there are only 3 options: **Edit**, **Hide** and **New**



- When **Edit** is selected the Create/Edit Viewports menu is displayed with 5 options; these options are explained in the following slides

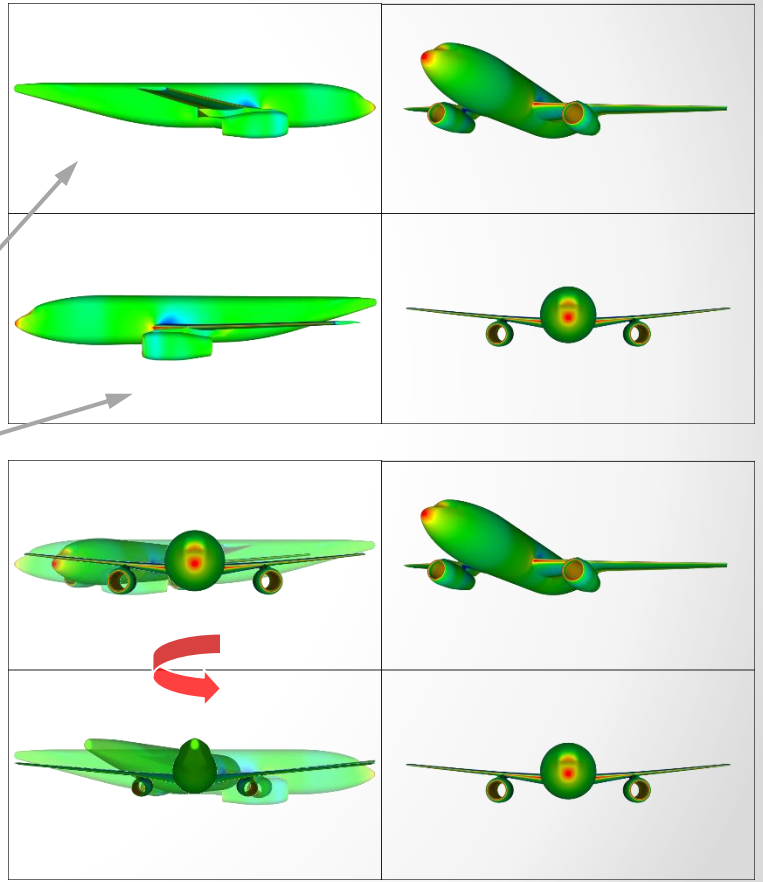


- When there are 2 or more viewports, a right click has 4 additional options: **Copy Transform**, **Paste Transform, Link** and **Exchange with Largest Viewport**
- **Copy Transform** and **Paste Transform** can be used to copy the view from one viewport to another; use **Copy Transform** in one viewport and **Paste Transform** in another to copy the view from the first viewport to the second



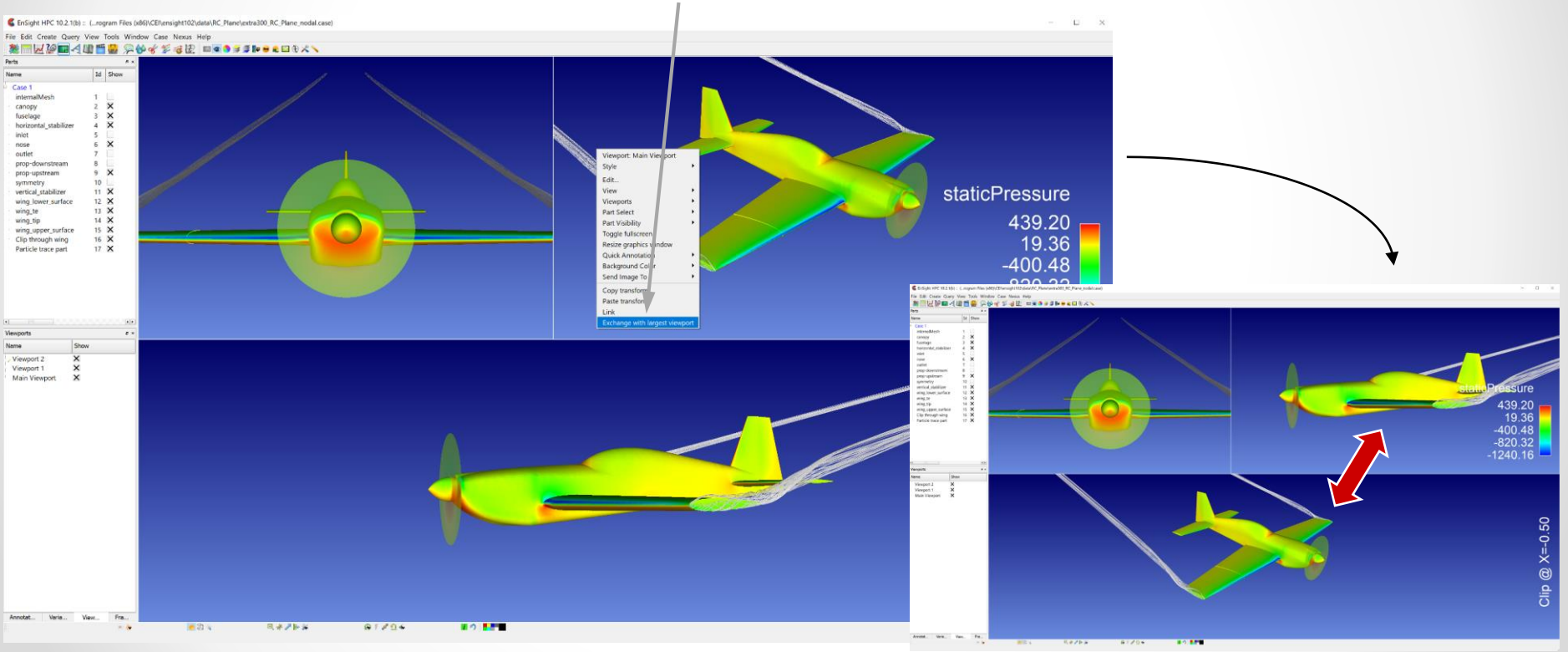
Linking Viewports

- The **Link** option can link 2 or more viewports that have the same or totally different views in them
- To link 2 viewports, for instance the right and left views of the airplane, right click in the first viewport and select **Link** – then perform the same action in the second viewport; the 2 viewports are now linked; more viewports can be linked in the same way
- To unlink the viewports, right click and select **Unlink** in each of the viewports that was linked



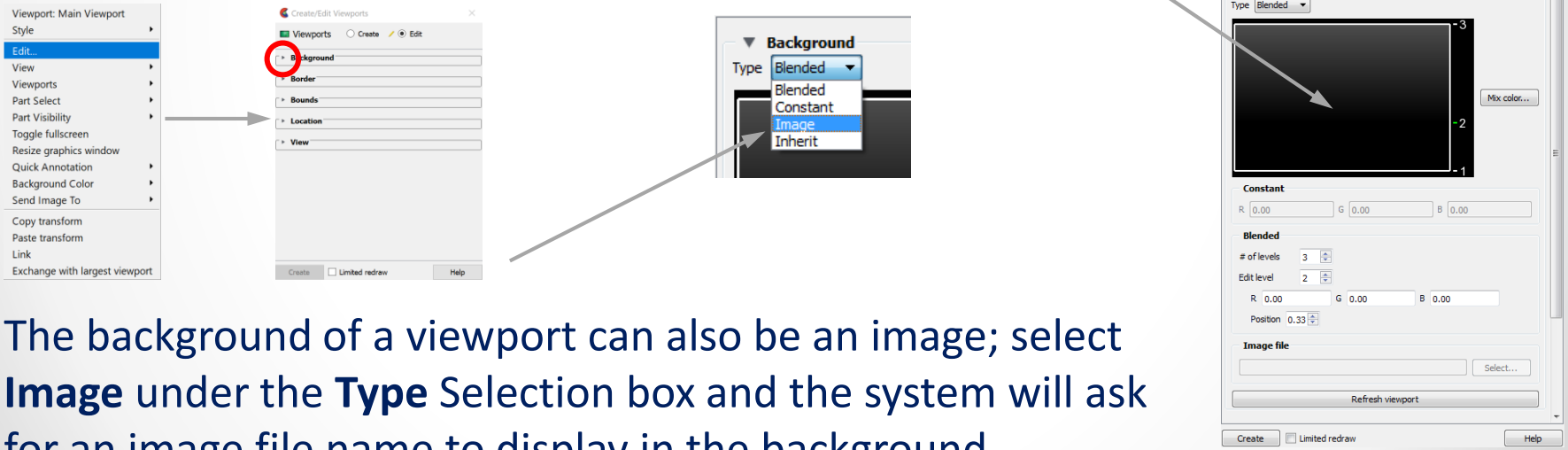
Exchange with Largest Viewport

- The **Exchange with Largest Viewport** option exchanges the view of a smaller viewport with the one in the largest viewport



Viewport Background Color 1

- Select a viewport and right click to edit the viewport; click the **Background** expand arrow ▶ to display the options - the expand button is now pointing downwards ▼ to indicate that the menu has expanded; the viewport background colors can be constant, blended or inherited from the default viewport; the default color is a blended color that changes from black at the bottom to gray at the top in 3 levels

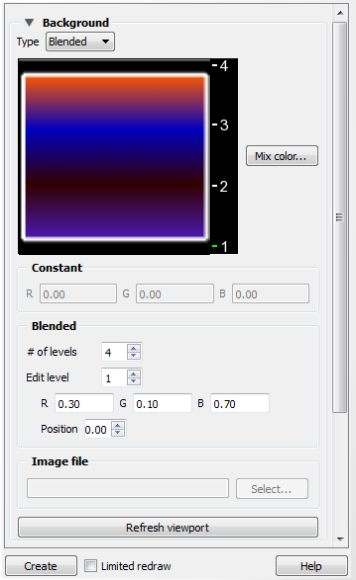


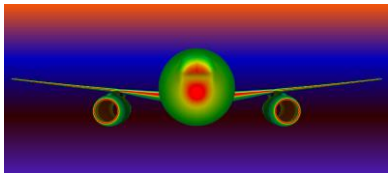
- The background of a viewport can also be an image; select **Image** under the **Type** Selection box and the system will ask for an image file name to display in the background

Viewport Background Color 2

- For a **Blended** background, up to 5 horizontal level colors can be specified with blending between each level

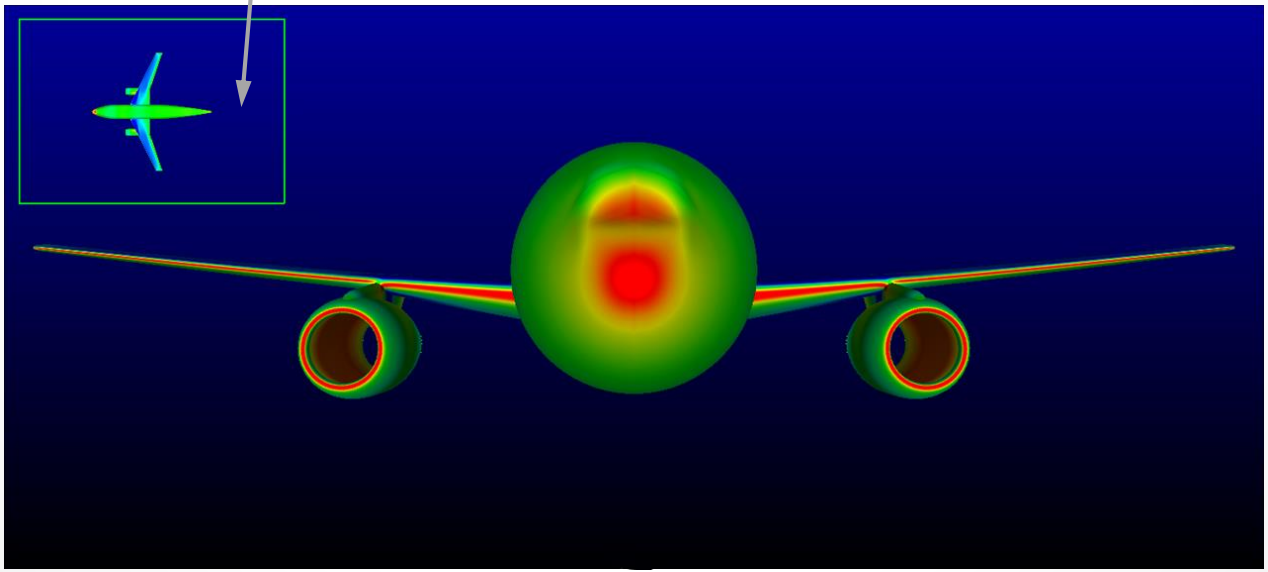
- To create a new blended background color, do the following:
  - Select **Blended** under **Type**
  - Enter the **# of Levels**, for example 4
  - Set **Edit Level** to 1 and enter the **RGB** values, for instance .3, .1 & .7 or click on **Mix Color…**
  - Do the same for the other levels and a new **Blended** background has been created





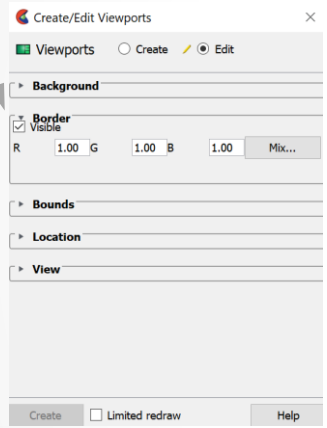
Viewport Background Color 3

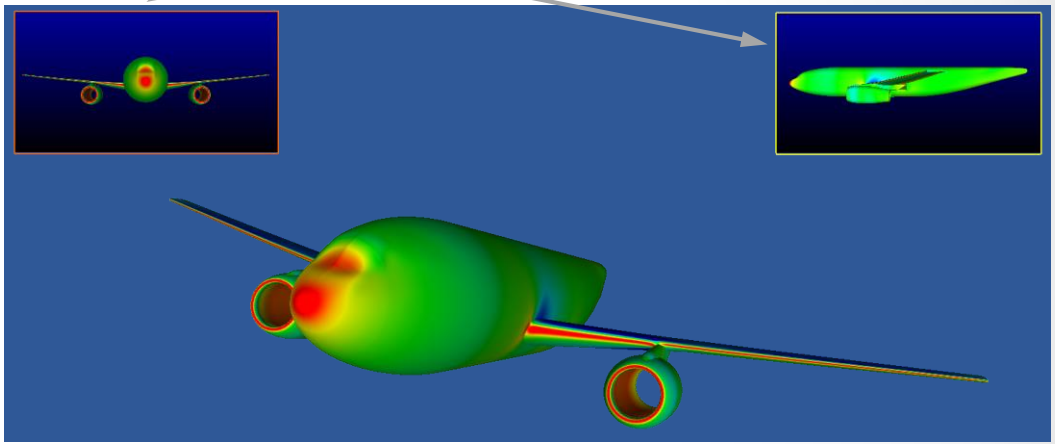
- Use the **Inherit** option so the selected viewport(s) inherit the background type and color from the Main viewport
- In this example the new viewport (# 1) has the default **Blended** colors and when that is changed to **Inherit**, it takes the local color of the default viewport



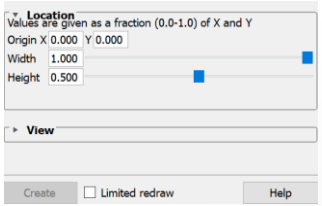
Viewport Border and Location

- The border of a viewport can be made visible by clicking the **Border** arrow; the border visibility can be toggled and the color of the border can be selected as well



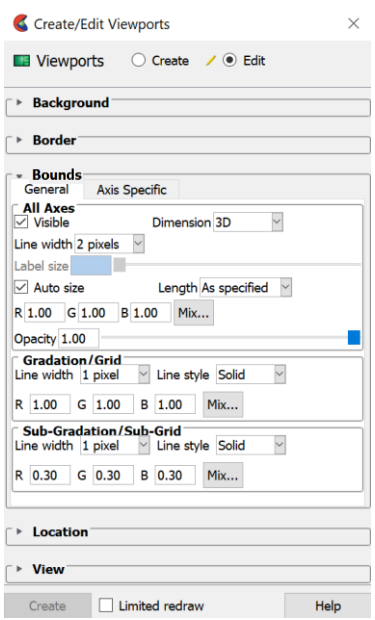


- The position and size of a viewport can be entered by clicking the **Location** arrow



Viewport Bounds Visibility

- Part bounds can be displayed per viewport; click on **View** -> **Bounds Visibility** from the Main Menu to toggle this feature on; right click in a viewport and click on the **Bounds** expand arrow
- The attributes of the **Bounds Visibility** feature can be modified with this panel

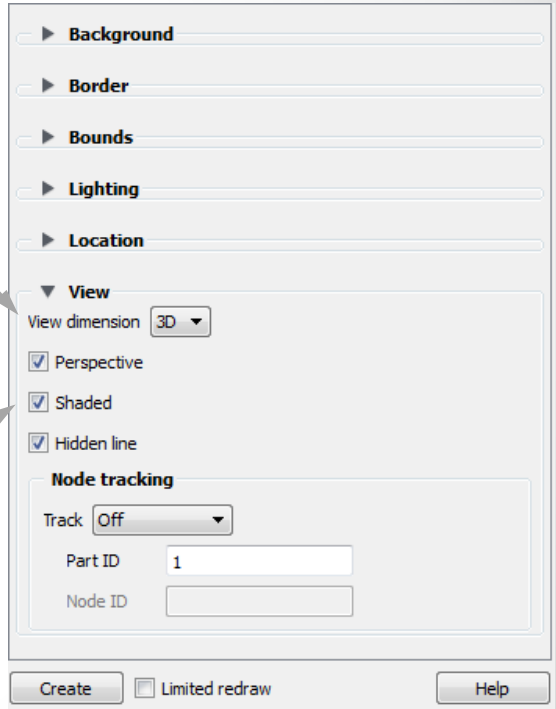


Selected Viewport Special Attributes 1

- Click the **View** arrow to see more options that can be set per viewport

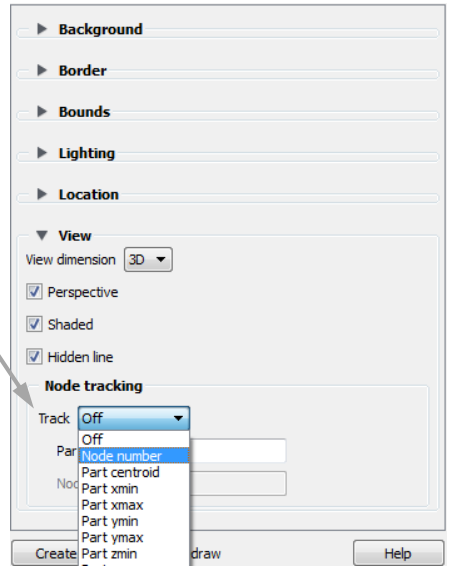
- A viewport can be set to either a 2D or 3D **View Dimension**; if the viewport is set to 2D, only planar parts will be displayed in the viewport and the rotations in that viewport can be only around the screen Z-axis

- Each viewport has it's own toggles for **Perspective**, **Shaded** and **Hidden Line** display modes



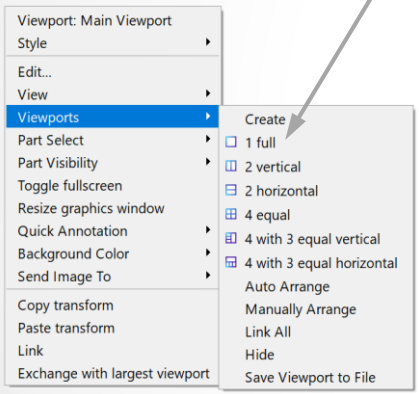
Selected Viewport Special Attributes 2

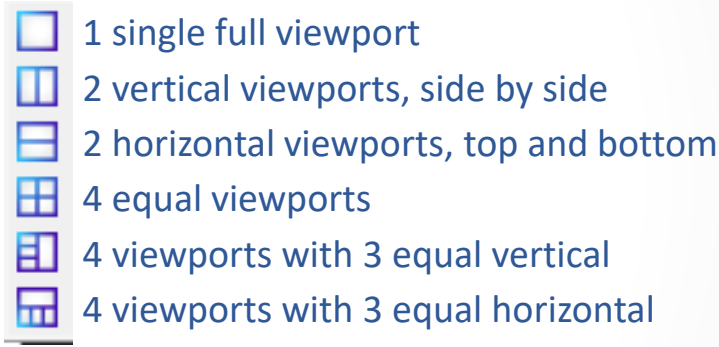
- A viewport can track a N**ode Number**, a **Part Centroid** or the **Part XYZ Min** or **Max** values; as a model changes over time the viewport will remain centered on that location; this can be very useful for transient datasets



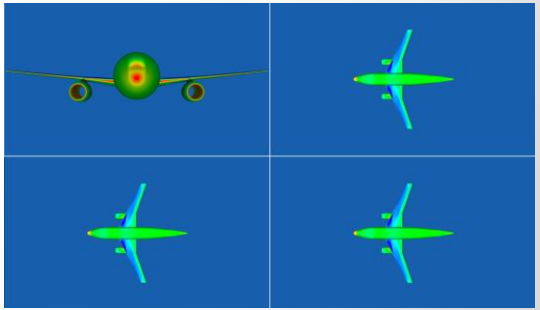
Viewport Layouts

- When EnSight is started, it creates a single default viewport (labeled **Main Viewport**) that fills the entire Graphics Window
- Right click in the viewport and 6 viewport layouts are displayed:



