CATIA V5 Training
Foils

Digitized Shape Editor

Copyright DASSAULT SYSTEMES

Version 5 Release 19
January 2009

EDU_CAT_EN_DSE_FI_V5R19

Instructor Notes:

About this course

Objectives of the course

Upon completion of this course you will be able to:

- **Import and process digitized point cloud data**
- **Create tessellated mesh on the point cloud data**
- **Extract characteristic curves from the data**
- **Export the result in the popular file formats**

Targeted audience

Shape Designers

Prerequisites

Students attending this course should be familiar with the CATIA V5 interface

8 hrs

Copyright DASSAULT SYSTEMES

Instructor Notes:

Table of Contents (1/2)

- Introduction to CATIA Digitized Shape Editor 5
  - An Overview of Digitized Shape Editor 6
  - Accessing the Workbench 7
  - User Interface 8
  - Digitized Shape Editor Terminology 10
- Car Door: Master Exercise Presentation 12
- Point Processing 13
  - Cloud Import 14
  - Cloud Operations 19
  - To Sum Up 27
- Master Exercise Step (1): Point Processing 28
- Mesh Processing 29
  - Mesh Creation 30
  - Mesh Correction 33
  - Mesh Edition 39
  - Mesh Operations 45
  - To Sum Up 52
- Master Exercise Step (2): Faceting the Cloud 53

Copyright DASSAULT SYSTEMES

Instructor Notes:

Table of Contents (2/2)

- Scans and Curves 54
  - Scan Creation and Edition 55
  - Curve Creation 67
  - To Sum Up 75
- Master Exercise Step (3): Creating the curves and exporting the result 76
- Advanced Tasks 77
  - Integration of Clouds 78
  - Mesh Offset 89
  - Cloud Export 92
- Added Exercise - Hair Dryer 94

Copyright DASSAULT SYSTEMES

Instructor Notes:

Introduction to CATIA Digitized Shape Editor

In this lesson, you will get an introduction to CATIA Digitized Shape Editor workbench, terminologies used and the user interface of the workbench.

- An Overview of Digitized Shape Editor
- Accessing the Workbench
- User Interface
- Digitized Shape Editor Terminology

Copyright DASSAULT SYSTEMES

Instructor Notes:

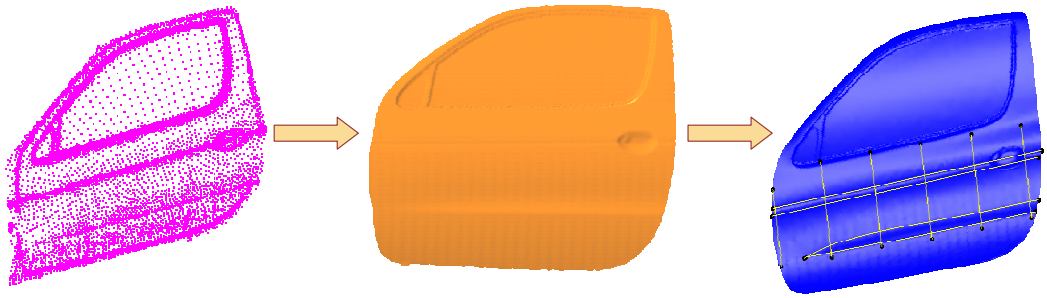
An Overview of Digitized Shape Editor

Digitized Shape Editor (DSE) is used at the initial stages of the Reverse Engineering cycle. In reverse engineering, geometry of existing physical object is captured by scanning. The result is obtained in the form of digitized data.
Digitized Shape Editor allows various operations on this digitized data.

With DSE you can import various forms of digitized data (meshes, grids, scans and clouds of points) and extract characteristic curves, create tessellated meshes.

The extracted digitized data can be further used for the following:

- **Designing parts in other workbenches such as FreeStyle workbench.**
- **Machining using the mesh created in DSE Workbench.**



Digitized data obtained from scanning a car door

Tessellated mesh created in DSE workbench

Characteristic curves extracted in DSE workbench

Copyright DASSAULT SYSTEMES

Instructor Notes:

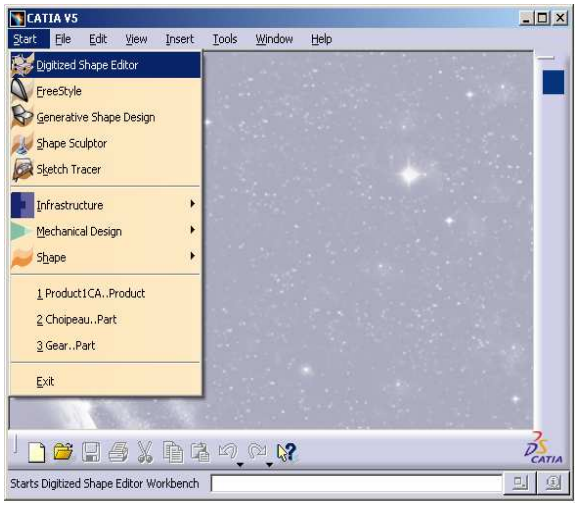
Accessing the Workbench

To access the Digitized Shape Editor workbench:

1. **Select Start > Shape > Digitized Shape Editor**

OR

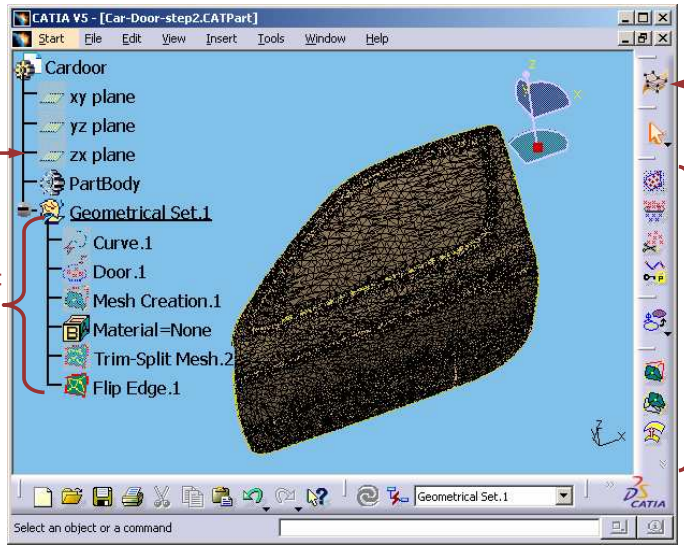
2. **Select 'Digitized Shape Editor' from the list of favorite workbenches (if you have earlier added it to your favorites).**



Copyright DASSAULT SYSTEMES

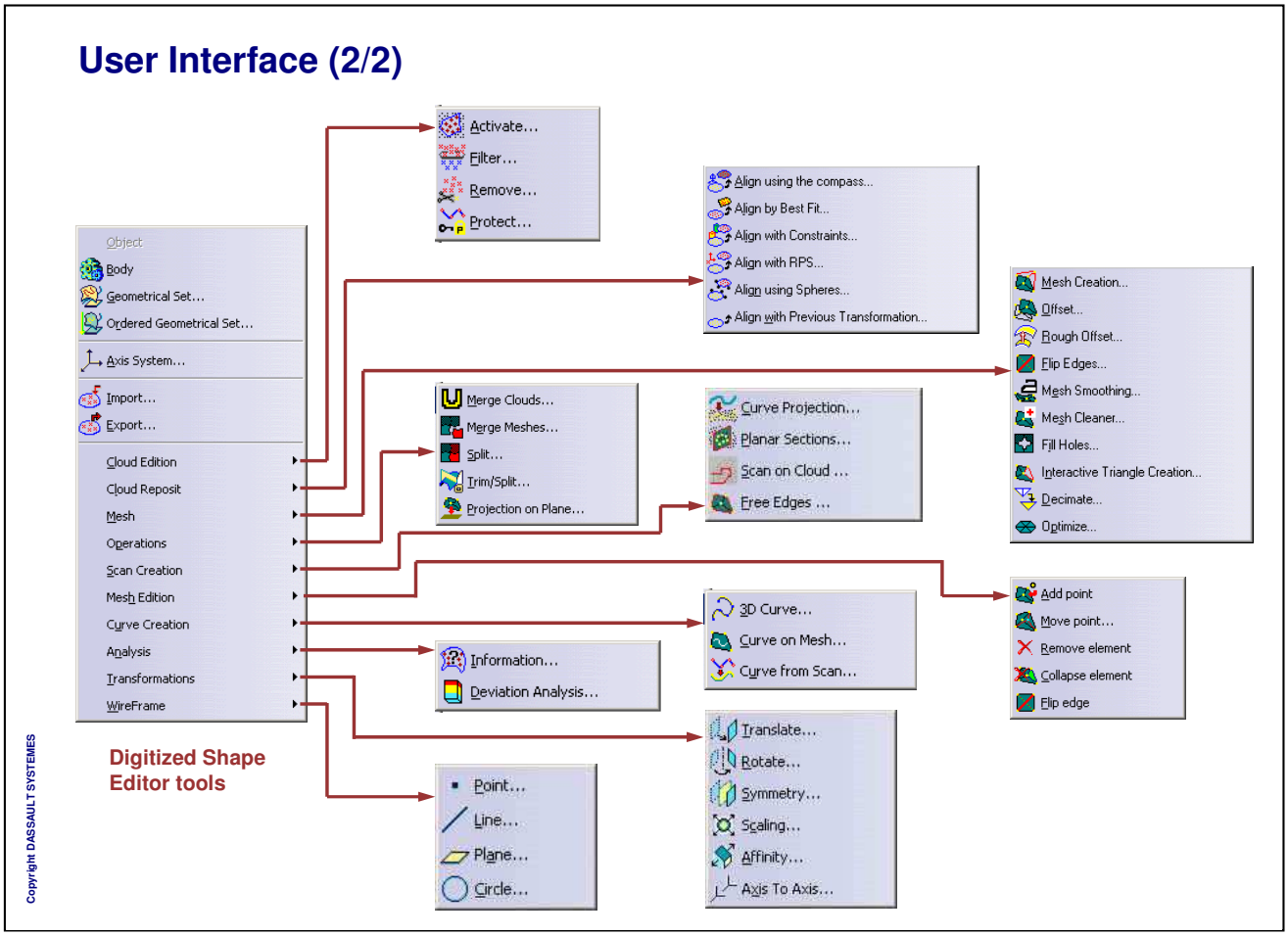
Instructor Notes:

User Interface (1/2)



Copyright DASSAULT SYSTEMES

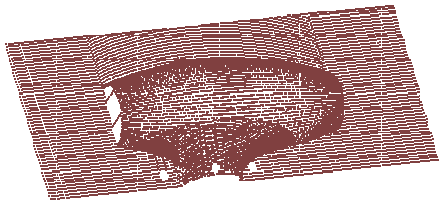
Instructor Notes:

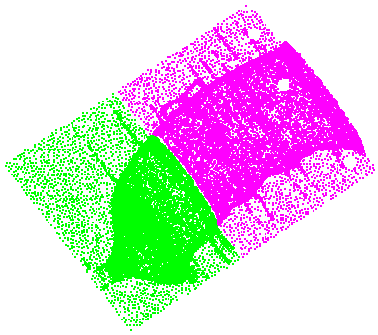


Instructor Notes:

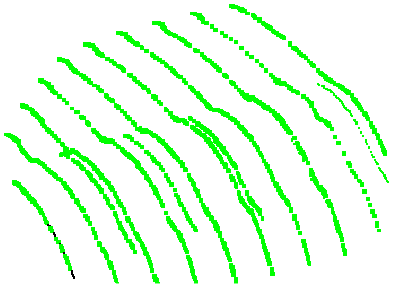
Digitized Shape Editor Terminology (1/2)

- Cloud of points: A cloud of points is defined as a set of points in 3D space. In this course, the term cloud of points refers to several representations:
  - representation as a set of points
  - representation as a set of scans
  - representation as a set of grids
  - representation as a mesh (or polygon)





- Cell: A cloud of points may consist of several sub-clouds which are called cells.



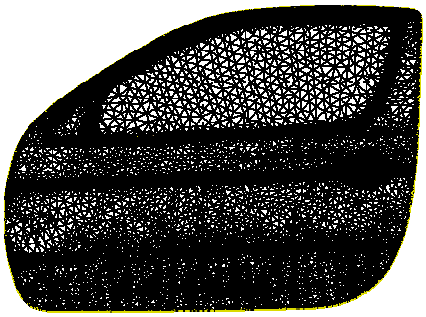
- Scan: A scan is an ordered series of points. It often represents curves via series of points. A cloud of points can be organized in consecutive scans.

Copyright DASSAULT SYSTEMES

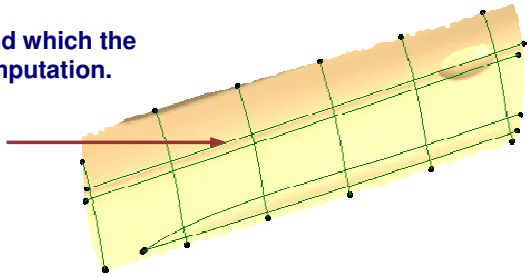
Instructor Notes:

Digitized Shape Editor Terminology (2/2)

- **Mesh: A mesh consists of a set of polygonal faces (triangles) which represent the surface of a 3D model.**



- **Neighborhood: Many functions in Digitized Shape Editor operate on the points in space. In these functions, you can specify a maximum distance (neighborhood) which will be considered around a point for the operation.**
- **Working Distance: This is the distance beyond which the elements are not taken into account for a computation.**
- **Characteristic lines: They are particular lines corresponding for instance to curvature variations (fillets start/end) or sharp edges.**



Copyright DASSAULT SYSTEMES

Instructor Notes:

Car Door

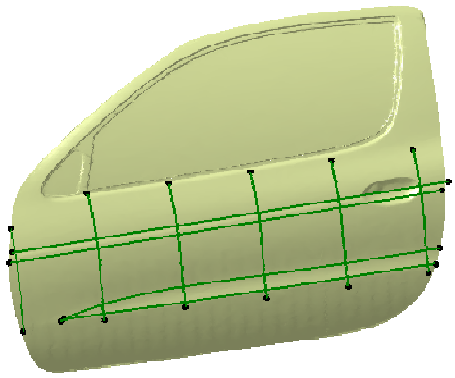
Master Exercise Presentation

90 min

The purpose of this exercise is to practice the functionalities that you will learn usingout the course. At the end of each lesson, there will be an exercise step based on the lesson.

In this exercise, you will extract characteristic curves of a Car Door. For that you have to perform the following steps.

- **Point processing**
- **Faceting cloud of points**
- **Creating curves and exporting the result**



Copyright DASSAULT SYSTEMES

Instructor Notes:

Point Processing

In this lesson, you will learn to import the point cloud data and to clean the cloud of points.

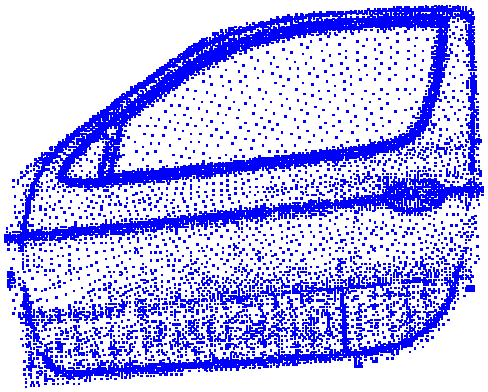
- Cloud Import
- Cloud Operations
- To Sum Up

Copyright DASSAULT SYSTEMES

Instructor Notes:

Cloud Import

You will learn how to import the point cloud data.



Copyright DASSAULT SYSTEMES

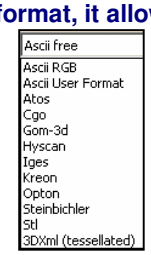
Instructor Notes:

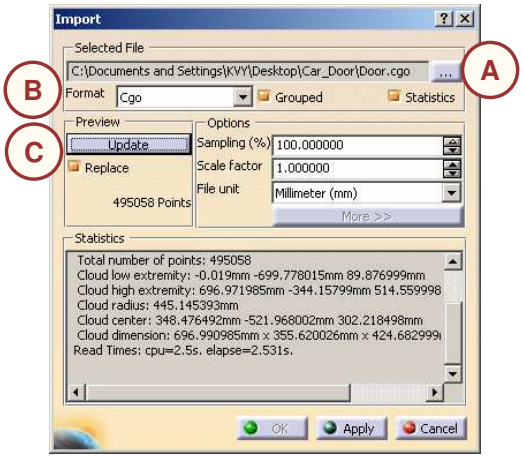
Importing the Cloud of Points (1/4)

You can import point cloud data using 'Cloud Import' tool. 

A. Select the files to import.

B. Format: You have the choice among a wide range of standard formats. A More button shows up according to the selected format, it allows more parameterization.

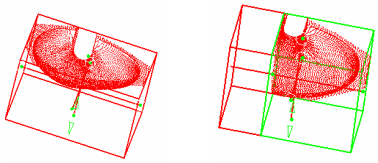




C. Preview:

a. Replace: It will replace the current cloud of points by a new one.

b. Update: It will display the cloud of points along with its bounding box. Then use the manipulators of this box to clip the part of the cloud you import



Copyright DASSAULT SYSTEMES

Instructor Notes:

Importing the Cloud of Points (2/4)

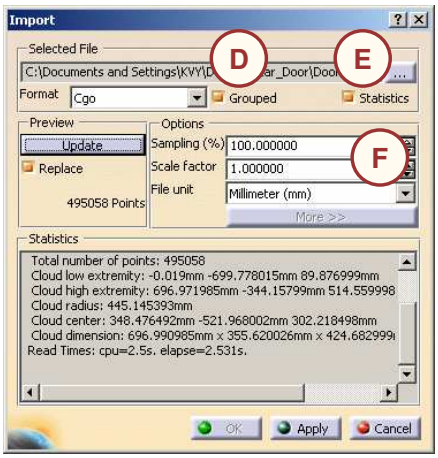
D. **Grouped:** If activated, when recalling several clouds of points at the same time you will get a single Cloud Entity in CATIA V5.

E. **Statistics:** This check box will allow you to get information in the statistics window (number of points, dimensions…)

F. **Options:**
   a. Sampling: It allows to import a certain percentage of the Digitized data. This will be done sequentially every x points will be kept.

   b. Scale Factor: The scan operation is very often performed on a scaled model. You can apply the given scale factor to work on real size model.

   c. File Unit: your CATIA session is set up for a given unit. The file you import comes from the outside world and may be digitized in another unit. Set up the proper unit before importing the file



- *An element is created in the specification tree, under the name of the cloud.x.*
- *If you import several clouds of points in the same action using option Grouped, the result will be a single cloud of points entity Cloud Import.x*

Copyright DASSAULT SYSTEMES

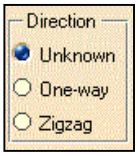
Instructor Notes:

Importing the Cloud of Points (3/4)

G. More: Select the More button and you will see the following menus specific to the file format you have chosen.

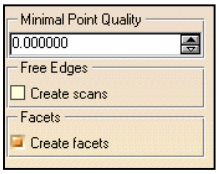
a. ASCII: Direction and Delimiters apply to scans. Type this data whenever you know it.

b. IGES:

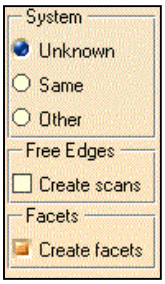


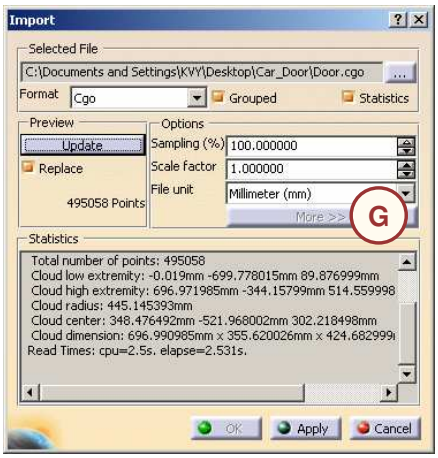
c. 3D XML:

d. Gom3D: Facets is used to create or not the facets of the imported cloud of points.

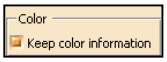


e. Stl: System applies to the operating system used to generate the binary data: select Same if you know you are using the same operating system as the one used to generate the binary data. Free Edges is used to create or not the scans representing the free edges of a cloud of points. Facets is used to create or not the facets of the imported cloud of points.





f. Leica:



g. ASCII User Format:

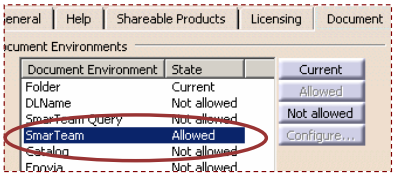
Copyright DASSAULT SYSTEMES

Instructor Notes:

Importing the Cloud of Points (4/4)

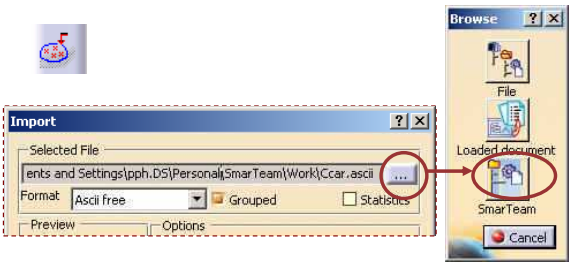
You can import data from your Smarteam vault

1 **Check that Smarteam is set at the "Allowed" (or "Current") state in Tools-Options-General-Document**

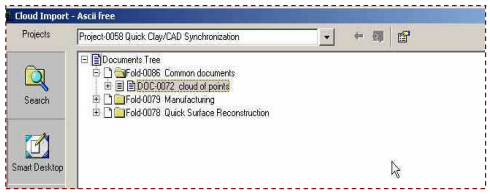


2 **Get connected to Smarteam:**

3 **Click Browse, a small window enables you to choose to open your document from the common Files or from Smarteam.**



4 **Smarteam is open so you can select your file**

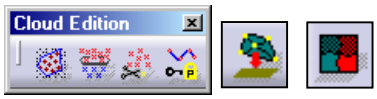


Copyright DASSAULT SYSTEMES

Instructor Notes:

Cloud Operations

You will learn to reduce the number of points in a cloud, so that further operations will be faster.



Copyright DASSAULT SYSTEMES

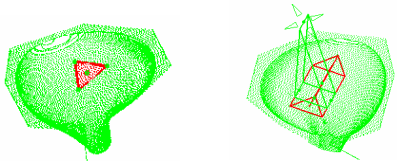
Instructor Notes:

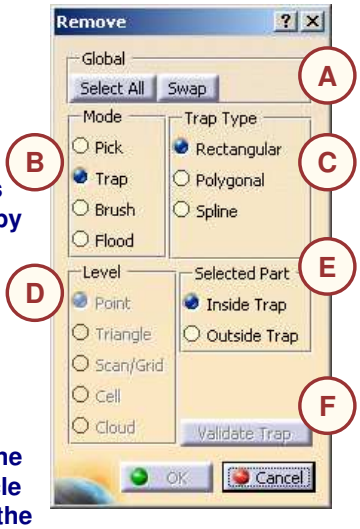
Removing Points (1/2)

You can remove points from the cloud using remove tool. 

A. **Global:**
   a. Select all: It will select the whole cloud of points.
   b. Swap: It will switch the selection by the 'not selected' elements.

B. **Mode:**
   a. Pick: It will allow selection by picking elements on screen. The element type is filtered by the options from Level
   b. Trap: It will allow the selection through a contour trap. The trap is drawn in a plane but has a depth. Then you may change the trap by the manipulators of the Trap.





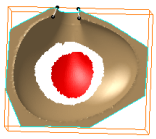
   c. Brush: It is an option to select an area on a mesh only. It allows the selection by keeping the left button pressed, and passing the circle over the desired area. The precision of the selection depends on the diameter of the circle, modifiable in the contextual menu in Edit Radius.

Copyright DASSAULT SYSTEMES

Instructor Notes:

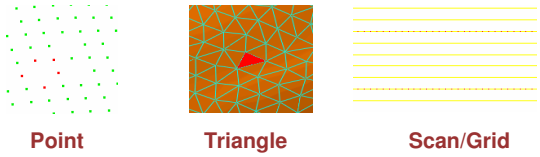
Removing Points (2/2)

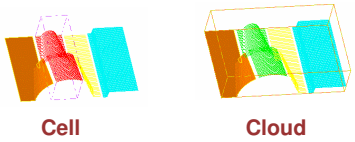
c. **Flood:** Its an option to select an area on a mesh only. It allows the selection by picking a triangle on the mesh. All triangles connected to this triangle are automatically selected.



C. **Trap Type:** It can be defined by a Rectangular or Polygonal contour or by a spline.

D. **Level:** In coordination with Pick can be set,
   a. Point
   b. Triangle
   c. Scan/Grid
   d. Cell
   e. Cloud





E. **Selected Part:** It can be,
   a. Inside Trap
   b. Outside Trap

F. **Valid Trap:** You can create as many areas as you wish by clicking Valid Trap and drawing another trap, until you click OK to validate and exit the action.

Copyright DASSAULT SYSTEMES

Removed points cannot be recalled! Select All and Swap apply only to the current removal action. They cannot be used to recall removed points, once you have clicked OK.

Instructor Notes:

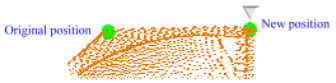
Filtering Points

When you filter scans or grids, filtered points are hidden, and new scans or grids are created. You can recall the points with Reset tab.

Principle:

- *Homogeneous*: The sphere passes over the cloud of points, starting on the first point met. All the points that are inside the sphere are then hidden. The sphere goes to the next remaining point and removes the points that it contains, and so on.
- *Adaptative*: will use a local chordal deviation criterion.

A. Homogeneous: Enter the value of the Sphere radius, used to filter the points. The filtering sphere is visualized by a green sphere. You can change its position by a simple mouse click at a desired location.

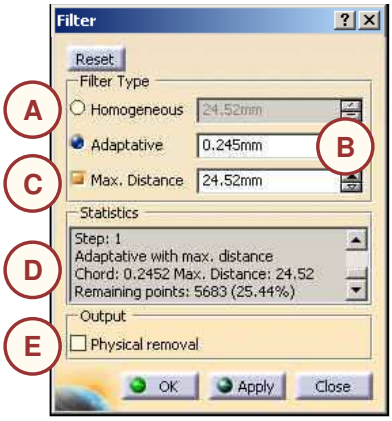


B. Adaptative: The value to enter represents the local chordal deviation. This filtering hides more points from the planar areas than from other areas. That way, you can highlight bent areas.

C. Max. Distance: Ensures that some points will be kept to satisfy the distance criterion

D. Statistics: It will be then displayed,
   a. Before application, the number of points to be processed.
   b. After application, the number, and the percentage of the remaining points.

E. Physical removal: When this option is checked, you will remove the points from your actual Cloud of Points.



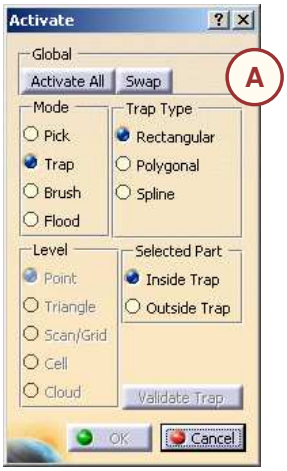
Copyright DASSAULT SYSTEMES

Instructor Notes:

Activating Points

You can define an active set of points from a Cloud of Points by using 'Activate' tool.

A. Global:



a. **Activate All: It will activate the whole Cloud of points.**
b. **Swap will switch the activated by the 'de-activated' elements.**

Selection: The selection of the points is exactly the same as with the Remove Points option. You have the same modes available which are Pick, Trap and Brush.

The activated points are those that appear in red during the selection.
Non active points are still in the cloud of points element, you can restore it when desired.
If displayed as a shaded mesh, the result will not be visible. You should display the mesh as triangles to visualize the result.

Copyright DASSAULT SYSTEMES

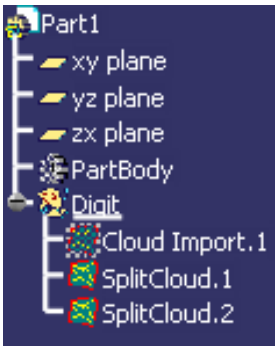
Instructor Notes:

Splitting Clouds

You can split a cloud into two using this command. 

The user interface is the same as in Activate or Remove.

New elements SplitCloud are created.



Copyright DASSAULT SYSTEMES

Instructor Notes:

Projecting Clouds

This action projects all points of the selected cloud to a plane.

A. Elements:
- **a. Clouds**
- **b. Meshes**
- **c. Scans**

B. You can use the multi-selection icon to select several elements.

C. Plane: Destination plane, the projection is always normal to the plane.

D. Distinct/Grouped: In case several elements are selected you can choose to group the projections into a single output cloud.

- ***The structures are preserved: projected scans are scans, projected meshes are meshes.***
- ***You can click a label to change the status of the element.***
- ***A pop up menu is also available to switch the lock status of one or all elements.***

This command is often used in aerodynamics to compute the area of the projected cloud.

Copyright DASSAULT SYSTEMES

Instructor Notes:

Protecting Points

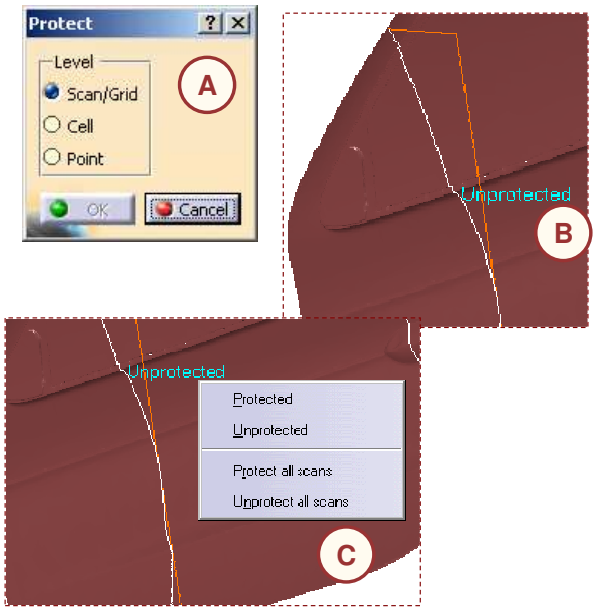
When working on a Cloud of points it may be useful to lock some points of them to protect them from any processing such as filtering or smoothing.
It is the case for example when working on a cloud which contains both dense points resulting from optic measurement and scans resulting from the accurate measurement of character lines.

A. Level: You can select the type of element that you want to protect.

   a. Scans and grids
   b. Whole cells
   c. Points

B. All elements of the selected type are displayed with a label showing their status.

C. You can click a label to change the status of the element. A contextual menu is also available to switch the lock status of one or all elements



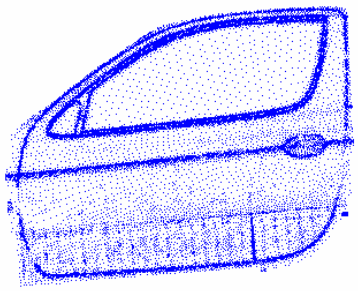
Copyright DASSAULT SYSTEMES

Instructor Notes:

To Sum Up

You have seen how to handle digitized Data . You learnt:

- **Importing Clouds of points.**
- **Removing points.**
- **Filtering points according to criteria.**
- **Activating part of the cloud of points.**



Point Processing

Copyright DASSAULT SYSTEMES

Instructor Notes:

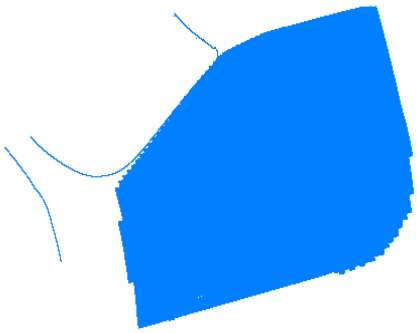
Car Door

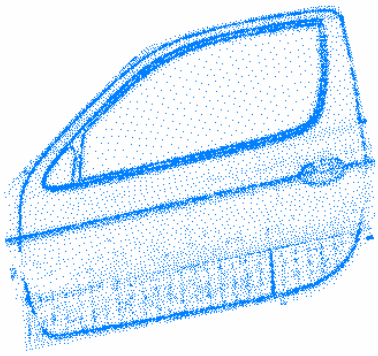
Step 1 - Point Processing

30 min

In this step you will:

- **Import the cloud of points.**
- **Remove unnecessary points.**
- **Activate Area / Filter the Data. From this Area preserve the characteristic lines.**
- **Activate all points to check the result.**





Copyright DASSAULT SYSTEMES

Instructor Notes:

Mesh Processing

In this lesson you will learn to mesh the clouds, to improve the mesh characteristics and to split a mesh.