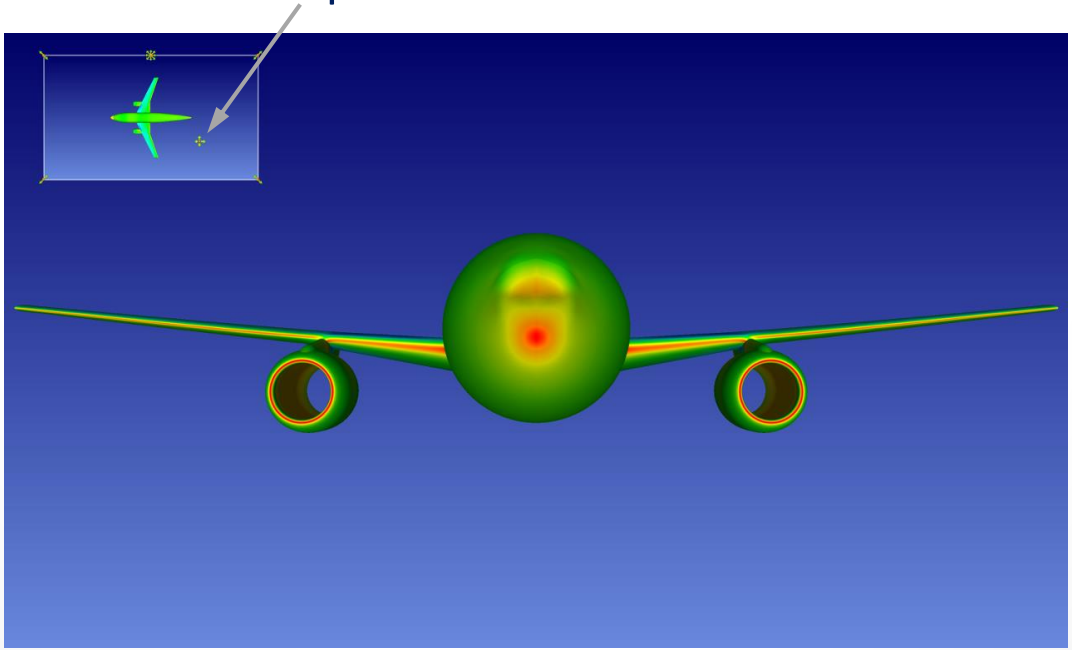


- Click the layout that is desired; in this example the 4 **equal viewports** layout is selected and the system resets the views in the new viewports



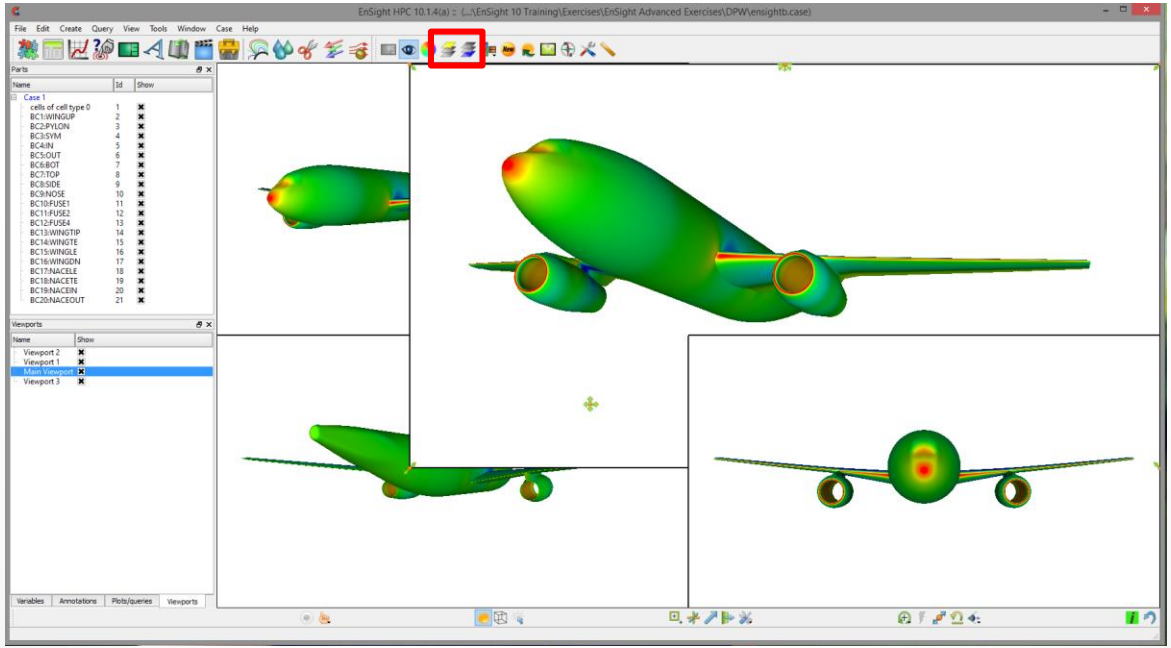
New Viewport

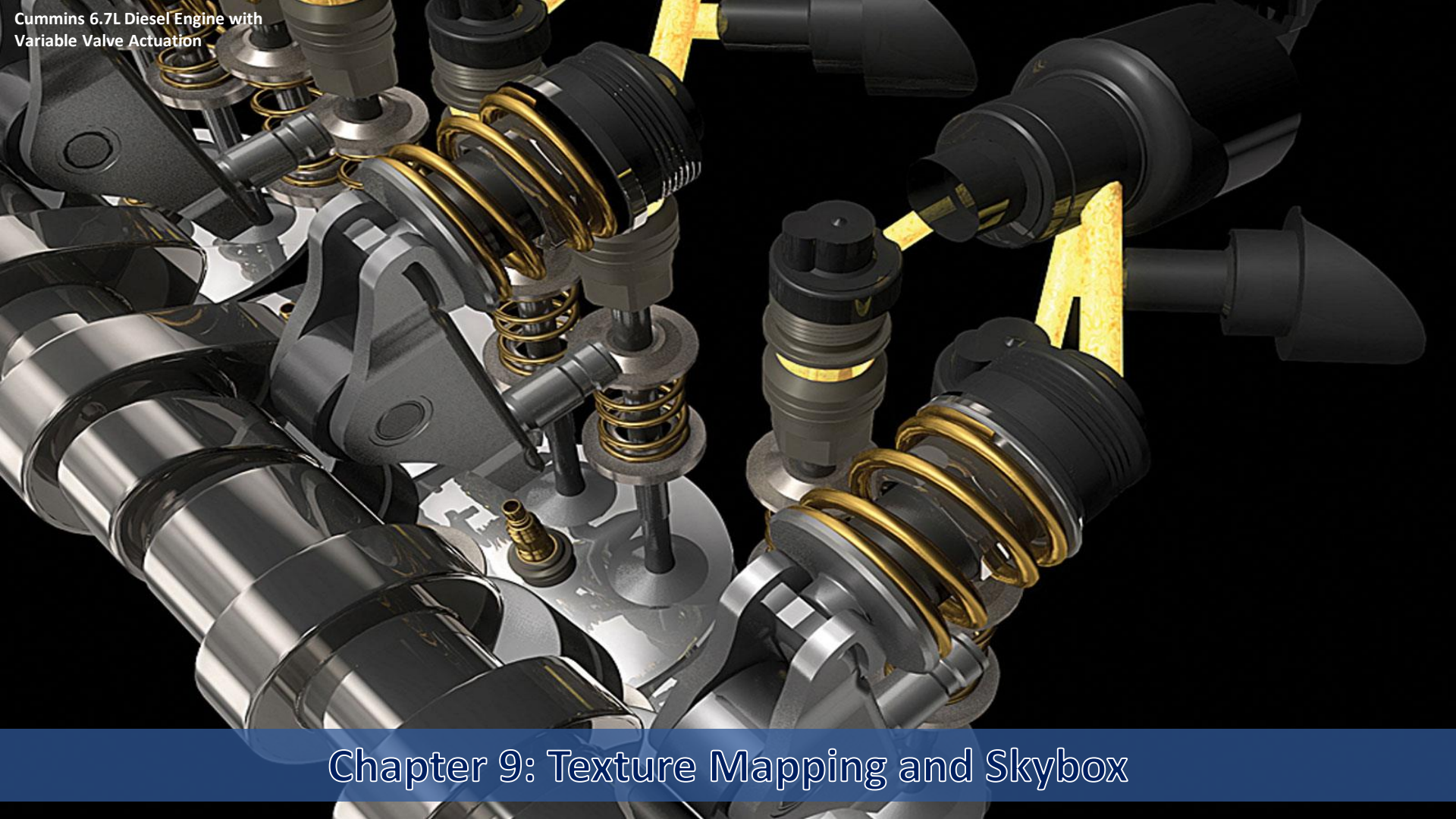
- Right click in the viewport and select **Viewports** -> **Create**; this will create a new viewport in the top left corner of the Graphics Window
- The new viewport can be resized by dragging the borders and moved by clicking and dragging in the new viewport



Pop/Push Viewport(s) Forward/Back

- EnSight allows overlapping viewports; the viewport order can be controlled with the **Pop Viewport(s) Forward** icon and **Push Viewport(s) Back** icon ; click in a viewport and click either the Pop or Push icons
- The Main Viewport can also be popped forward or pushed backwards

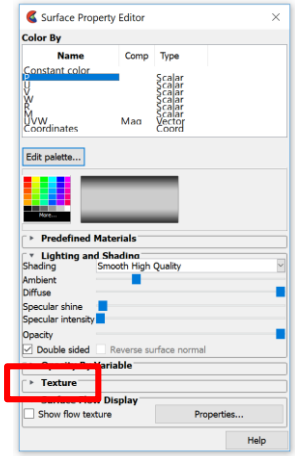


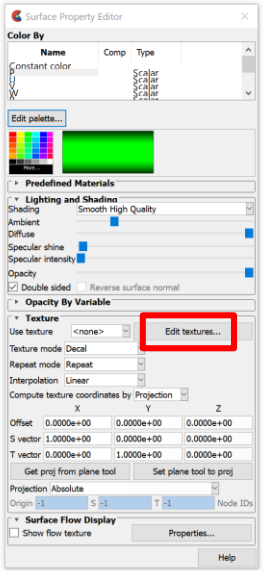


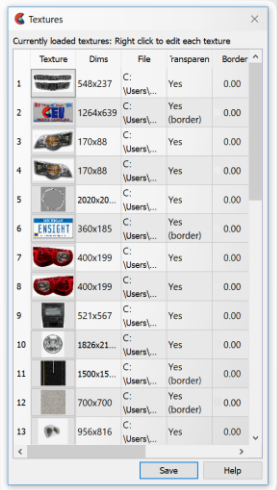
Chapter 9: Texture Mapping and Skybox

Texture Mapping Overview

- Texture Mapping is a technique whereby a 2D picture is projected onto a 3D object or even wrapped around a 3D object
- The easiest way to use Texture Mapping is to take a picture and use it as a decal (a sticker) and place it somewhere on a 3D object; Texture Mapping can also be used to display a pattern over part of the 3D object or the entire 3D object
- To access the Texture Mapping panel, select a part, click the **Surface Property Editor** and click the **Texture** arrow; click the **Edit Textures** button to display the **Textures** menu





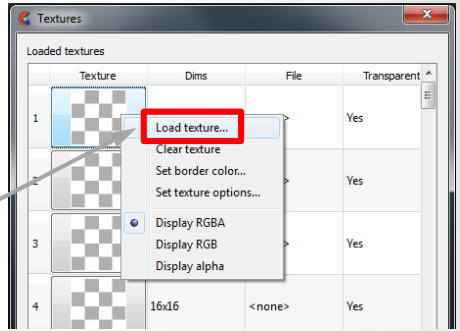


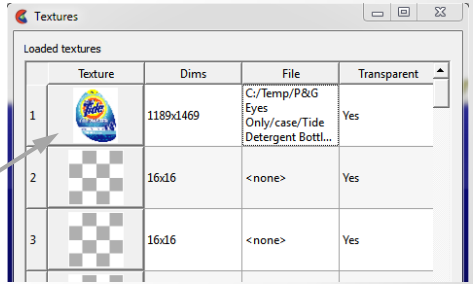
Texture Mapping – 1st Method - 1

- The following example projects the front label of a Tide detergent bottle onto the front of a 3D model of that bottle

- On the **Textures** menu, load a picture by right clicking on the first texture and selecting **Load Texture**; this will display a file browser where a picture file can be selected (.png, .jpg, .bmp etc); a high resolution image will give best results

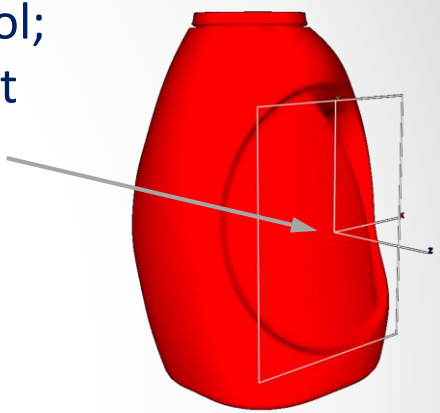
- As soon as a picture is selected, a thumbnail will be displayed on the **Textures** menu; in this case the front label of the Tide detergent bottle





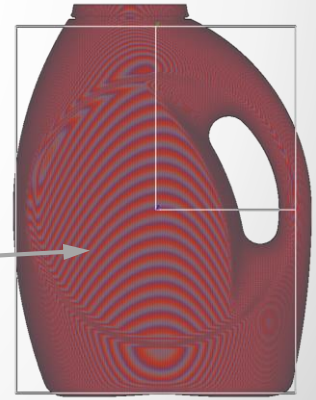
Texture Mapping – 1st Method - 2

- The fastest way to apply a texture map is to use the Plane Tool; in this example the XY-plane of the tool is parallel to the front of the Tide bottle with the Z-axis sticking out the front of the display



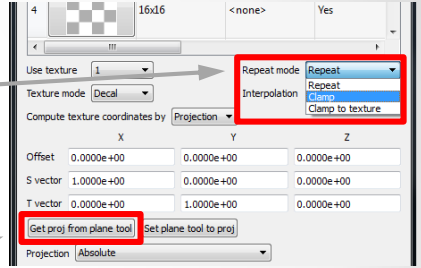
- It's easiest to look at it from the front view with **View** -> **Perspective** switched off

- Select the bottle in the Parts List and left click on the first texture with the Tide front label; this will project the texture along the Z-axis onto the bottle; the strange pattern on the bottle is caused by the **Repeat Mode** on the **Textures** menu



Texture Mapping – 1st Method - 3

- **Repeat Mode** is set by default to **Repeat** which is typically used for patterns; for this example change it to **Clamp**



- To make the system use the current size, position and orientation of the Plane Tool, click on the **Get Proj from Plane Tool** button; this will update the texture map to use the current Plane Tool

- Adjust the size and position of the Plane Tool and click the **Get Proj from Plane Tool** button after each change; after several changes the fit will be near perfect

Texture Mapping – 1st Method - 4

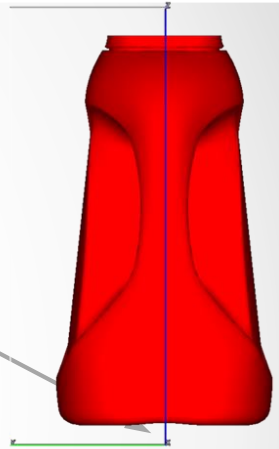
- Rotate and zoom in on the model (watch it full screen for instance by hitting F9) and see that the texture map adds a great amount of realism to the model

- When the model is rotated to the back of the bottle it will be clear there's a problem; the front label is also mapped on the back of the bottle and because of the Z-axis projection, it's in mirror image

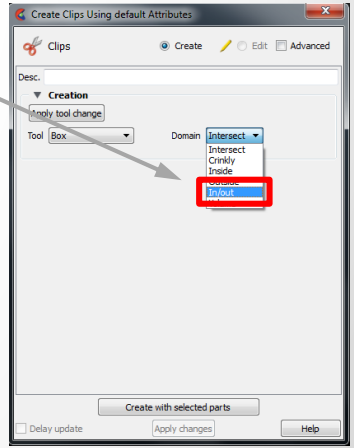
- This can be fixed by cutting the bottle in 2 halves; click the **Tools** button on the top **Main Menu** and select **Box**; position the box in the middle of the bottle and make sure it covers the entire half of the bottle; click the **Clip** icon and select **Box** as the **Use Tool**

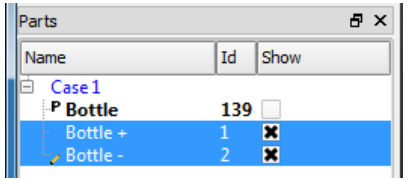
Texture Mapping – 1st Method - 5

- The model looks like this with the Box tool covering half the model; the Box tool's axis can be resized and the entire tool can be moved by dragging the origin point of the 3 axes



- With the bottle still selected in the Parts List, cut the bottle with the Box tool using the **Clip** feature; select **In/Out** as the **Domain,** this will create 2 new bottle halves in the parts list, named **Bottle +** and **Bottle -** ; at the same time it will hide the original Bottle part

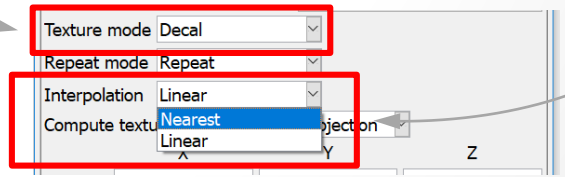




Texture Mapping – 1st Method - 6

- Now there are 2 bottle halves that can be texture mapped individually using the same Z-axis projection method as described before to texture both the front and the back label onto the Tide bottle

- Under **Interpolation** select **Nearest** or **Linear,** whichever looks best
- On the **Textures** panel, set **Texture Mode** to **Decal**



Texture Mapping Exercise

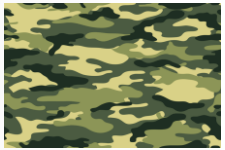
- Load bottle.case and cut the bottle so the front and back labels can be texture mapped on it
- Texture map both labels onto the bottle
- Create 2 viewports side by side and show the front and back view at the same time; link the viewports and make the viewport borders invisible

Texture Mapping – 2nd Method - 1

- The second method of using Texture Mapping is to use an image as a pattern over the entire 3D model or a portion of it; patterns like wood, brick, fabric etc can be used to give a 3D object more realism
- The Plane Tool is again used to specify the Z-axis that will define the way the pattern is projected onto the 3D object
- The Comanche helicopter body will be used to texture map a pattern onto; the color of the body of the helicopter is military green

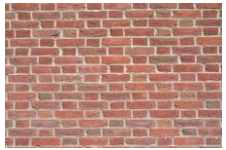
Texture Mapping – 2nd Method - 2

- The following patterns are textures that were found on the internet and will be used to texture map the Comanche body


camouflage


water drops


wood

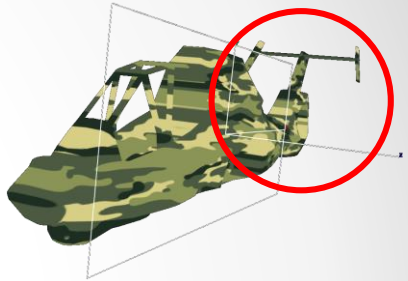

bricks


$100 bill

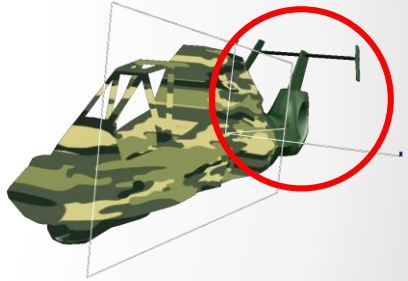
- To use one of the above textures, do the following:
  - Load the Comanche model, select the Comanche body (Part ID 4) and open the **Textures** menu
  - Toggle the **Plane Tool Visibility** icon on
  - Right click the first texture field and select **Load Texture…** and load the camouflage image from the file browser
  - Left click the thumbnail that just appeared on the **Textures** menu and the texture will appear on the body of the Comanche

Texture Mapping – 2nd Method - 3

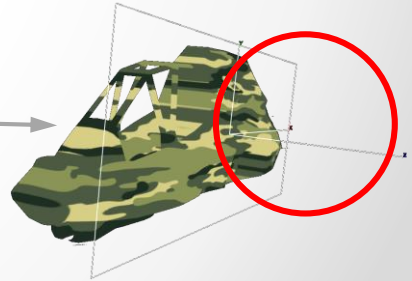
- On the **Textures** menu the **Texture Mode** is by default set to **Decal**; the **Repeat Mode** is by default set to **Repeat**; the Comanche model will look something like this



- Set **Repeat Mode** to **Clamp** and only the portion of the object that is under the Z-axis projection will be textured



- Set **Texture Mode** to **Replace** and only the textured portion of the object will be displayed



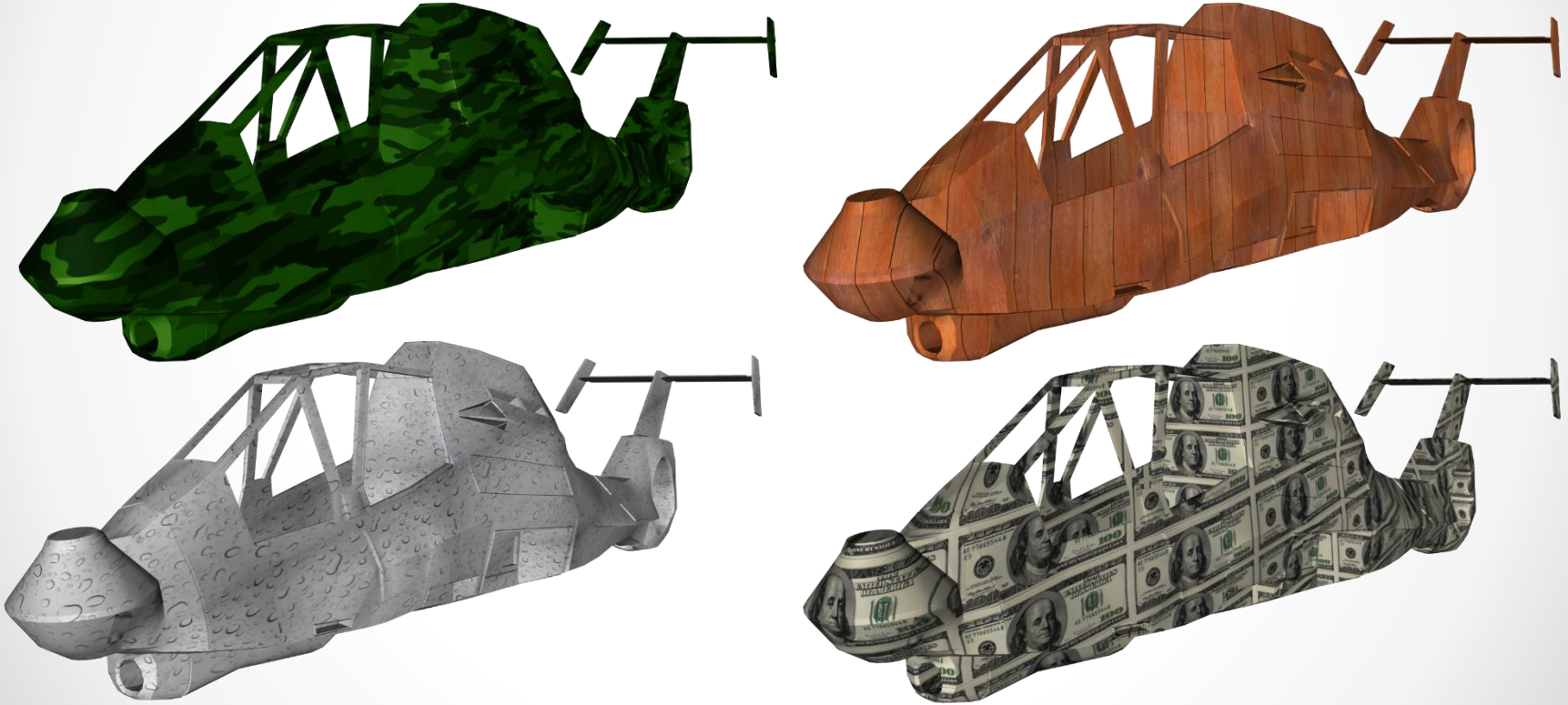
Texture Mapping – 2nd Method - 4

- Still on the **Textures** menu, set the **Texture Mode** to **Modulate** and the color of the object (military green in this example) will shine through the texture and it will preserve the shading of the model underneath the texture map; the **Repeat Mode** has been set back to **Repeat**