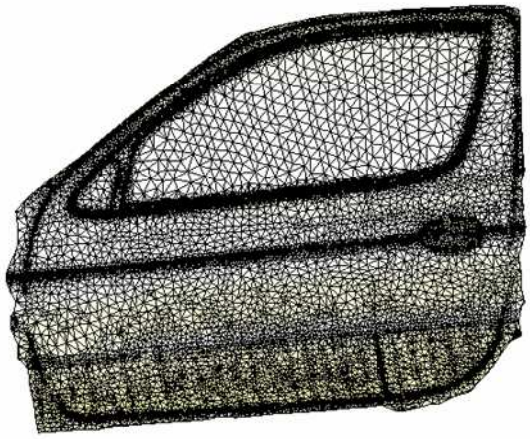
- Mesh Creation
- Mesh Correction
- Mesh Edition
- Mesh Operations
- To Sum Up

Copyright DASSAULT SYSTEMES

Instructor Notes:

Mesh Creation

You will learn how to create a mesh on the cloud of points.



Copyright DASSAULT SYSTEMES

Instructor Notes:

Meshing the Cloud (1/2)

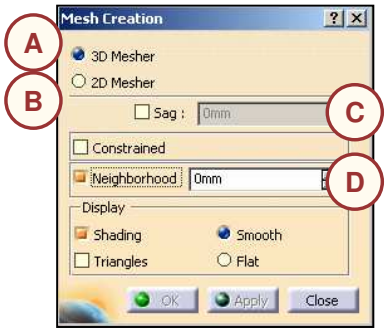
You can create a mesh on all the active points of the cloud using this tool.

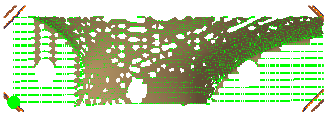
A. **3D Mesher: Applicable for any situation.**

B. **2D Mesher: Applicable only to clouds with no hidden area in a given direction which must be specified; then the connections are computed in a plane normal to that direction.**

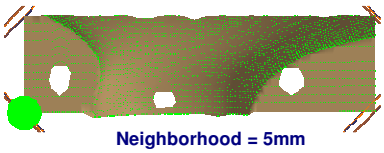
C. **Sag: This is used to mesh dense cloud option without filtering it prior to mesh. When used, the resulting mesh will not pass through all the active points of the Cloud of Points. It will then generate a mesh to the given tolerance.**

D. **Neighborhood: Maximum length of the facet edge: if one edge of the facet is greater than this maximum, the facet will not be created. Example:**

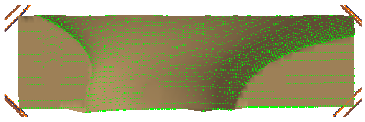




Neighborhood = 2mm



Neighborhood = 5mm



no Neighborhood

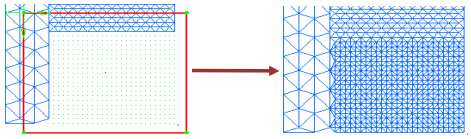
Increase the Neighborhood value to close unwanted holes of the mesh. In some cases, it may be difficult to find a Neighborhood value that will fill holes without creating unwanted triangles. You can use the green sphere to set up this parameter.

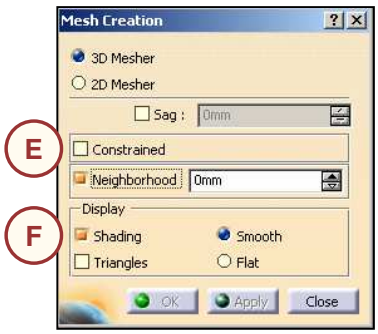
Copyright DASSAULT SYSTEMES

Instructor Notes:

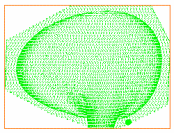
Meshing the Cloud (2/2)

E. **Constrained:** When meshing part of a Cloud of Points, if some of the active points are already used in another mesh, the computation of the new mesh will take this mesh into account to create a 'seamless' result. Constrained meshes are compatible (same nodes on common boundaries) and can be merged by Merge Meshes.

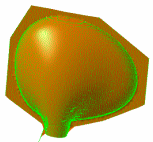




F. **Display:**
   a. Triangles
   b. Shading
      - Flat: Each triangle is lighted according to its normal; this mode shows the exact facets
      - Smooth: Triangle are lighted according to the average normal at each vertex. The result is smoothed and the display looks more realistic.



Triangles



Shading

- *When computing a constrained mesh, type 0 as the Neighborhood value to check the boundaries of the mesh. If the boundaries are not satisfactory, modify the meshing plane to improve them.After the computation of a constrained mesh, two mesh elements are visible in the specification tree: the constrained mesh and the initial mesh. You can select one and then the other to make sure they are complementary.*
- *A Mesh may require big computation time when based on a very dense cloud of points.*

Copyright DASSAULT SYSTEMES

Instructor Notes:

Mesh Correction

Mesh created has defects and flaws in it. You will learn how to correct it.

Copyright DASSAULT SYSTEMES

Instructor Notes:

Filling Holes

The mesh obtained from using 'Create Mesh' tool may contain some holes. You can fill such areas using this tool.

1. **Select the polygon to analyze.**

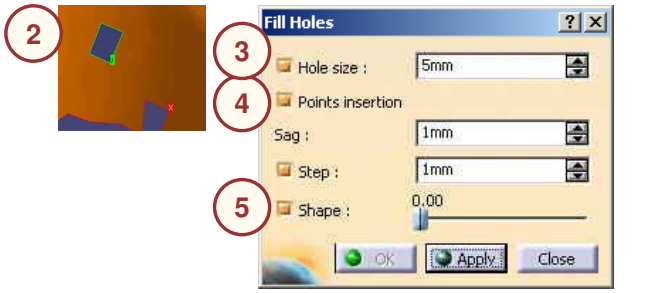
2. **The system displays all the holes as follow:**
   a. **X Holes not selected**
   b. **V Holes selected**

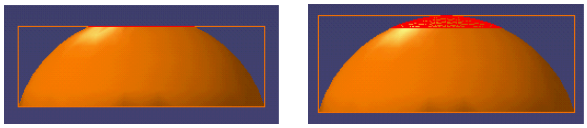
3. **You can check Hole Size to control which hole you want to fill.**

4. **Check Point Insertion to allow the computation of addition mesh points, with this option you can also access to a sag value and a step value to control the points you will insert.**

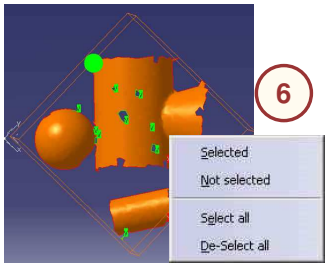
5. **By default the fill is Flat, use Shape option to create the mesh on a virtual tangent surface.**

6. **You may click a label to switch it from X to V (or vice versa) or use the contextual menu.**





Mesh created with the help of shape option.



Copyright DASSAULT SYSTEMES

Instructor Notes:

Creating Facets

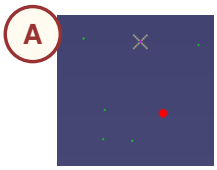
You can create facets on the cloud of points, triangle by triangle, using 'Interactive Triangle Creation' tool.

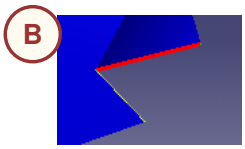
Three ways of creating triangles are:

A. **Select three points or vertices.**

B. **Select two neighboring edges of an existing mesh**

C. **Select an edge of an existing mesh and a point**





Then click Apply and OK to confirm.

Copyright DASSAULT SYSTEMES

- ***The points are not necessarily points from a cloud but they could also be CATIA points, curve extremities.***
- ***This function can be used to fill in holes in the mesh.***

Instructor Notes:

Cleaning a Mesh (1/2)

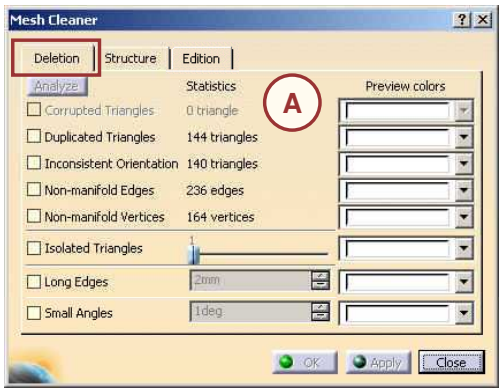
You can remove defects in the mesh and improve the quality of the mesh, using 'Mesh Cleaner' tool.

3 Actions:

A. Deletion of corrupted / duplicated triangles, triangles with inconsistent orientations, non-manifold edges or non-manifold vertices, isolated triangles, triangles with long edges.

B. Structure problems: global orientation problems.

C. Detection and deletion of thin triangles to make local modification on meshes.



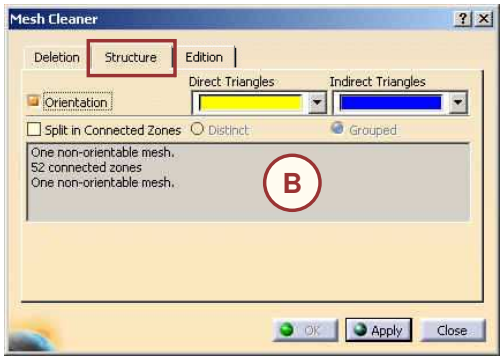
A. Deletion:

a. **Select the Smooth Polygon icon and a mesh (or an activated portion). The dialog box is displayed.**

b. **Select Analyze. The systems displays the stats on the defect of the mesh.**

c. **Select the defects type you wants to erase, and eventually the color you want to display it with.**

d. **Click Apply.**



B. Structure:

a. **You may reorient the mesh facets automatically.**

b. **You may split the mesh into connex (continuous) domains.**

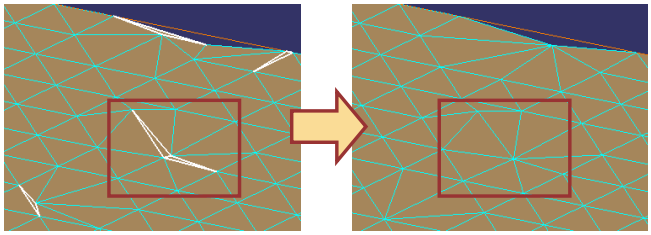
Copyright DASSAULT SYSTEMES

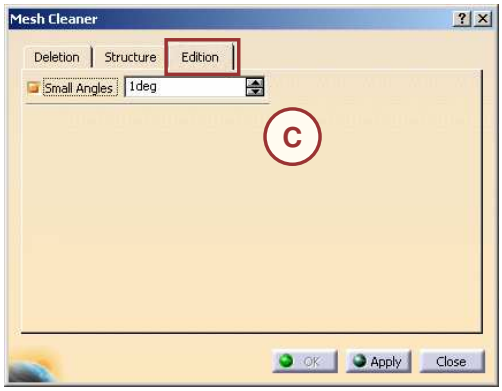
Instructor Notes:

Cleaning a Mesh (2/2)

C. Edition:

a. Select a mesh.
b. Select Edition tab, check Small Angles box and type the value for small angles.
c. Select Apply to collapse the displayed triangles with an angle equal to or lower than the specified value.





Copyright DASSAULT SYSTEMES

Instructor Notes:

Smoothing a Mesh

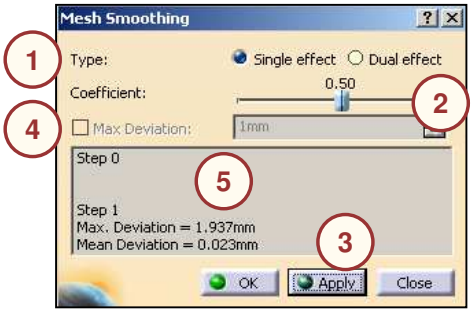
You can improve the quality of a mesh that comes from a digit of poor quality, with the help of 'Mesh Smoothing' tool.

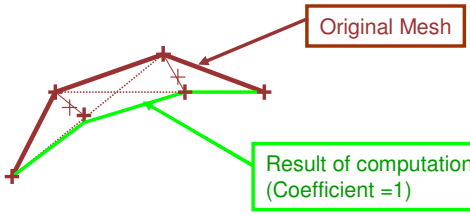
1. **Type: single effect if no sharp edge, dual effect to minimize the volume reduction.**

2. **Type the value of Coefficient.**
   a. **When the value of Coefficient is 0, vertices are not moved.**
   b. **When the value of Coefficient is 1, the vertices are moved at the full extent of the computation result.**
   c. **This may lead to a temporary degradation of the energy function of the polygon, but results in a final optimal solution.**
3. **Click Apply. The mesh is first displayed in Flat mode and then the computation is applied. The action is iterative a new Apply will open a new computation.**

4. **Select max deviation if you want to control accuracy.**

5. **The cumulative deviation statistic is displayed.**





- ***You may first activate only part of the mesh to focus on some area only.***
- ***You may need to alternate the use of this action with the flip edge action.***
- ***This action cannot be used on the mesh containing Non-Manifold edges.***

Copyright DASSAULT SYSTEMES

Instructor Notes:

Mesh Edition

Mesh created has defects and flaws in it. You will learn how to edit the mesh to modify its structure or content.

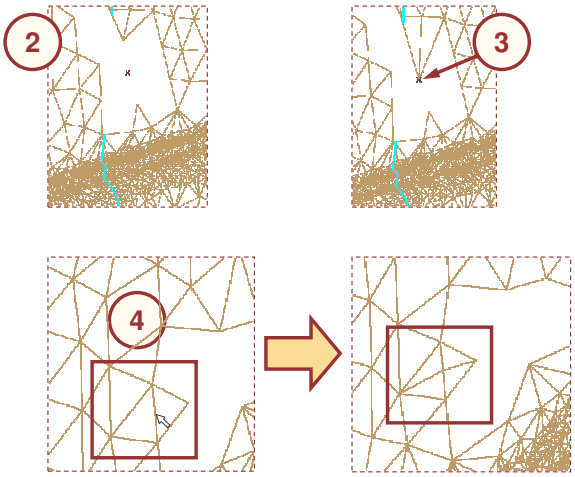
Copyright DASSAULT SYSTEMES

Instructor Notes:

Adding Points

You can add a new vertex either by picking an existing point or indicating a location on the mesh. You can flip several edges successively by double-clicking on the icon.

1. **Click the command.**
2. **Select the mesh to modify.**
3. **Select the GSD point and it will be automatically inserted in the mesh.**
4. **You can also pick anywhere on the mesh without any underlying existing point. The vertex will be created and added to the mesh.**



Copyright DASSAULT SYSTEMES

It is better to choose the Wireframe graphic visualization mode to have a nice preview during mesh edition.

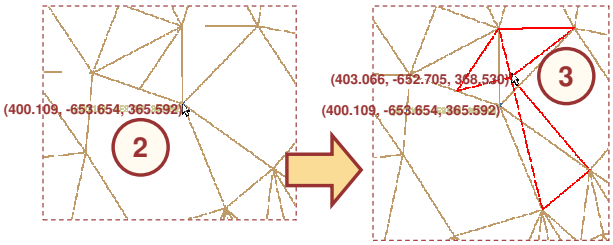
Instructor Notes:

Moving Points

You can move a vertex of a mesh to define the new position. You can view the current final coordinates of the point. There are two possibilities to move the vertex.