- By changing the color of the Comanche body and by modifying the size and orientation of the Plane Tool, the texture map will be influenced; after every change to the Plane Tool, click the **Get Proj from Plane Tool** button to update the image

Texture Mapping – 2nd Method - 5

- Here are some examples of texture maps on the Comanche



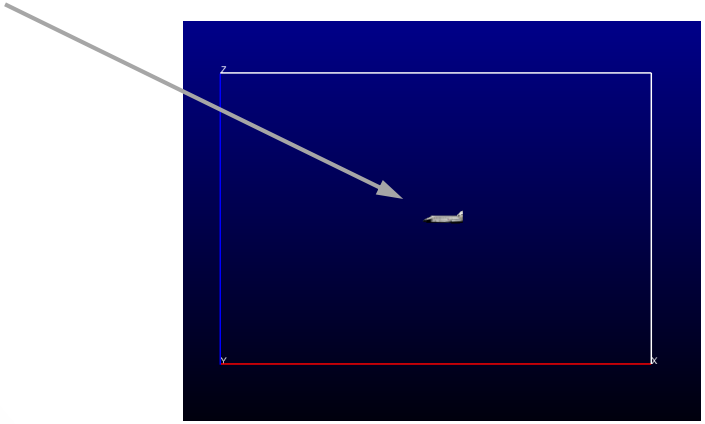
Skybox Overview

- A Skybox is a cube that is placed around the entire model and the 6 sides of the box are texture mapped with 6 images that are continuous at the seams; this creates the illusion of a 3D world around the model that rotates as the model is being rotated
- The cube has to be significantly bigger than the limits of the model, so the user can zoom in to the model and see the 3D world in the background

Click the movie to play

Skybox 1

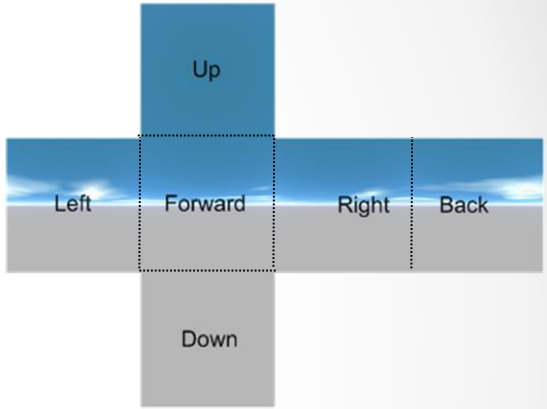
- To create a Skybox around the Space Shuttle model, do the following:
  - Load the model and texture map the shuttle for even more realism (set **View** -> **Perspective** to off to make it easier)
  - Zoom out so the model is relatively small at the center of the screen and click **Tools** -> **Box** (resize the box if needed)



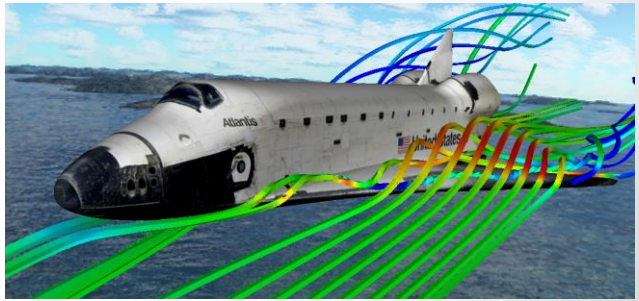
  - Select the Space Shuttle (Part ID 2) in the Parts List
  - Click the Skybox icon and click **Ok**

Skybox 2

- Select a Skybox picture in the file browser; a Skybox picture has the following layout



- Switch of the **Box** tool and switch **Perspective** back on
- Zoom into the Skybox so the model can be seen; switch on any particle traces, clip planes, isosurfaces etc



- Click F9 and rotate and zoom the model full screen
- A Skybox will be recorded in an animation

Texture Mapping Exercise

- See the EnSight 10 Advanced Training Exercises handout and do Exercise 4


Embraer Legacy 600 Business Jet
LEGACY 600
BY EMBRAER
Chapter 10: Keyframe Animation

Keyframe Animation Basics 1

- EnSight can create smooth and complex animations using a process called Keyframe Animation
- In a keyframe animation, a particular start view and a different end view (called keyframes) are defined and EnSight can automatically generate frames to interpolate between the two views creating a smooth animation
- A keyframe animation will record not only all rotations, pans and zooms that are used but also changes in transparency, auxiliary clipping, particle traces, isosurfaces, clip planes, plots, texture maps, a skybox etc
- The length of the animation is determined by the number of frames generated between the two keyframes

Keyframe Animation Basics 2

- A keyframe animation can also be used to record a transient model; the time line of the dataset can be controlled
- The results of a keyframe animation can be written to various movie formats such as MPEG, AVI, QuickTime, Flash etc
- The resolution of the animation can be controlled from NTSC, PAL, DVD, HD 720p, HD 1080p up to User Defined; anti-aliasing can be used to get the highest possible quality for the frames used in the animation
- Once EnSight is starting to generate frames, the process can be aborted by placing the cursor in the Graphics Window and pressing the 

key on the keyboard (

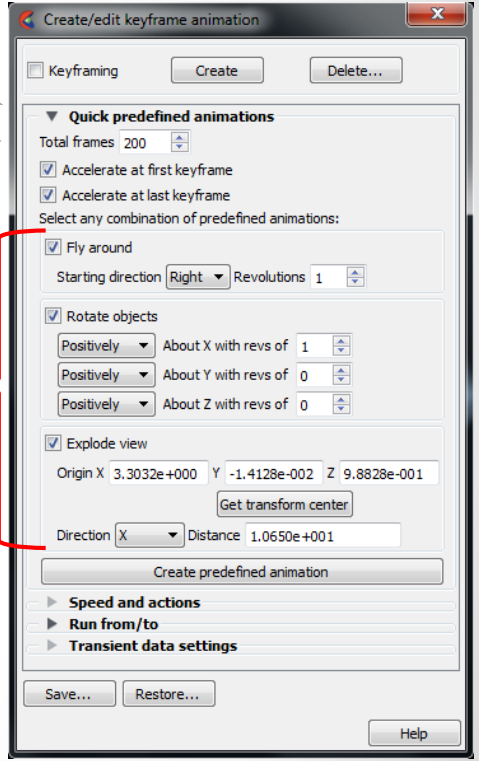
for abort)

Quick Animations

- A fast way of generating a keyframe animation is to click the **Keyframe Animation** -> **Quick Predefined Animations** toggle; this displays a panel where 3 predefined options can be selected

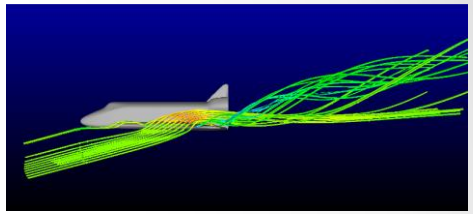
- Enter the total number of frames, one or more of the predefined options (**Fly Around**, **Rotate Objects** and **Explode Views**) and type in the desired parameters for each option

- Click the **Create Predefined Keyframe** button and the keyframe sequence will be created



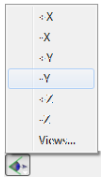
Keyframe Animation Creation 1

- The Space Shuttle model with the particle traces will be used as example

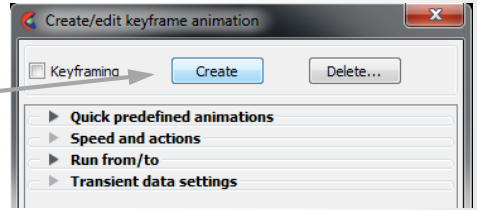


Keyframe 1

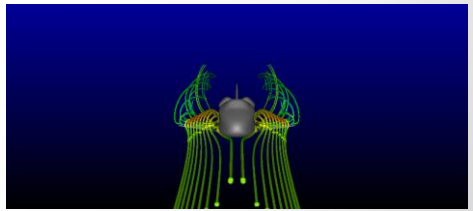
- Once the model is loaded, select the -Y view and press the ‘Fit Geometry’ icon so the Shuttle is facing left



- Click the **Keyframe** icon ; on the menu click the **Create** button – this will create the first keyframe



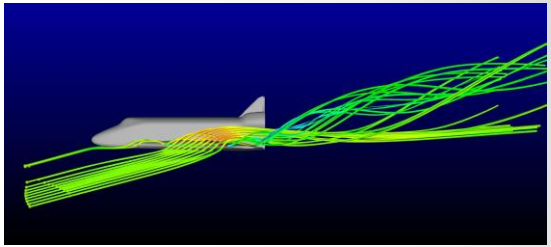
- Then rotate the model to the front view by pressing F2 twice and click **Create** again - this will create the second keyframe



Keyframe 2

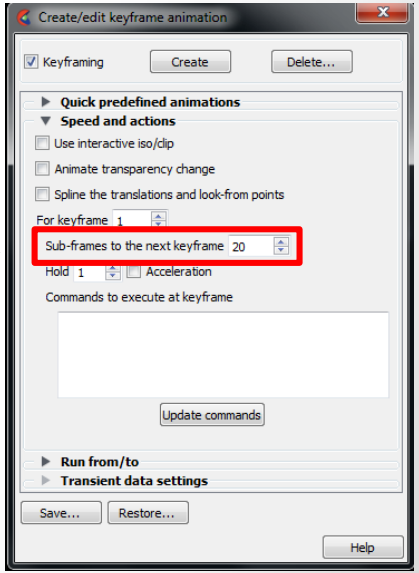
Keyframe Animation Creation 2

- EnSight just created a keyframe animation and it can be played back; click on the **Run** button and the animation will be played



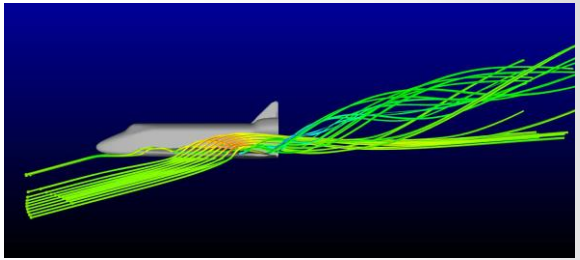
Click the movie to play

- This animation is not very useful because the playback time is less than a second; this is controlled by the **Speed and Actions** button; clicking this toggle will display a menu where the number of **Sub-frames to the Next Keyframe** can be entered; this number controls the speed of an animation

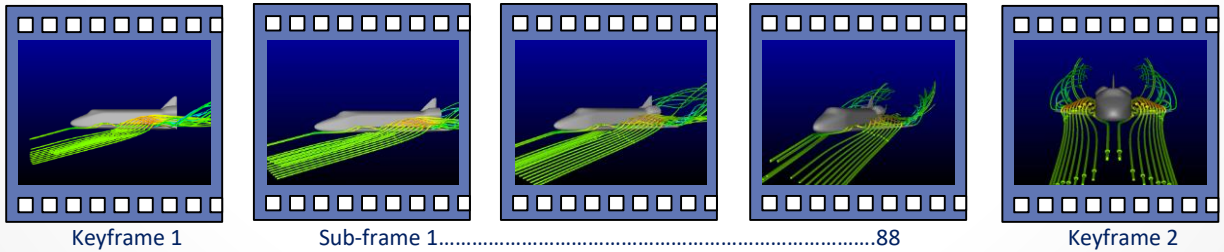


Keyframe Animation Creation 3

- In this next movie the number of subframes has been set to 88; the animation now takes exactly 3 seconds to play - why 3 seconds? Well, an animation has a default frame rate of 30 frames per second and since there are 2 keyframes that were created (the start view and the end view) plus 88 sub-frames, there are a total of 90 frames - at 30 frames per second, that's exactly 3 seconds

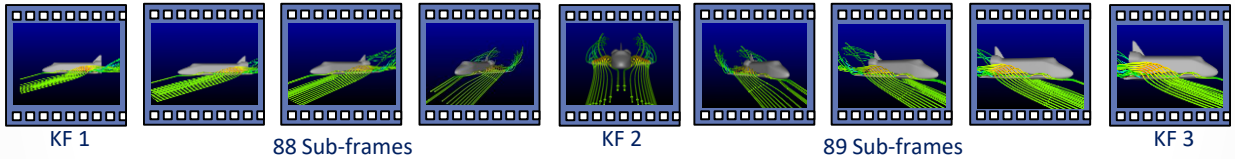


Click the movie to play

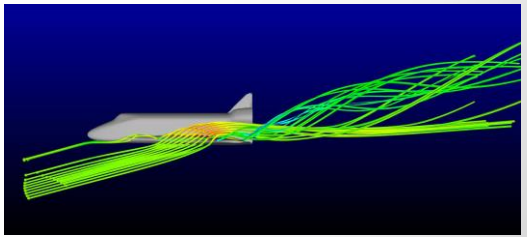


Keyframe Animation Creation 4

- After the first 2 keyframes, more keyframes can be created; press F2 two more times and zoom in on the Space Shuttle, then click **Create** again; there are now 3 keyframes and 2 sets of sub-frames; to make the segment from the second keyframe to the third keyframe also 3 seconds long, the number of sub-frames have to be 89 - here's why:



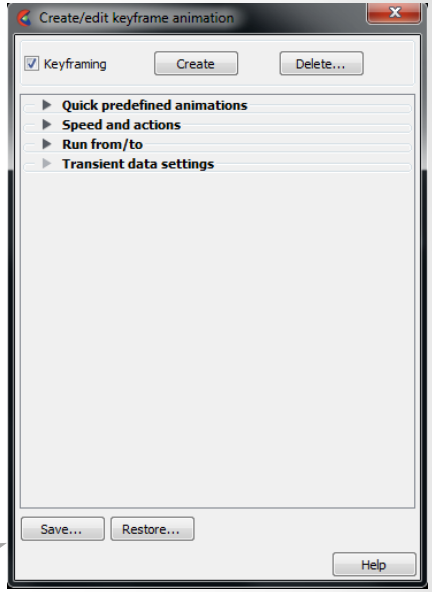
- Note that the animation between keyframes 2 and 3 not only rotates but also zooms at the same time (noticed the animated particles?)



Click the movie to play

Keyframe Animation Creation 5

- As soon as 1 or more keyframes are defined, the **Keyframing** toggle will be checked; when this toggle is checked, it means there are 1 or more keyframes defined

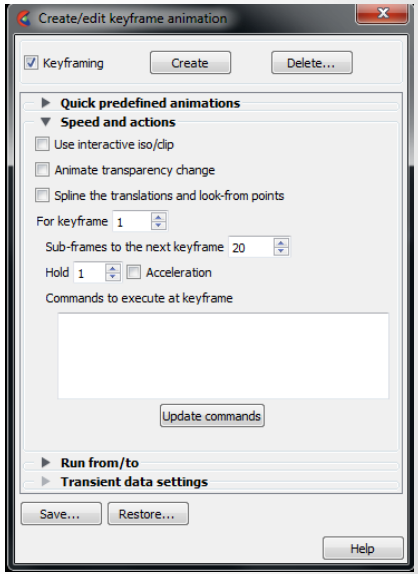


- When the toggle is unchecked, the entire keyframe sequence is deleted; click on **Save** to file a keyframe sequence and press **Restore** to read it back in from a file

Keyframe Run Attributes

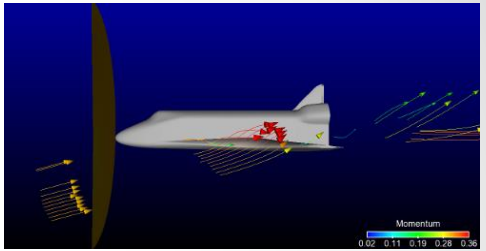
- After 2 or more keyframes have been created, the **Speed and Actions** panel will become available; on this panel besides the number of sub-frames, options to animate Isosurfaces, Clips and Transparency changes can be toggled on

- Click the **Transient Data Settings** toggle if the dataset is transient and the motion should be included in the animation (if the dataset is not transient this feature is grayed out)

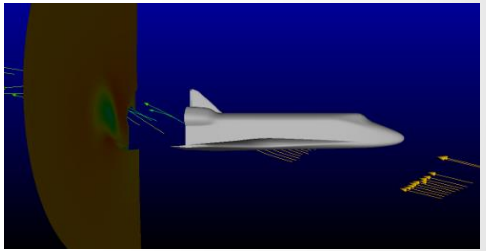


Animating a Clip Plane

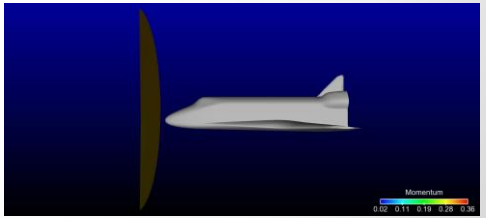
- Create another Space Shuttle keyframe animation starting with the shuttle facing left with a clip plane in X



- Click the **Create** button then rotate the model 180° so it's facing right; left click the clip plane and drag it so it's just behind the Space Shuttle; now click the **Create** button again



- The animation now rotates and moves the clip plane at the same time

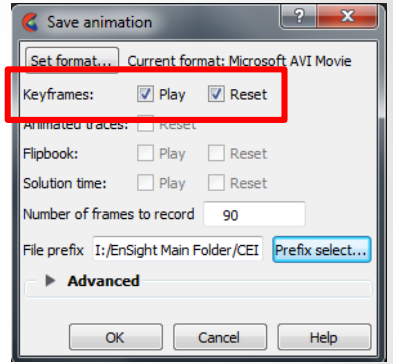


Click the movie to play

Recording a Keyframe Animation 1

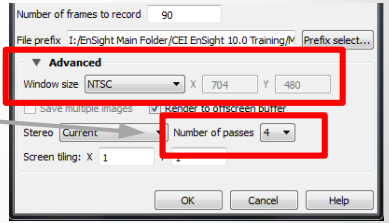
- To make a movie from a keyframe animation, click the record icon or click **File** -> **Export** -> **Animation**; the Save Animation menu is displayed

- Make sure to select the toggles for **Keyframes Play** and **Reset**



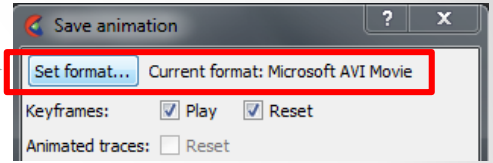
- Enter the name for the movie

- Click the **Advanced** toggle to display a menu where the **Window Size** and anti-aliasing (**Number of Passes**) can be set

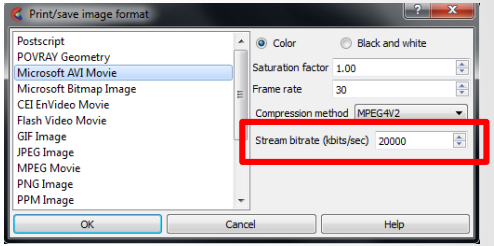


Recording a Keyframe Animation 2

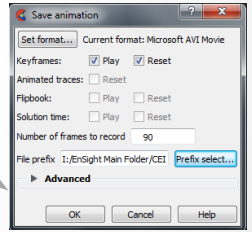
- The **Set Format** button displays a variety of movie formats that can be selected; choose a format, in this example AVI, and see the options for that format



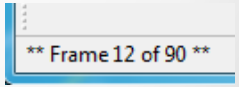
- The **Stream Bitrate** controls the quality of the movie; a higher number results in a higher quality but at the cost of a bigger file size



- Click the **OK** button on the Save Animation menu and the movie will be created



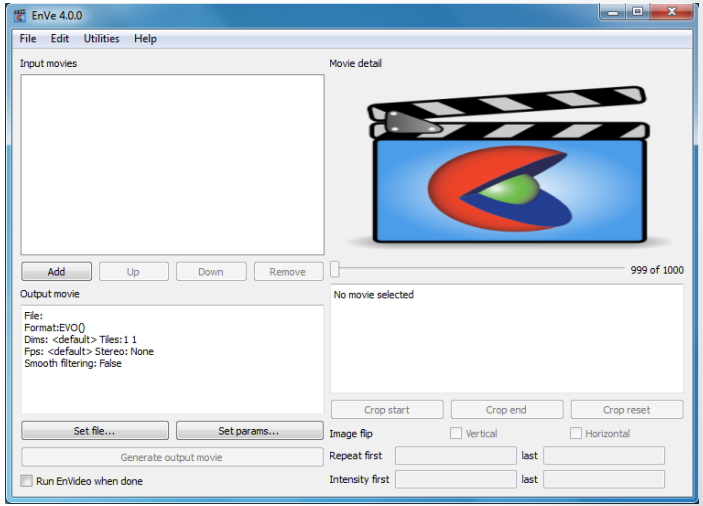
- The Message Area in the lower left corner of the EnSight window will display the frame numbers that are being generated