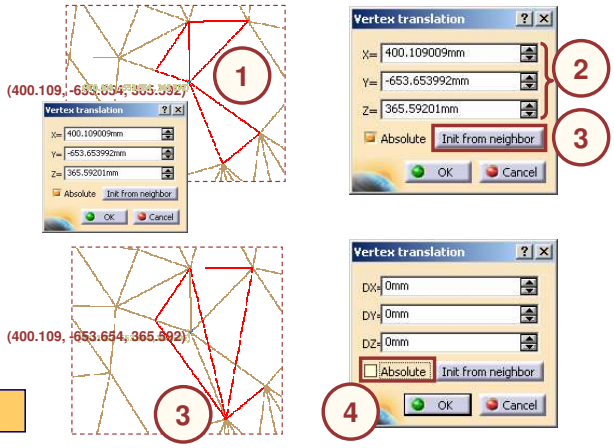
A. Drag and drop the point:

1. **Click the command.**
2. **Select the point and hold the mouse button.**
3. **Release the mouse button when you want to validate the new position.**



B. Edit the values in a dialog box:

1. **Select the point. The Vertex translation dialog box appears.**
2. **Change the coordinates to reposition the point.**
3. **Click 'Init from neighbor' to set the coordinates of a neighbor vertex in the coordinate fields.**
4. **Uncheck the Absolute check box to translate the point with respect to the coordinates of the selected point.**



You cannot move a vertex on another existing vertex.

Copyright DASSAULT SYSTEMES

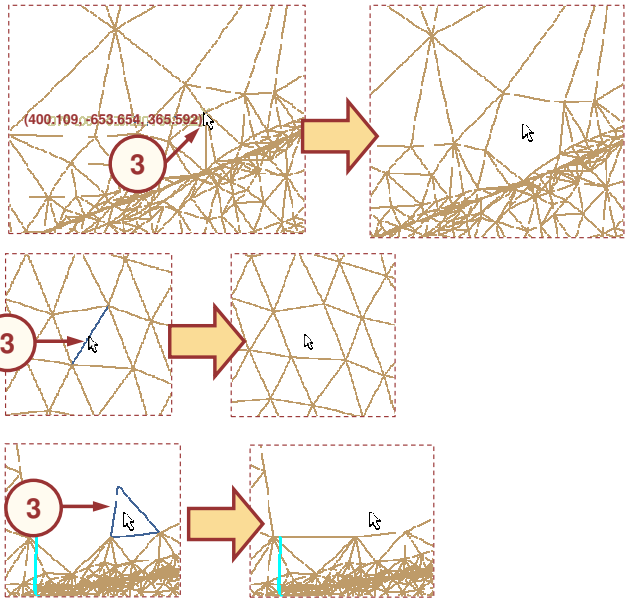
Instructor Notes:

Removing Elements

You can remove a vertex, an edge or a triangle from the mesh. It will remove all connected edges or triangle and create a hole in the mesh.

1. **Click the command.**

2. **Choose the kind of element you want to remove.**
3. **Pick the element to remove from the mesh.**



Copyright DASSAULT SYSTEMES

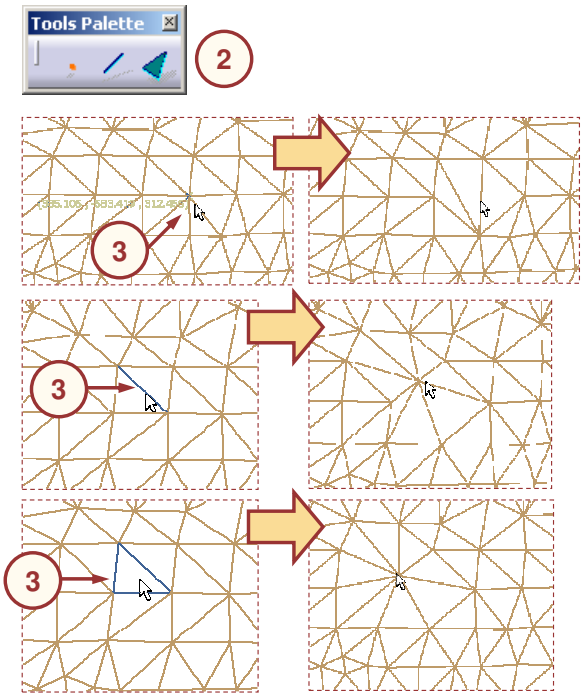
Instructor Notes:

Collapsing Elements

You can remove a vertex, an edge or a triangle from the mesh. It will automatically fill the created hole by local remeshing.

1. **Click the command.**

2. **Choose the kind of element you want to remove.**
3. **Select the element to remove from the mesh.**



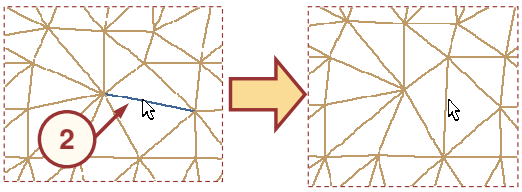
Copyright DASSAULT SYSTEMES

Instructor Notes:

Flipping Edges

You can turn over the edges of the mesh to see the alternate position.

1. **Click the command.**
2. **Pick the edge to flip.**



- *You cannot flip free edges.*
- *You cannot flip an edge if it produces a non-manifold result.*

Copyright DASSAULT SYSTEMES

Instructor Notes:

Mesh Operations

You will learn to improve the characteristics of a mesh. You will also learn to split a mesh.

Copyright DASSAULT SYSTEMES

Instructor Notes:

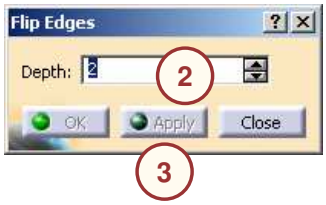
Improving Edges

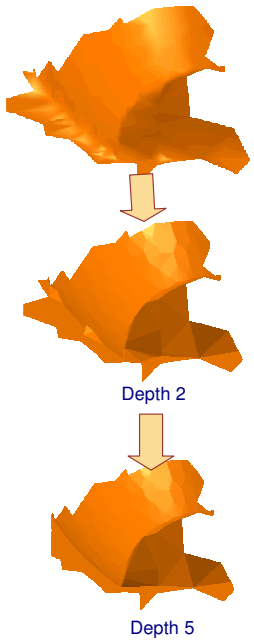
You can flip the common diagonal of two triangles, to better respect the sharp edges, using 'Flip Edges' tool.

1. Select the Flip Edges icon and a cloud or an activated portion of the cloud.

2. Type the value of Depth, this parameter identifies the candidate triangles.
   a. Value of 'Depth' determines the amplitude of the reorganization of the polygon and ranges from 0 to 10.
   b. When the value of Depth is 0, the action processes a triangle and its direct neighbors.
   c. When the value of Depth is 1, the action processes a triangle, its direct neighbors and their direct neighbors, and so on as you increase the value of Depth.
   d. This may lead to a temporary degradation of the energy function of the polygon, but results in a final optimal solution.

3. Click Apply. The edges that could be flipped are highlighted in their eventual new position. Click OK to validate.





- *You may first activate only part of the mesh to focus on some area only.*
- *This action reorganizes the meshing without modifying the geometry because the vertices are not recomputed.*
- *This action cannot be used on the mesh containing Non-Manifold edges.*

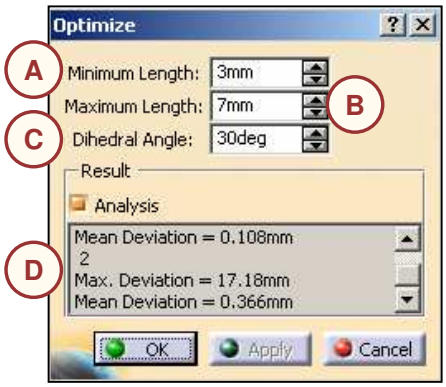
Copyright DASSAULT SYSTEMES

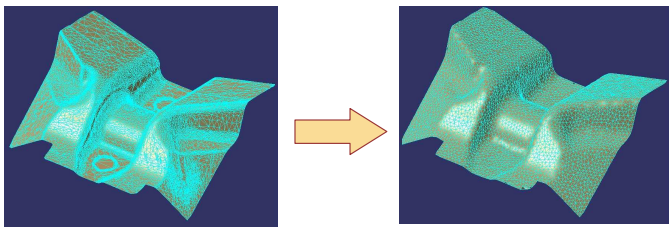
Instructor Notes:

Optimizing Density

You can obtain more homogeneous triangle calculation for the mesh which will be required for analysis purpose.

A. **Minimum edge length for decimation.**
B. **Maximum edge length for refinement.**
C. **Dihedral angle filter to preserve sharp edges. The smaller the angle value is, the sharper the edges are.**
D. **Possibility to display the deviation with the original mesh.**





- ***Use the Cloud Display option to display the triangular tessellation, so that you can check the differences.***
- ***After this operation, the mesh is homogeneous; so it is useful to do analysis. However, generally the mesh is less accurate after an optimization because the density of the facets is not adapted to the curve of the surface.***

Copyright DASSAULT SYSTEMES

Instructor Notes:

Reducing the Number of Facets

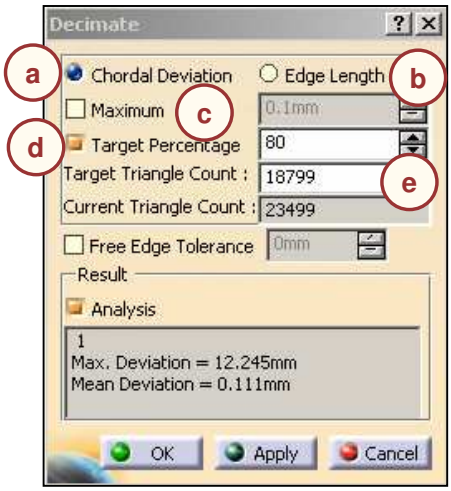
You can reduce the number of facet in the polygon, using 'Mesh Decimation' tool. 

1. Select the Decimate Polygon icon and a polygon (or an activated portion of the polygon).

2. Choose the appropriate method for decimation.
   a. Chordal Deviation: It allows to preserve the shape.
   b. Edge Length: focusing on removing smaller triangles
   c. Maximum for Chordal Deviation option: It is the max deviation allowed, for Edge Length it is the minimum edge length for kept triangles.
   d. Target Percentage: Type the percentage of the number of facets you want to get from the original mesh.
   e. Target Triangle count: Type the number of triangles to keep.

3. Click Apply.

4. Cumulative deviation statistic is displayed.



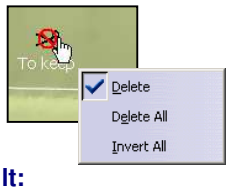
Copyright DASSAULT SYSTEMES

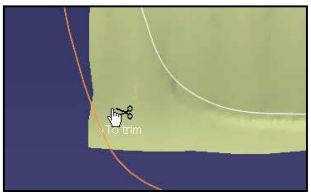
Instructor Notes:

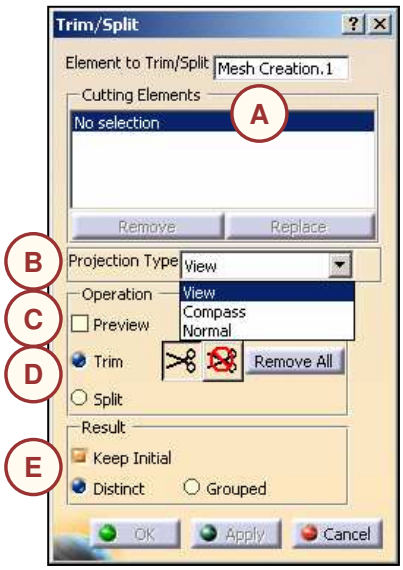
Splitting a Mesh

A mesh may be cut by 3D elements (Curves, Scans, Planes, Surfaces). The action may be Trim or Split. Result may be one or several entities.

A. **List of Cutting Elements: If you want to remove one from the list, select it then hit Remove button.**
B. **Projection Type: When a curve is used as a cutting element a projection direction should be input, chosen among:**
   a. **View**
   b. **Compass**
   c. **Normal**
C. **Check preview if you want to see the projection.**
D. **If you use the Trim option, the scissors will be used to decide what to keep and what to remove. A contextual menu is available for global modifications.**







E. **Result:**
   a. **Distinct or Grouped may be used to get one result or several meshes.**
   b. **Keep Initial if not checked the original mesh is removed from the specification tree.**

- ***To define an area the intersection curves must intersect each other.***
- ***Do not use Preview option if not needed, otherwise performance will drop.***

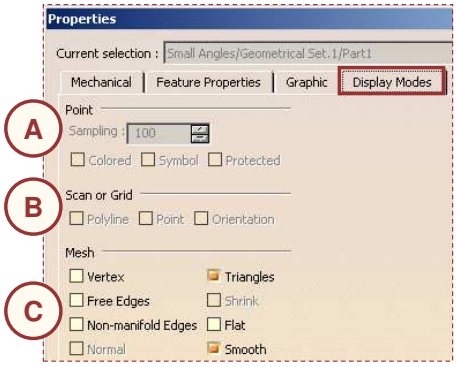
Copyright DASSAULT SYSTEMES

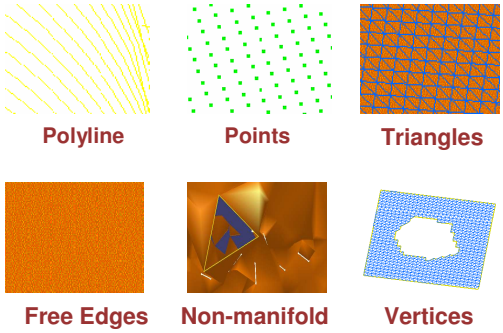
Instructor Notes:

Display Options (1/2)

You can display clouds in various styles using 'Display Modes' from Properties.

A. **Sampling:** You can choose to display only a percentage of the points, using the Sampling option. By default, 100% of the points are visible. You can change this value with the associated spinner.
B. **Scan or Grid:**
   a. **Polyline:** Displays the scan and grid selected connecting the ordered points by segments of lines.
   b. **Point:** whether yes or not the points are displayed.
C. **For Polygons only following options are available:**
   a. **Triangles:** Display of the facets (shaded display); the display can be flat (each facet has a unique normal direction) or smooth (also known as Gouraud shading).
   b. **Free Edges:** The free edges displayed are those of the complete cloud of points (if you activate only a portion of a cloud of points, the free edges of that portion are not displayed).
   c. **Non-manifold:** Edge have their edges displayed as regular white lines.
   d. **Vertices:** To display only the vertices of a mesh. Do not forget to deactivate the Flat or Smooth option.
   e. **Flat/Smooth:** The common edges to be displayed as flat or smooth.





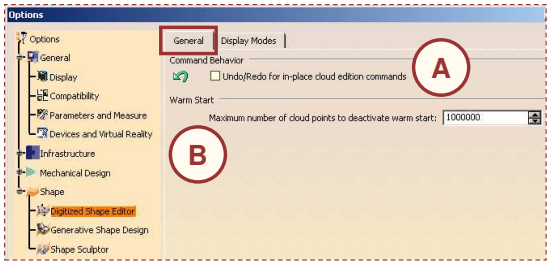
Copyright DASSAULT SYSTEMES

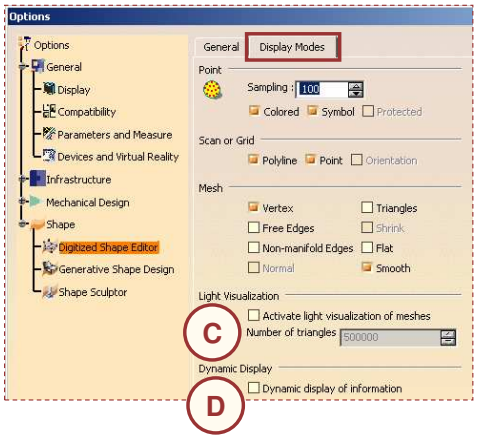
Instructor Notes:

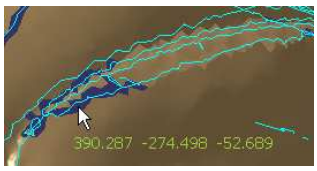
Display Options (2/2)

You can display clouds in various styles using Tools>Options>Digitized Shape Editor.

A. **Command Behavior: You can undo/redo the cloud edition commands.**
B. **Warm Start: You can specify maximum number of cloud points to deactivate the warm start.**
C. **Light Visualization: You can choose to display only a percentage of the points, using the Sampling option. By default, 100% of the points are visible. You can change this value with the associated spinner.**
D. **Dynamic Display: You can choose to display the coordinate information.**







Dynamic Display

Copyright DASSAULT SYSTEMES

Instructor Notes:

To Sum Up

You have seen how to use digitized data to:

- **Create a Mesh (faceting) of your cleaned Point Data**
- **Clean a Mesh.**
- **Improve characteristics of a Mesh.**

Faceting the cloud of points

Copyright DASSAULT SYSTEMES

Instructor Notes:

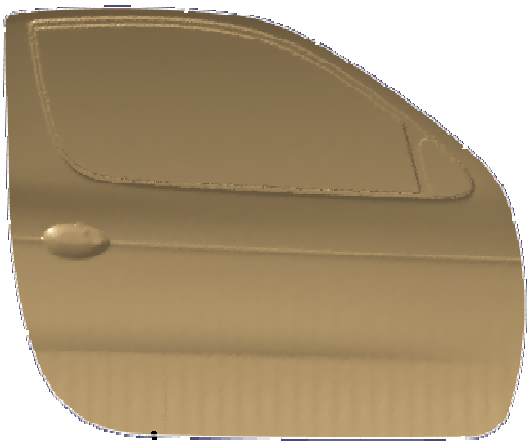
Car Door

Step 2 - Faceting the Cloud

30 min

In this step you will:

- **Mesh the model.**
- **Clean the Mesh.**
- **Optimize the Mesh.**



Copyright DASSAULT SYSTEMES

Instructor Notes:

Scans and Curves

You will learn how to extract the characteristic curves from a mesh.

- Scan Creation and Edition
- Curve Creation
- To Sum Up

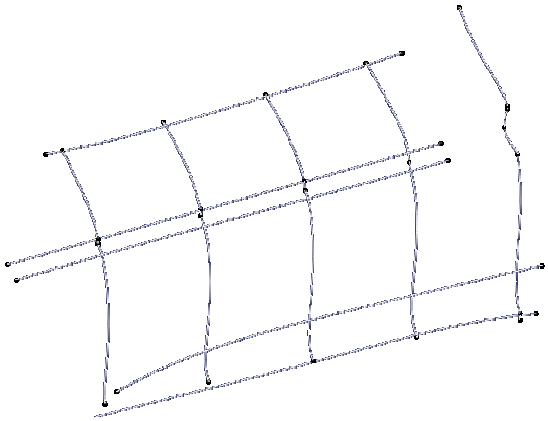
Copyright DASSAULT SYSTEMES

Instructor Notes:

Scan Creation and Edition

In this lesson, you will learn:

- To create sections.
- To project curves on a mesh.
- To create scans freely on a mesh.
- To create boundaries of a mesh.
- To create scan according to discretization mode.
- To edit the scans.



Copyright DASSAULT SYSTEMES

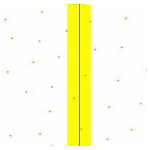
Instructor Notes:

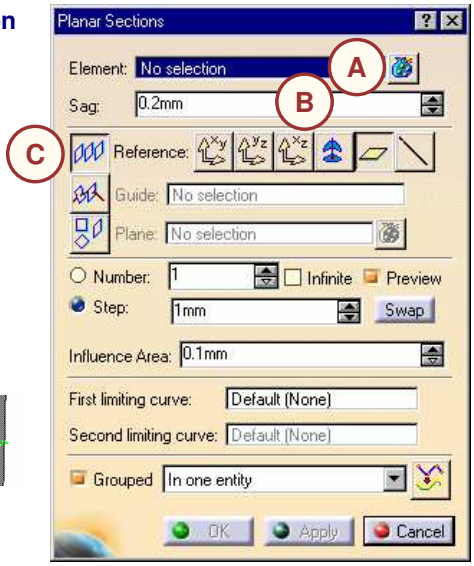
Creating Sections (1/3)

You can create intersections of clouds of points, meshes, surfaces and volumes with the planes, using the 'Planar Sections' tool.

Principle:

- **From each plane using the Influence Area distance the system computes a volume (yellow on the drawing) then select the points of the cloud inside this volume. From these points the system interpolates an intersection point.**
- **For a meshed model, the intersection is directly performed on the facets. Sag: If input elements are surfaces or volumes, a tessellation of these elements is done.**
- **The result is a Scan typed as Planar Section.**



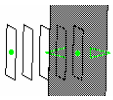


A. Element: You can multi-select the elements to create sections from.

B. Sag: You can choose and modify the sag used for the tessellation of surfaces or volumes.

C. Plane definition: sets the reference plane

- **a. use main plane (YZ, XZ, XY), use the compass or use an existing plane.**
- **b. a manipulator is then available to move the reference plane along its normal.**
- **c. another manipulator is available for the last plane.**



Copyright DASSAULT SYSTEMES

Instructor Notes:

Creating Sections (2/3)

D. **Section guide:** Select a curve, the sections will be perpendicular to this curve. The degree of the section guide must be greater than 2.

E. **Independent Planes:** If you choose this option, you can select several planes. The sections will be computed on these planes.

F. **Number:**

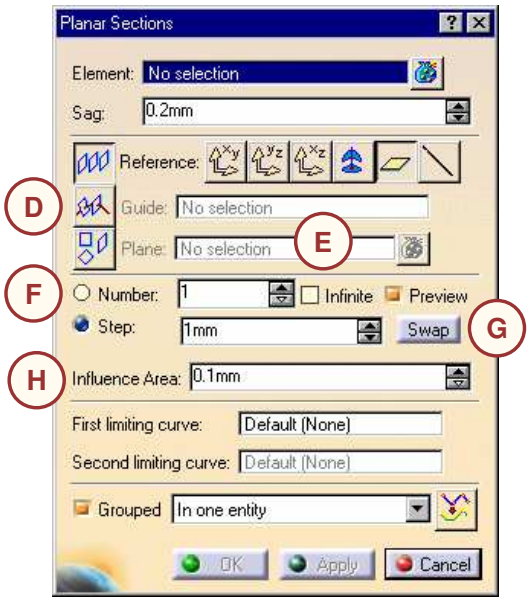
   a. Number: Type the number of planes you want to process.

   b. Infinite: The whole model will be processed using the fixed parameter.

   c. Step: Type the distance between 2 planes.

G. **Swap:** it inverts the direction for the planes.

H. **Influence Area:** 'thickness' of the plane (N.A for meshed model).



Copyright DASSAULT SYSTEMES

Instructor Notes:

Creating Sections (3/3)

I. If required, you can select one or two limiting curves. The section guide curve can be selected as second limiting curve (not as the first).

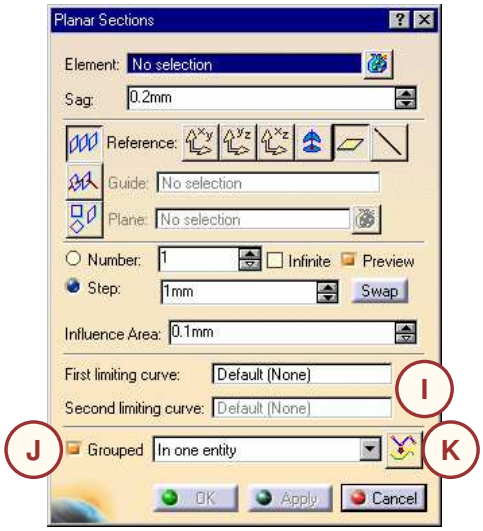
J. You can choose to group the resulting scans

   a. In one entity:

   b. By Element: The scans which cut the same element are grouped into one entity.

   c. By Plane: The scans which belong to the same plane are grouped into one entity.

K. Curve creation: The result will be curve obtained using the function Curve from scan.



Although cutting a cloud of points is quicker (no need to mesh first), creating planar sections on a mesh rather than on a cloud of points has some advantages:
1- the action is dynamic on mesh: no need to apply to visualize the modifications (position of the reference plane, step, number of planes,...),
2- In the case of a cloud of points, the intersection may be interpolated, since the plane does not necessarily intersect points. That problem is reduced with mesh since the plane intersects facets, providing a better accuracy.

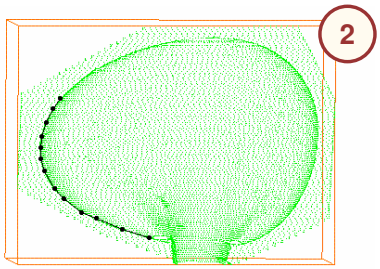
Copyright DASSAULT SYSTEMES

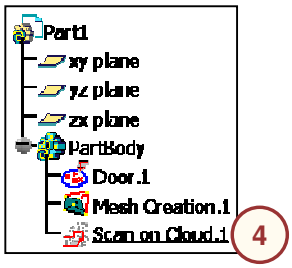
Instructor Notes:

Creating Scans Manually

You can manually draw characteristic lines on the cloud of points, using 'Create Scans on Cloud' tool.

1. **Select the Scan on Cloud icon and a cloud.**
2. **Click points on the cloud to create the scan.**
3. **Double-click to exit the action.**
4. **A Scan_on_Cloud.x element is created in the specification tree.**





- *One single scan cannot be created over several clouds.*
- *If you press the Ctrl key while moving the pointer on the cloud, the creation of the scan is displayed interactively.*

Copyright DASSAULT SYSTEMES

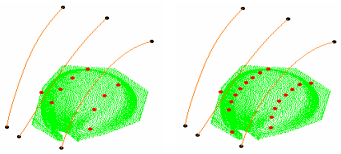
Instructor Notes:

Projecting Curves

You can project one or several curves onto a Cloud of Points using 'Project Curves' tool.

Principle:

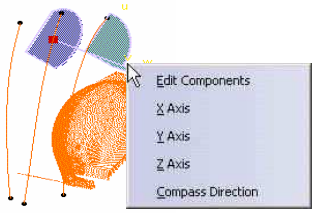
The curve is broken down in a number of points that are projected on the cloud. The 3 closest points from the cloud are located. The result is the intersection of the closest facet with the direction of projection.



A. Projection type:

   a. Normal For mesh only.

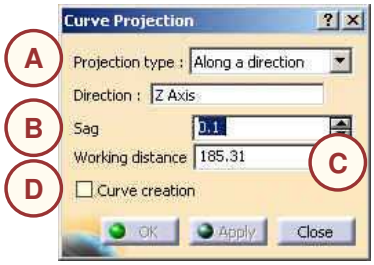
   b. Along a direction: then select the direction of projection



B. Sag: Computed on the curve according to the Sag value then projected on the Cloud of Point (non-faceted cloud only).

C. Working Distance: The distance used to locate the points eligible to compute the 3 closest points to the projection (non-facetted cloud only). Try increasing this number if the result does not contain enough points.

D. Curve Creation: When selected the result of the action will be curves. It will then also run the action Curve from scan (see dedicated function).



- *You can use trap for curve selection.*
- *Try increasing the Working Distance if the result does not contain enough points.*

Copyright DASSAULT SYSTEMES

Instructor Notes:

Creating Free Edges

You can create scans or curves that represent the free edges of a polygon, using 'Create Free Edges' tool.

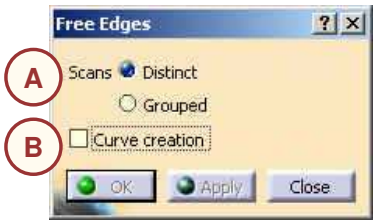
Principle:
The system analyzes the active part of the mesh and checks which facet has free edges and create the resulting curves from scan.

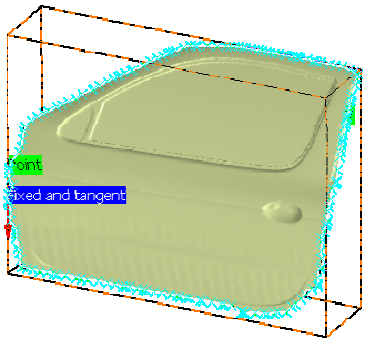
A. Scans:

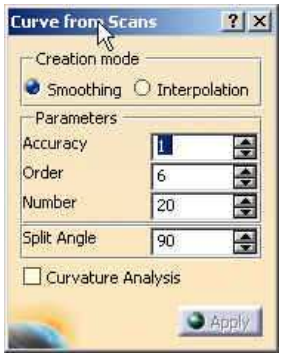
a. **Distinct:** Creates one entity per continuous set of edges.

b. **Grouped:** Creates only one entity.

B. Curve creation: When activated, Curve from Scan interface will be activated. You can then create the curves directly.







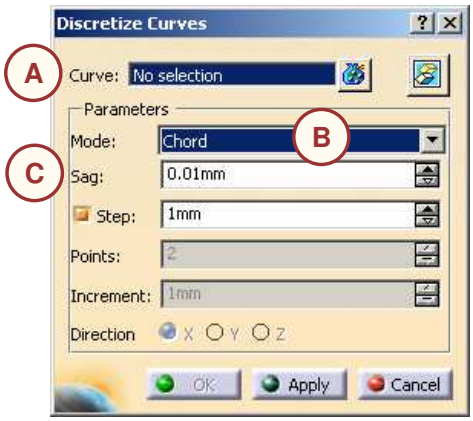
- ***Available for mesh only.***
- ***Use Activate function to limit the portion of Free edge.***

Copyright DASSAULT SYSTEMES

Instructor Notes:

Discretizing Curves (1/2)

You can create a scan using a curve. Each scan position is located on the curve and the repartition of the positions is determined by the user according to a criteria.
Several sampling criteria are available; each criterion has specific input parameters.



A. Curve: You can multi-select the curves to create the scans. You can also hide the selection.

B. Mode:

   a. Chord: You specify the Sag and the Step.

   b. Length + Positions: You specify the number of positions.

   c. Length + Increment: You specify the curve-length increment.

   d. Parameter + Positions: You specify the number of positions.

   e. Direction + Increment: You specify the main direction to define a reference plane. You also define the increment between the planes.

C. Sag: The sag represents the maximum distance between the input curve and the theoretical chord connecting two successive positions of the scan.

Copyright DASSAULT SYSTEMES

Instructor Notes:

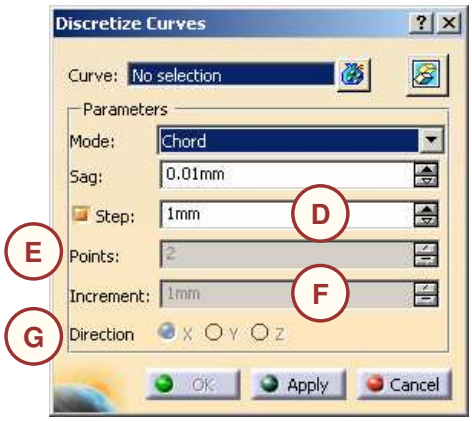
Discretizing Curves (2/2)

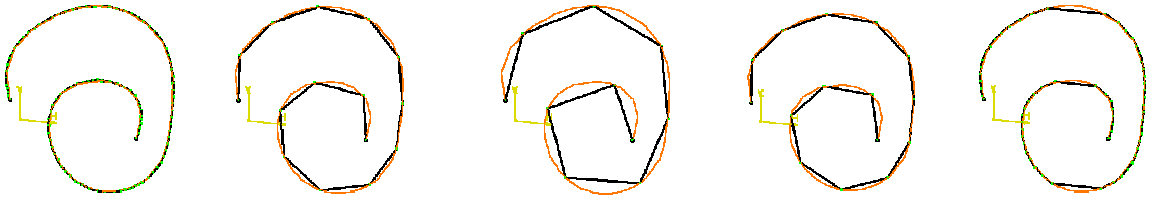
D. **Step:** The step represents the maximum distance between two successive positions of the scan. It's an optional parameter.