Keyframe Animation Tips 1

- Creating many keyframes does have its drawbacks; for instance if a certain keyframe sequence has to be modified, there's no way to just go back to that one sequence and modify it; for this reason it's better to create a keyframe sequence between only 2 keyframes and then write it as an animation file

- After all the individual movie segments have been created, a free program supplied by CEI called EnVe is used to create one complete movie from all these segments



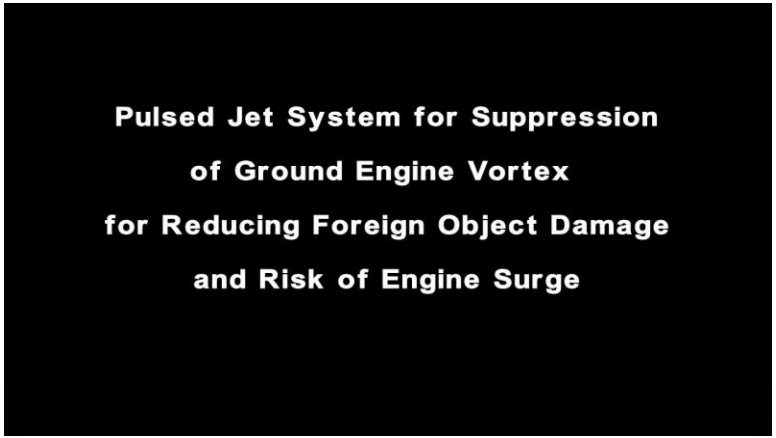
Keyframe Animation Tips 2

- If an animation is needed that can be continuously looped, make sure the start frame and the last frame are the same
- Remember that the graphics on a workstation run at a different frame rate (normally slower) than a movie (30 frames per second); this means that an animation that looks fine on a workstation could be too fast when played back as a movie
- When an animation is created using a large window size (resolution) and/or anti-aliasing, the generation time of the movie file will be significantly longer
- Creating a keyframe animation is pretty easy; however, creating a great movie using keyframe animation takes skills, creativity, a plan of what the movie should look like and experience

Keyframe Animation Examples

HCCI Engine In-cylinder

Contact Lens Packaging



Boeing C-17 Ground Vortex

Keyframe Animation Exercise

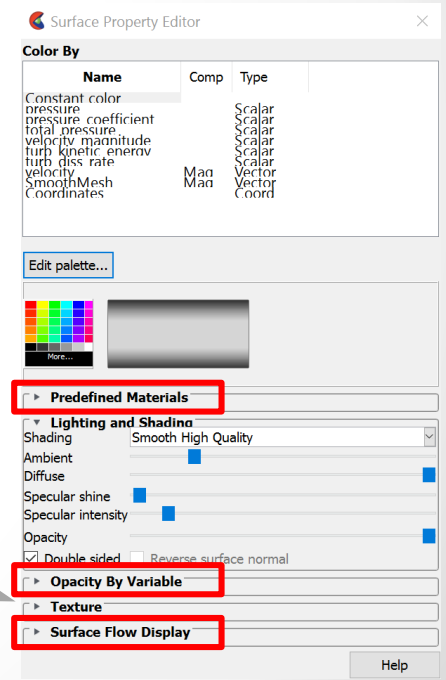
- See the EnSight 10 Advanced Training Exercises handout and do Exercise 5

Boeing 787-10 Dreamliner

Chapter 11: Using Predefined Materials

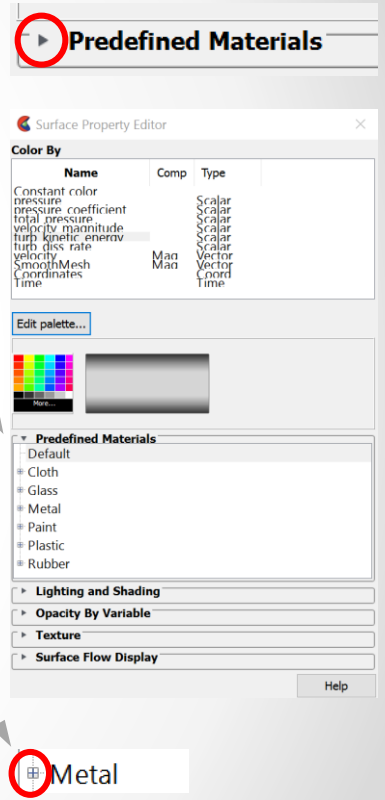
Materials Library Overview 1

- EnSight 10.2 introduces a predefined materials library that can be used to enhance the realism of parts
- The Surface Property Editor is used to assign a material to a part – click the Color Wheel icon to display the menu
- The Surface Property Editor menu is similar to the previous Color Editor menu but it has more lighting controls and it contains several new menus as well:
  - Predefined Materials
  - Opacity By Variable
  - Surface Flow Display
- The Texture menu is now directly accessible on the Surface Property Editor menu



Materials Library Overview 2

- The materials library is accessible by clicking the expand button ▶ to the left of the **Predefined Materials** label
- When clicked, the materials library opens up and displays 6 types of materials - the expand button is now pointing downwards ▼ to indicate that the menu has expanded
- By default, parts are not assigned a material when they are created or read in from a dataset
- To see the available metal materials for instance, click the + sign to the left of the **Metal** label; this will open the Metal group and it will display several metals
- Select a part in the Parts List and select a material, for instance **Metal** -> **Chrome** – this assigns the metal chrome to the part that is selected



Materials Assignment Example

- Use the ‘Chevy Traverse Wheel.case’ dataset as example
- Assign **Chrome** as the material for the rim, rim center logo edge and rim center, **Rubber** -> **Hard** for the tire, use **Paint** -> **High-gloss** for the caliper, **Aluminum** for the rotor, brake line and the valve stem, **Iron** for the brake housing, **Gold** for the GM logo and **Plastic** -> **Hard** for the valve cap – modify some of the slider settings and see what the effects are; adjust colors as needed

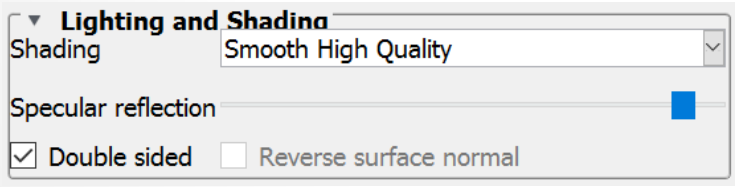
Without Materials

With Materials

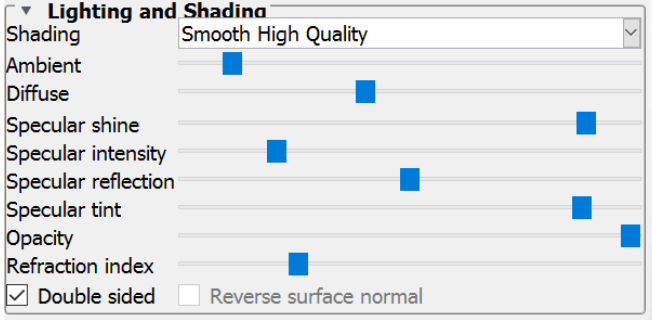
With Materials, texture map & road

Lighting and Shading per Material

- The sliders in the **Lighting and Shading** menu have different options depending on the material that is selected
- The **Metal** and **Glass** group and the **Paint** –> **High-gloss** and **Semi-gloss** have reflections – on the **Lighting and Shading** menu there's an additional slider for **Specular Reflections** for these materials – some of the **Glass** and the **Gloss Paints** have a **Refraction Index** as well; all other materials are matte and do not have reflections
- Depending on the selected material, the Lighting and Shading menu can have from 1 to 8 sliders that control how the materials look



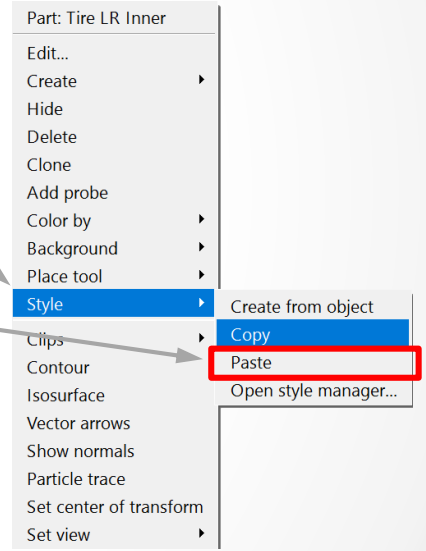
Lighting and Shading menu for Glass -> Mirror



Lighting and Shading menu for High-gloss & Semi-gloss paint

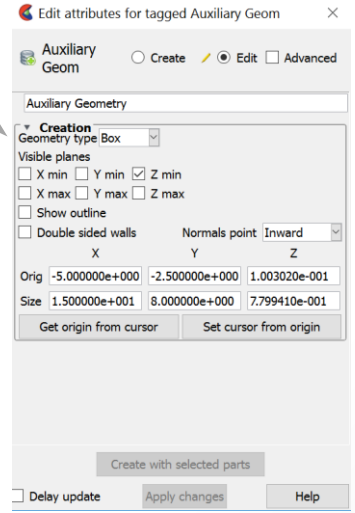
Style Feature to Copy Materials

- The color and material settings of a certain part can be copied to other parts by using the **Style** feature in the right-click menu
- Put the cursor on the part with the color and material that needs to be copied and right-click; then select **Style** and click **Copy**
- Now put the cursor on the part that needs the new color and material settings and right-click; then select **Style** and click **Paste**
- **Style** -> **Paste** can be used for multiple parts and groups as well
- Please note that the **Shading Type** is not copied from one part to another

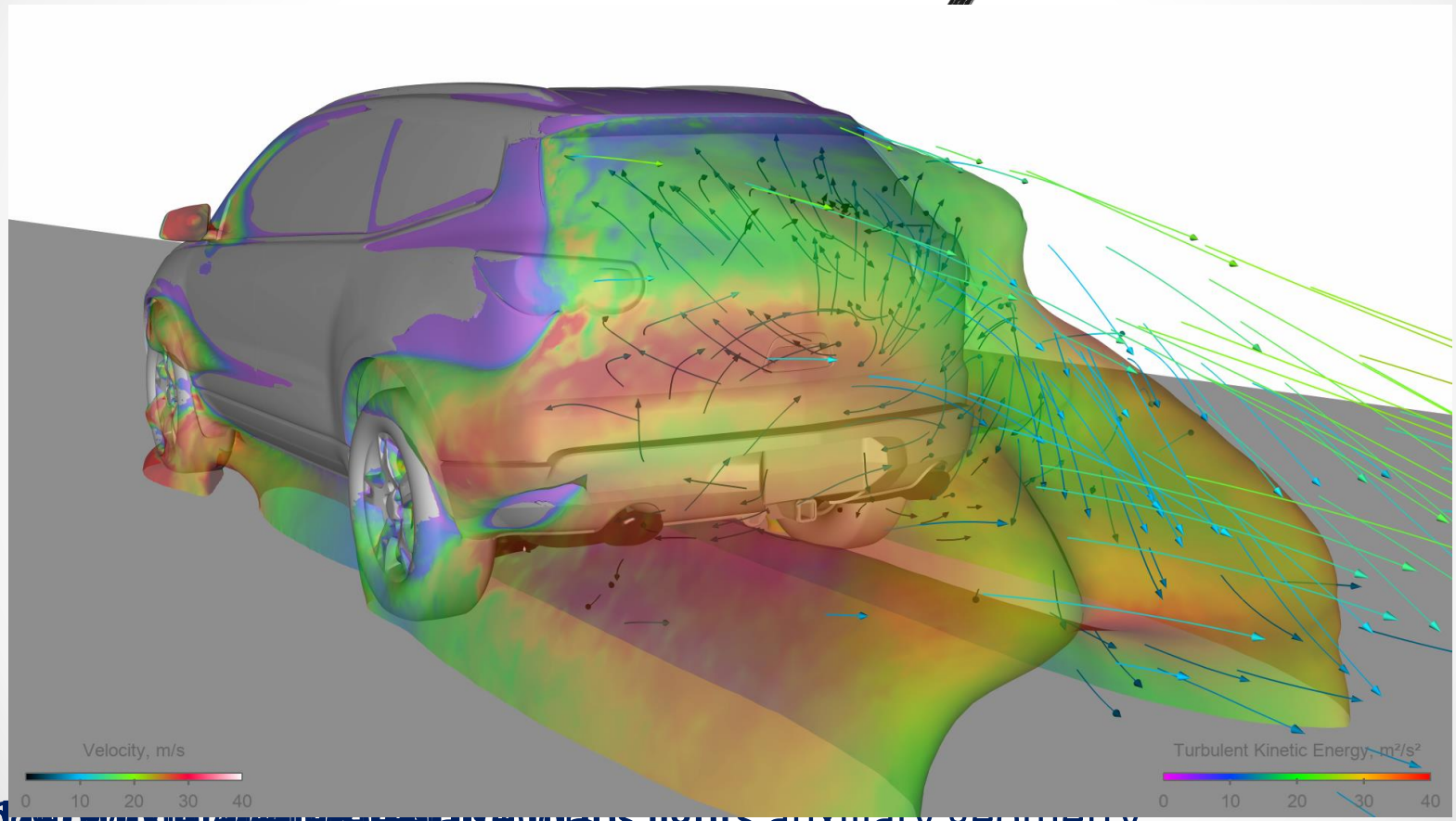


Notes About the Materials Library

- Some notes about using materials:
  - For **Glass** -> **Mirror**, there is only 1 slider for **Specular Reflection** in the **Lighting and Shading** menu; there is no color assigned to a mirror
  - Auxiliary geometry can serve as reference geometry to increase the realism of the scene; click on **Create** -> **Auxiliary Geometry** to select the plane that's needed; the road surface in the image on the right was created using Auxiliary Geometry with a texture map
  - To create even more realistic scenes, use multiple light sources and Ray Tracing; this is discussed in the next 2 chapters
  - For high quality output, use antialiasing and make sure the **Shading** is at least set to **Smooth**; switch off the Model Triad

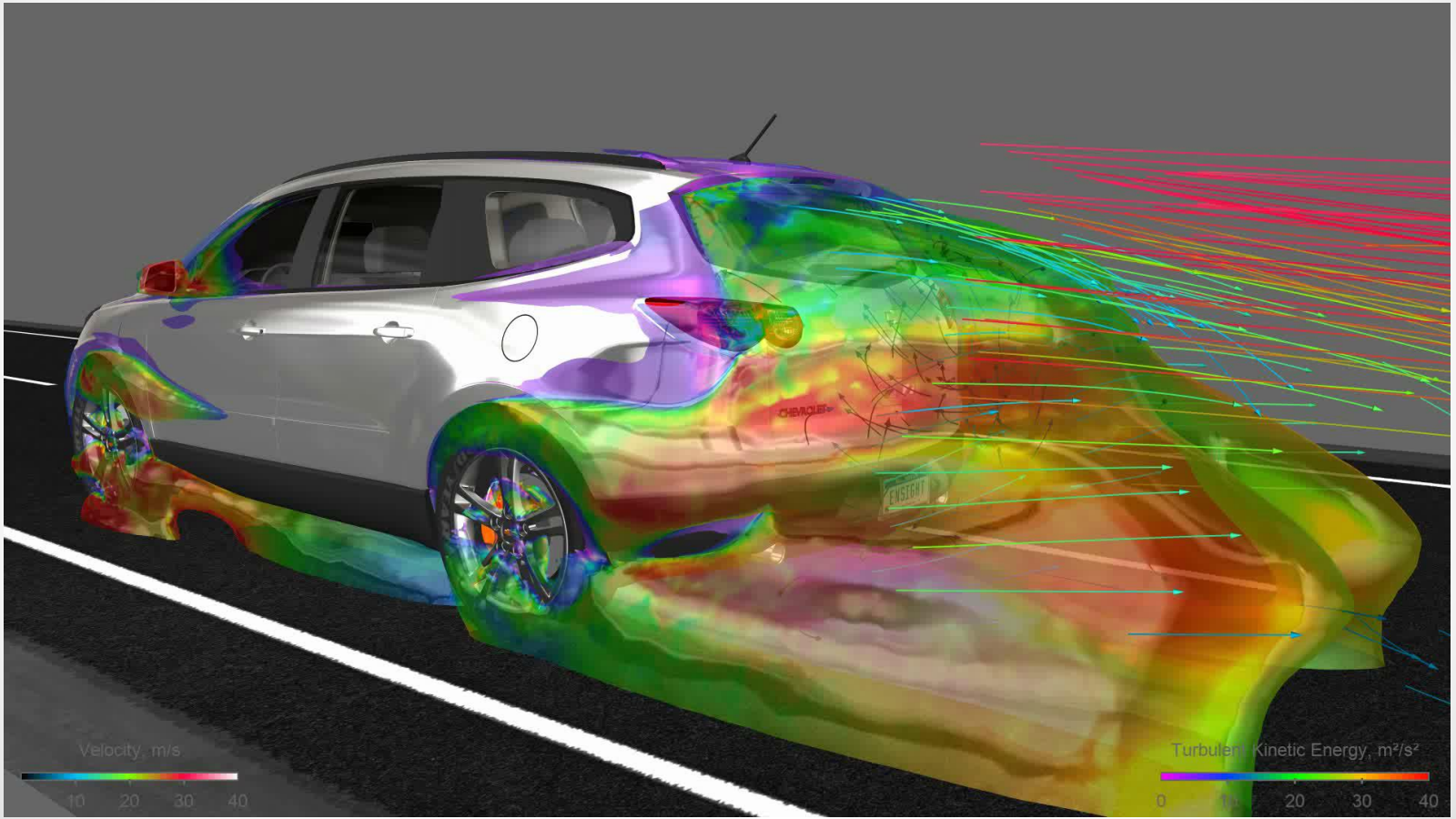


Using Materials on the Chevy Traverse



CFD CAE [illegible] lights auxiliary geometry

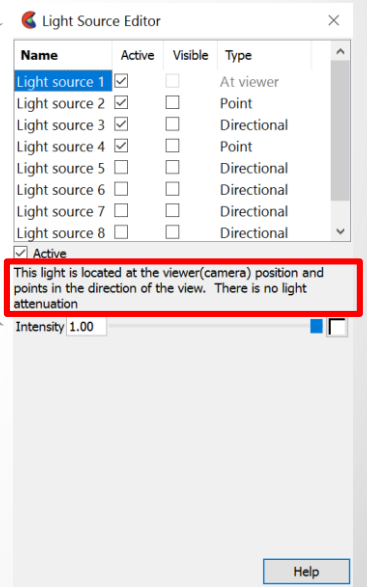
Chevy Traverse Animation



Chapter 12: Using Multiple Light Sources

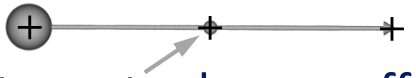
Multiple Light Sources Overview

- EnSight 10.2 Has 7 additional light sources besides the standard light
- Using multiple light sources has a significant effect on the realism and overall visual quality of the model, particularly when Ray Tracing will be used
- There are 3 different types of lights: Directional, Spot and Point
- Each light source can be placed anywhere in 3D space
- Click **View** -> **Lighting** to display the Light Source Editor menu
- The intensity and color of the lights can be controlled
- The first light source is the original light source that has always been present in EnSight; the light is from the user in the direction of the –Z screen axis, towards the model; there is no light attenuation (gradual loss in light intensity) and the type of light can not be modified

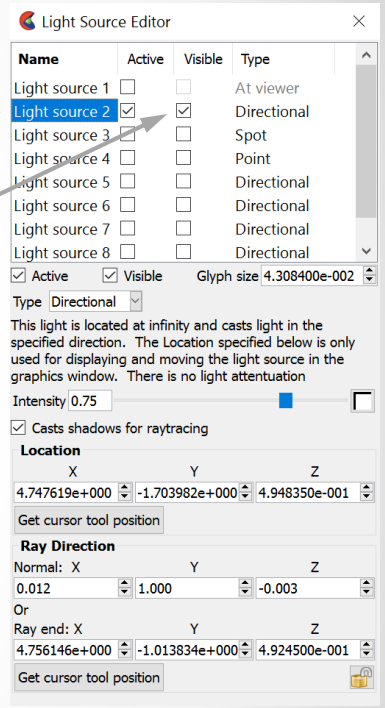


Types of Light 1

- Each of the 7 additional light sources can be active or inactive – use the toggle to switch a light source on or off
- Each light source, except light source 1, has a Glyph (an icon) in the graphics window that indicates its position and orientation; the visibility of a glyph can be toggled on or off; a glyph has 3 hot points: at both ends and at its center
- There are 3 types of light:
  - Directional - the light is located at infinity and points towards its end; there is no light attenuation; the glyph and hot point locations for this light are:



Moving the glyph at its center has no effect on the lighting of the model; only moving its end points changes the lighting effects

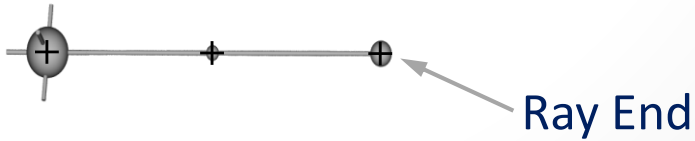


Types of Light 2

- Spot - the light shines towards its point; this light has attenuation; this light type also has a spot angle as well as a falloff angle which defines how blurry the light cone will be; the glyph and hot point locations for this light are:

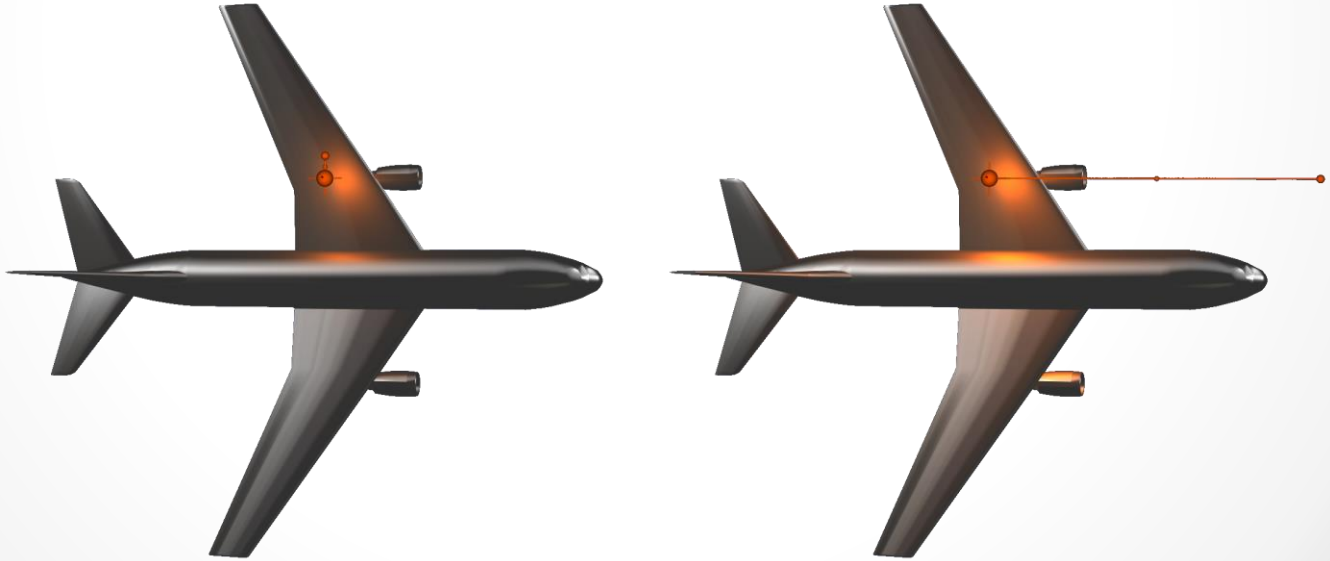
  Moving any of the hot points has an immediate effect on the lighting of the model; use the **Falloff %** slider to control the blurriness of the light; use the **Intensity** slider and the length of the arrow to control the light intensity

- Point - the light shines in all directions and is located at the specified point; this light type has light attenuation; the glyph and hot point locations for this light are:



Types of Light 3

Moving any of the hot points has an immediate effect on the lighting of the model; moving the **Ray End** (the small sphere at the end of the glyph) towards the light source will dim the light – moving it away from the light source will increase the intensity of the light until at a certain distance it's not noticeable anymore; the direction of the light source has no effect on this type of light



Notes About Light Sources

- Using multiple light sources of various types can greatly enhance the overall visual quality of a scene
- Start by switching off all available light sources
- Add the first light source and don’t use the first default light since this is often overpowering and it will make the scene look flat
- Use **Light Source 2** as the first light source to light the overall scene; experiment with the position, color and intensity – click and drag the glyph to various positions to see the effect
- Use additional light sources to fill or back light objects you want to focus on
- When multiple light sources are used, make sure not to oversaturate the scene with too much light – experiment with the intensity slider
- Lighting a scene properly is as much art as it is science; the more experience someone has in lighting, the better the scene will look

Example of Using Multiple Light Sources

Adding 1 Spotlight and 1 Directional light

Default Light vs 3 Light Sources

Default light

3 Light sources, no default light

Lockheed Martin F-35
Joint Strike Fighter with
F-16 in the background

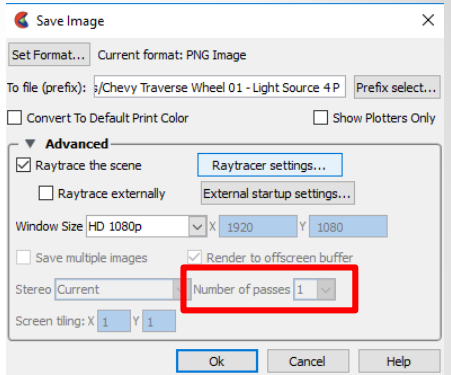
Chapter 13: Ray Tracing

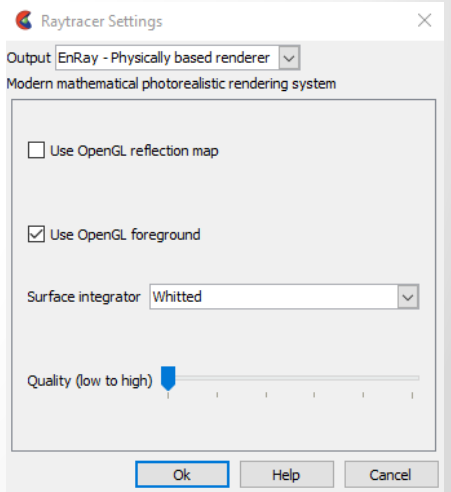
Raytracing Basics

- Raytracing is an advanced rendering technique that traces the path of light and simulates the effects of its encounters with virtual objects
- This technique is capable of creating highly realistic images; however these types of images have to be calculated by the system and this takes time
- Raytracing can accurately create a wide variety of optical effects including multiple lights, shadows and reflections
- EnSight uses a build in raytracer called enRay – this Raytracer can be accessed directly from within EnSight
- The implementation of the Raytracer in terms of the GUI has been kept to a minimum to ensure ease of use
- Depending on the quality of the output and the complexity of the scene, Raytraced images can take several seconds to several minutes to calculate

Creating a Raytraced Image 1

- Click on **File** -> **Export** -> **Image** and click on the **Raytrace The Scene** toggle; this informs EnSight to use the Raytracer to create the image; the anti-aliasing options are disabled since they are automatically set in the Raytracer
- Click the **Raytracer Settings** button to see the available options for the Raytracing process
- The Surface Integrator has 2 options: **Whitted** or **Ambient Occlusion**
  - **Whitted** will produce images with sharp shadow edges that typically occur on a sunny day without any clouds
  - **Ambient Occlusion** will create much softer, blurry shadows that occur on a sunny but slightly cloudy day



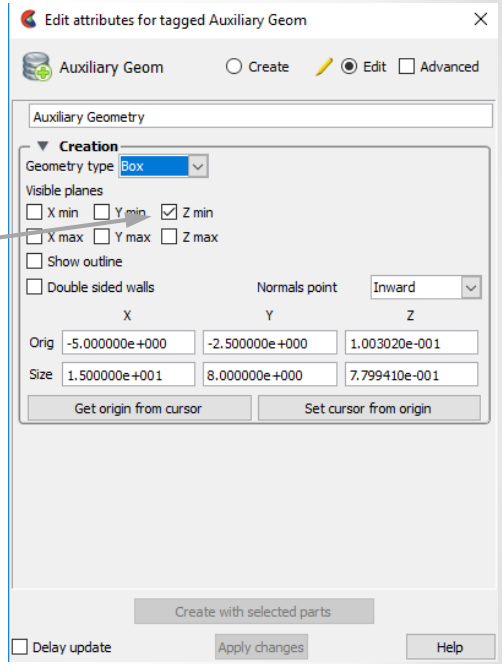


Creating a Raytraced Image 2

- There are 6 Quality settings for both Surface Integrators; on setting 1 (the lowest quality setting) the image on the left takes about 7 seconds to create on a Dell Precision 7710 laptop; the image to the right on quality setting 6 (the highest quality setting) takes about 160 seconds to create using the same laptop

Create Auxiliary Geometry

- Click on **Create** -> **Auxiliary Geometry** to create the road surface for the Chevy Traverse model for instance
- This uses the sides of the bounding box of the model to create up to 6 sides; in this case only the bottom Z-plane of the bounding box is needed so that is selected
- This plane can be used to texture map or to assign a material for instance
- Shadows from the Raytracer will be cast onto the Auxiliary Geometry



Wheel - Whitted Quality 3

- The following image, using Whitted on Quality 3, takes about 20 seconds to generate

Wheel - Ambient Occlusion Quality 3

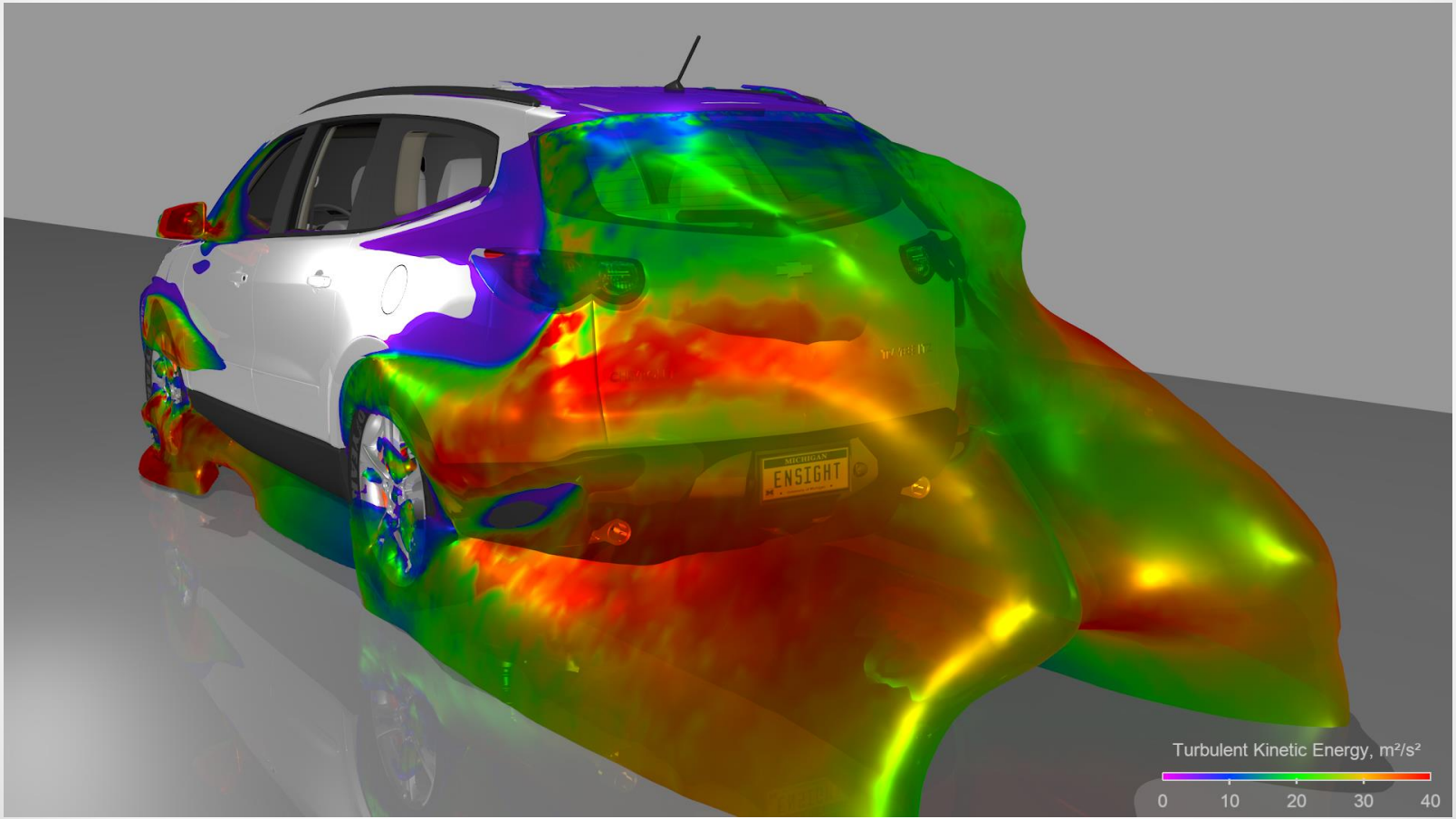
- Here's the same image with Ambient Occlusion on Quality Setting 3 (70 seconds)

Ray Traced Images Comparison

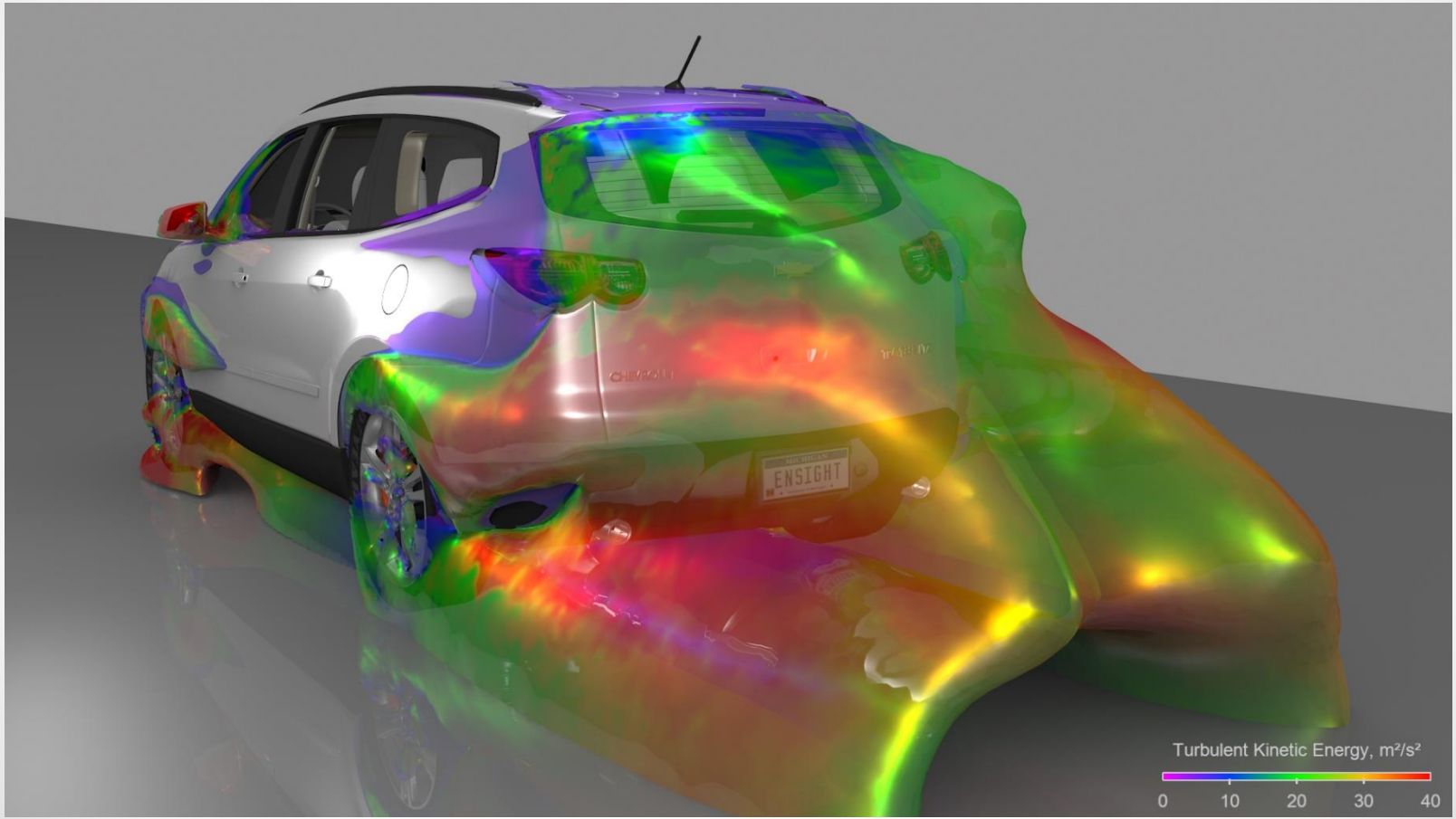
Whitted, Quality 3

Ambient Occlusion, Quality 3

Chevy Traverse - Whitted Quality 3



Chevy Traverse – Ambient Occlusion Quality 3



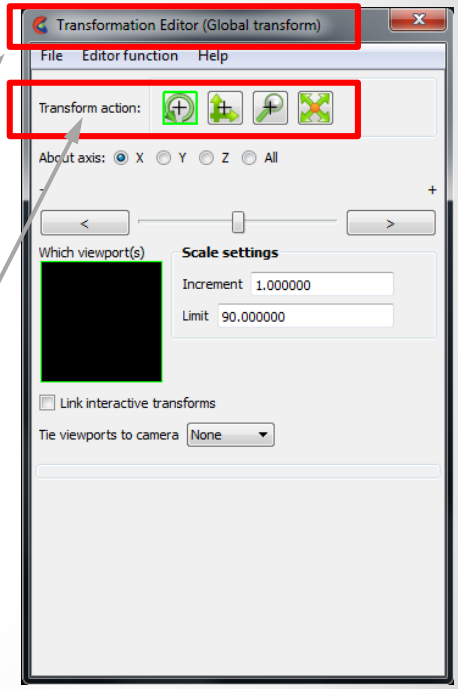
Chapter 14: The Transformation Editor

The Transformation Editor 1

- The **Transformation Editor** can do transformations with exact values; it can also perform several other useful tasks

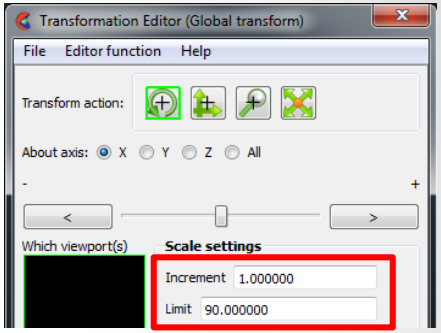
- Click on **Edit** -> **Transformation Editor** and a panel is displayed that has Transformation Editor in the title bar and then in brackets **Global Transform**; the phrase in brackets indicates the current function the Transformation Editor is executing, so in this case it is a global transformation

- There are 4 transformations that can be done: Rotate, Translate, Zoom and Scale around any of the axes

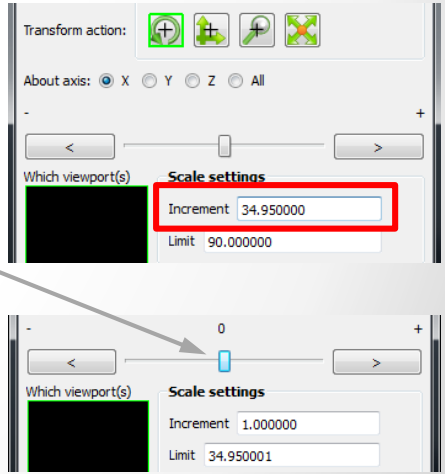


The Transformation Editor 2

- To do a global transformation with exact values, use either the slider, the forward and backward increments or type in a number in the **Increment** or **Limit** field



- For instance, if a rotation is needed around the X-axis with an angle of 34.95°, type in 34.95 in the **Increment** field and click the forward or backward button or type in 34.95 in the **Limit** field and move the slider to its limit in either positive or negative direction; the total angle will be + or - 34.95°

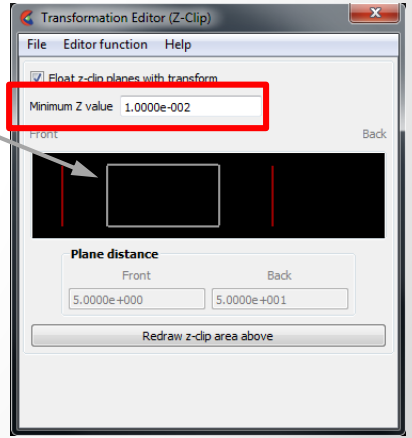


The Transformation Editor 3

- When parts are zoomed in very closely, the model can progressively disappear; this is called Z-clipping; there are 2 Z-clip planes: the front and back planes and they are in the Z-axes of the display so coming out and going into the screen

- The Z-clipping in EnSight can be controlled by clicking on **Editor Functions** -> **Z-Clip** to display a panel that shows the Front and Back Z-clip planes (the red lines); the white box represents a bounding box of the geometry

- By default the Z-clip buffers float with any transformation; however, if Z-clipping occurs anyway, the user can modify the **Minimum Z-Value** by making it smaller or larger



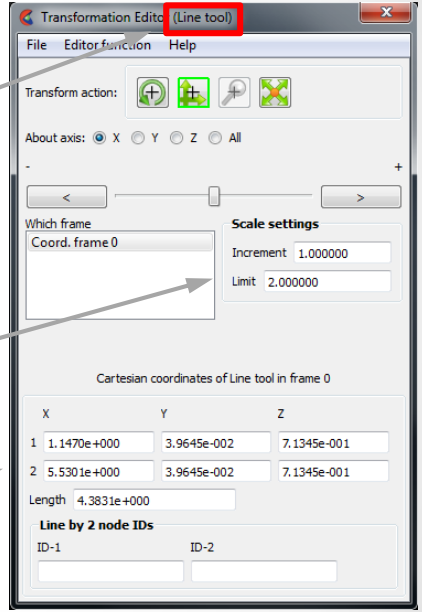
The Transformation Editor 4

- The tools in EnSight (Cursor, Line and Plane) can also be manipulated using the Transformation Editor; click on **Editor Functions** -> **Tools** -> **Line** and the following panel is displayed; please note that the title bar displays **Line Tool** in brackets

- The transformation tools on this panel can be used to Rotate, Translate and Scale the Line Tool by exact values

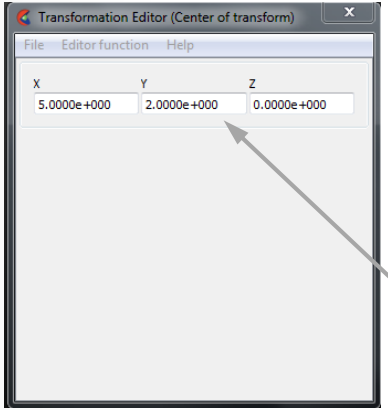
- For the Line Tool, this panel lists the start and end points

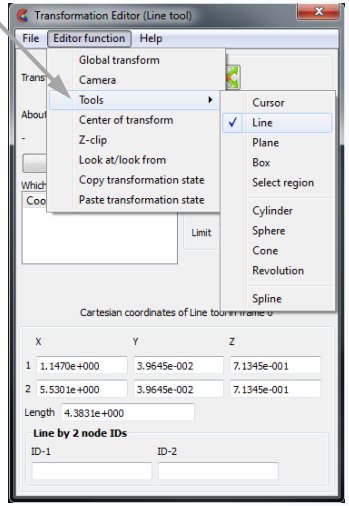
- The Line Tool can also be positioned using 2 Node ID's



The Transformation Editor 5

- A similar panel is available for all other tools (Cursor, Plane, Box and the quadratic tools as well)

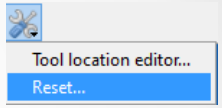




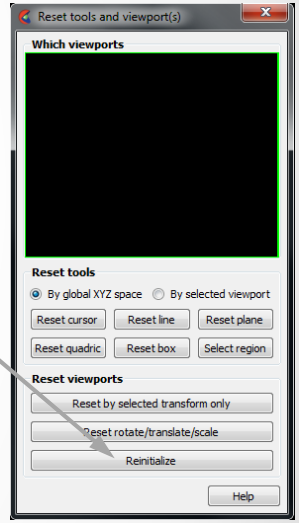
- Click on **Editor Function** -> **Center of Transform** to display or <u>set</u> the coordinates of the center around which the model transforms; this can also be done by clicking the Pick icon and selecting ‘Pick Center of Transform’ and then selecting a node of a part using the cursor and the P key on the keyboard

The Transformation Editor 6

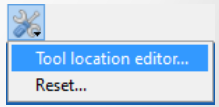
- Click the Tool Location Settings icon and then **Reset** to display a menu that allows tools and viewports to be reset



- When the **Reinitialize** button is clicked, the center of transform is modified to the geometric center of the visible parts on the screen

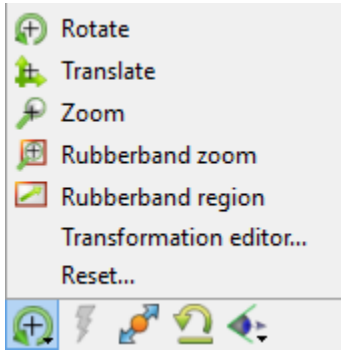


- Clicking the Tool Location Settings icon and then **Tool Location Editor** brings up the **Transformation Editor** displaying one of the **Tool** menus



The Transformation Editor 7

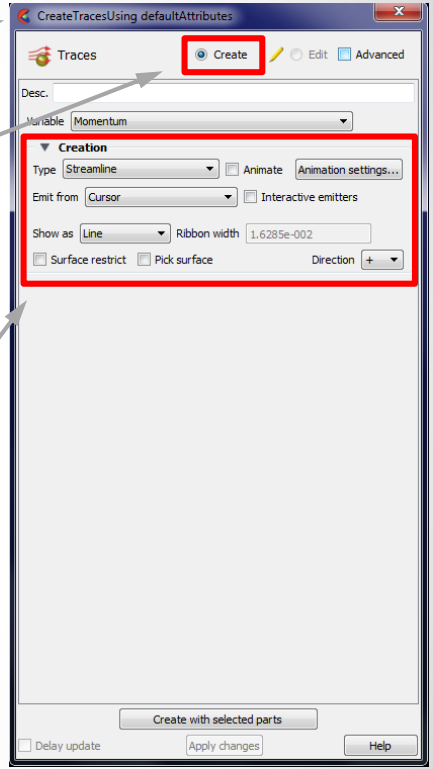
- Another method of displaying the Transformation Editor is to click the Graphics Window Transforms icon and then select Transformation Editor; the system will display the Transformation Editor in the Global Transform mode



Chapter 15: Create and Edit Attributes (Parts)

Create Attributes (Parts) 1

- When a function on the Feature Toolbar is clicked, for instance Particle Traces in this example, EnSight will display the basic Create Traces menu
- Please note this menu is to create a new particle trace part as can be seen by the **Create** selection box
- This menu displays the basic creation features that are available for this operation and are displayed under the **Creation** toggle



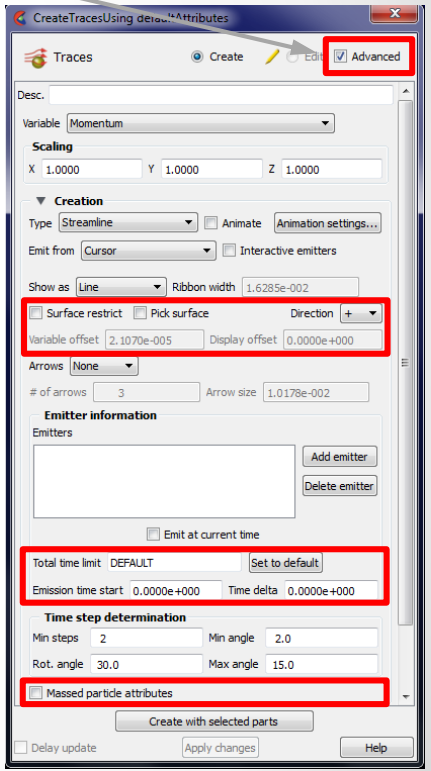
Create Attributes (Parts) 2

- For more advanced features, click on the **Advanced** toggle
- The system displays several more options to create particle traces

- For instance, for **Surface Restricted** particles, a **Variable Offset** and a **Display Offset** can be specified

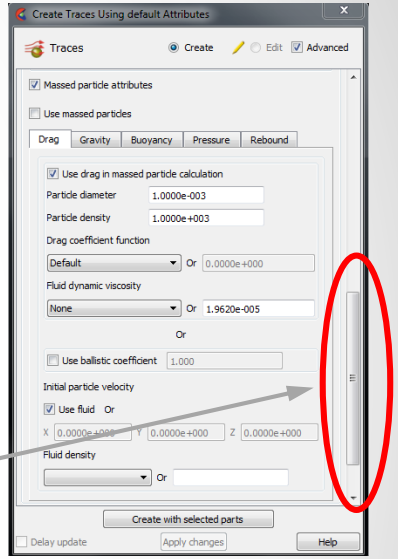
- For pathlines the **Total Time Limit**, **Emission Time Start** and **Time Delta** can be entered on this menu

- At the bottom of this panel there's a toggle called **Massed Particles**; when this toggle is switched on, a new panel will be displayed that allows particle traces to have a mass (regular particle traces are massless)

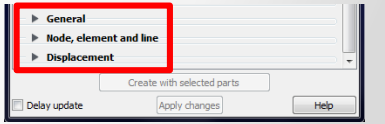


Create Attributes (Parts) 3

- The **Massed Particle Attributes** panel has several tabs that specify how to define the massed particles; see the User Manual for more information about the mathematical background of massed particles



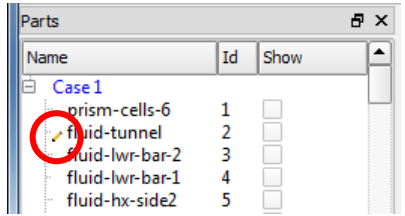
- Scroll down on the Edit Attributes menu and click the other toggles on the Edit Attributes menu to modify **General Attributes**, **Node**, **Element & Line Attributes** and **Displacement Attributes;** for some parts it will also display **IJK Axis Display Attributes**

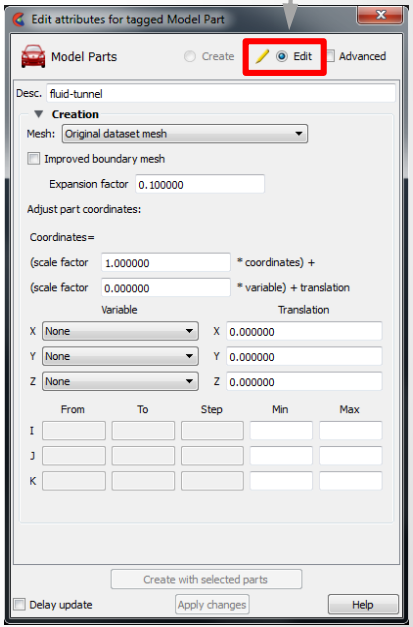


Edit Attributes (Parts) 1

- When a part in the Parts List is double clicked, the Edit Attributes menu is displayed; this menu is very similar to the Create Attributes menu but on this menu the **Edit** selection box is active

- Please note that in the Parts List the part that is being edited has the icon in front of the name to indicate this is the part that is being edited; even if another part is selected, the icon will stay in front of the part that was edited last



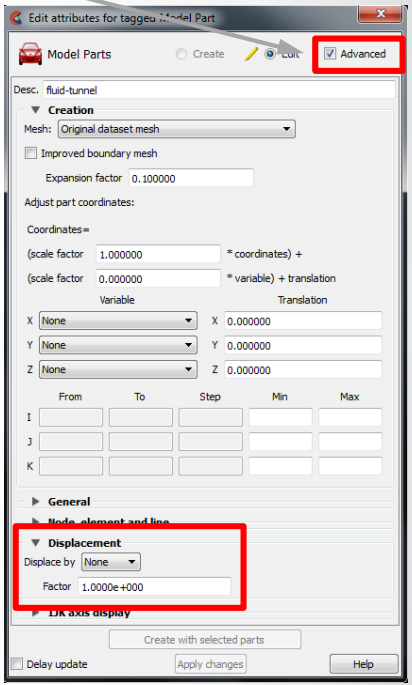


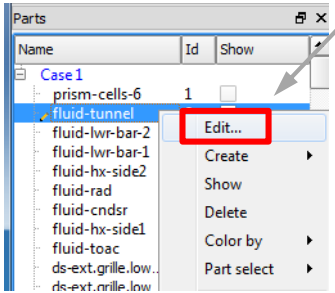
Edit Attributes (Parts) 2

- Toggle the **Advanced** button and EnSight will display several more options

- Click for instance on the **Displacement** toggle to display several options for this feature; check out the others as well (**General**, **Node** & **IJK**)

- The Edit Attributes menu can also be displayed by right clicking on a part in the Parts List and selecting **Edit**





Chapter 16: Case Linking

Case Linking

- As an extension to EnSight's ability to Load Multiple Datasets at once, Case Linking attempts to provide an easier manipulation of the analysis of multiple datasets
  - Operations performed on one case are automatically done on all other open datasets
  - Synchronization of part creation, coloring, queries across all of the cases
  - EnSight 10.1 has introduced this capability (version 1)

- Make Comparisons between cases easier

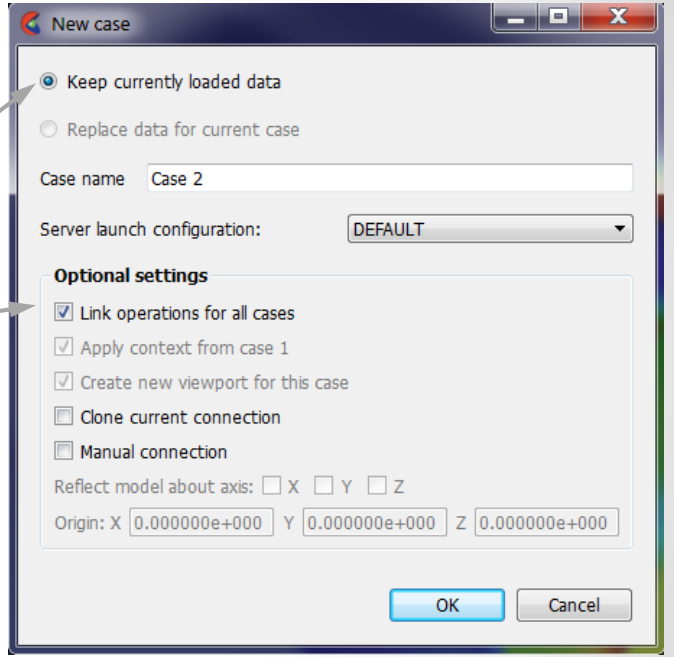
Case Linking Requirements and Caveats

- Current Case Linking Requirements:
  - The Parts List for all cases must be identical in number and order
  - The Variable list for all cases must be identical and comparable
  - Must be turned on when you load 2nd case

- Caveats:
  - Only 4 datasets can be case linked
  - Once turned Off, case linking cannot be turned back on (no catch up)
  - Uses context capability – whatever limitations on context files will apply to case linking

Case Linking – How To

- Load up for first dataset, and perform as many operations as you'd like
- Upon Loading the 2nd case
  - Select to **Keep currently loaded data**
  - Select **Link Operations for All Cases**
- The new case is loaded and all subsequent operations are linked to the other datasets
- All else is take care of by EnSight - there is no other user involvement in Case Linking

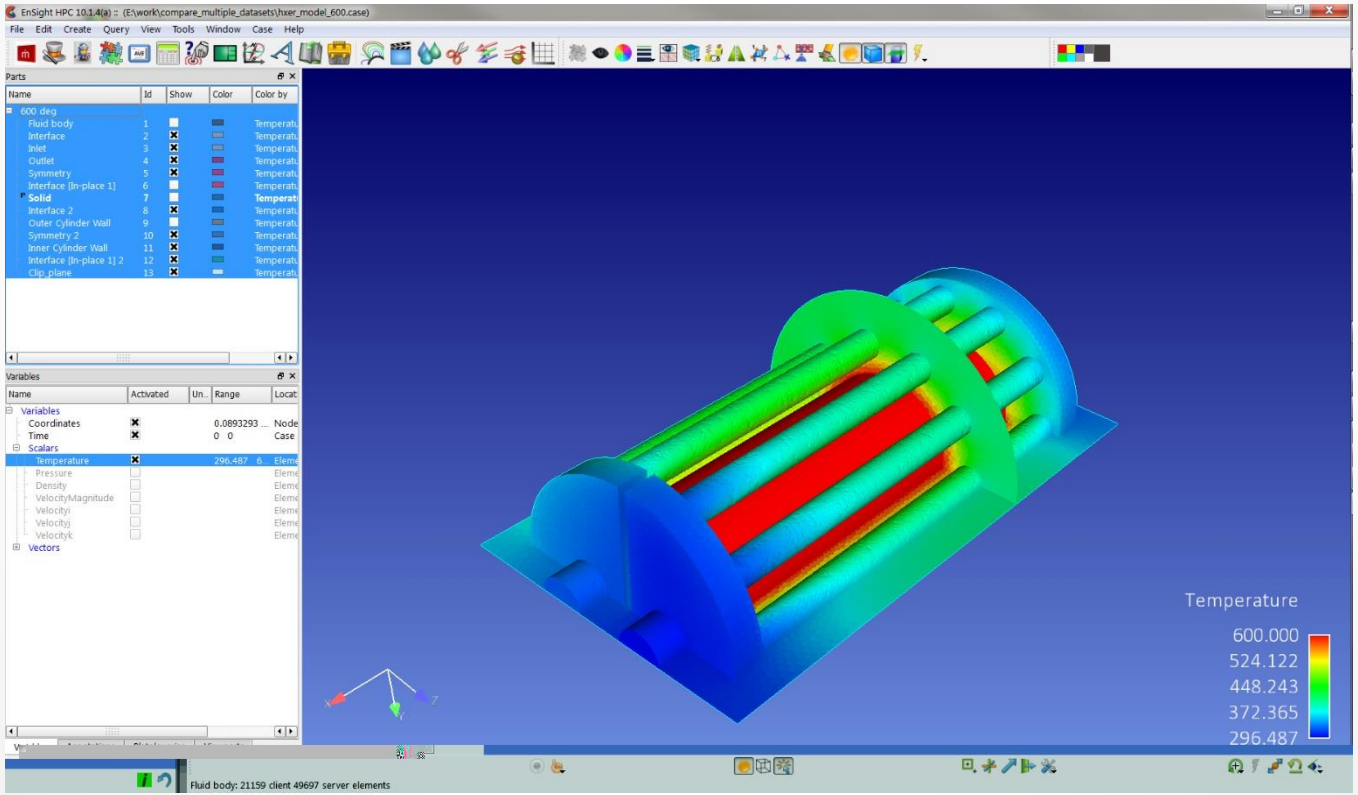


Case Linking – Operation 1

- In Case Linking mode:
  - Separate Viewports for each case are made
    - Viewports are linked for transformations
  - Part Edits and creations are done across all cases
  - Variables are created across all cases
    - Constants are created for each case as well
  - Queries are created across all cases
  - Interactive Probes are performed across all cases

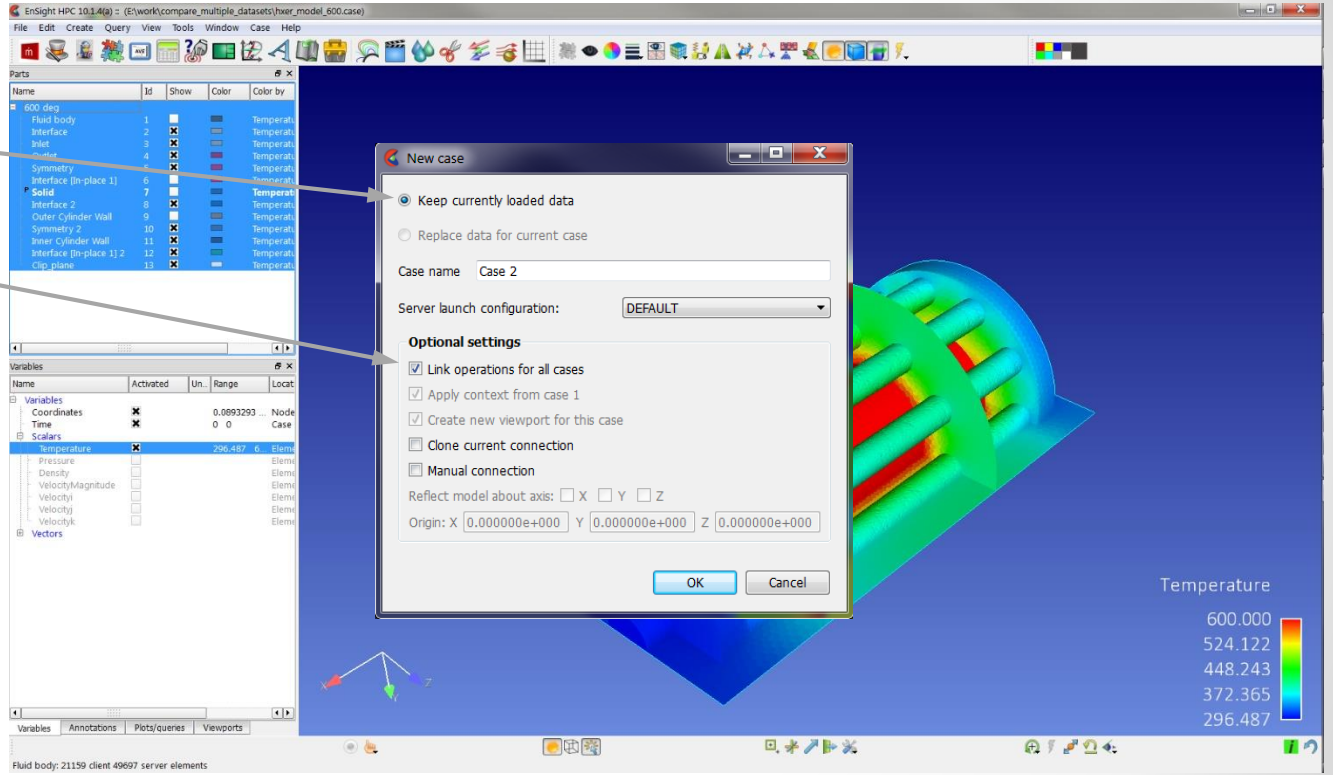
Case Linking – Operation 2

- Load the first model, and perform the operations you desire



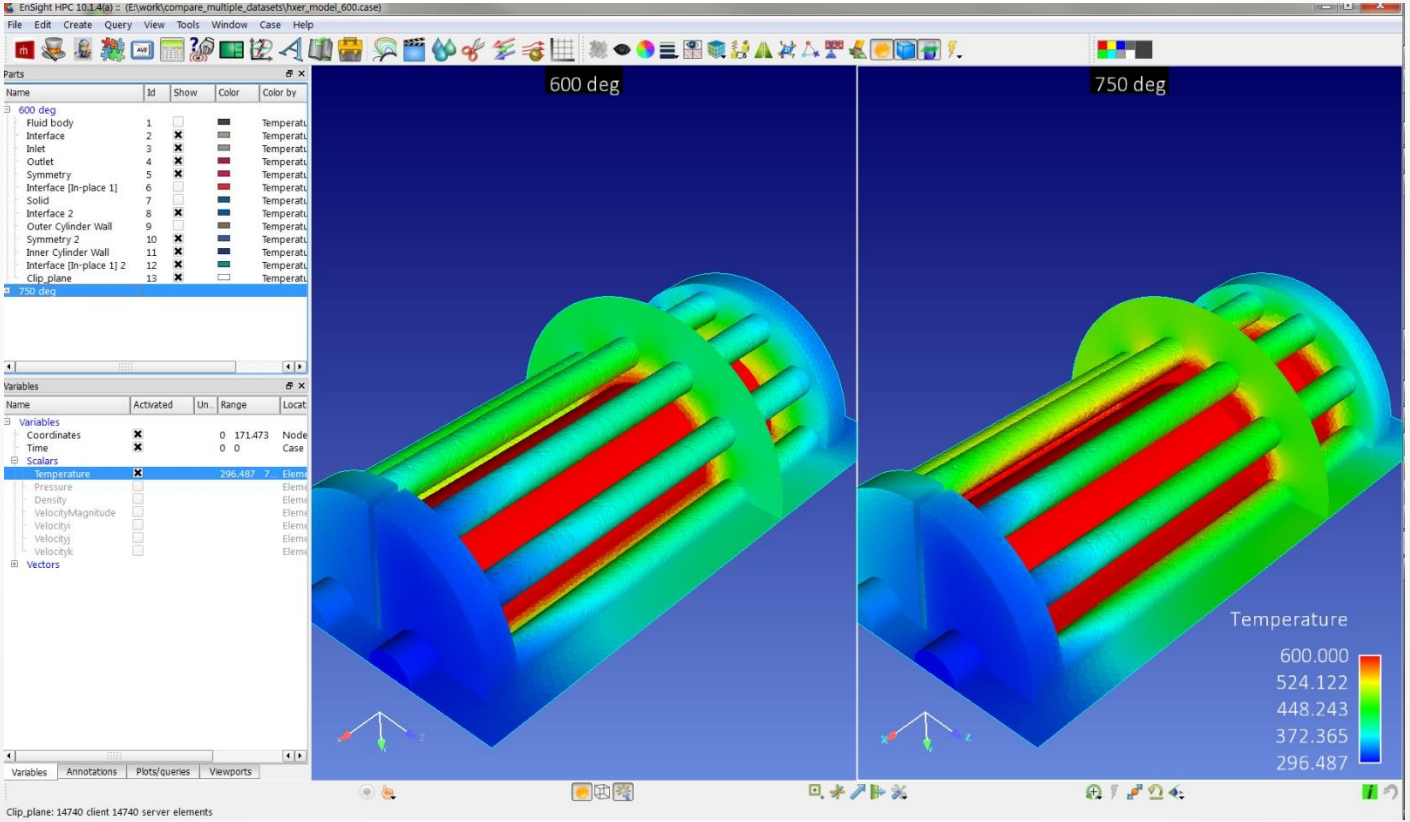
Case Linking – Operation 3

- Load the second case, choosing to (a) keep currently loaded data, and (b) Link operations



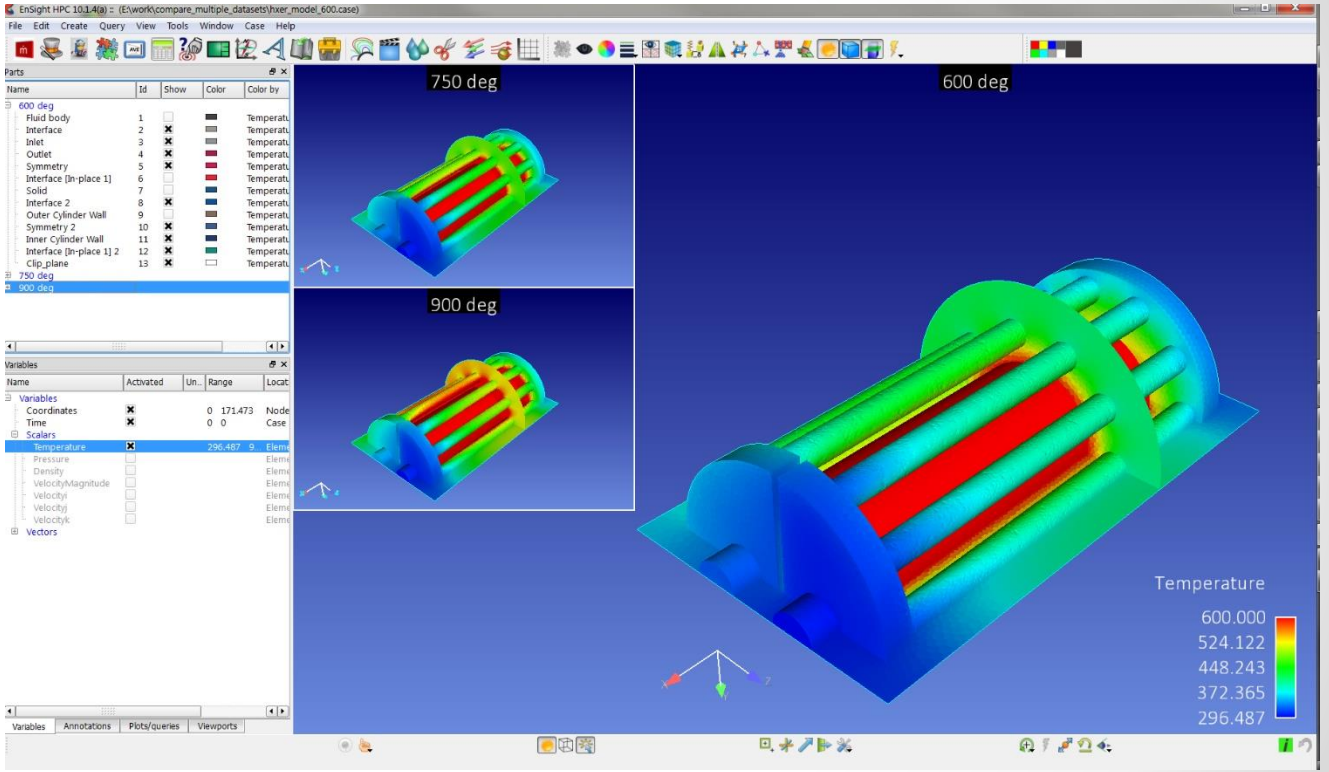
Case Linking – Operation 4

- Second Case is loaded, and all operations are automatically performed



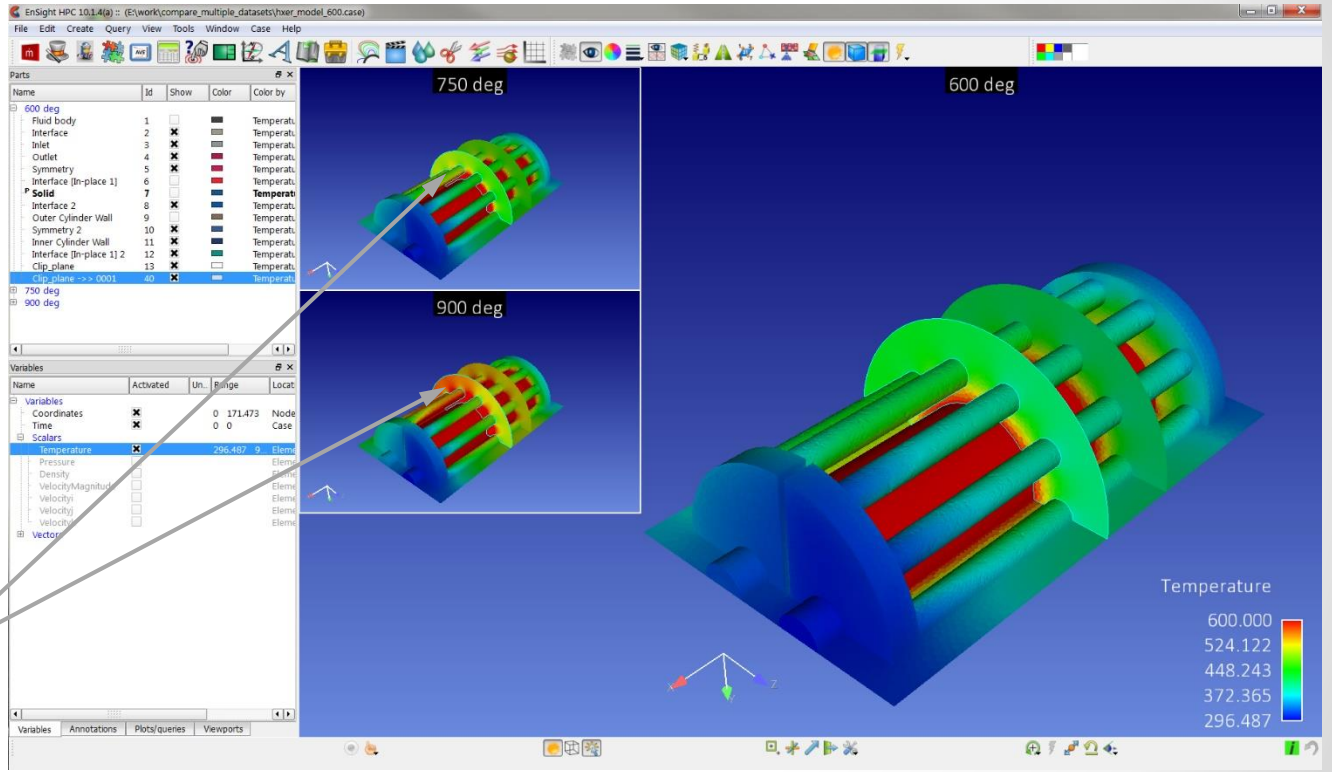
Case Linking – Operation 5

- As subsequent cases are loaded, operations are automatically performed



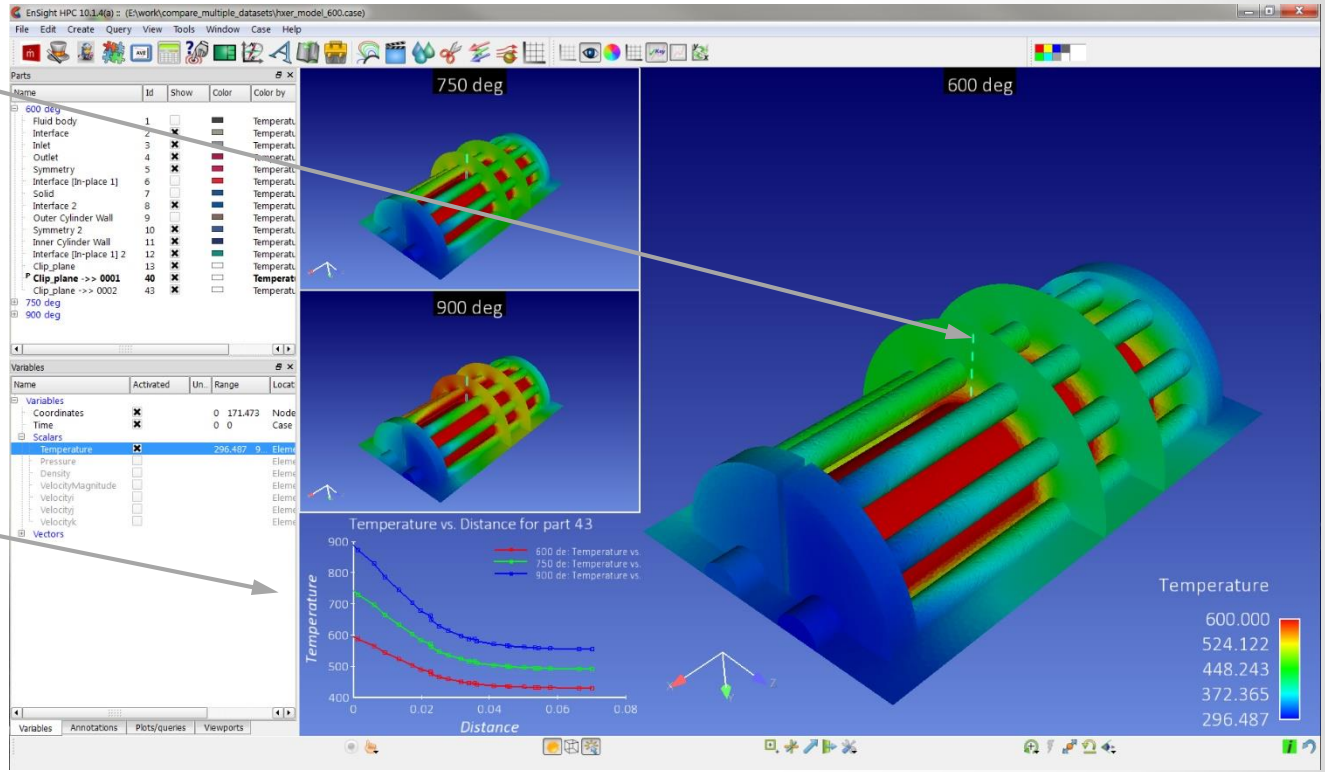
Case Linking – Operation 6

- Now, as you create new parts in a single case, that operation is automatically done on all other cases
- The second clip plane is automatically created & colored



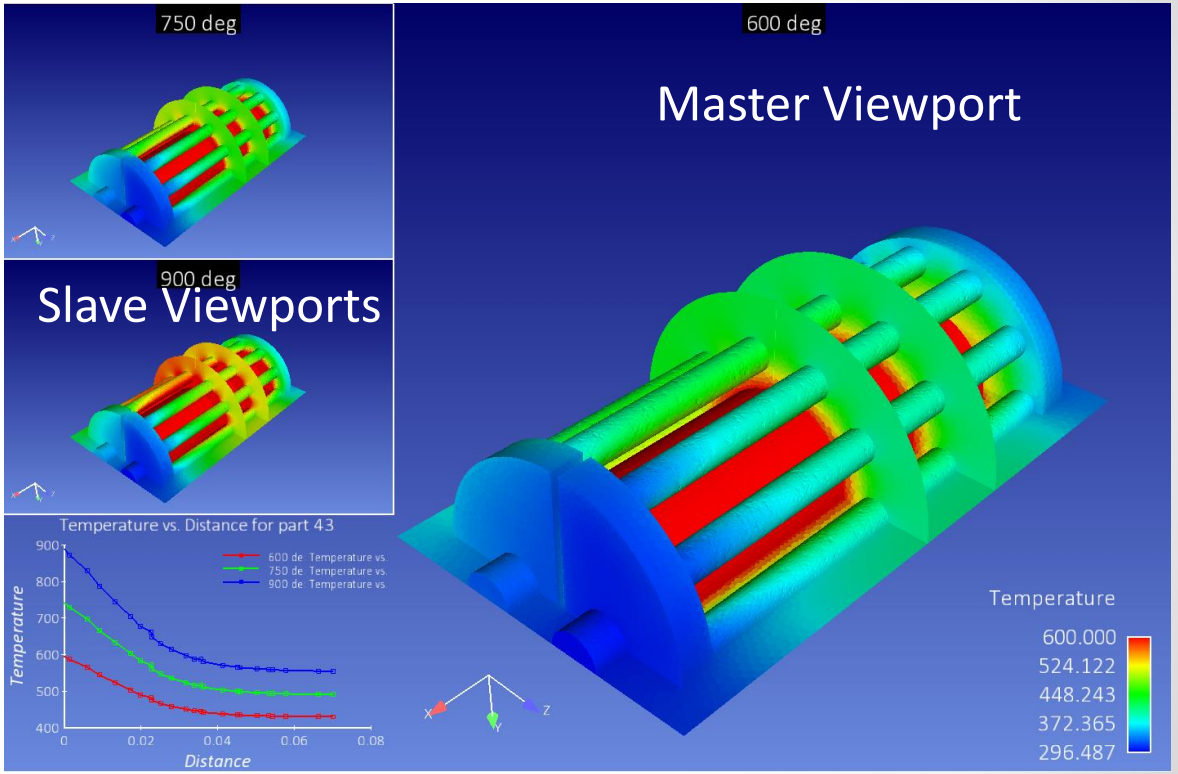
Case Linking – Operation 7

- Creating a single Query in one model
- The queries are automatically created in the other models, and added together into the graph automatically



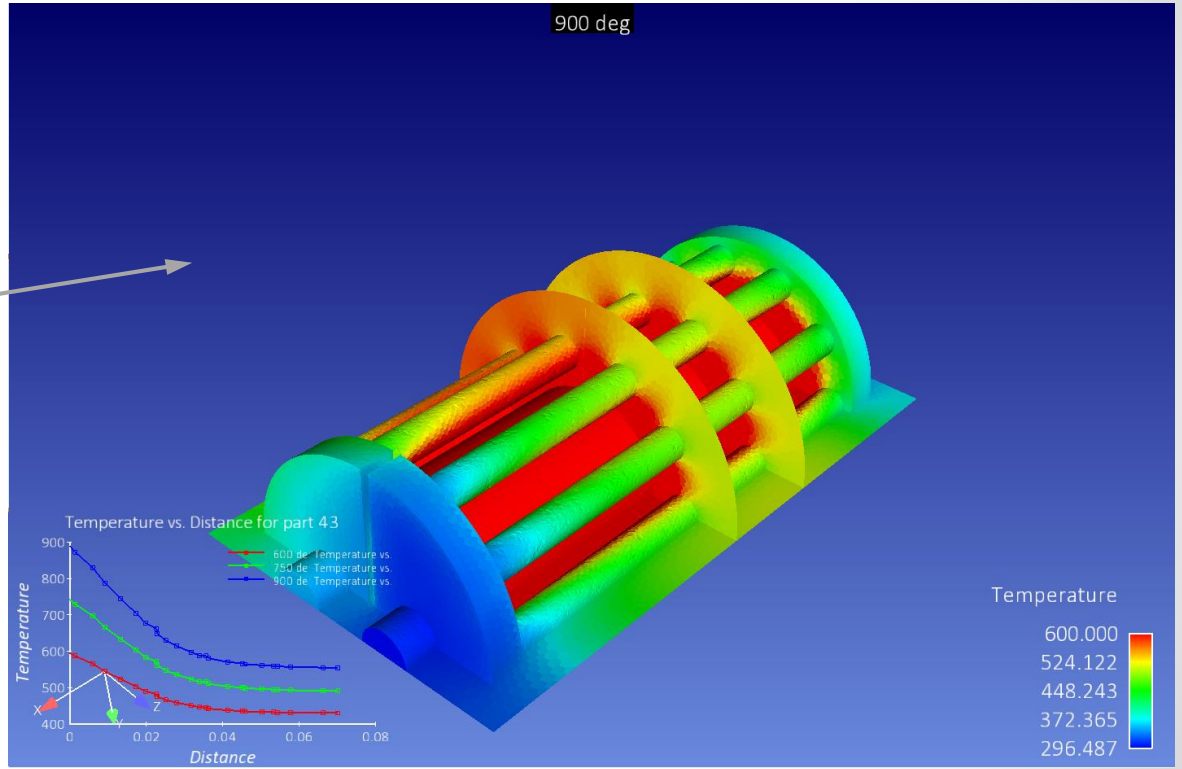
Case Linking – Viewport Capabilities

- While in Case Linking mode, there is a ‘master viewport’ and ‘slave viewports’
- There are two new functions with regards to viewport operation while in Case Linking Mode



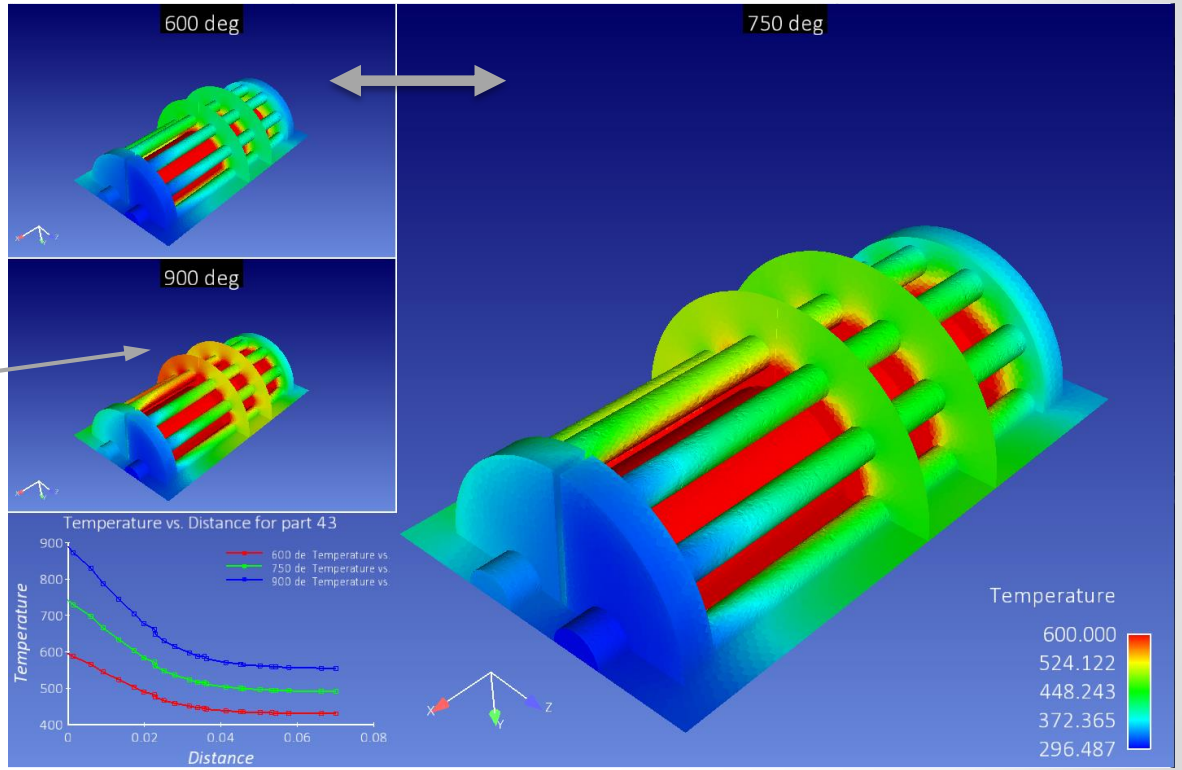
Case Linking – Viewport Toggle Full View

- Left Click in the viewport to bring up the selection handles
- Click on the handle at the top/middle of the viewport
- This toggles this viewport to be Full View
- Same operation to return to its original position



Case Linking – Viewport Exchange

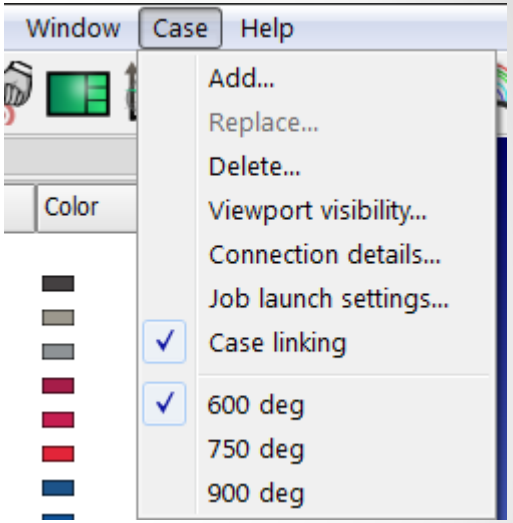
- You can Exchange the current viewport with the largest viewport
- Right Click in a Viewport
- Choose **Exchange with Largest Viewport**



Case Linking - Off

- To turn OFF Case Linking, simple toggle off under the main menu: Case -- > Case Linking

- Once you have turned it Off, you cannot re-activate it



Case Linking - Notes

- Saving Journal Files, Context Files, Session Files, should all work* with Case Linking
- Restoring from these files will also maintain Case Linking
- Improvements in Case Linking, expansion of capability, and removal of limitations is planned for future work

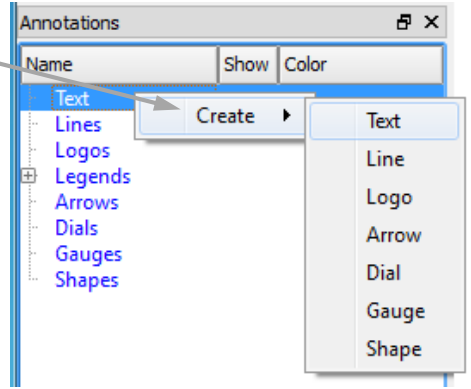
* Context and Session Files do not always restore, but this is independent of Case Linking

Caterpillar 797F Mining Truck

Chapter 17: More Types of Annotations

Create/Edit Annotations

- EnSight can create any annotation by right clicking in the Annotations Objects List and selecting any of the 7 annotations that are listed



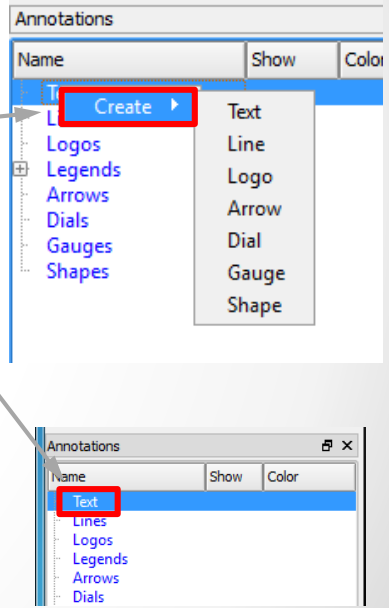
- EnSight will create the selected annotation immediately; please note that 2D annotations overlay the graphics window and are not associated with any viewport
- When a new text is created, the Create/Edit Annotation (Text) menu is displayed as well

Create/Edit Text Strings 1

- Text strings can be created in 2 ways:
  1. By right clicking in the background of the graphics window and selecting **Quick Text** (see the 'Annotations' chapter in the EnSight 10 Basic Training)

  2. By right clicking the **Text** keyword in the Annotations Tab of the Object Lists and selecting **Create** -> **Text**

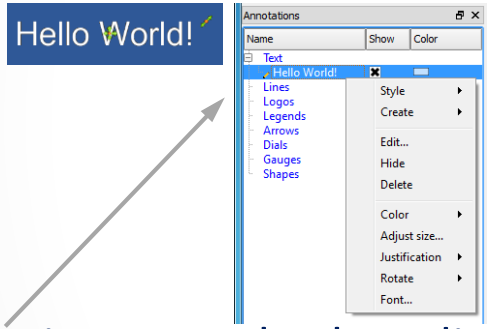
     By double clicking the **Text** keyword in the Annotations Tab of the Object Lists, the Create/Edit Annotations (Text) menu is displayed; click the **Create** button to create a text

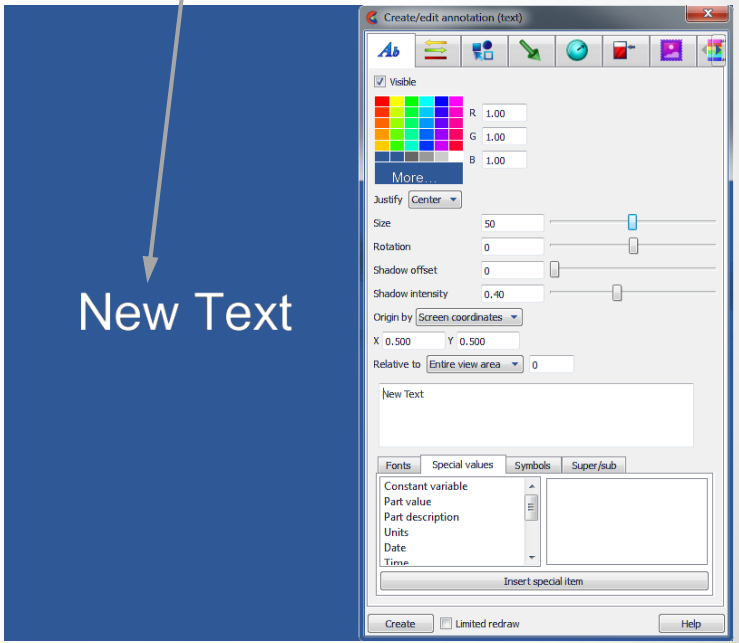


- Dynamic text that changes over time can be created as well

Create/Edit Text Strings 2

- In all text creation methods, the Create/Edit Annotation (text) menu is displayed and a ‘New Text’ annotation is created in the location of the mouse when the right mouse button was clicked or in the center of the graphics window; this menu is used to both create and edit text annotations



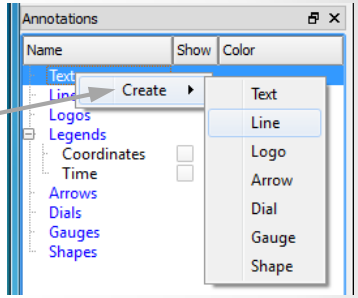


- Text annotations can also be edited by right clicking on an annotation in the graphics window or in the text in the Annotation Tab of the Object Lists

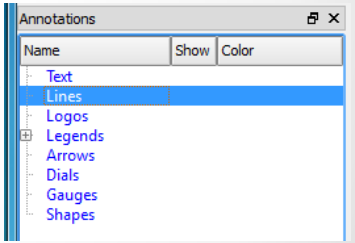
Create/Edit Lines 1

- Lines can be created in 2 ways:

1. By right clicking in the background of the graphics window and selecting **Quick Line** (see the 'Annotations' chapter in the EnSight 10 Basic Training)



2. By right clicking one of the Annotations keywords in the Annotations Tab of the Object Lists and selecting **Create** > **Line**



3. By double clicking the **Line** keyword in the Annotations Tab of the Object Lists, the Create/Edit Annotations (Lines) menu is displayed; click the **Create** button to create a text

Create/Edit Lines 2

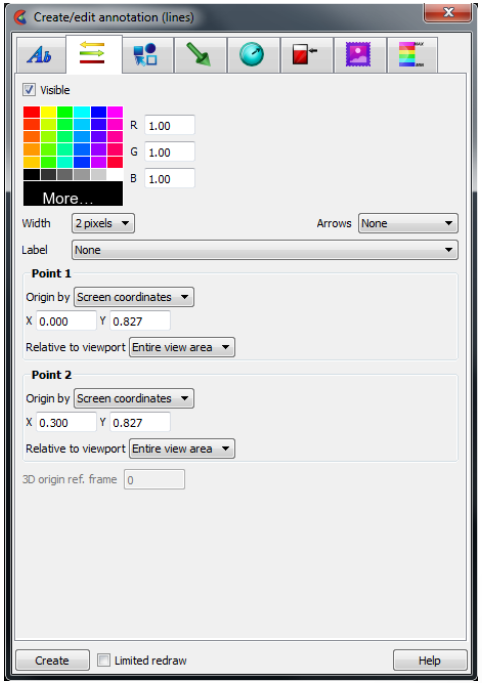
- EnSight can display annotation lines that can either be specified in 2D screen space or in 3D world space

Select a color for the line

Select **Width**

Attach a previously made text to the line

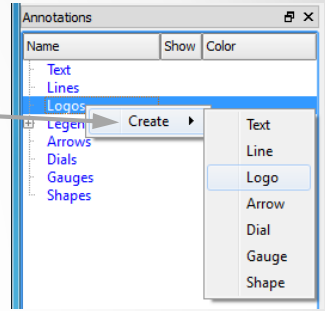
Select **Origin** settings for **Point 1** and **Point 2** in either **Screen Coordinates** or **3D Coordinates**



Select **Arrows**

Create/Edit Logos

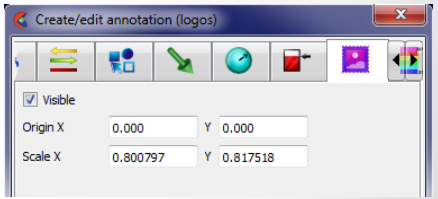
- EnSight can display a bitmap file such as a logo Display a logo on the screen by right clicking one of the Annotations keywords in the Annotations Tab of the Object Lists and click **Create** -> **Logos**, then select a logo bitmap file



- Right click on the logo and select **Edit** to display the Create/Edit Annotation (Logo) menu or right click the **Logos** keyword in the Annotations Objects List or double click the **Logos** keyword

Specify the **Origin** of the logo

Specify the **Scale** of the logo



- A logo has move and resize handles just like any annotation
- Bitmaps are drawn over all geometric objects in the Graphics Window but under all other annotation entities

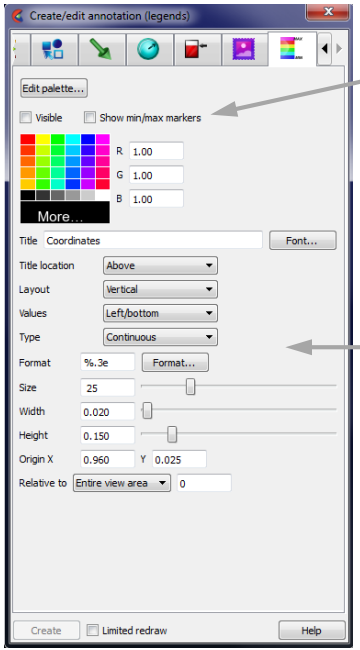
Create/Edit Legends

- Right click the **Legends** keyword in the Annotations Objects List or double click the **Legends** keyword
- The size and position of the color legend can be changed and the legend itself can be edited as well

Click **Edit Palette** to modify the colors and range of the legend

Select the color of the legend

Edit the **Title** of the legend



Toggle **Show Min/Max Markers** on or off

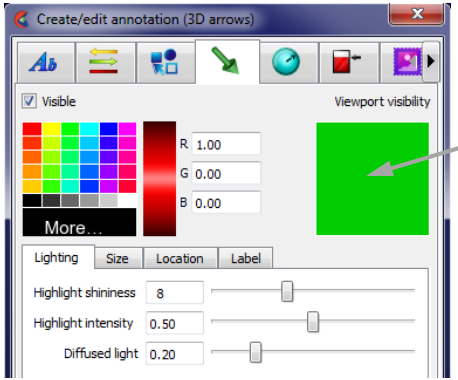
Edit many details of the legend such as **Layout**, **Title**, **Values**, **Value Format**, **Type**, **Font Size**, **Font**, **Origin**, **Width** and **Height**

Create/Edit 3D Arrows with Text

- The 3D arrow is defined in model space and transforms with the scene
- Display an arrow on the screen by right clicking one of the Annotations keywords in the Annotations Tab of the Object Lists and clicking **Create** -> **Arrow**
- Right click on the shape to display the Create/Edit Annotation (Shapes) menu or right click the **Shapes** keyword in the Annotations Objects List or double click the **Shapes** keyword

Select color of the 3D Arrow

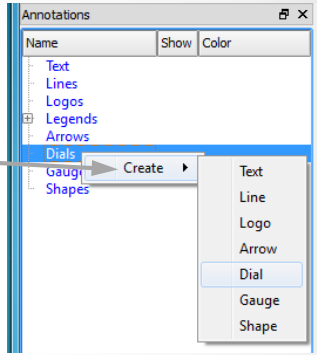
Click the 4 tabs (**Lighting**, **Size**, **Location** and **Label**) to specify more details of the 3D Arrow and text



The 3D arrow is visible in the specified viewport(s)

Create/Edit Dials 1

- EnSight can display a 2D dial (such as a clock to measure time) tied to a constant variable
- Create a dial by right clicking one of the Annotations keywords in the Annotations Tab of the Object Lists and selecting **Create** -> **Dial**

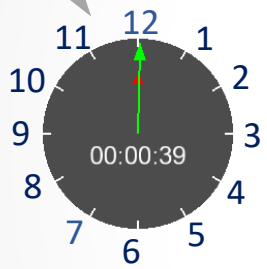


- A dial is created in the center of the screen by default

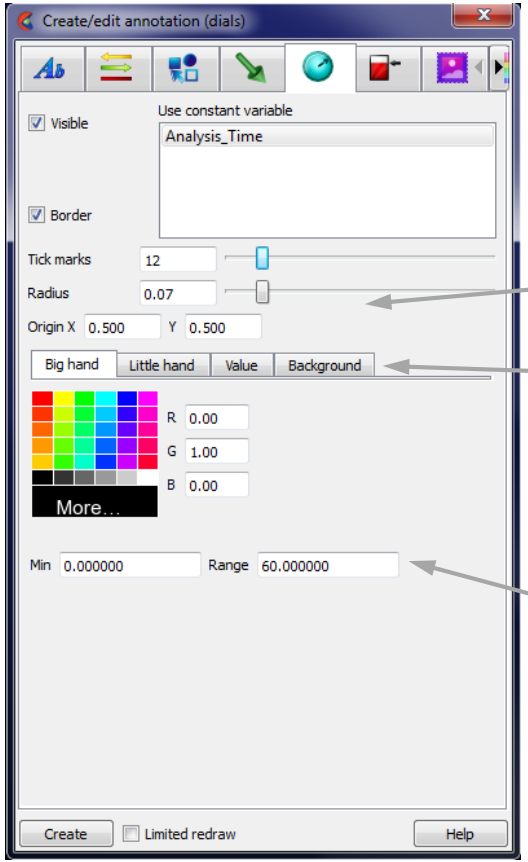
- Right click on the dial and select **Edit** to display the Create/Edit Annotation (Dials) menu or right click the **Dials** keyword and select **Edit** in the Annotations Objects List or double click the **Dials** keyword

Create/Edit Dials 2

Enter the number of **Tick Marks** for the dial; in this example there are 12



Select color of the dial



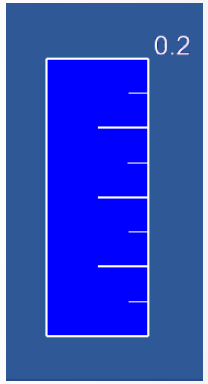
Specify the **Radius** and **Origin**

Click the 4 options (**Big Hand**, **Little Hand**, **Value** and **Background**) to specify more details of the dial

Enter the **Min**imum value and the **Range** of the dial

Create/Edit Bar Gauges 1

- Similar to a dial, EnSight can display a 2D gauge tied to a constant variable
- Create a gauge by right clicking one of the Annotations keywords in the Annotations Tab of the Object Lists and selecting **Create** -> **Gauge**
- A gauge is created in the center of the screen by default



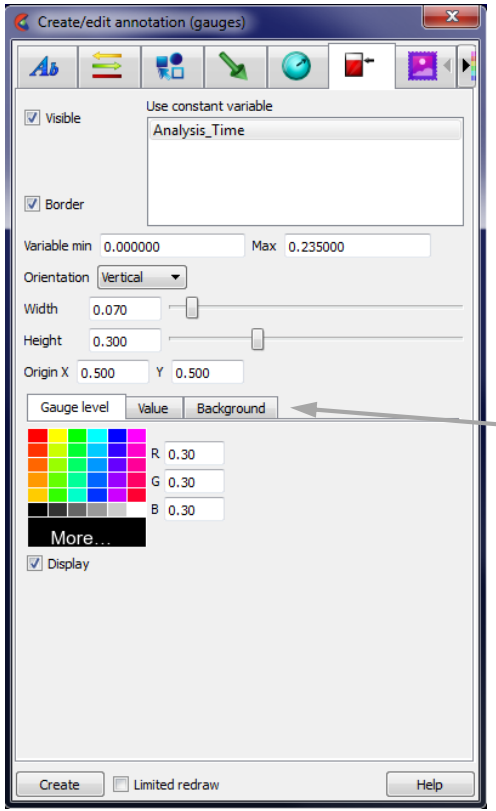
- Right click on the gauge and select **Edit** to display the Create/Edit Annotation (Gauges) menu or right click the **Gauges** keyword and select **Edit** in the Annotations Objects List or double click the **Gauges** keyword

Create/Edit Bar Gauges 2

Select the **Orientation** of the gauge

Specify the **Origin** point

Select the color of the gauge



Specify the **Variable Min** and **Max**

Specify the **Width** and **Height**

Click the 3 options (**Gauge Level**, **Value** and **Background**) to specify more details of the gauge

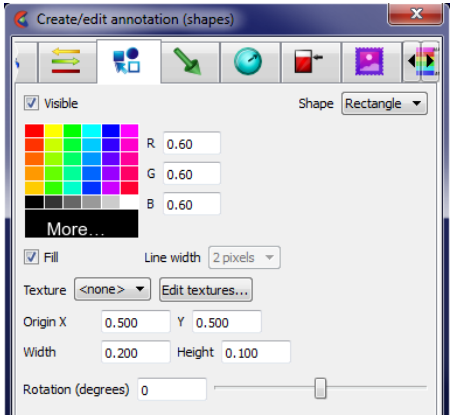
Create/Edit 2D Shapes

- EnSight can create 2D shapes (2D arrow, circle and rectangle)
- Create a 2 D shape by right clicking one of the Annotations keywords in the Annotations Tab of the Object Lists and selecting **Create** -> **Shape**
- A rectangle is created in the center of the screen by default
- Right click on the shape and select **Edit** to display the Create/Edit Annotation (Shapes) menu or right click the **Shapes** keyword in the Annotations Objects List or double click the **Shapes** keyword

Select color of the shape

Select **Fill** or **Texture**

Select **Origin, Width, Height** and **Rotation**



Select **a** Shape (2D arrow, circle or rectangle)

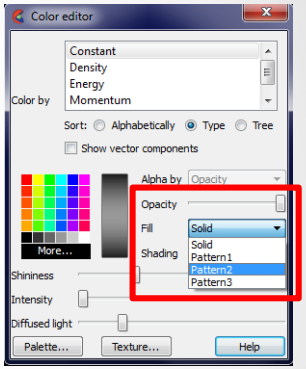
Buzz Aldrin walking on the moon in 1969 during the Apollo 11 mission – photo taken by Neil Armstrong

Chapter 18: Actions That Will Slow Down Your System

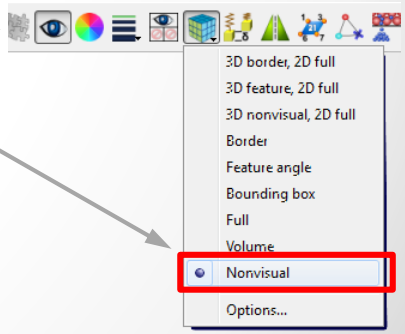
Actions that will slow down your system 1

- Displaying many parts in **Hidden Line** mode

- Displaying many parts with an **Opacity** setting smaller than 1.0 when the graphics card is not powerful enough to do so - instead use **Fill Patterns 1**, **2** or **3**

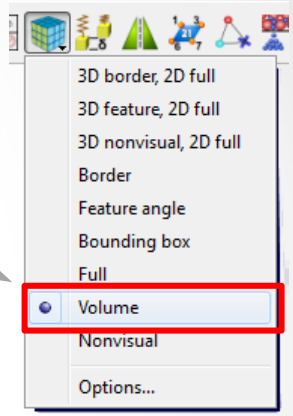


- Making the Element Representation of a big part on the server such as the flowfield of a CFD model, anything other than **NonVisual** - hiding it using the **Visibility** icon is not going to do any good because it's still using RAM on the Client side

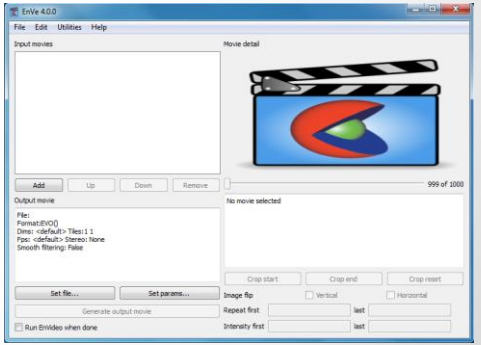


Actions that will slow down your system 2

- Using **Volume** rendering on too many parts; this will take a lot of power from the graphics card - instead make sure the graphics card is powerful enough to do volume rendering

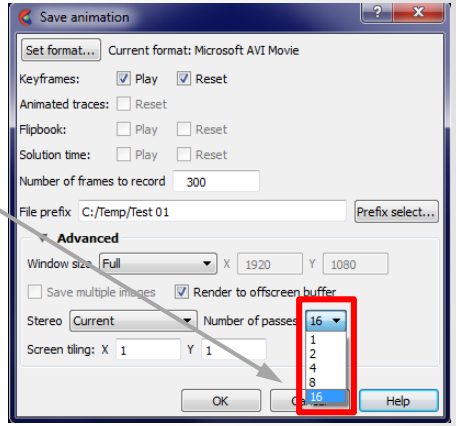


- Creating too many keyframes in the **Keyframe Animator** with many subframes while using transient data - instead make movie segments between 2 keyframes and create the complete movie using EnVe

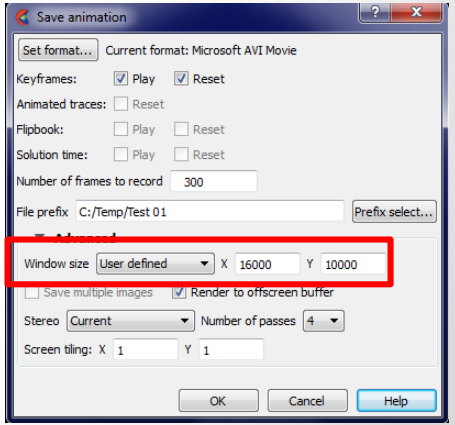


Actions that will slow down your system 3

- Using the maximum number of anti-aliasing (**Number of Passes** = **16**) when pictures or movies are created - instead start by using **Number of Passes** = **1** and when a test picture or movie is ok, crank up the quality

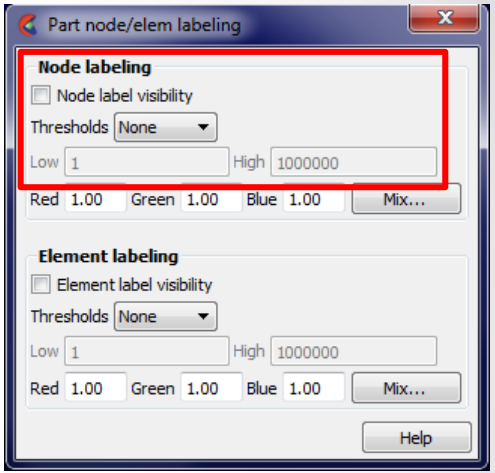


- Using a large resolution when creating a picture or movie - instead start by using a smaller size resolution (for instance NTSC or PAL) and when a test picture or movie is ok, use the desired higher resolution



Actions that will slow down your system 4

- Displaying labels on parts with thousands or millions of nodes or elements can degrade display performance significantly as well as obscure both the geometry and the labels of interest - instead use a small range of nodes and element labels of interest then toggle the **Node** or **Element Label Visibility** on



Enjoy working with EnSight 10.2!