E. **Points:** This is the number of positions N. The length of the curve is divided by (N-1), providing a constant curve-length increment or curve-parameter increment.

F. **Increment:** This is the distance between two consecutive points or planes.

G. **Direction:** You specify a main direction (x, y or z) in order to define a reference plane.





Chord | Length + Positions | Length + Increment | Parameter + Positions | Direction + Increment

Copyright DASSAULT SYSTEMES

Instructor Notes:

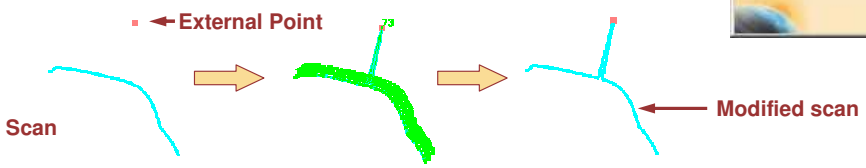
Editing Scans (1/3)

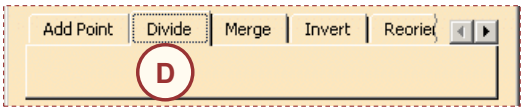
You can modify the scan using this command. A scan may need to be modified, to adjust its structure or content. Several options are available to edit the scan.

A. **Element:** You can multi-select the scans to modify it.

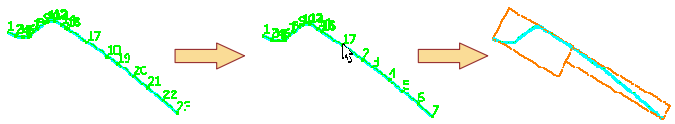
B. **Display rank:** The scan is displayed with points positions.

C. **Add Point:** You can select the existing point to modify the shape of the scan.





D. **Divide:** You have to pick a position of the scan. The scan is divided into two parts.



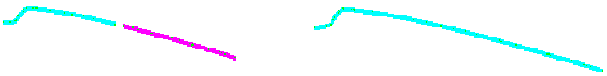
Single scan Pick the position Divided scan

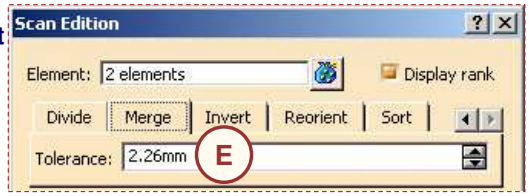
Copyright DASSAULT SYSTEMES

Instructor Notes:

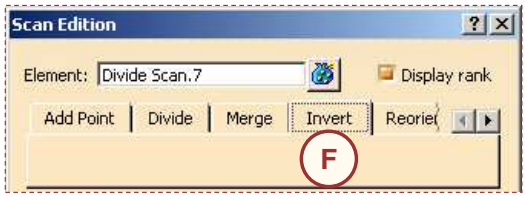
Editing Scans (2/3)

E. **Merge:** You defines the tolerance value. The tolerance value will get modified if the merge cannot be performed with the defined value.

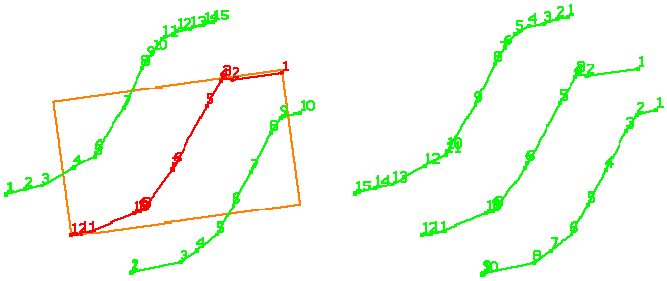


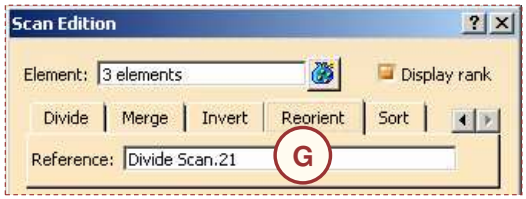


F. **Invert:** You can change the orientation of the scan.



G. **Reorient:** You can modify the orientations of the other scans to match with the reference.



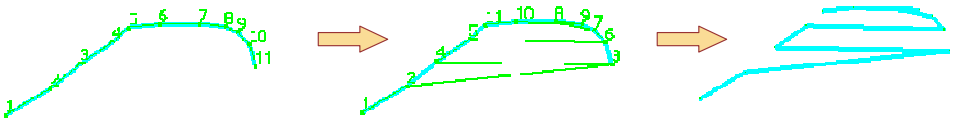


Copyright DASSAULT SYSTEMES

Instructor Notes:

Editing Scans (3/3)

H. **Sort: You can sort the positions inside each scan against the selected direction.**



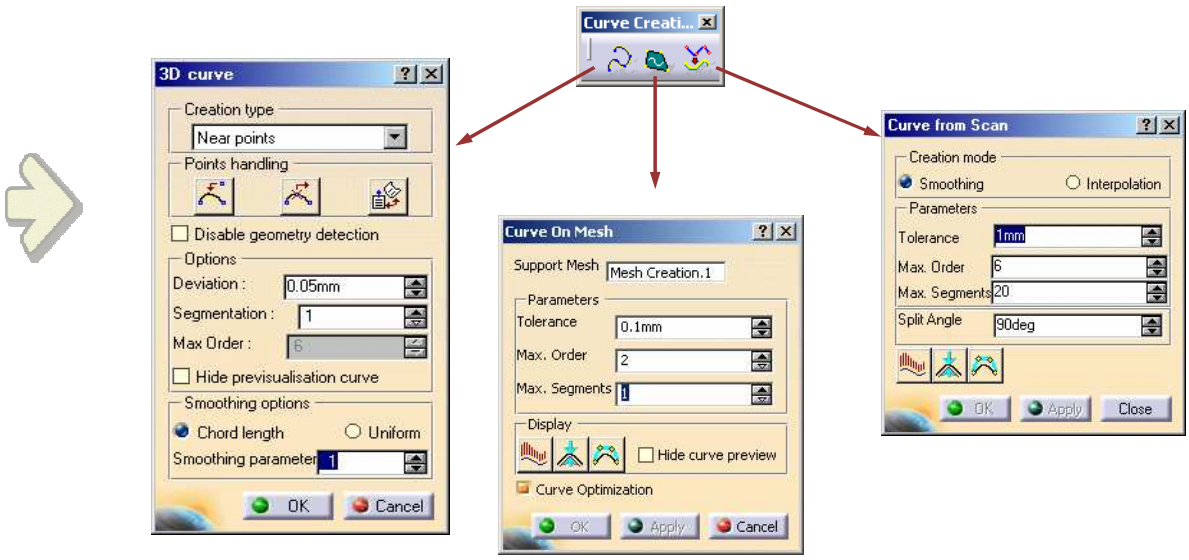
Copyright DASSAULT SYSTEMES

Instructor Notes:

Curve Creation

In this lesson, you will learn:

- To create curves from scans.
- To draw curves on a Mesh
- To draw 3D curves



Copyright DASSAULT SYSTEMES

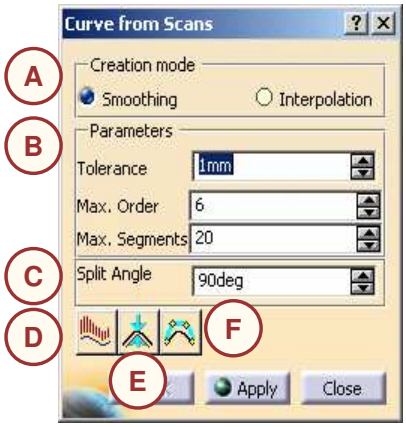
Instructor Notes:

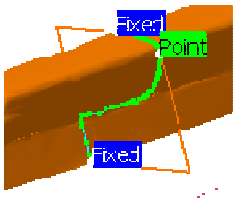
Creating Curves From Scans (1/3)

Curves can be generated from scans or set of scans using 'Curve from Scan' tool. 

A. Creation mode:
   a. Smoothing: Fit the points from the Scan by curve to a given tolerance.
   b. Interpolation: Creation of a bi-spline passing through the points of the scan.

B. Parameters
   a. Tolerance: Sets the maximum distance between the original Scan points and the resulting curve.
   b. Max. order: It is the maximum order of the curves created, i.e. the number of control points of those curves.
   c. Max. Segment: It is the maximum number of spans between two cutting points.





Copyright DASSAULT SYSTEMES

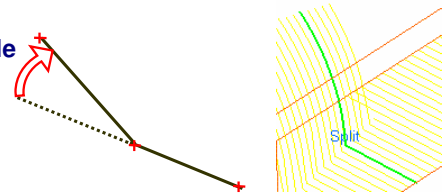
Instructor Notes:

Split Angle is very useful to detect the sharp edges.

Curves created can be edited in other CATIA workbenches.

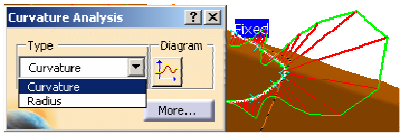
Creating Curves From Scans (2/3)

C. **Split Angle: This parameter is used to divide a computed curve into two curves according to an angle criterion. A split angle is proposed by default at 90 degrees. Whenever the computed curve forms an angle greater than this value, it is split automatically into two curves.**

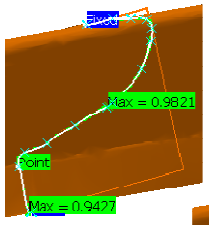


Split Angle

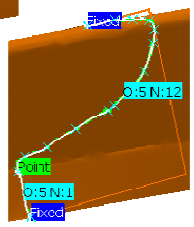
D. **Analyzes the curvature of the curves. You have the choice between a curvature analysis or a radius analysis.**



E. **Display the maximum deviation.**



F. **Display the order and the number of segments.**



Copyright DASSAULT SYSTEMES

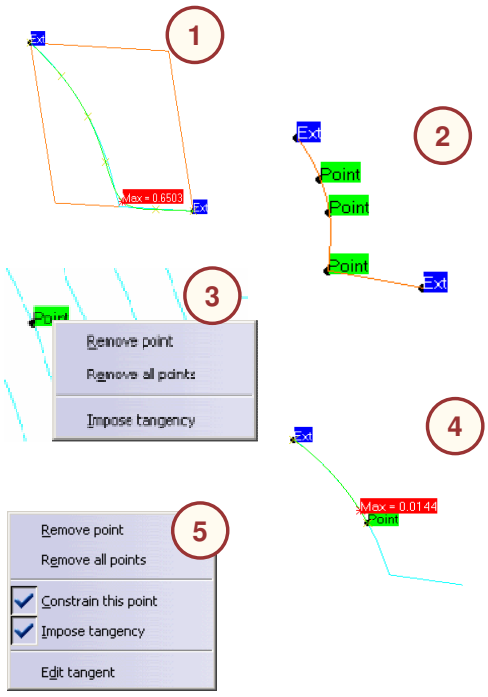
Instructor Notes:

Split Angle is very useful to detect the sharp edges.

Curves created can be edited in other CATIA workbenches.

Creating Curves From Scans (3/3)

1. When the curve computed is segmented, the segmentation is displayed as yellow x symbols. This color and symbol are not editable.
2. You can also add split points by picking points on the scan
3. The default constraint on a split point is "Point", i.e. passage. Click on the green square to change it to "Tangent". A second click will return it to "Point". You can also use the contextual menu of the constraint. Use the contextual menu of the constraint to remove the split point.
4. You can remove the extremity point of a computed curve, and replace it with a new or existing split point.
5. You can free or constrain the extremity point by checking the appropriate option in the contextual menu. You can also decide to remove the endpoint to select another point from the scan to be the endpoint. You may align in tangency the extremity of the curve to create with an existing curve or set a given tangency direction



Copyright DASSAULT SYSTEMES

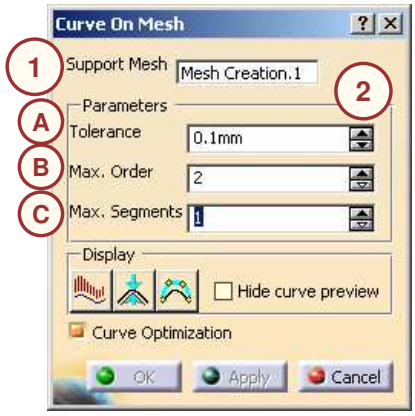
Instructor Notes:

Creating Curves on Mesh

1. **Mesh Support: Displays the name of the mesh on which the curve is created. It can be a multi-cells mesh.**

2. **Parameters: The parameters become active once the mesh has been selected.**

   A. **Tolerance: It is the tolerance used to smooth the scan formed by the points picked and the intermediate points, i.e. the maximum allowed distance between the resulting curve and the points of the scan.**
   B. **Max. Order: Maximum order of the curves created, i.e. the number of control points of those curves.**
   C. **Max. Segments: Maximum number of spans between two cutting points.**



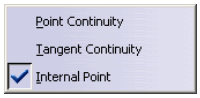
The symbol under the pointer indicates the current type of constraint. Picking a point on the mesh will create a new point with that constraint.

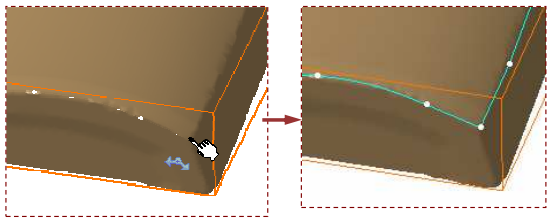
indicates a point continuity (G0 continuity),

indicates a tangent continuity (G1 continuity),

indicates an internal point.

Use the contextual menu to select a type of constraint





Point continuity

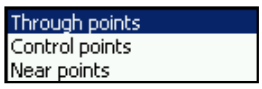
Copyright DASSAULT SYSTEMES

Instructor Notes:

Drawing Curves (1/3)

You can also create curves directly with '3D Curve' tool. 

A. Define creation option:



B. Insert a point

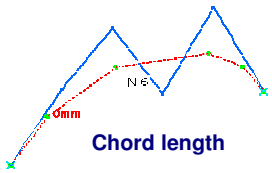
C. Remove a point

D. Constraint point on an element (curve, cloud…)

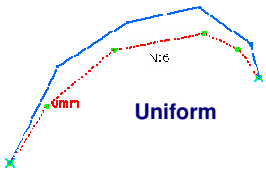
E. Define the smoothing accuracy if near points option is active

F. Define maximum number of segments of the curve if near points option is active

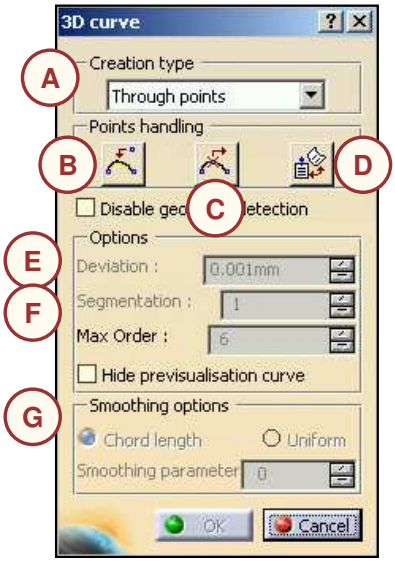
G. Choose a parameterization type:



Chord length



Uniform



Copyright DASSAULT SYSTEMES

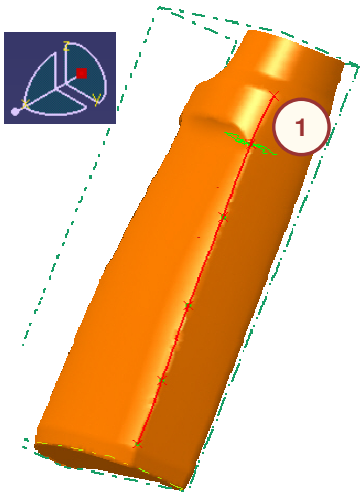
Instructor Notes:

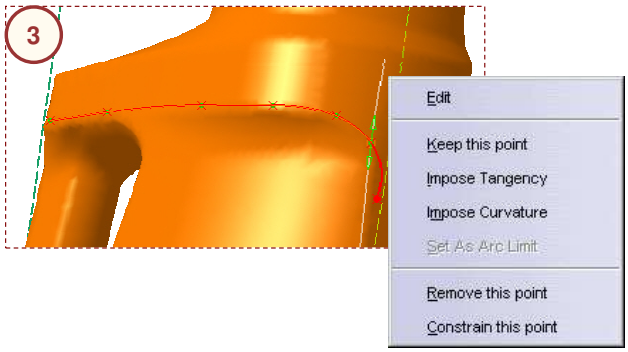
Drawing Curves (2/3)

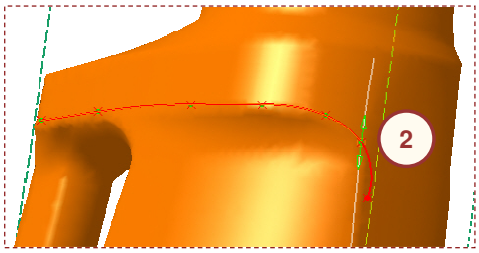
1. Select existing points on a cloud or on any existing element (curve, point…) or select new points taken in the current privileged plane (defined by the compass)

2. When a point is constrained on a curve you can move it along the curve with manipulators.

3. Right-click a point:
   a. to edit its position in space or on its support element
   b. to impose a tangency or a curvature
   c. to remove or constrain the point





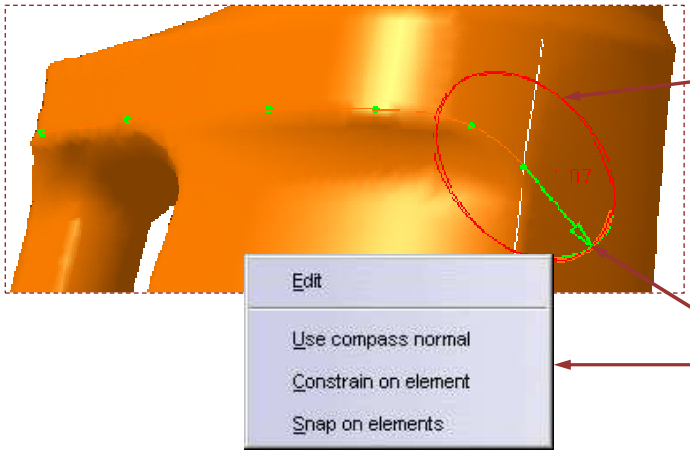


Copyright DASSAULT SYSTEMES

Instructor Notes:

Drawing Curves (3/3)

4



Define the tangency direction at a point using the circles as manipulators

Right-click the curve tangent arrow to access tangent definition options

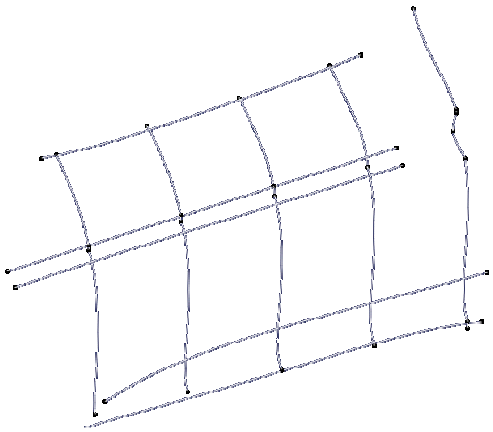
Copyright DASSAULT SYSTEMES

Instructor Notes:

To Sum Up

You have seen how to :

- **Create Scan Sections on a mesh.**
- **Create curves on the clouds.**
- **Draw Characteristic lines on a mesh.**



Creating curves and exporting the result

Copyright DASSAULT SYSTEMES

Instructor Notes:

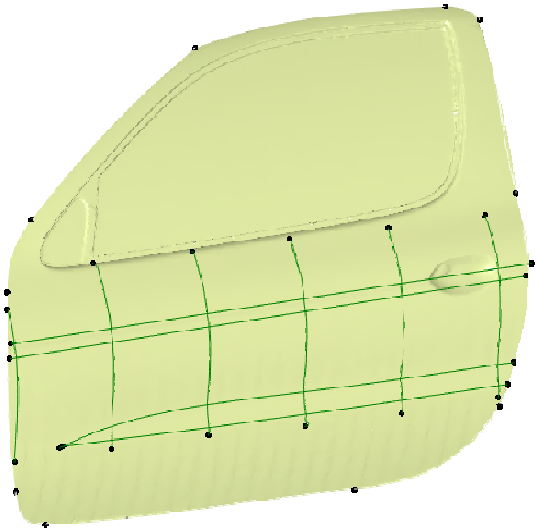
Car Door

Step 3– Creating the curves and exporting the result.

30 min

In this step you will:

- **Create planar sections on the mesh.**
- **Create curves from these sections.**
- **Draw characteristic lines on the mesh.**



Copyright DASSAULT SYSTEMES

Instructor Notes:

Advanced Tasks

You will learn advanced operations in Digitized Shape Editor.

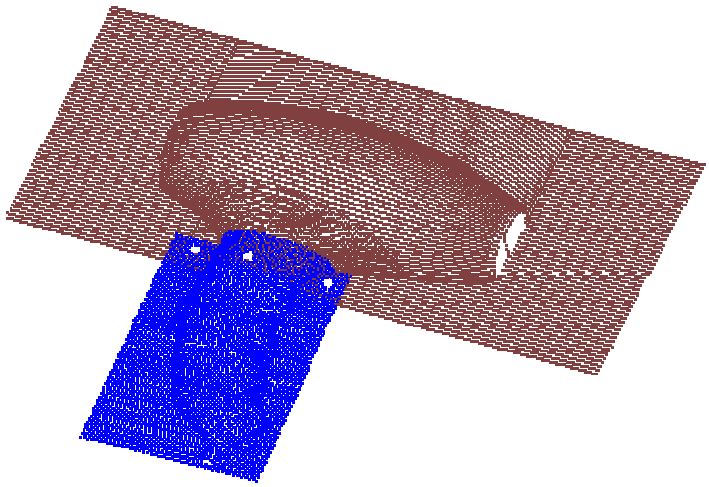
- Integration of Clouds
- Mesh Offset
- Cloud Export

Copyright DASSAULT SYSTEMES

Instructor Notes:

Integration of Clouds

In this lesson you will learn how to align several Clouds of Points, how to merge them, how to extract and how to disassemble them.



Copyright DASSAULT SYSTEMES

Instructor Notes:

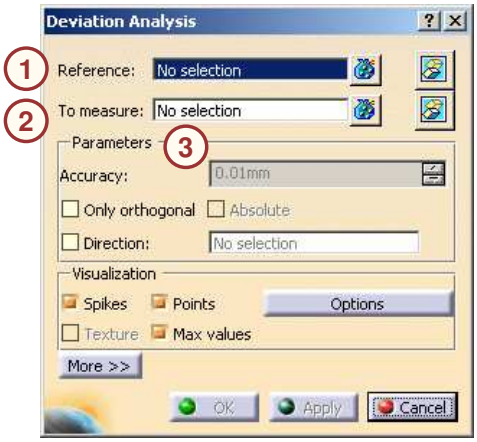
Checking Results (1/3)

Analyze the distance between any two geometric element.
1- Select the reference.
2- Select the features to analyze (Curve, Surface, Set of Surfaces).
3- Set the parameters.
4- Tune displays options.

1. **Reference: Select a volume, a surface, a plane, a point, a curve or a cloud of points.**

2. **To measure: Select clouds of points, points, curves, surfaces or volumes.**

3. **Parameters:**
   a. **Accuracy: Define computation accuracy when at least one volume or surface is selected as Reference.**
   b. **Only orthogonal: Select this check box to eliminate points that are not projected orthogonal on the Reference.**
   c. **Absolute: Select this check box to perform the analysis with absolute values only.**
   d. **Direction: select this check box to define a projection direction by picking a plane or a line.**



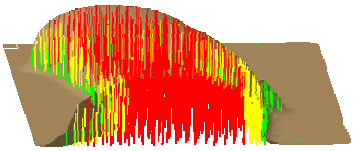
View mode must be set as "Materials"

Copyright DASSAULT SYSTEMES

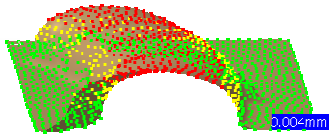
Instructor Notes:

Checking Results (2/3)

4. **Visualization:** Set the Visualization from the following options
   a. Spikes
   b. Points
   c. Texture
   d. Max values



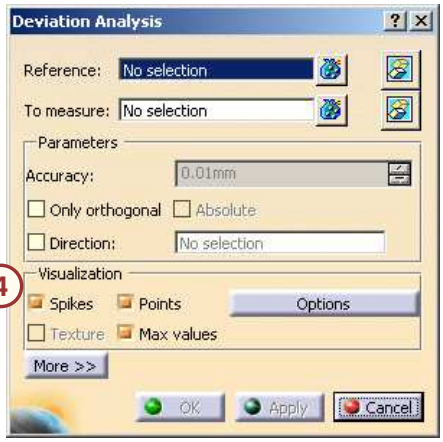
Spikes



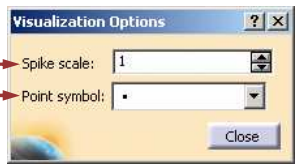
Points

Texture

Max values



Options:
Change the length of the spikes
Change the display symbol of points



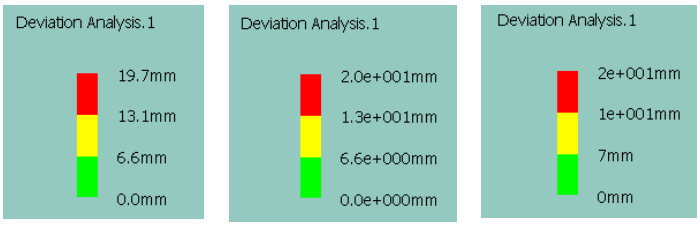
Copyright DASSAULT SYSTEMES

Instructor Notes:

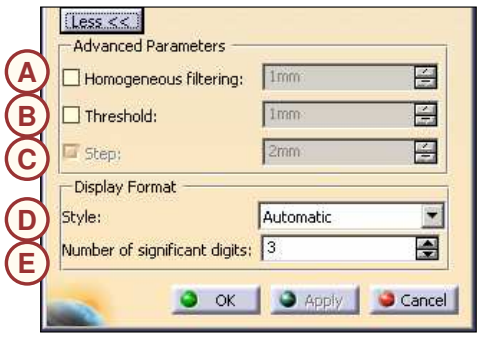
Checking Results (3/3)

More options:

A. **Homogeneous filtering: select this check box to reduce the number of points of the data**
B. **Threshold: select this check box to remove points with a deviation higher than this threshold.**
C. **Step: this parameter is used to divide the data to measure in small elements when they are curves, surfaces or volumes.**
D. **Style: Define the numerical display style of the scale. Three options are:**



Decimal **Scientific** **Automatic**



E. **Number of significant digits: Define the number of significant digits for numerical display of the values.**

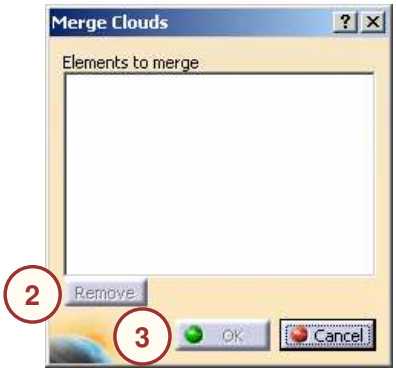
Copyright DASSAULT SYSTEMES

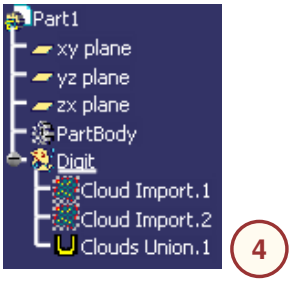
Instructor Notes:

Merging Clouds

You can merge several cloud of points into one using this command. 

1. **Select the clouds to merge.**
2. **You can remove a cloud from the selection using the Remove button.**
3. **Use OK to create the new Cloud of Points.**
4. **A new entity Cloud Union is created.**





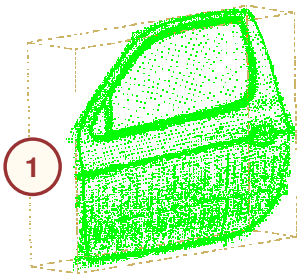
Copyright DASSAULT SYSTEMES

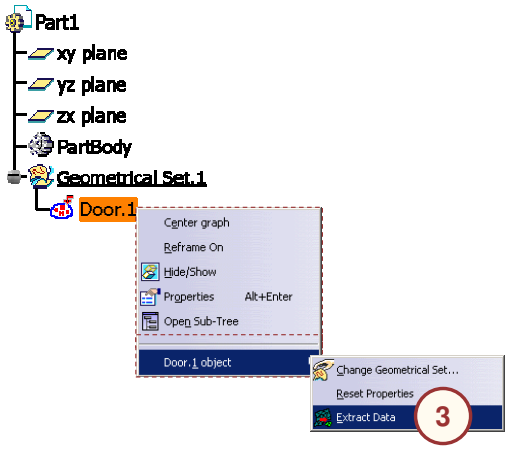
Instructor Notes:

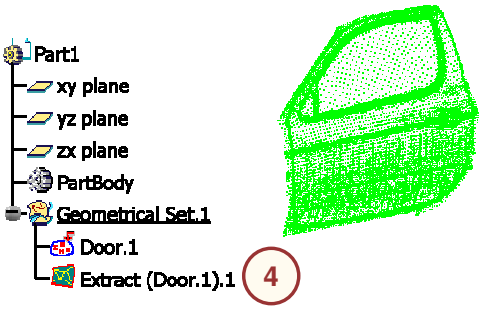
Extracting Data

You can extract the visible points of an existing cloud of points.
You can also extract the existing mesh.

1. Select the clouds to extract.
2. You can access the command from Cloud Operations toolbar.
3. You can also access the command from the contextual menu of cloud entity.
4. A new Cloud entity is created.







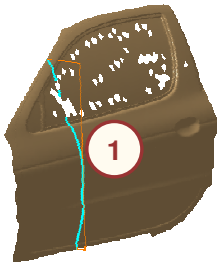
Copyright DASSAULT SYSTEMES

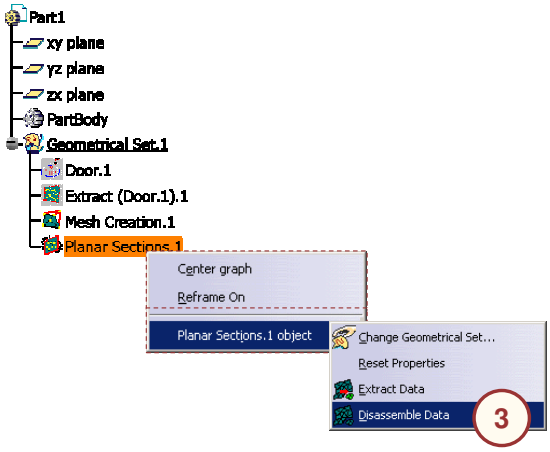
Instructor Notes:

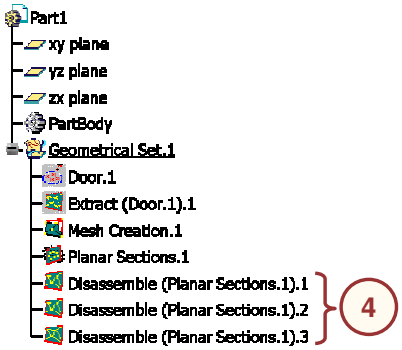
Disassembling Data

You can disassemble multi-cell clouds into mono-cell clouds. 

1. Select the clouds to extract.
2. You can access the command from Cloud Operations toolbar.
3. You can also access the command from the contextual menu of cloud entity.
4. New three Cloud entities are created.







Copyright DASSAULT SYSTEMES

Instructor Notes:

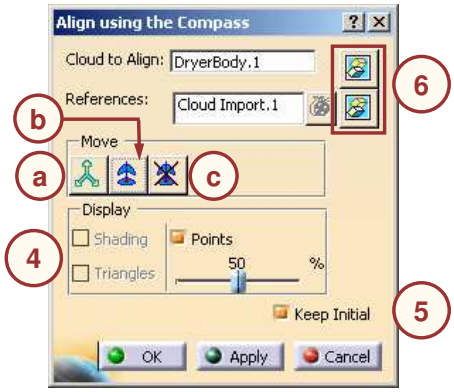
Aligning Point Data (1/4)

Move a cloud of points to position it relatively to another by using dynamic translation/rotation using the compass.

1. Select the cloud to align.
2. Select the reference cloud (not mandatory).
3. Move:
   a. If you selected a target cloud. Allows a first move using axis of inertia.

   b. The compass is moved at the center of gravity of the cloud. You can act on the compass to move the cloud.

   c. Return to the last move validated by Apply.
4. Display: For meshes Shading or Triangle display mode are available. Percentage of points is available for Clouds of points.
5. Keep Initial: Clear this check box if you do not want to create a copy of the initial Cloud to Align.
6. Select the respective Hide/Show icon to hide Cloud to Align or reference cloud.



Copyright DASSAULT SYSTEMES

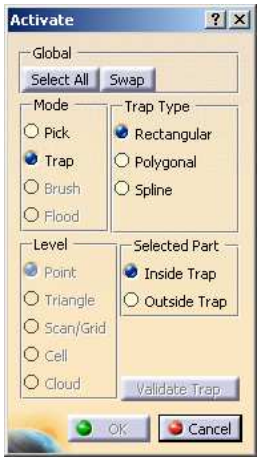
A new cloud is created.
Two axis systems are created: AxisRef.X (from the input cloud) and AxisTrs.x (to the output cloud).
Align with previous transformation is available.

Instructor Notes:

Aligning Point Data (2/4)

When the model has been digitized in several steps, you need to align the different clouds. In such case you can use 'Align by Best Fit' tool. The first picked cloud is moved to be fit to the reference. Then select a trap of points on each cloud that the system will try to fit.

1. **Select the cloud to move.**
2. **Select the reference element.**
3. **Draw the area to match on the cloud to move: To draw the area click the Activate icon.**
4. **Similarly draw the corresponding area on the reference cloud: To draw the area click the Activate icon.**
5. **Keep Initial: Clear this check box if you do not want to create a copy of the initial Cloud to Align.**
6. **Select apply to preview the result and then click OK.**
7. **A new cloud is created.**



- ***It is better to use the denser cloud as the reference cloud.***
- ***The result entity has the same structure as the input entity: scans, grids or polygons.***
- ***Select areas of uniform size and shapes on both clouds of points.***

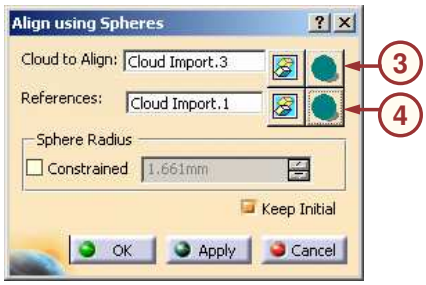
Copyright DASSAULT SYSTEMES

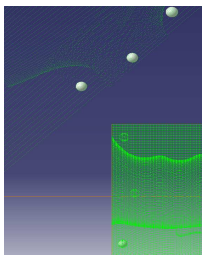
Instructor Notes:

Aligning Point Data (3/4)

When the model has been digitized in several steps and there are some spherical features, you can use 'Align using Spheres' tool. The first picked cloud is moved to be fit to the second picked cloud (reference).

1. Select the cloud to move.
2. Select the reference cloud.
3. Click on the right of Cloud to Align, and pick the spheres on the Cloud to Align.
4. When you are done, click on the right of References and pick the spheres on the References
5. Use OK to compute the alignment.





- *While aligning clouds, you can use the function Deviation analysis to check the output accuracy. The target will be the output cloud.*
- *We recommend that you pick the sphere in a direction orthogonal to the part to process, i.e. along the green axis and not along the black axis in our example.*
- *For an easier sphere recognition, we recommend that you pick in the middle of the sphere, not at the edge.*

Copyright DASSAULT SYSTEMES

Instructor Notes:

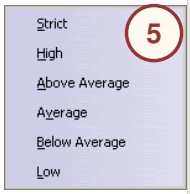
Aligning Point Data (4/4)

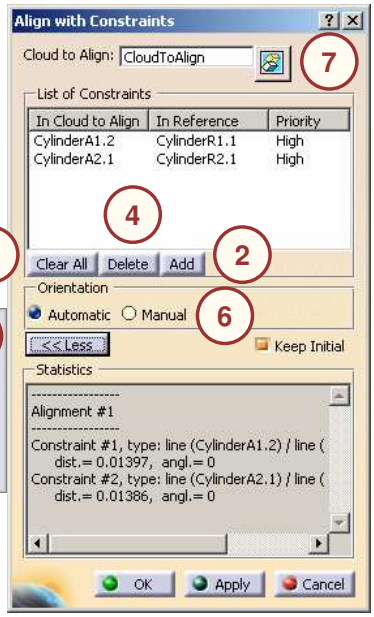
If the cloud has distinctive constraints (points, planes, cylinders), you can use 'Align Clouds with constraint' tool.

Principle:

Match canonic shapes by pair to define the transformation to apply. The number of constraints is not limited, you need to define enough constraints to solve the 6 degrees of freedom but if you define more, the alignment will be optimized.

1. **Select the cloud to move.**
2. **Select Add to input the constraints, then pick the pair of constraints to set, starting on the cloud you want to align. Select Add again to input more constraints.**
3. **Use Clear All to delete all constraints in one shot.**
4. **Use Delete to remove a constraint from the list.**
5. **Right-click a constraint to choose its priority level.**





6. **Select 'Manual' to display the directions of the constraints to be able to invert them.**
7. **Use the Hide/Show icon to hide the cloud of points to make the environment more clear.**

- ***When an alignment cannot be computed, an information message is displayed, you can then delete or add constraints, check that the constraints are consistent, or modify the constraint elements.***

Copyright DASSAULT SYSTEMES

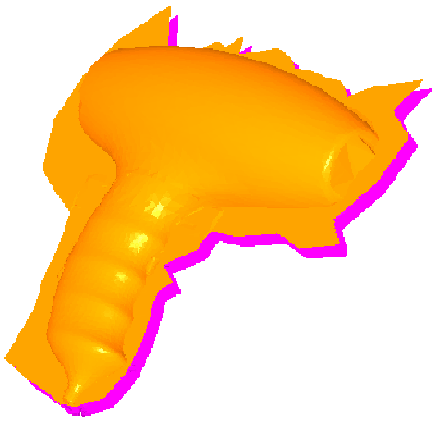
Instructor Notes:

When confirming the action, a new cloud is created and an axis system is created on both original and transformed cloud of points.

Temporary colors/transparency are assigned to the constraints to identify them.

Mesh Offset

You will learn how to offset a mesh.



Copyright DASSAULT SYSTEMES

Instructor Notes:

Creating an Offset

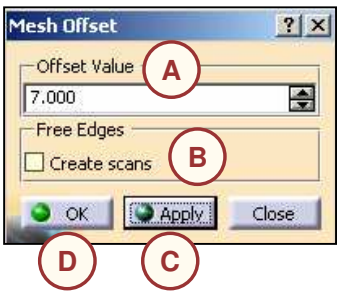
If you want to offset the mesh by small values, use 'Offset' tool. 

A. **Offset Value:** Type the value of the offset.

B. **Free Edges:** Create scans, will allow you to create the free edges of your new mesh as scan entities.

C. Click Apply to check the result.

D. Use OK to create the new mesh.



- *The offset is computed in the direction of the weighted normals of the points.*
- *For better results, you should avoid to enter a high offset value because no control of auto-intersection is performed, no control is performed for disappearing facets neither.*
- *For large offset values, you should use Rough Offset.*

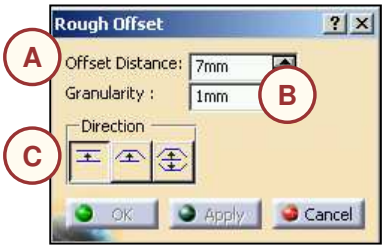
Copyright DASSAULT SYSTEMES

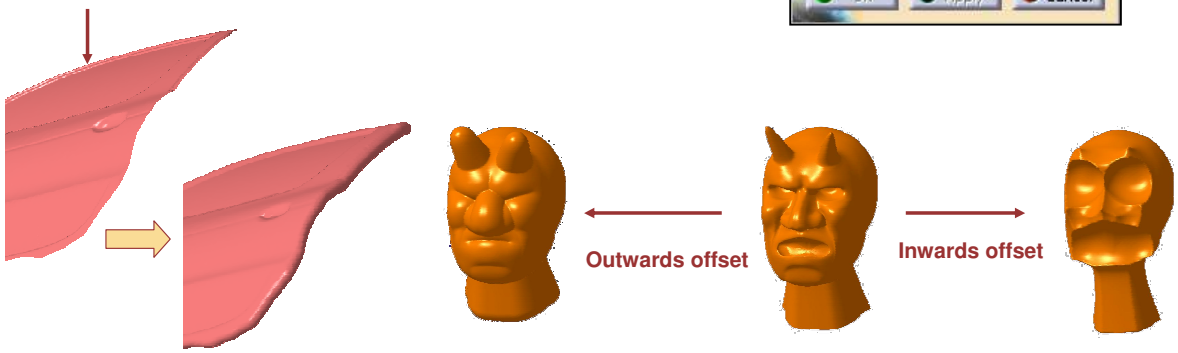
Instructor Notes:

Creating a Rough Offset

It enables to offset a complex mesh with auto-intersecting processing.

A. **Offset Value:** Type the value of the offset.
B. **Granularity:** Type a value to set the precision of the offset. The smaller the value of granularity, more accurate will be the offset mesh.
C. **Direction:** This option enables to round the borders. Besides you can offset and extend in both directions.





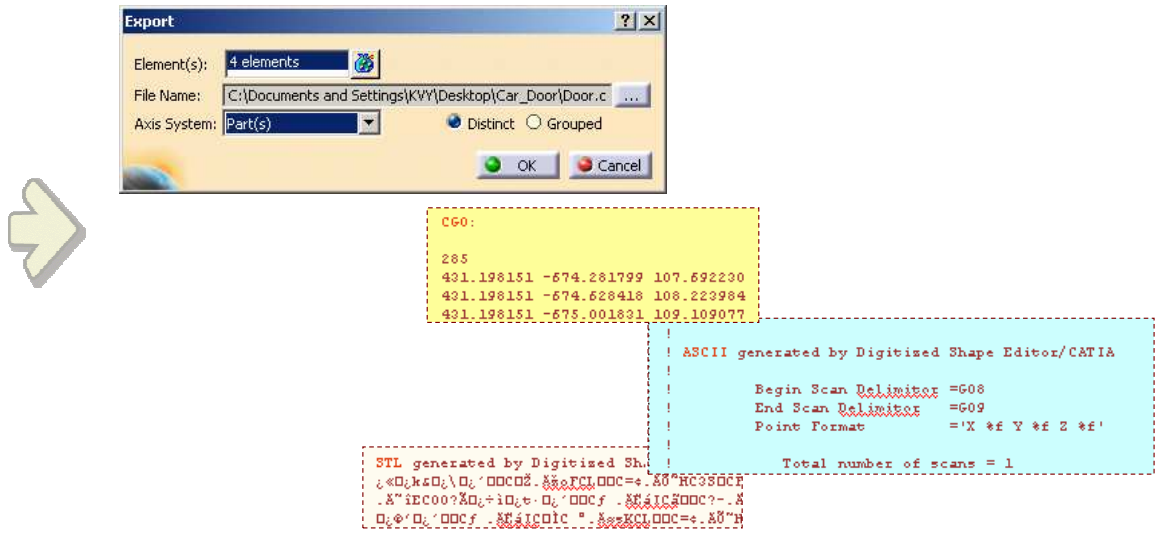
- *If you want to do an offset with a small offset value and if you don't have to figure out auto-intersection problems, you better use the other Offset option (previous slide) which is more accurate in these cases.*
- *The two Offset options can be useful when it comes to create rough stock.*

Copyright DASSAULT SYSTEMES

Instructor Notes:

Cloud Export

You will learn to export the data in various formats.



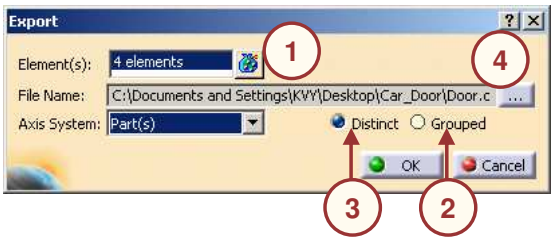
Copyright DASSAULT SYSTEMES

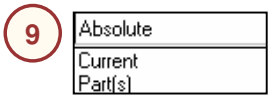
Instructor Notes:

Exporting the Result

You can create external format files from the clouds, scans or meshes using 'Export' tool.

1. **Select the elements to export. The number of elements selected is displayed in the 'Element(s)' field.**
2. **Click Grouped if you want to export all elements into a single file.**
3. **Click Distinct if you want to export each element into a distinct file.**
4. **Click '...' to enter the name and the path of the file to be created.**
5. **The Save As dialog box is displayed:**
6. **Type the name of the file you want to create**
7. **Select the format of the file you want to create.**
8. **Click Save to revert to the main dialog box.**
9. **Select in which Axis System you want to export the selection:**
   a. **Absolute: The selection will be exported using the axis system of the root of the document,**
   b. **Current: The selection will be exported using the current axis system,**
   c. **Part(s): The selection will be exported using the current axis system of each CATPart containing the selected elements.**





Copyright DASSAULT SYSTEMES

Instructor Notes:

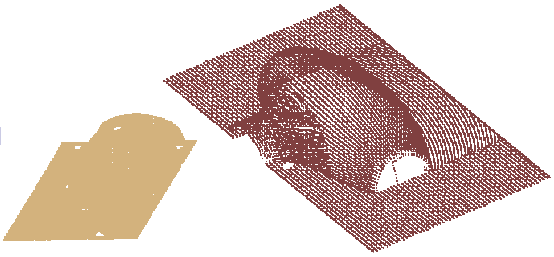
IN ASCII mode, the scans exported have the following delimiters: G08 for the start and G09 for the end.

You may get a STL file through the menu Save As.

Added Exercise - Hair Dryer

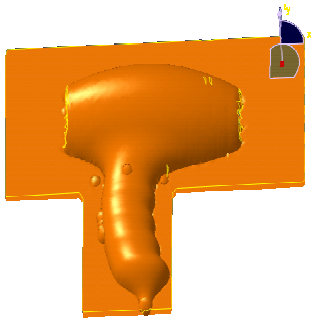
Exercise Presentation

75 min



In this exercise, you will create mesh from the Digitized data of Hair Dryer. For that you have to perform following steps.

- Import several clouds of points.
- Align the cloud of points.
- Create mesh.
- Offset the mesh.



You will require the following CATIA licenses to perform the exercise:
1. HD2. slt
2. DSE. prd
3. DSS. prd

Copyright DASSAULT SYSTEMES

Instructor Notes:

To Sum Up

In this course you have seen:

- **How to import and process point data.**
- **How to create and process meshes.**
- **How to create scans and curves on meshes.**
- **How to align, offset and export clouds.**

Copyright DASSAULT SYSTEMES

Instructor Notes: