



# CATIA Training Exercises

# FEM Surface

Version 5 Release 15  
May 2005  
EDU-CAT-E-FMS-FX-V5R15

## Table of Contents (1/3)

|                                         |           |
|-----------------------------------------|-----------|
| <b>1. Meshing of a Cross-member</b>     | <b>5</b>  |
| Mesh specifications                     | 9         |
| Constrain edges and mesh                | 14        |
| Mesh modifications                      | 19        |
| Mesh quality                            | 25        |
| <b>2. Floor Mesh</b>                    | <b>28</b> |
| <b>3. Fuselage Door Static Analysis</b> | <b>30</b> |
| <b>4. Spot welding Analysis</b>         | <b>33</b> |
| Spot welding Analysis connections       | 36        |
| Meshing                                 | 38        |
| Pre-processing in GAS                   | 42        |
| Analysis and Post-Processing            | 44        |
| <b>5. Seam welding Analysis</b>         | <b>47</b> |
| Seam welding Analysis connections       | 50        |
| Meshing                                 | 52        |
| Seam Welds Meshing                      | 54        |



**Table of Contents (2/3)**

|                                                  |           |
|--------------------------------------------------|-----------|
| <b>6. Surface welding Analysis</b>               | <b>57</b> |
| Surface welding Analysis connections             | 60        |
| Meshing                                          | 62        |
| Surface Welds Meshing                            | 64        |
| <b>7. Compatible Seam Welds</b>                  | <b>67</b> |
| <b>8. Spot Welds With Hemming</b>                | <b>73</b> |
| <b>9. Wing Structure Analysis</b>                | <b>78</b> |
| Rib meshing                                      | 81        |
| Spars and central box meshing                    | 85        |
| Skins meshing                                    | 89        |
| Pressure with Data Mapping                       | 94        |
| Static Analysis and Post-Processing              | 96        |
| <b>10. Meshing of Parametric Conceptual Body</b> | <b>99</b> |
| Creating a mesh according to specifications      | 102       |
| Modify and update an existing mesh               | 104       |
| Analyze Mesh Quality                             | 107       |
| Static Analysis Performance                      | 109       |

**Table of Contents (3/3)**

|                                                      |            |
|------------------------------------------------------|------------|
| <b>11. Meshing of a Fuselage</b>                     | <b>112</b> |
| <b>12. Fuselage connections</b>                      | <b>123</b> |
| <b>13. Pan Floor Mesh</b>                            | <b>131</b> |
| <b>14. Octree Mesh on Block Cylinder</b>             | <b>139</b> |
| <b>15. Hybrid 1D and 2D Meshes</b>                   | <b>148</b> |
| <b>16. Extrude 1D Elements into 2D Mesh</b>          | <b>156</b> |
| <b>17. Mesh Parameters Exposition to Knowledge 1</b> | <b>162</b> |
| <b>18. Mesh Parameters Exposition to Knowledge 2</b> | <b>169</b> |
| <b>19. Mesh Parameters Exposition to Knowledge 3</b> | <b>173</b> |
| <b>Hints and Tips</b>                                | <b>178</b> |

# Master Exercise

## 1. Meshing of a Crossmember



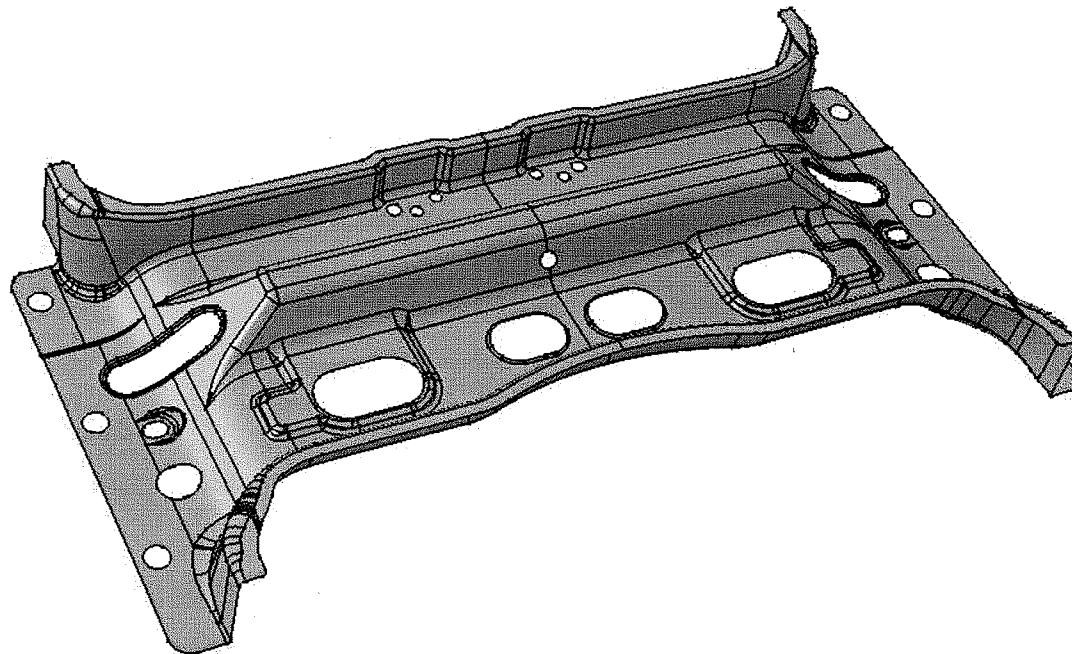
60 min.

In this exercise you will learn to mesh a cross-member with FEM Surface tools.

You will use :

Mesh definition

- ◆ Mesh specifications
- ◆ Mesh modifications
- ◆ Mesh quality

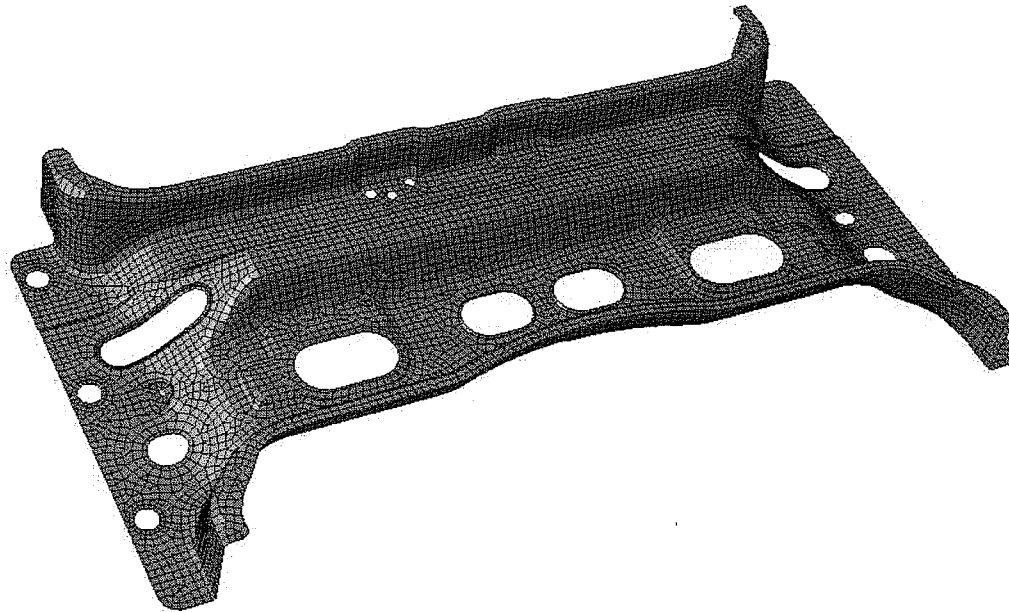


## Fundamental steps

- **Creating a mesh according to specifications**
  - ◆ Create a new surface mesh object.
  - ◆ Remove holes and gaps.
  - ◆ Define edge constraints and node distributions.
  - ◆ Define Domain specifications
  - ◆ Create the mesh.
  
- **Modify and update an existing mesh**
  - ◆ Modify the geometry.
  - ◆ Update the geometry and the mesh.
  - ◆ Re-mesh a domain.
  
- **Analyze Mesh Quality**
  - ◆ Analyse the mesh quality.
  - ◆ Edit an analysis report.
  - ◆ Modify the mesh to increase quality.
  
- **Static Analysis Performance**
  - ◆ Perform a torsional stiffness analysis.
  - ◆ Compute and display the displacements.

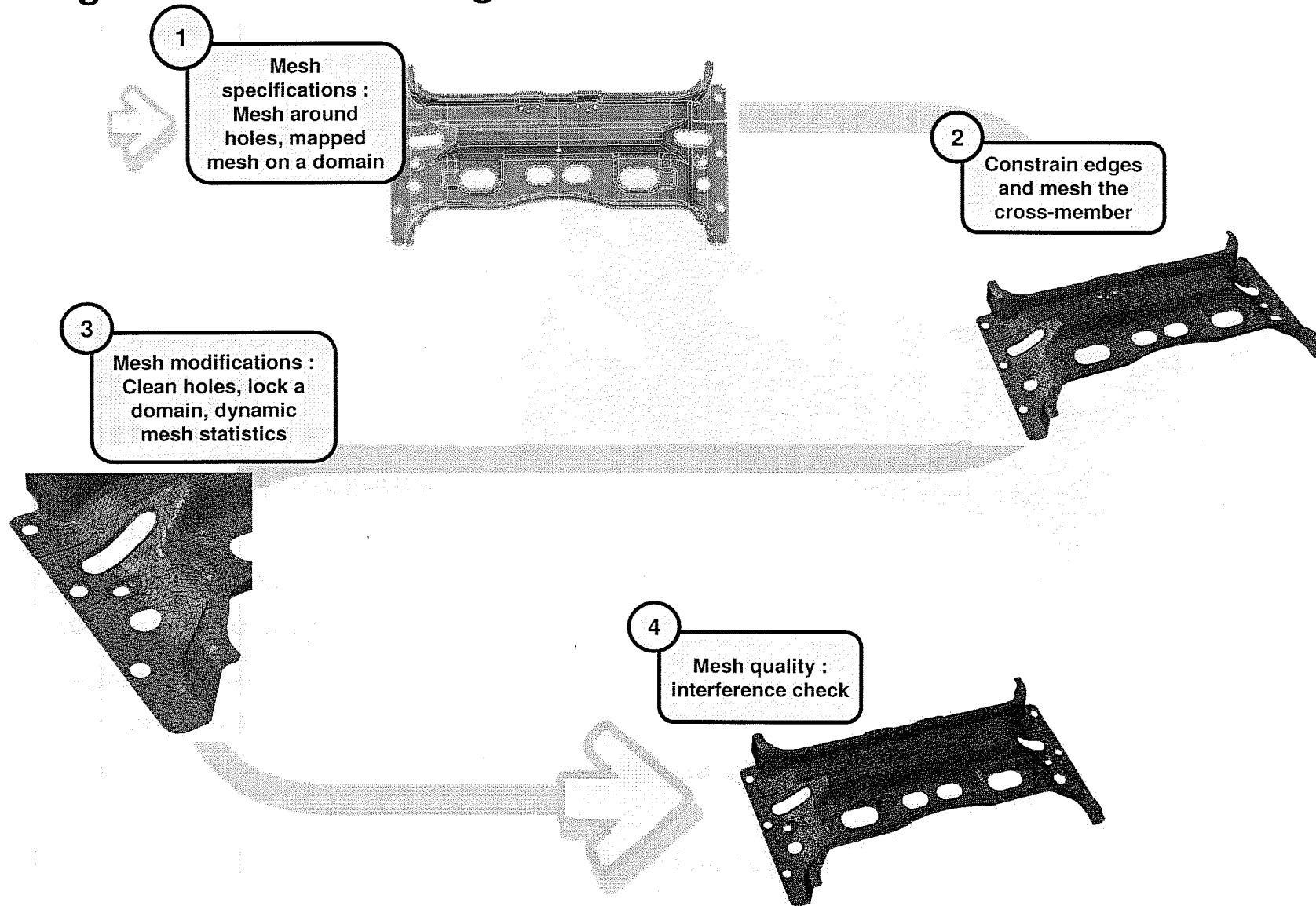
## Design Intent : Meshing of a Cross-member

- Mesh around holes with elements heights specifications
- Impose elements around filled hole
- Define mapped mesh on a domain
- Constrain an edge over a hole
- Edit a mesh-part
- Clean holes
- Dynamic mesh statistics
- “Quad to tria” conversion
- Lock a domain
- Interference check



⚙ Load : Crossmember.CATPart

## Design Process : Meshing of a Cross-member



# Master Exercise

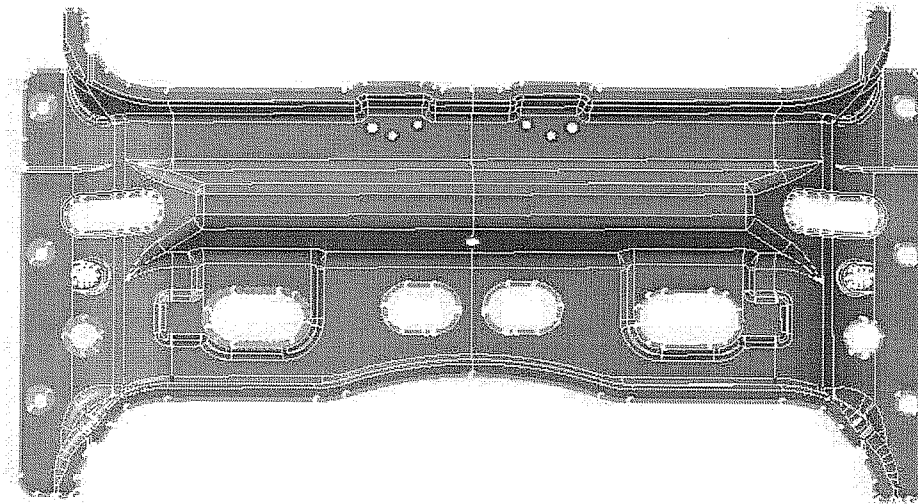
## Meshing of a Cross-member : Mesh specifications



15 min.

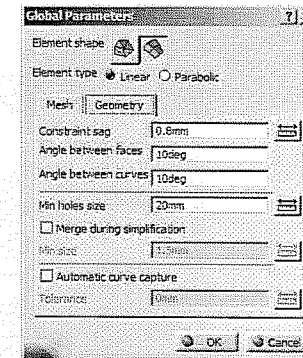
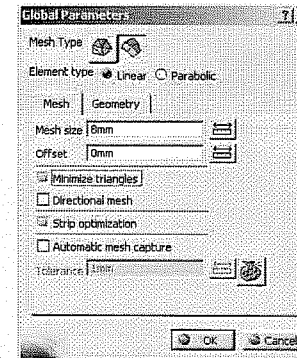
In this step you will :

- ◆ Define mesh specifications.
- ◆ Impose elements around filled hole.
- ◆ Mesh around holes with elements heights specifications.
- ◆ Define a Mapped mesh on a specific domain

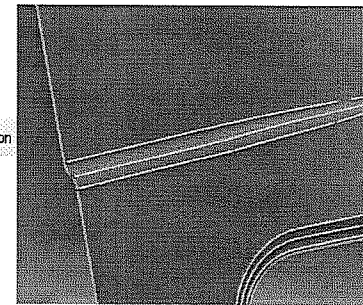


## Do It Yourself (1/4)

- Go to Advanced meshing tools workbench
- Select the **Advanced Surface Mesher** icon
  - Select the part and enter a 8mm global size and a 0.8mm sag, and 20mm as min holes size.
  - Uncheck *Merge during simplification* option, click OK and select *Geometry simplification*. Visualize constrained edges.
  - Go back in specifications toolbar with *Remove simplification* icon. Click on *Global meshing parameters* and enter 5mm as min size for *Merge...* option. Check constrained edges : what are the differences ?
  - Go back in specifications toolbar with *Remove simplification* icon.

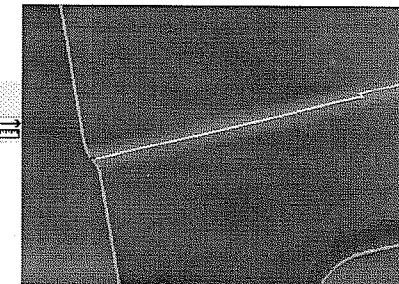


☐ Merge during simplification



☒ Merge during simplification

Min size 5mm





## Do It Yourself (2/4)

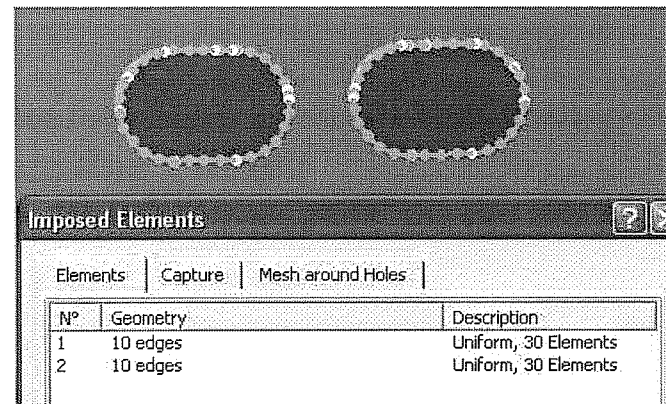
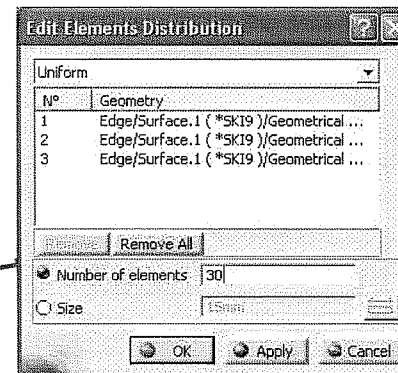
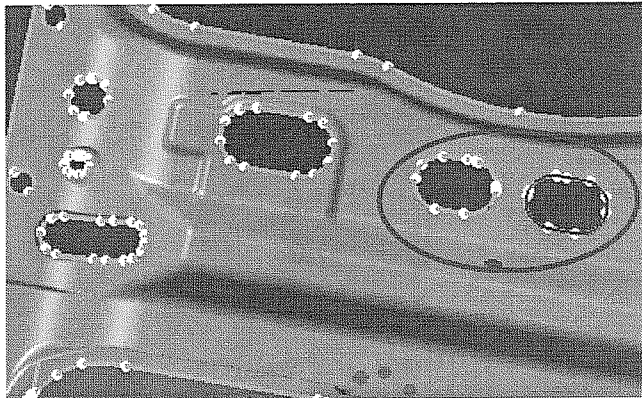
- Select *Imposed elements* icon.



- Click on *Elements* tab and multi-select the one of the central hole edges (12 edges).

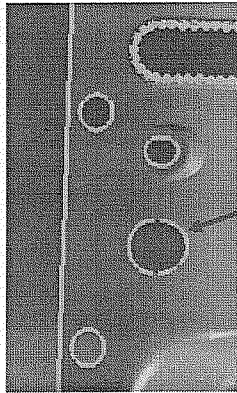
To multi select: click on first edge with SHIFT key on, then select a highlighted edge to end the path

- Enter 30 as number of elements.
- Idem for second central hole



## Do It Yourself (3/4)

- Select *Imposed elements* icon .
- Select *Mesh around holes* and click on the bigger hole, on the left side of the part. Enter 20 as number of elements with 2 rows :
- Click on *Specify heights of elements* and enter 5mm for *Height of row 1* and 10mm for *Height of row 2*.



**Mesh Around Holes**

Number of elements: 20

Number of rows: 2

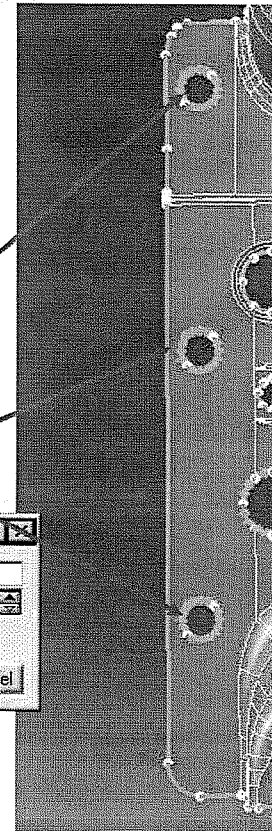
☒ Specify heights of elements

Height of row 1: 5mm

Height of row 2: 10mm

OK Apply Cancel

Choose *Specify heights of elements* and enter heights values.



- Select a small hole, close to the left edge of the part, and enter 16 number of elements with only one row. Repeat for the 2 others holes.

**Imposed Elements**

Elements | Capture | Mesh around Holes

| N° | Geometry                           | Description         |
|----|------------------------------------|---------------------|
| 1  | Edge/Surface.1 (*SK19)/Open_body.1 | 2 rows, 20 Elements |
| 2  | Edge/Surface.1 (*SK19)/Open_body.1 | 1 rows, 16 Elements |
| 3  | Edge/Surface.1 (*SK19)/Open_body.1 | 1 rows, 16 Elements |
| 4  | Edge/Surface.1 (*SK19)/Open_body.1 | 1 rows, 16 Elements |

Edit Remove Remove All Zoom

OK Cancel

**Mesh Around Holes**

Number of elements: 16

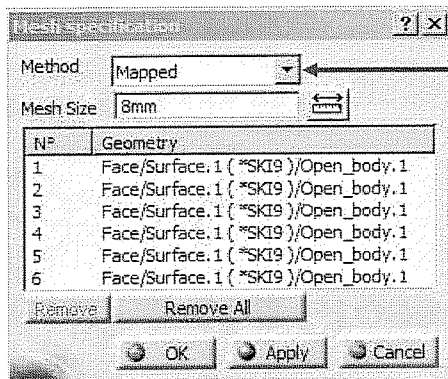
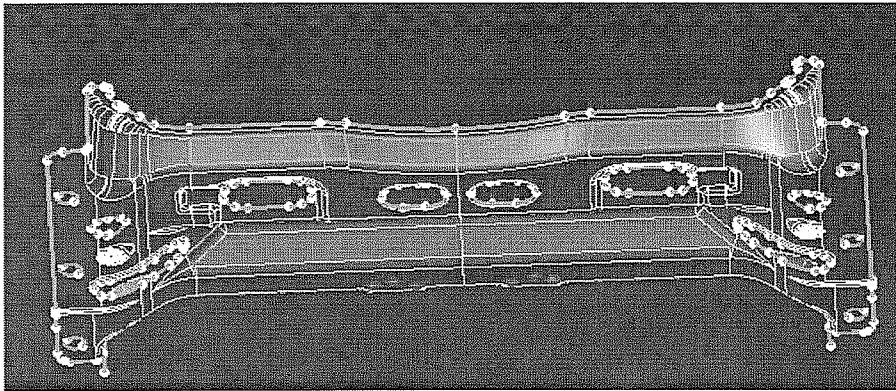
Number of rows: 1

☐ Specify heights of elements

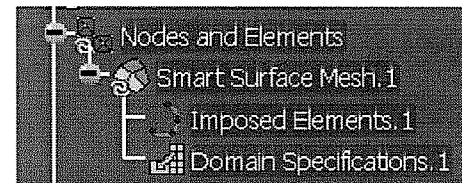
OK Apply Cancel

## Do It Yourself (4/4)

- Select **Domain Specifications** icon .
- Select 6 side faces as shown.
- Choose Mapped as meshing method. Enter 8 mm as mesh size



Meshing method list



Domain specifications are stored in the tree, will be applied for each mesh update.

They can be edited, modified, removed.

# Master Exercise

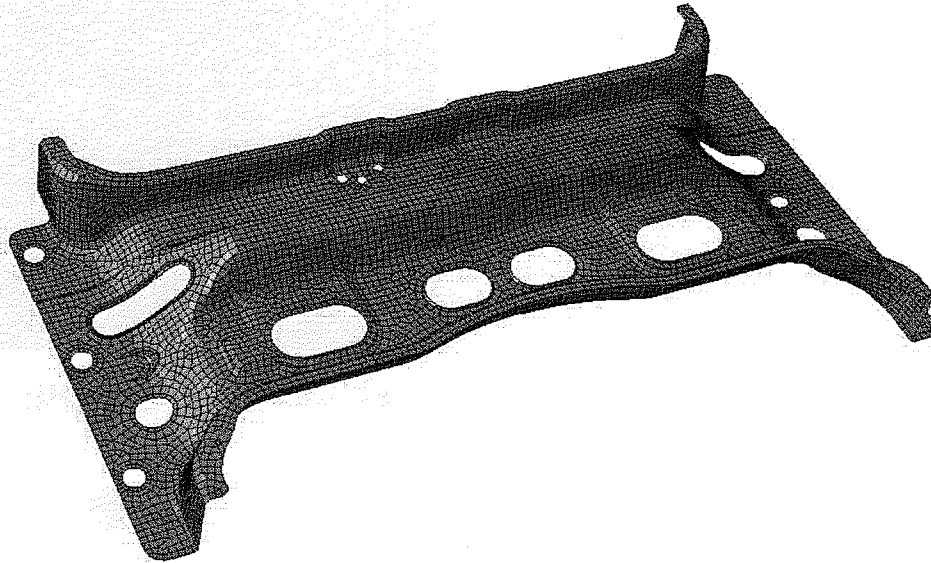
## Meshing of a Cross-member : Constrain edges and Mesh



15 min.

In this step you will :

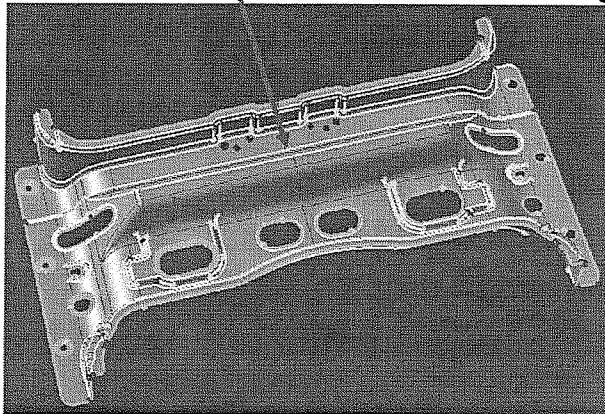
- ◆ Constrain an edge over an ignored hole.
- ◆ Mesh the part and see elements distributions around holes.
- ◆ Check mesh statistics.
- ◆ Edit a mesh-part.



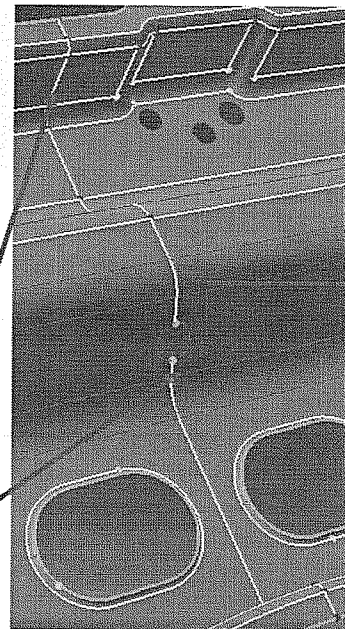
## Do It Yourself (1/4)

- Select the **Geometry simplification** icon 

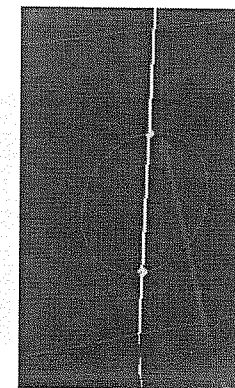
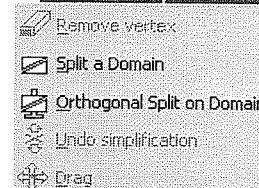
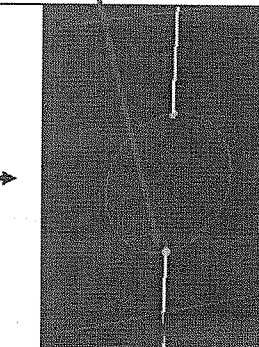
yellow = constrained edges and blue = ignored edges are displayed.
- Edit simplifications : 
  - Deactivate **Auto Remesh on modification**
  - Constrain all the middle edges.
  - Select a vertex of the middle ignored hole : contextual menu, choose **Split a Domain**. Select the other vertex : left click. The constrained edge is created over the hole.
  - All discontinuous edge paths can be constrained like this.



Constrain all the middle edges.



1. Right click on the first vertex.

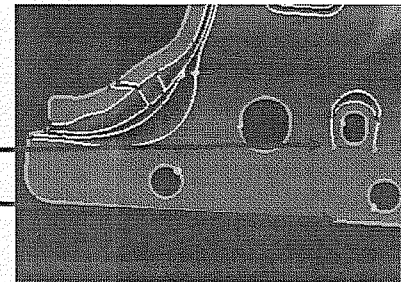
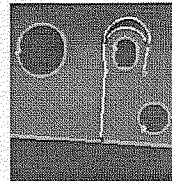


2. Left click on the second vertex. The constrained edge is created.

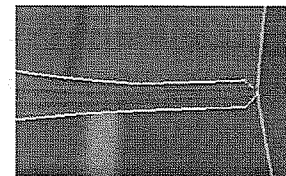
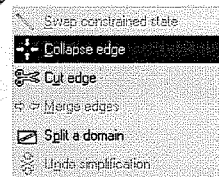
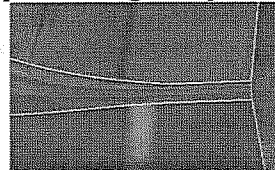
## Do It Yourself (2/4)

### Edit simplifications :

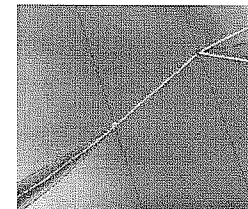
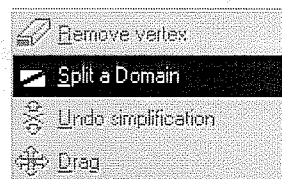
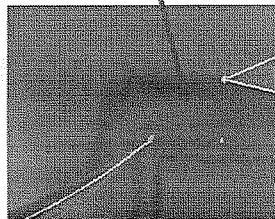
- Click on the *Edit simplifications* icon.
- On green vertex, choose *Orthogonal Split on Domain*
- Click on this green vertex
- Click on green edge: Split will be orthogonal



- Click right between the two vertices (on green edge) of close constrained edges.
- Choose *Collapse edge* option.



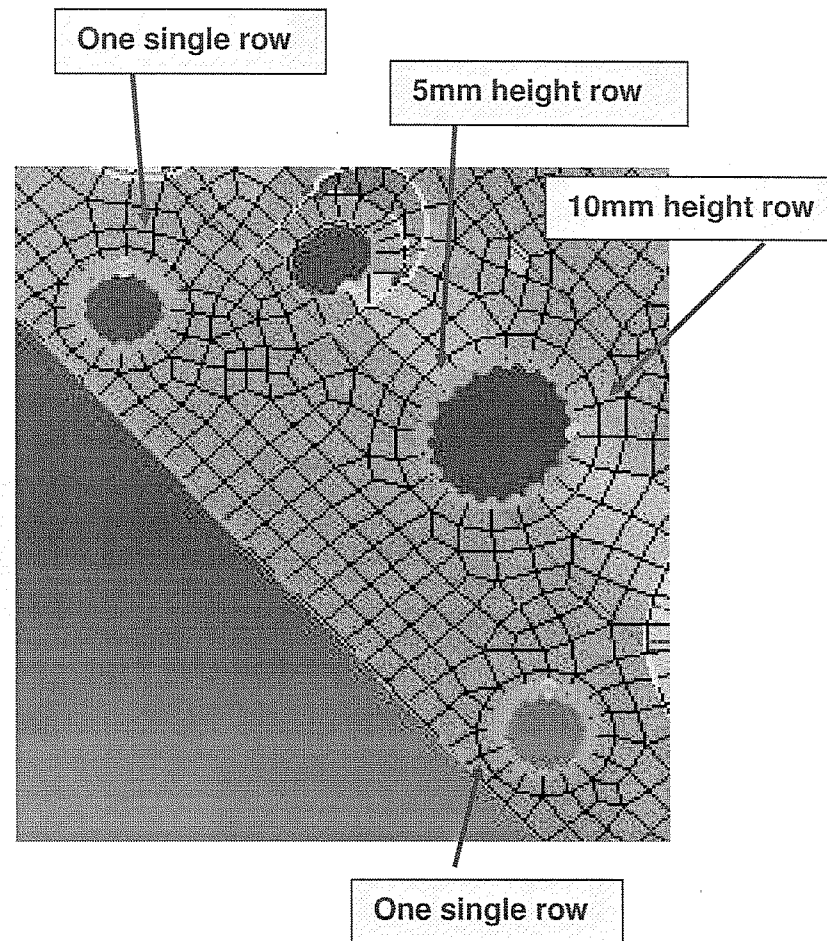
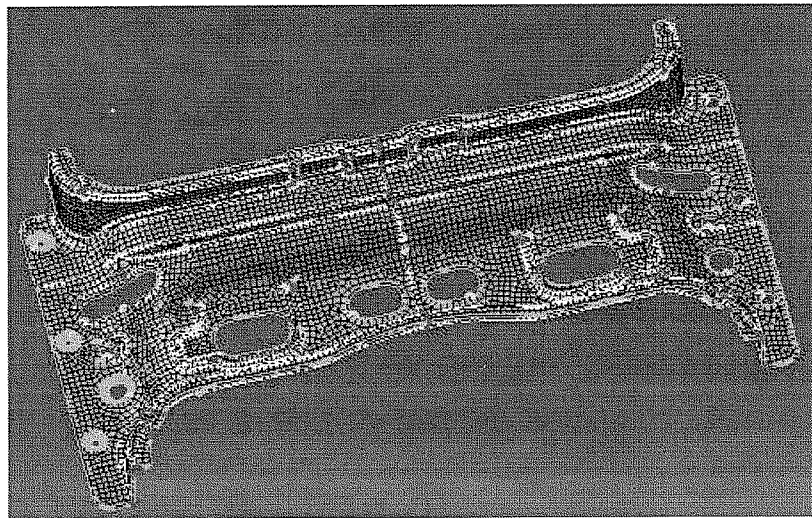
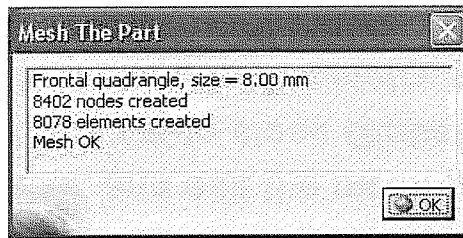
- Search a discontinuous constrained edges path : choose *Split a domain*.





## Do It Yourself (3/4)

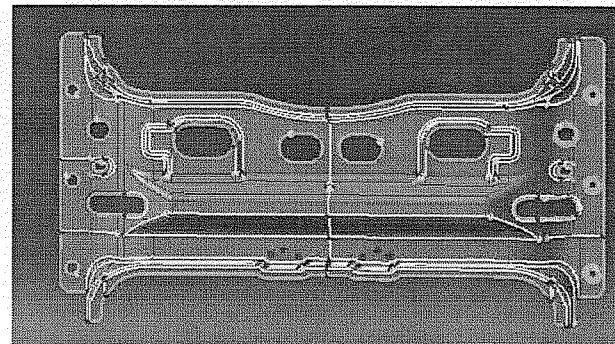
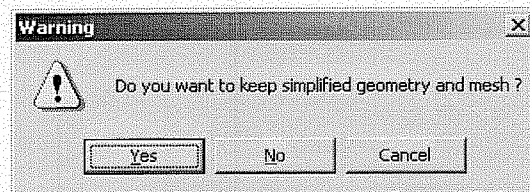
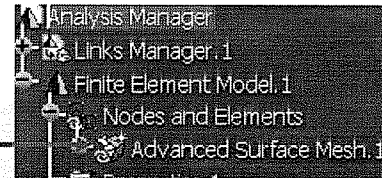
- Select *Mesh the part* icon . Click *OK*.
- See the elements distribution around holes.
- Exit .



## Do It Yourself (4/4)

### ■ Edit the created mesh-part :

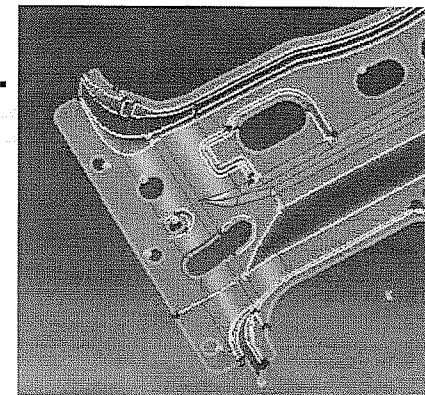
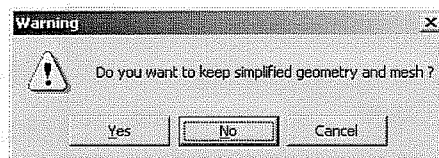
- Double click on the mesh-part in the tree.
- Click *OK* to close *Global Parameters* panel
- Choose *Yes*. What happens about the mesh and the specifications ?



- Exit, and edit the mesh-part again.
- Click *OK* to close *Global Parameters* panel and Click *No*
- And click on *Geometry simplification* icon to visualise constrained and unconstrained edges.



What happens ? What is the difference with the first case.





# Master Exercise

## Meshing of a Cross-member : Mesh modifications



15 min.

In this step you will :

- ◆ Clean holes.
- ◆ Modify mesh manually.
- ◆ Do dynamic mesh statistics.
- ◆ Lock a domain.
- ◆ Quad to tria conversion.



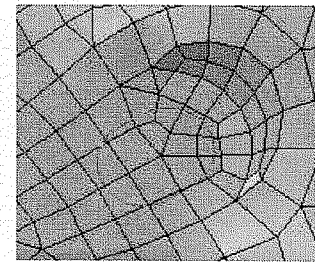
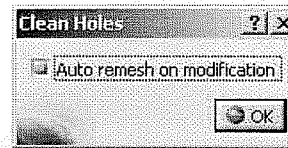
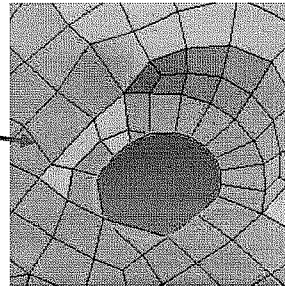
## Do It Yourself (1/5)

- Re-mesh the part. 

- Select the *Clean holes* icon 

- Select hole to ignore. *Auto-remesh on modification* option is activated.

Select the hole to ignore.



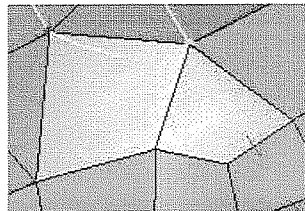
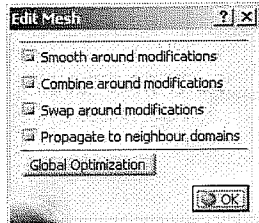
- Click *OK* to close the panel

## Do It Yourself (2/5)

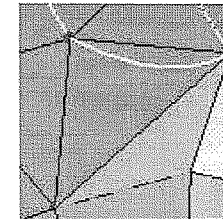
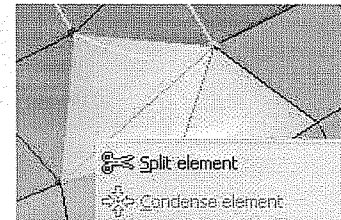
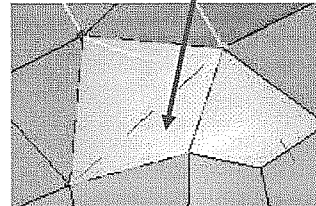
- Select the *Edit Mesh* icon.



- Split a quad with *Smooth* option on



Highlighted  
element to  
be split



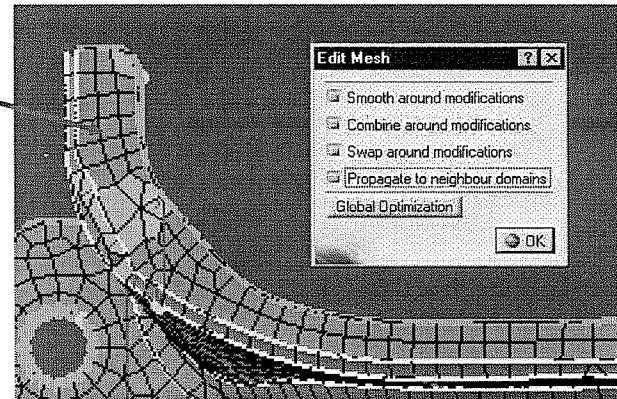
- Click OK to close the panel

- Modify manually a complex area :



- We want to only have 2 rows of quads elements in this domain.

- Activate *Smooth around modifications* option to have a better mesh after modifications.
- Right click on quads edges selecting *Condense nodes* option.
- Or right click inside quads elements selecting *Condense element* option.
- Use *undo* function if necessary.
- See methodology example next page.

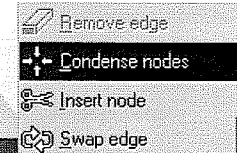


Student Notes:

## Do It Yourself (3/5)

1

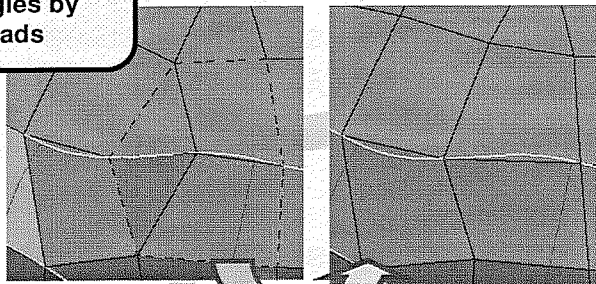
Condense nodes so as to have two rows of elements



Put the mouse pointer between the two nodes to condense

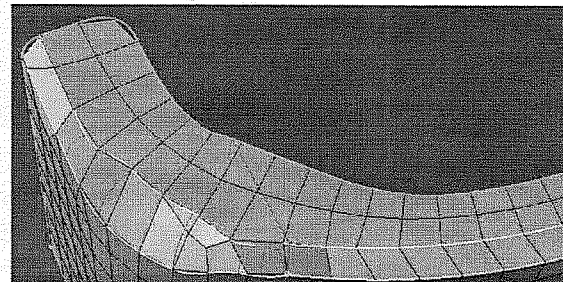
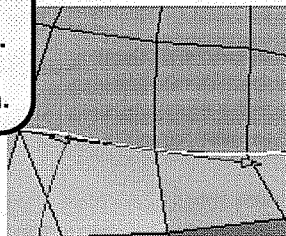
2

Condense nodes to replace triangles by quads

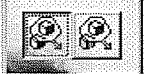


3

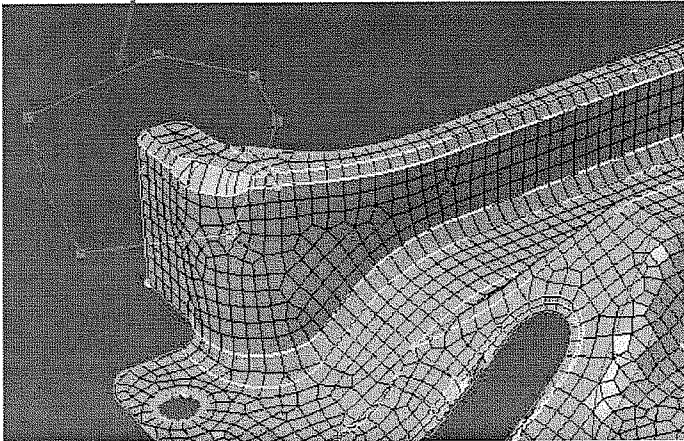
Move nodes manually to have better quads shape. Use *Global Optimization* option.



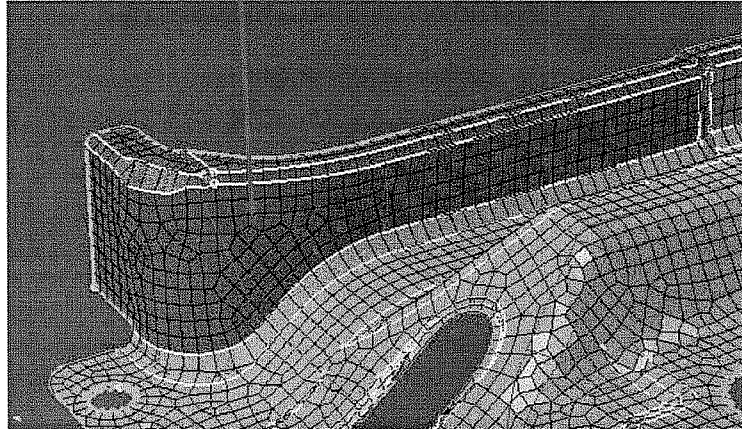
## Do It Yourself (4/5)

- Select the **Lock Domain** icon  .
  - Choose **Select objects by an intersection polygon trap**  .
  - Define your trap as the following captured image.
  - Try to modify the locked mesh with **Re-mesh Domain** and **Edit Mesh** : verify you are not able to do it.
- Exit  .

Polygon trap

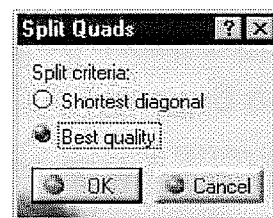
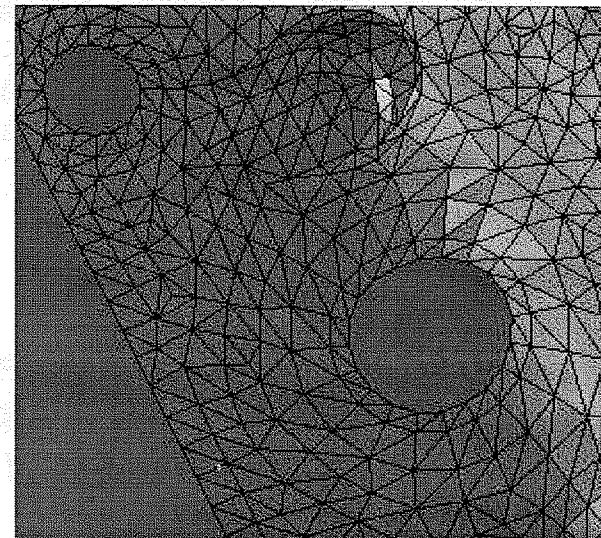
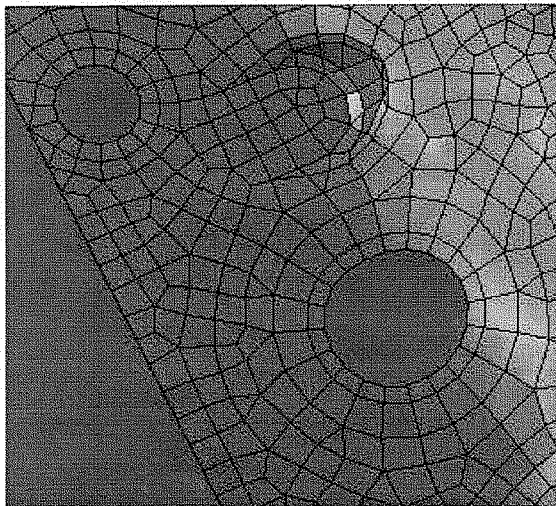
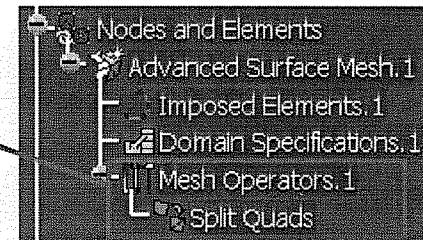


Locked domain  
(in blue)



## Do It Yourself (5/5)

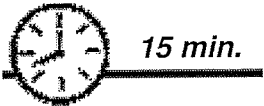
- Select the *Split quads* icon .
- Select the mesh.
- Choose the *Best Quality* option.



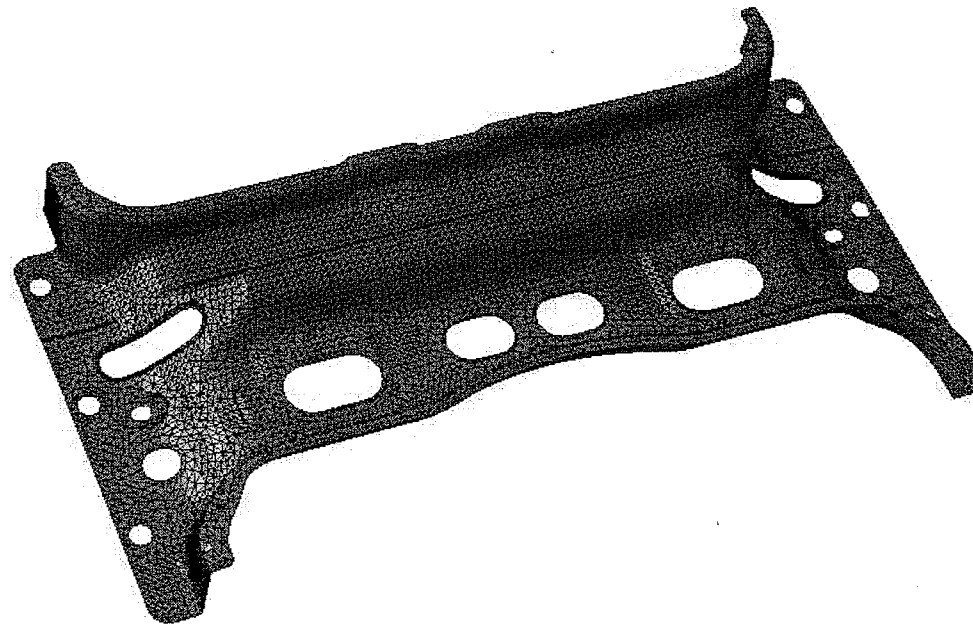


# Master Exercise

## Meshing of a Cross-member : Mesh Quality

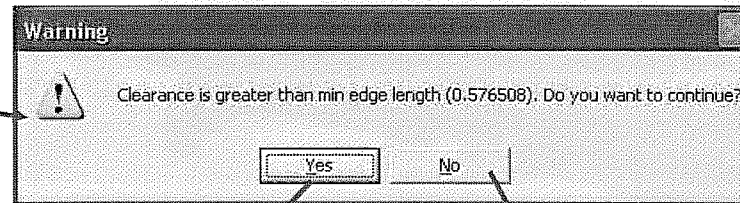
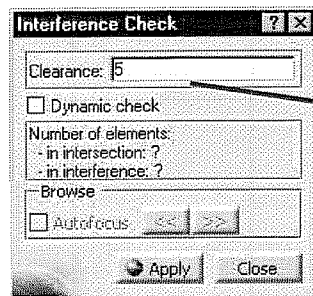


In this step you will check intersections and interferences.



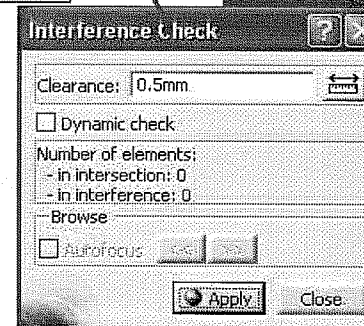
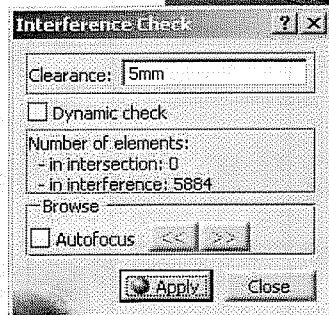
## Do It Yourself

- Select the *Intersections/interferences* icon .
  - Enter a *clearance* value (it must be less than the smallest edge element length). Notice that there is no elements intersection and interference in that case.
  - Select the *Dynamic Check* option and edit the mesh : verify that *interferences check* results are updated in real time.



Blue elements : no  
intersections/interferences

Pink elements : interferences





## Fundamental steps

### Mesh specifications

- ◆ Define global mesh specifications.
- ◆ Elements distribution around filled holes.
- ◆ Mesh around holes with elements heights specifications.
- ◆ Define mapped mesh on a domain

### Constrain edges and Mesh

- ◆ Constrain an edge over an ignored hole.
- ◆ Collapse, split and merge constrained edges.
- ◆ Mesh and check mesh statistics.
- ◆ Edit a mesh-part and recover data.

### Mesh modifications

- ◆ Clean holes.
- ◆ Edit mesh and mesh statistics.
- ◆ Lock a domain.
- ◆ Quad to tria conversion.

### Mesh Quality

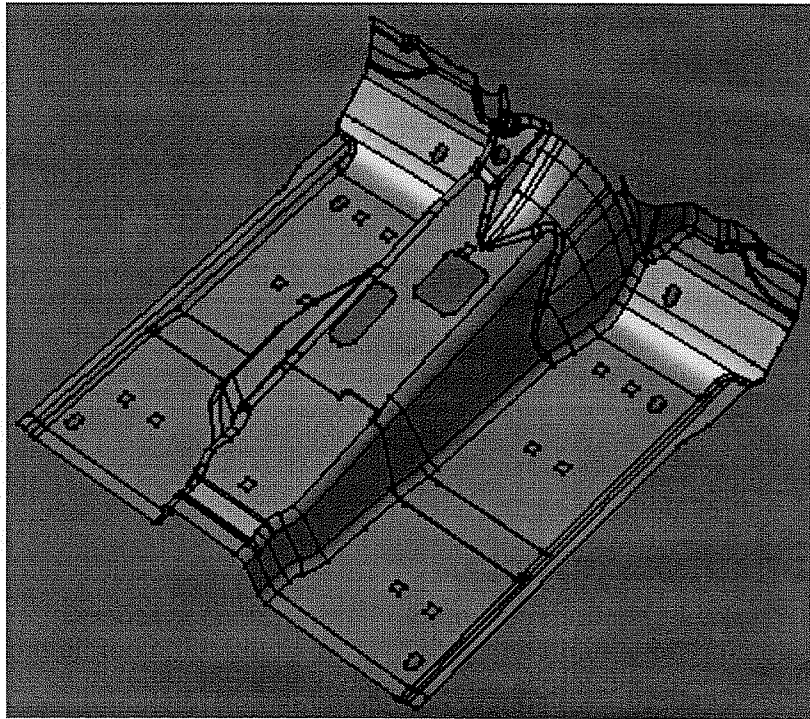
- ◆ Interferences check.

# Additional Exercise

## 2. Floor Mesh

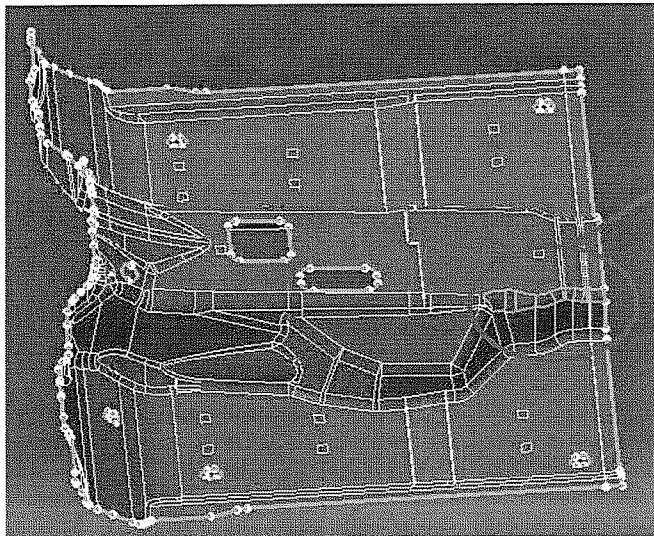
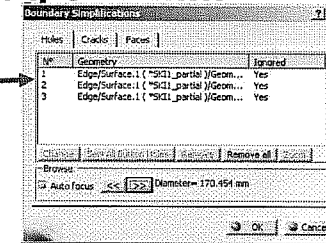


In this exercise, you will create a surface mesh on a part.

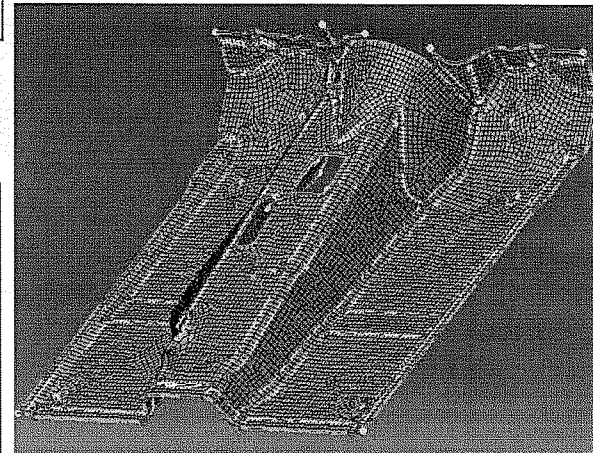
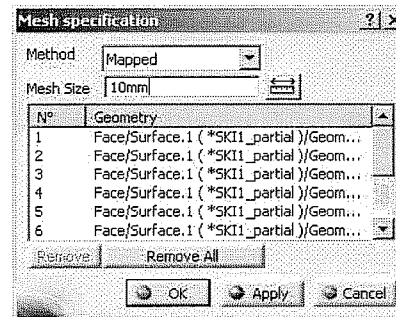


## Design Intent: Floor Mesh

- Create the Surface mesh, mesh size of 15 mm, constraint sag of 1 mm and minimum holes size of 10 mm.
- Boundary simplifications : ignore Pink holes (geometrical default)
- Apply *Domain Specifications* on indicated domain:  
*Mapped quads mesh method* and with a mesh size of 10mm.



Domain specifications on this domain: 11 faces



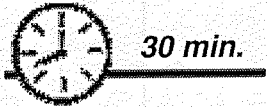
Load: FMS\_AddEx\_1.CATPart



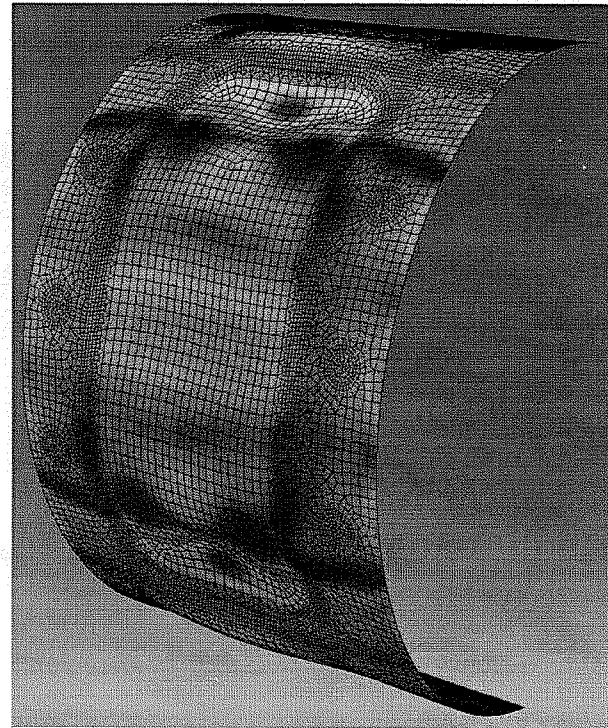
Load: FMS\_AddEx\_1\_end.CATAnalysis

# Additional Exercise

## 3. Fuselage door Static Analysis



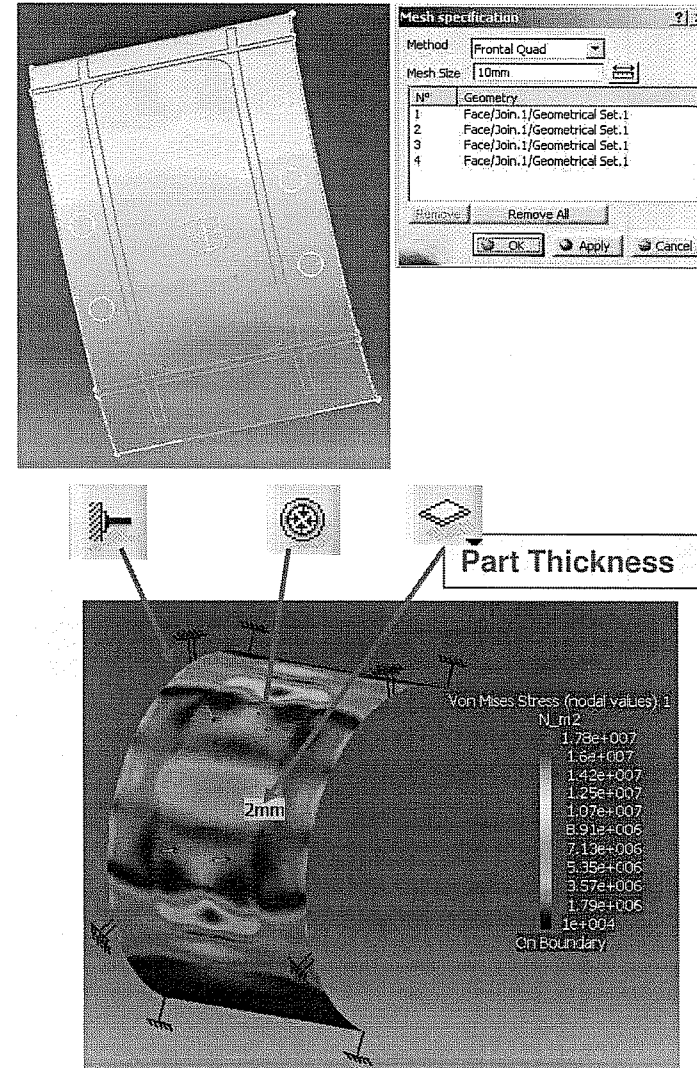
In this exercise, you will create a mapped surface mesh on the fuselage and create a static analysis.



## Design Intent: Fuselage Door Static Analysis

- ❏ Create the Surface mesh, mesh size of 35 mm, constraint sag of 2 mm.
- ❏ Modify the mesh around the door.
  - Click on *Domain Specifications* icon.
  - Select 4 areas around the door
  - Frontal method and 10mm size.
  - Idem for each circular faces
- ❏ Enter Generative Structural Analysis workbench. Apply Clamps all around the part, apply a pressure of 5000 N/m<sup>2</sup>, only on the door, apply a thickness property of 2 mm and then compute all the structure.
- ❏ Compute and display the Von Mises constraints.
- ❏ Change the thickness of the part from 2 mm to 5 mm and display Von Mises constraints.

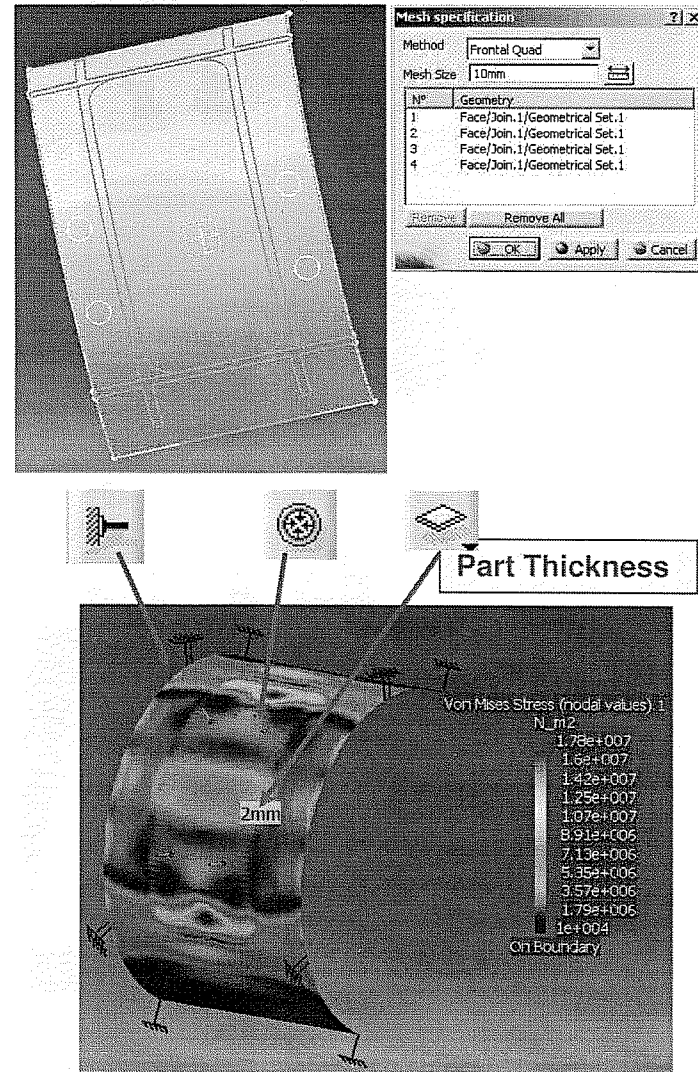
- ❏ Load: FMS\_AddEx\_2.CATPart
- ❏ Load: FMS\_AddEx\_2\_end.CATAnalysis



## Design Intent: Fuselage Door Static Analysis

- Create the Surface mesh, mesh size of 35 mm, constraint sag of 2 mm.
- Modify the mesh around the door.
  - Click on *Domain Specifications* icon.
  - Select 4 areas around the door
  - Frontal method and 10mm size.
  - Idem for each circular faces
- Enter Generative Structural Analysis workbench. Apply Clamps all around the part, apply a pressure of 5000 N/m<sup>2</sup>, only on the door, apply a thickness property of 2 mm and then compute all the structure.
- Compute and display the Von Mises constraints.
- Change the thickness of the part from 2 mm to 5 mm and display Von Mises constraints.

- Load: FMS\_AddEx\_2.CATPart
- Load: FMS\_AddEx\_2\_end.CATAnalysis





# Additional Exercise

## 4. Spot Welding Analysis

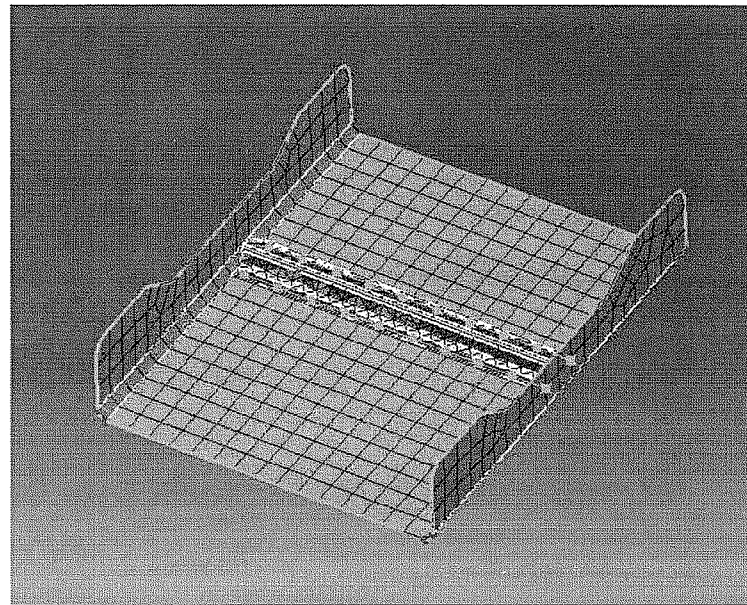


60 min.

In this exercise you will perform a Spot Welding meshing and static analysis.

You will use :

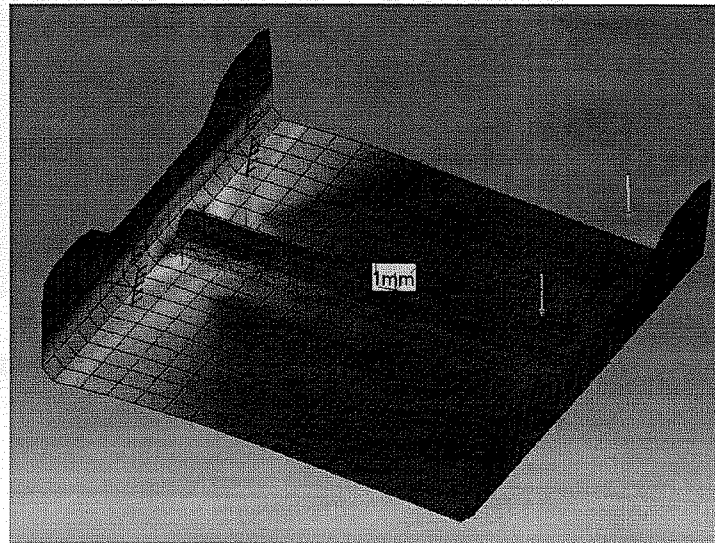
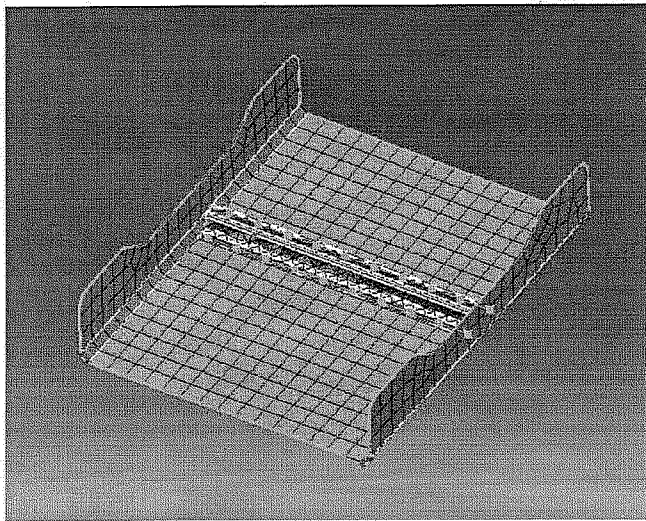
- ◆ Analysis connections
- ◆ Mesh Definition Tools
- ◆ Static Analysis
- ◆ Post-processing





## Design Intent: Spot Welding Analysis

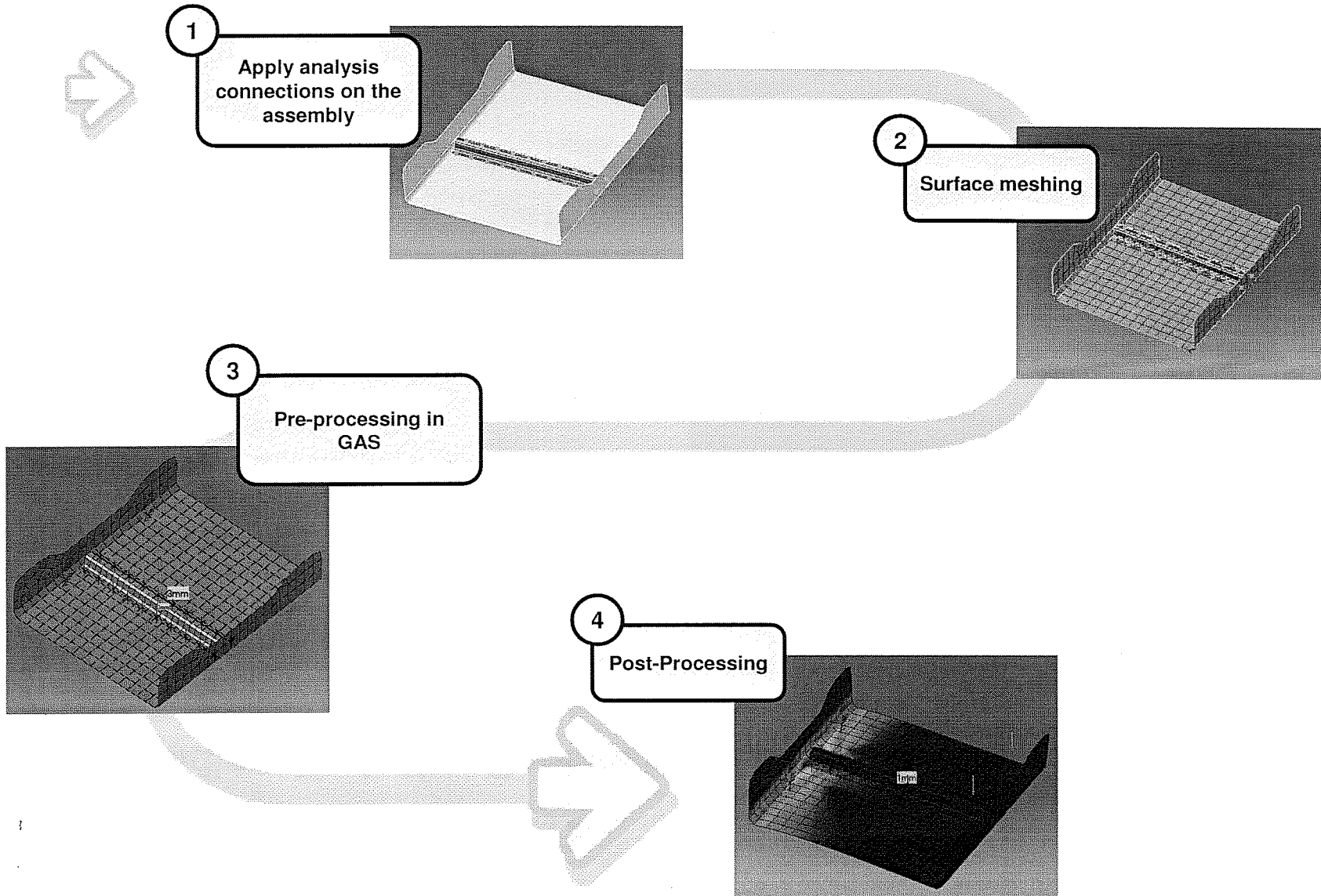
- Create analysis welding points connections.
- Mesh all the structure with FMS tools.
- Apply boundary conditions and spot welding connections.
- Perform a static analysis.
- Post-processing : point force vectors.



⚙ Load: Soudure.CATProduct

Student Notes:

## Design Process : Spot Welding Analysis



Student Notes:

# Additional Exercise

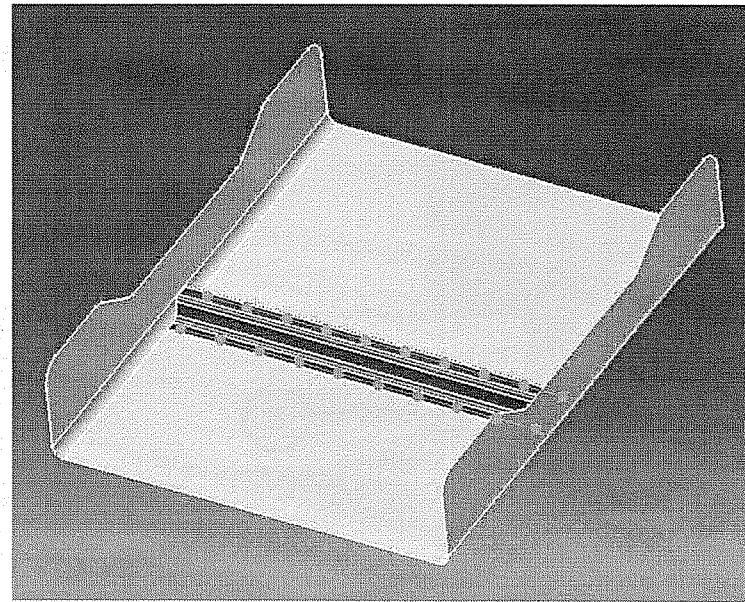
## Spot Welding Analysis : Spot welding Analysis connections



15 min.

In this step you will :

- ◆ Apply analysis welding point connections
- ◆ Save the product document



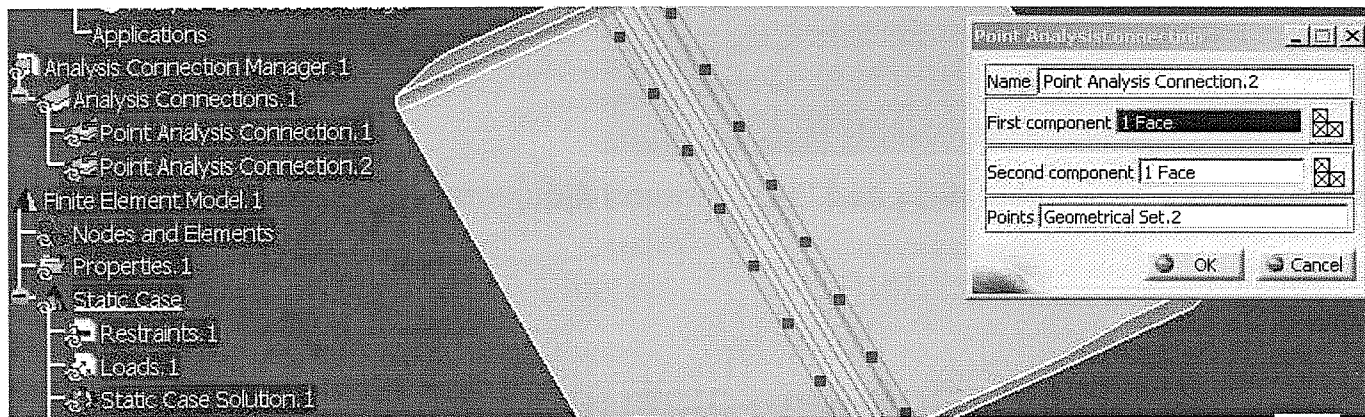
## Do It Yourself

- Enter the Advanced Meshing Tools workbench : Static analysis
- Choose *Point Analysis Connection* icon.
- Select *Stiffener* geometry as first component.
- Select *Panel* geometry as second component.
- Select *Geometrical Set.2* points as welding spots (contained in Stiffener CATPart).



Or select one point → geometrical set is automatically selected

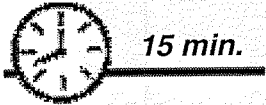
- Repeat the same sequence replacing *Geometrical Set.2* by *Geometrical Set.3*



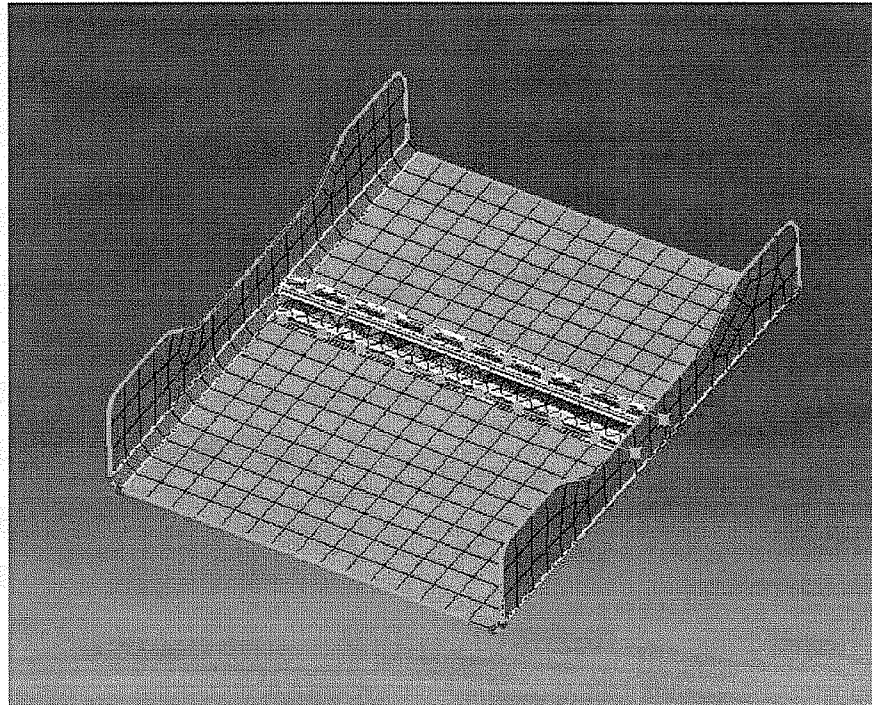
 Load: Soudure.CATProduct

# Additional Exercise

## Spot Welding Analysis : Surfacic meshing



In this step you will mesh the assembly components, use FMS mesh tools and mesh spot welds.

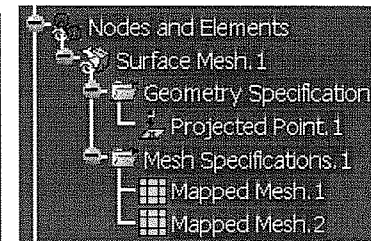
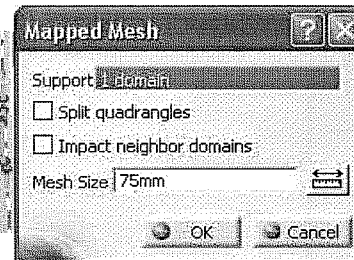
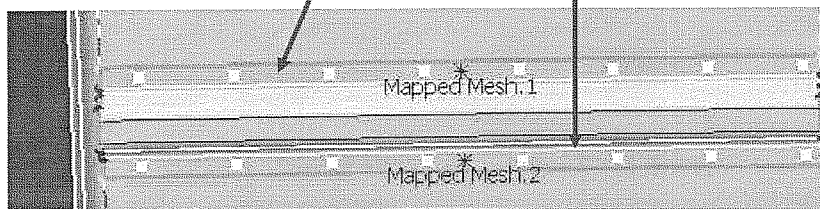
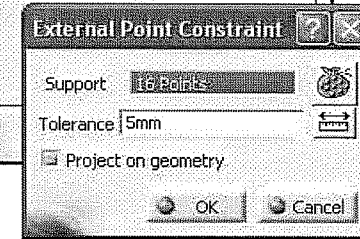
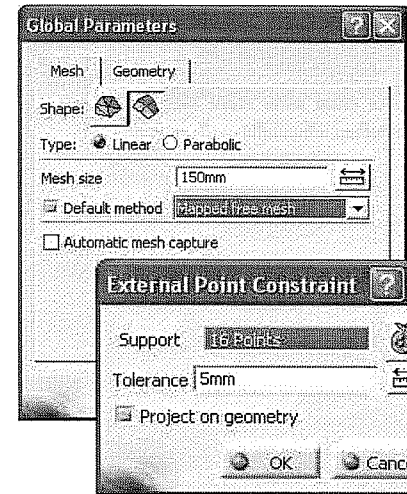




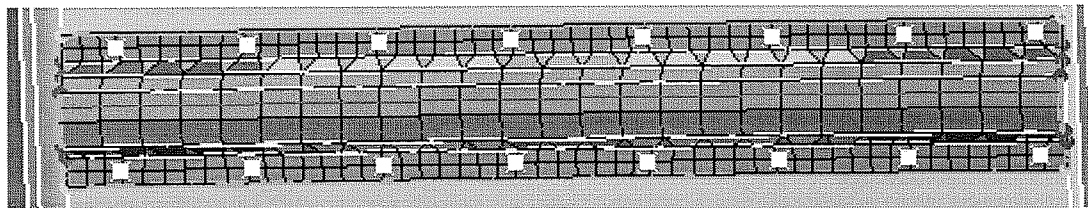
## Do It Yourself (1/3)

### Mesh Stiffener :

- Select *Surface Mesher* icon. 
- Choose the *Stiffener* part
- Enter a 150 mm Mesh size and a 20 mm Constraint sag
- As default method choose Mapped free mesh
- Select Project External point 
- Select all 16 points
- And Enter a 5 mm tolerance
- Create a Mapped mesh, size = 75mm 
- Select 2 sides of the stiffner



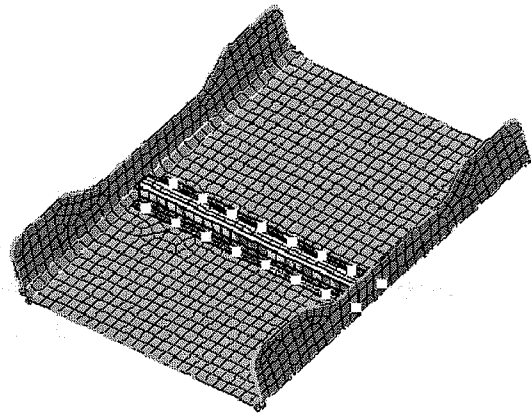
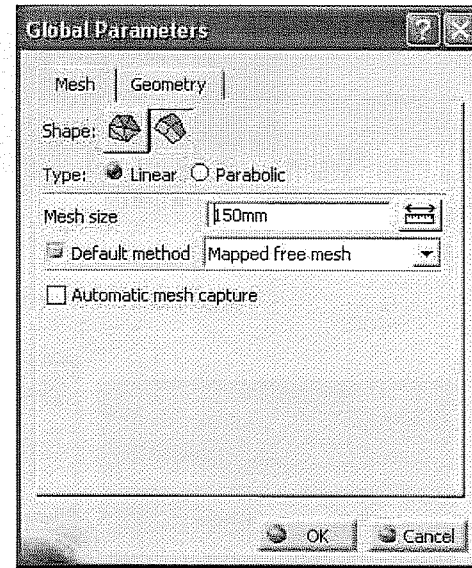
- Select *Mesh the part* icon and exit. 



## Do It Yourself (2/3)

### Mesh Panel :

- Repeat the sequence for the Panel, 
- Enter 150 mm as Mesh size, 10 mm as sag
- Mapped free as meshing method
- Constrain same 16 points, tolerance = 30 mm 
- Select *Mesh the part* icon and *Exit*

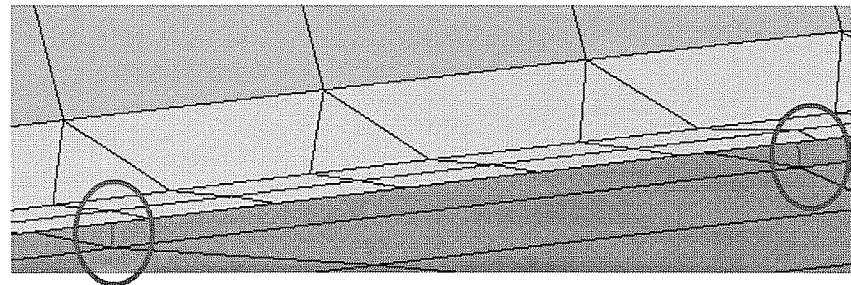
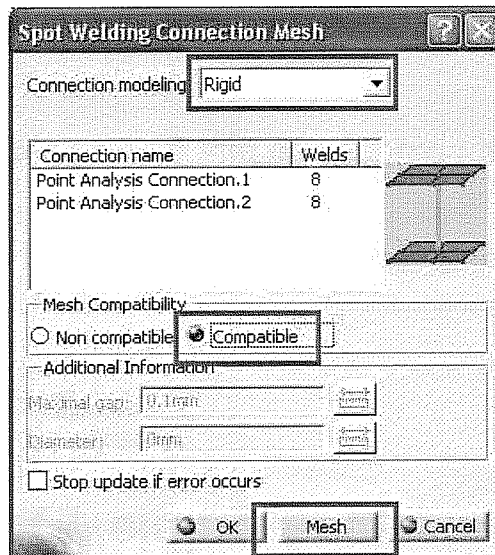
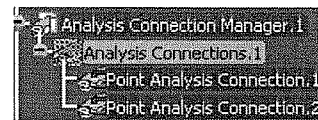




## Do It Yourself (3/3)

### Spot Welding connections :

- Click on the Spot Welding connection icon, select Analysis connection.1 branch in the tree: both point connection are selected
- Choose *Rigid* type
- And *Compatible* connectivity
- Click on *Mesh*



# Additional Exercise

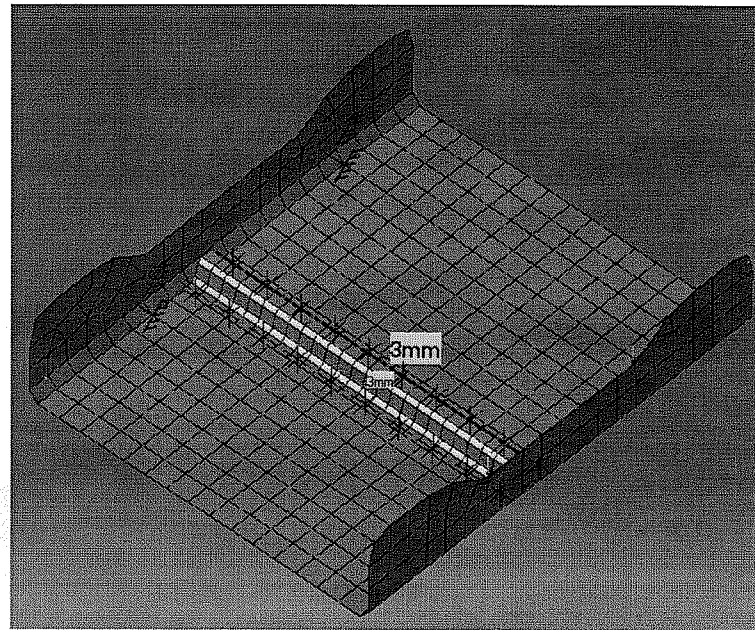
## Spot Welding Analysis : Pre-processing in GAS



15 min.

In this step you will :

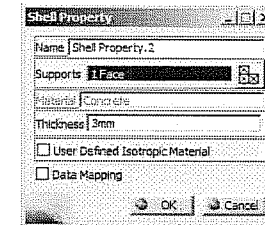
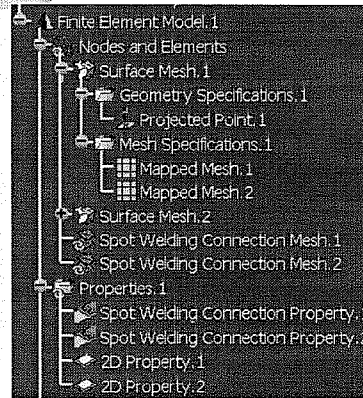
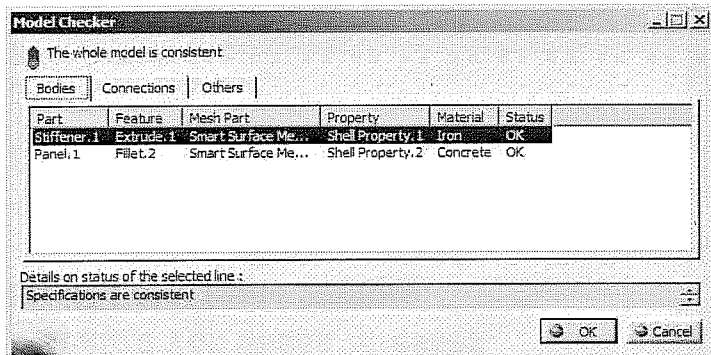
- ◆ Apply clamp and distributed load.



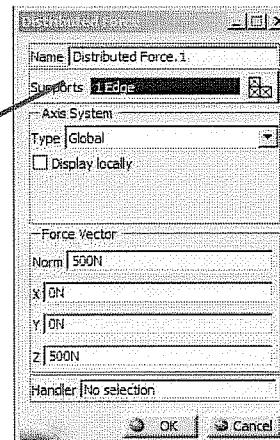
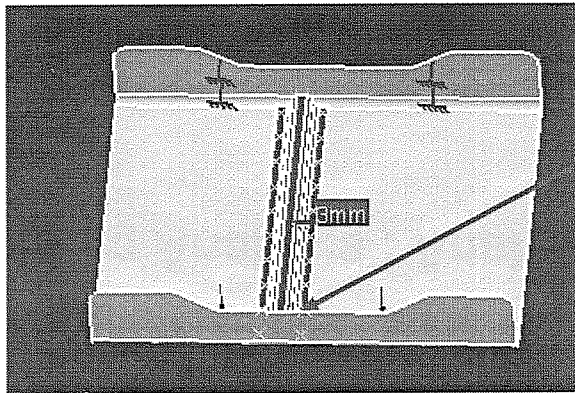
Student Notes:

## Do It Yourself

- Enter the Generative Structural Analysis workbench.
- Create Shell properties for the 2 parts (thickness of 3mm).
- Check the consistency of your model.



- Apply clamp on a *Panel* side.
- Apply a 500 N distributed force on the opposite middle edge side along Z axis.



# Additional Exercise

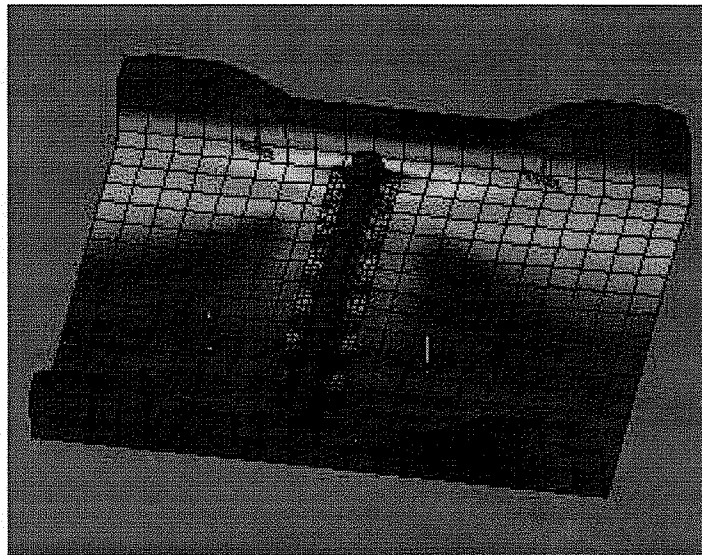
## Spot Welding Analysis : Analysis and Post-Processing



15 min.

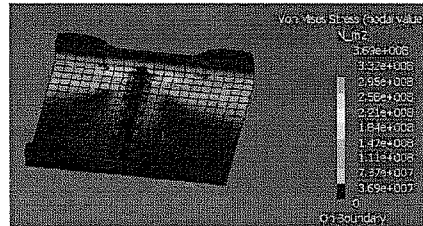
In this step you will :

- ◆ Compute a static analysis.
- ◆ Visualize Von Mises Stresses.
- ◆ Show Point force vectors.

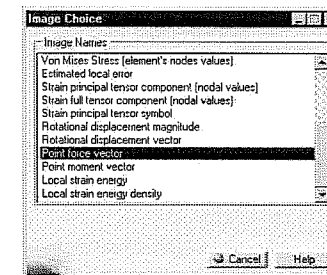


## Do It Yourself

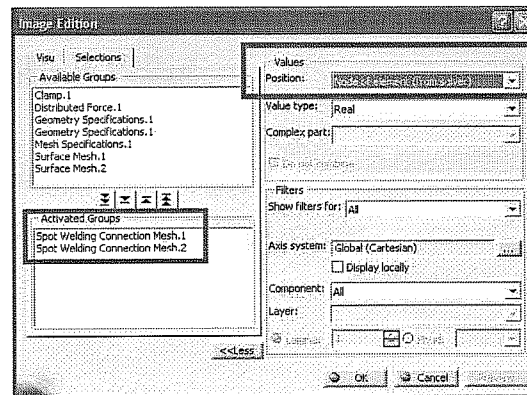
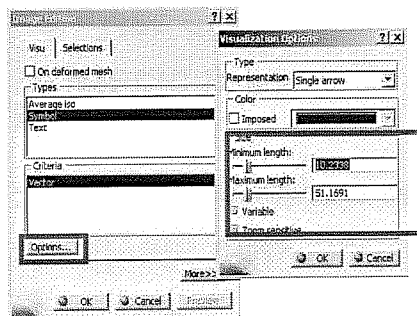
- Launch a static analysis.
- Visualize Von Mises stresses,
- then de-activate this image.



- Create a Point Force vector image:
  - Using Static case Solution contextual menu, select **Generate Image**.
  - Choose **Point force vector**.



- Edit **Point force vector** image by double clicking
  - In **Visu Panel**, Choose **Symbol** as **Types**, adjust the vectors size using **Options...**
  - In **Selections** panel, choose **Weld Spot Connection Mesh 1&2** as **Activated Groups**
  - Click on **More ...**, as **Values Position Node of element**.



- Load: spot welding before compute.CATAnalysis

## Fundamental steps

- **Analysis connections**
  - ◆ Apply analysis welding point connections.
- **Surfacic meshing**
  - ◆ Use of FMS mesh definition tools.
- **Boundary conditions and Spot Welding connections**
  - ◆ Apply clamp and distributed load.
- **Analysis and Post-Processing**
  - ◆ Launch a static analysis.
  - ◆ Visualize Von Mises Stresses.
  - ◆ Visualize Point force vectors.

# Additional Exercise

## 5. Seam Welding Analysis

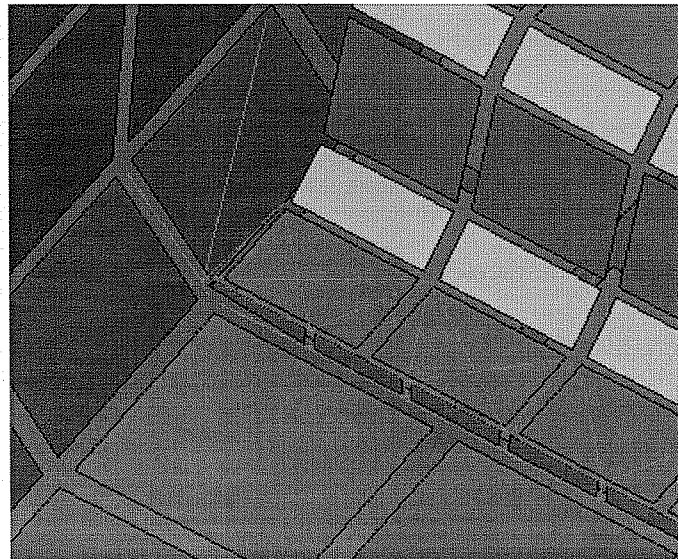


60 min.

In this exercise you will perform a Seam Welding meshing and static analysis.

◆ You will use :

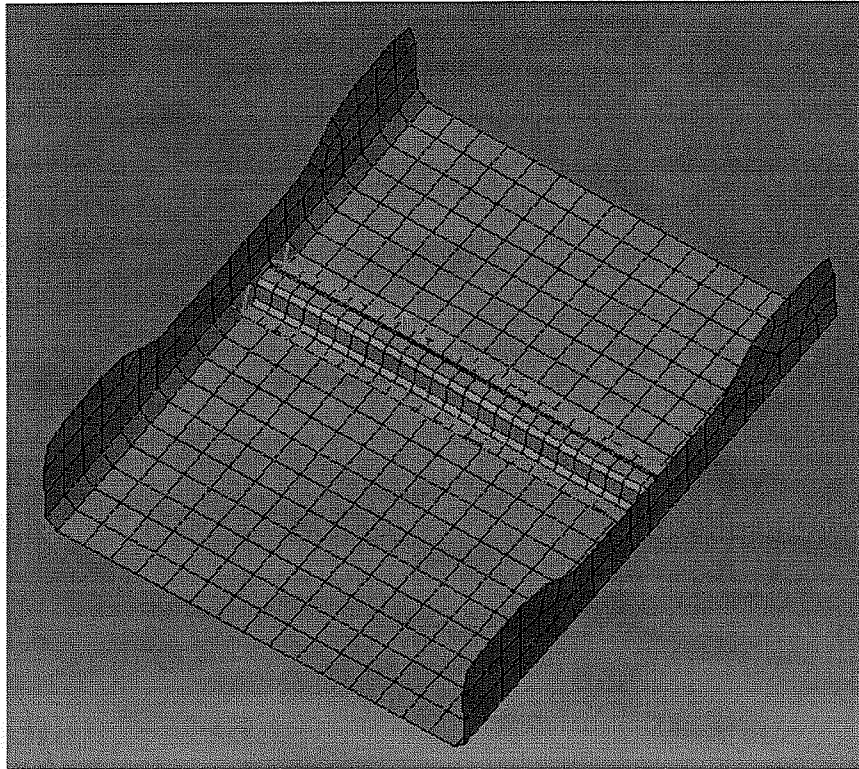
- ◆ Analysis connections
- ◆ Mesh Definition Tools
- ◆ Mesh Seam welds





## Design Intent: Seam Welding Analysis

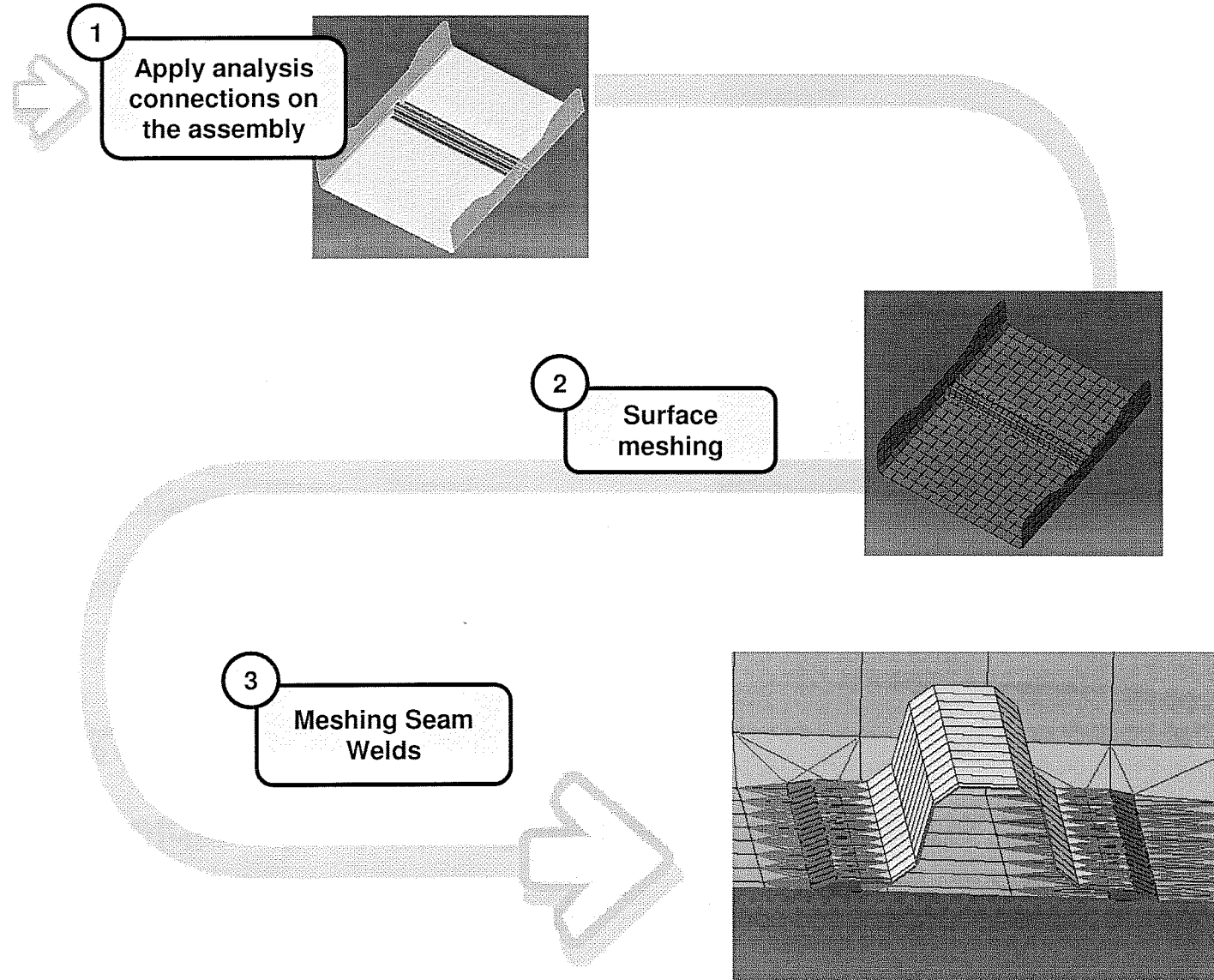
- Create analysis welding line connections.
- Mesh all the structure with FMS tools.
- Mesh seam welding connections.



 Load: Seam\_Welding.CATProduct

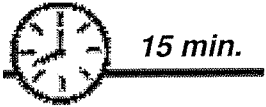
Student Notes:

## Design Process : Seam Welding Analysis



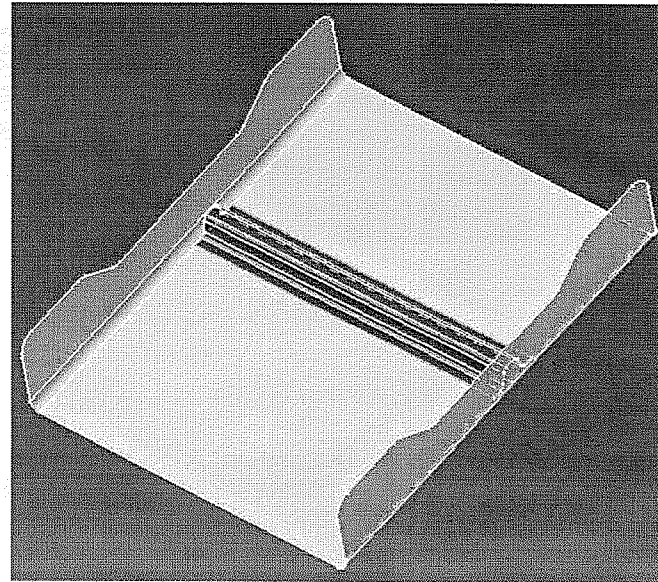
# Additional Exercise

## Seam Welding Analysis : Seam welding Analysis connections



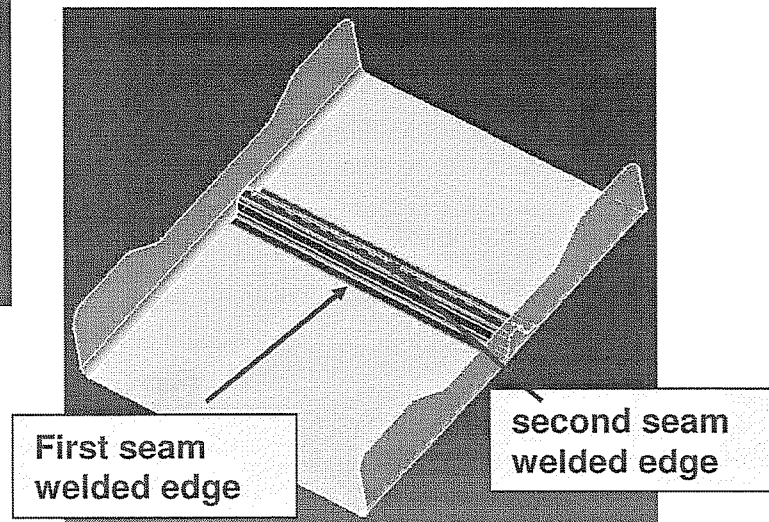
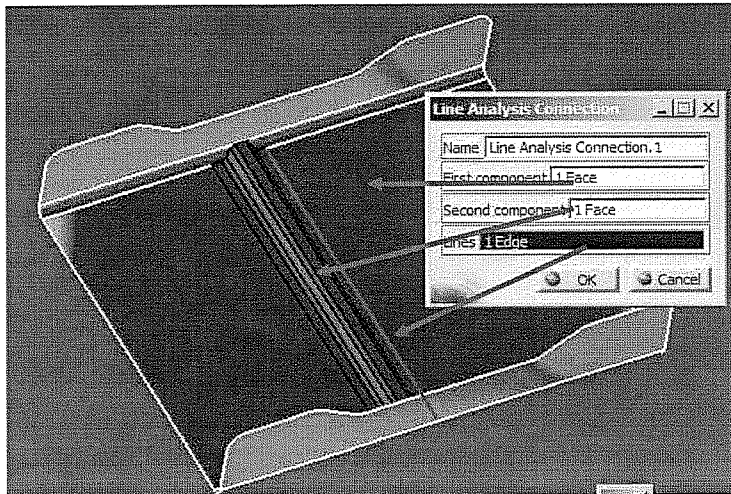
In this step you will :

- ◆ Apply analysis welding line connections



## Do It Yourself

- Enter in *Advanced Meshing Tools* workbench : Static analysis
- Choose *Line Analysis Connection* icon.
- Select Stiffener geometry as first component.
- Select Panel geometry as second component.
- Select welded edge as shown
- Idem to define line connection on the second welded edge



 Load: Seam\_Welding.CATProduct

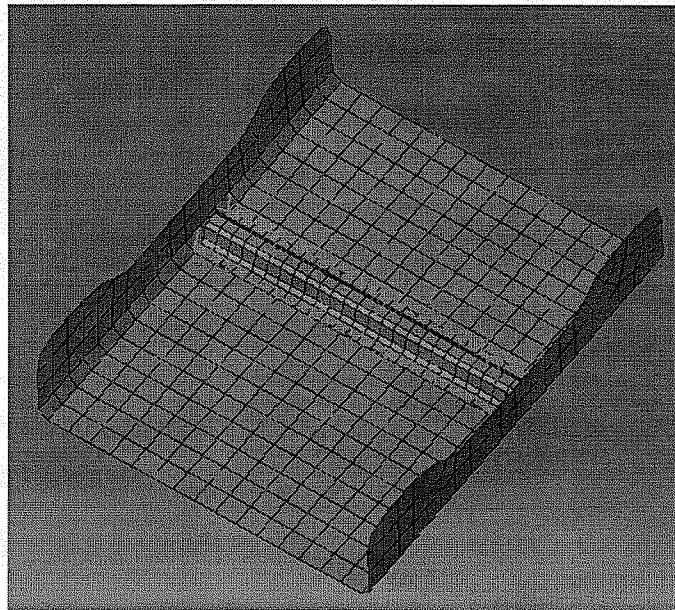
# Additional Exercise

## Seam Welding Analysis : Surfacic meshing



15 min.

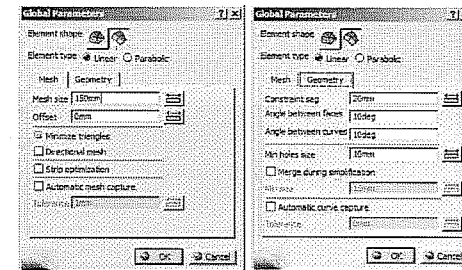
In this step you will Mesh the assembly surfaces



## Do It Yourself

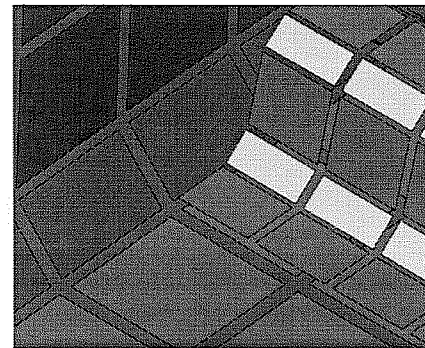
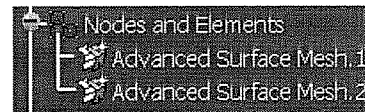
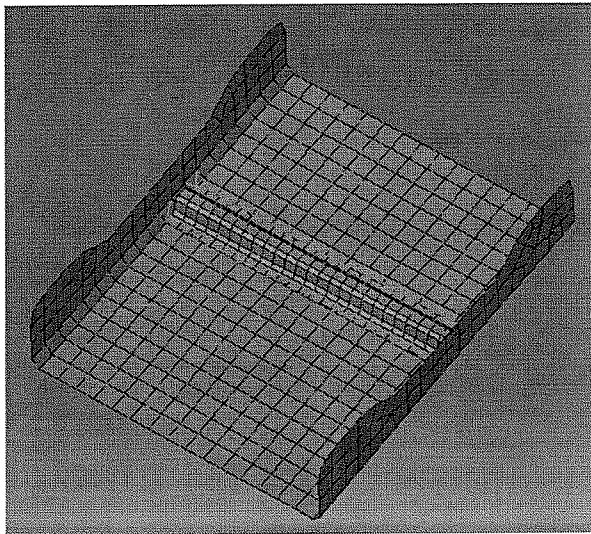
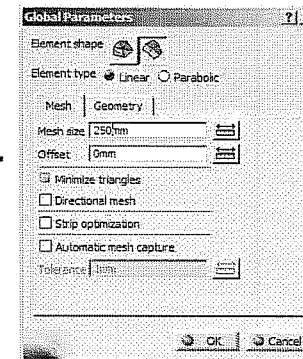
### Mesh Stiffener :

- Select **Advanced Surface Mesher** icon. 
- Select the **Stiffener** geometry, **quadrangle** element shape.
- Enter a **150 mm** Mesh size and a **20 mm** Constraint sag. **Validate**.
- Select **mesh** icon and exit.  



### Mesh Panel :

- Repeat the sequence for the **Panel** geometry, but enter a **250 mm** Mesh size.





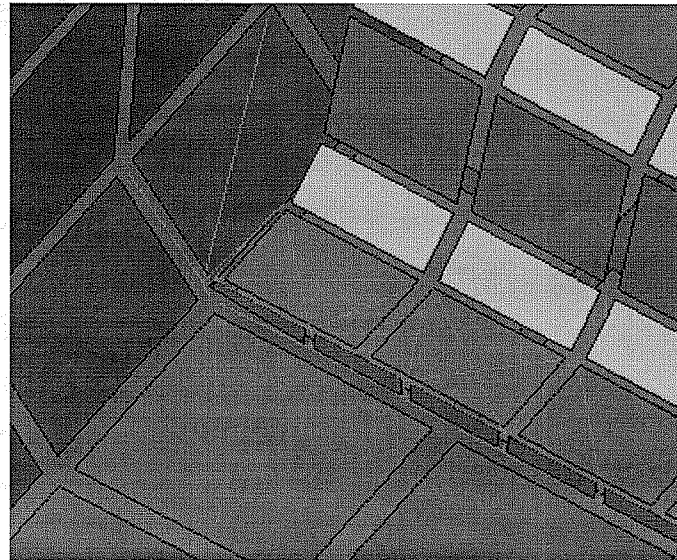
# Additional Exercise

## Seam Welding Analysis : Seam welds meshing



15 min.

In this step  
you will Mesh the seam Welds Connection

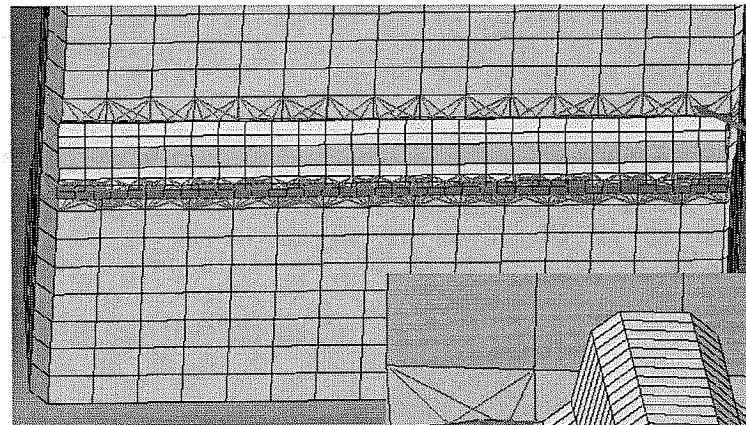
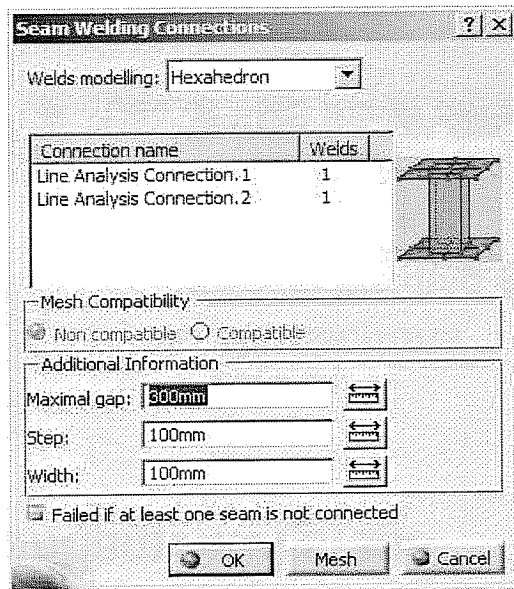
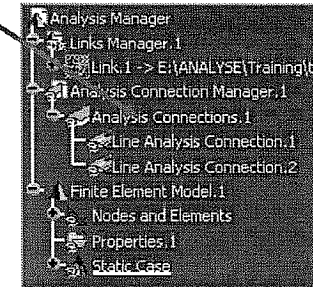




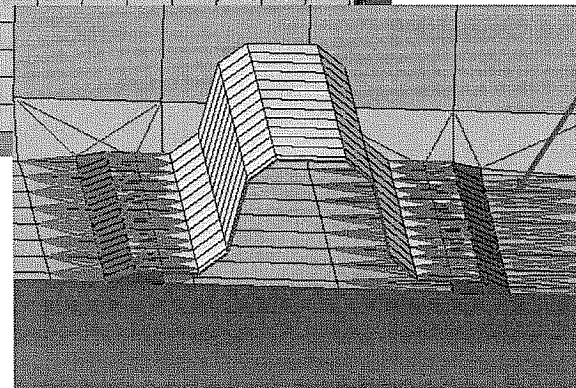
## Do It Yourself

### Mesh Seam welds:

- Click on the Seam Welding connection icon, and select Analysis connection.1 branch in the tree: both line connections are selected
- Choose Hexahedron type
- Enter parameters:  
300 mm as maximal gap for connection  
100 mm as step between elements,  
100 mm as seam width
- Activate 'Failed if at least one seam is not connected' to detect any error
- Click on *Mesh*



Notice that incompatible meshes are handle thanks to join type elements



## Fundamental steps

- **Analysis connections**
  - ◆ Apply Seam Welding Analysis connections.
- **Surfacic meshing**
  - ◆ Use of FMS mesh definition tools.
- **Mesh Seam Welding connections**
  - ◆ Use of Line analysis connection
  - ◆ Define type of elements and mesh parameter.

# Additional Exercise

## 6. Surface Welding Analysis

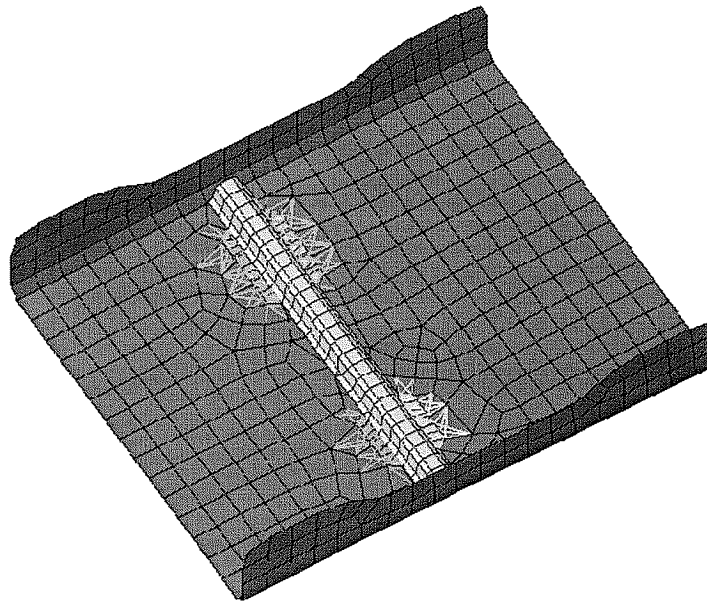


60 min.

In this exercise you will perform a Surface Welding meshing and static analysis.

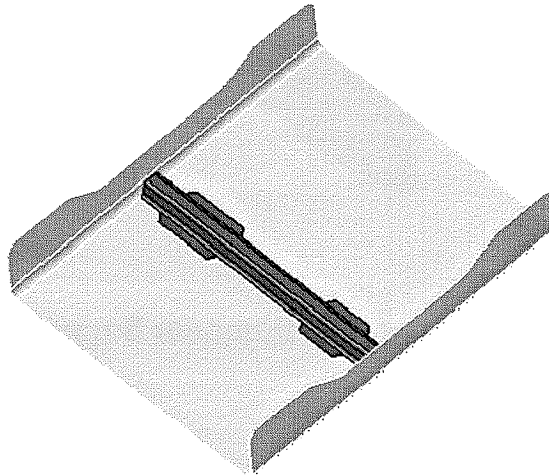
◆ You will use :

- ◆ Analysis connections
- ◆ Mesh Definition Tools
- ◆ Mesh Surface welds



## Design Intent: Surface Welding Analysis

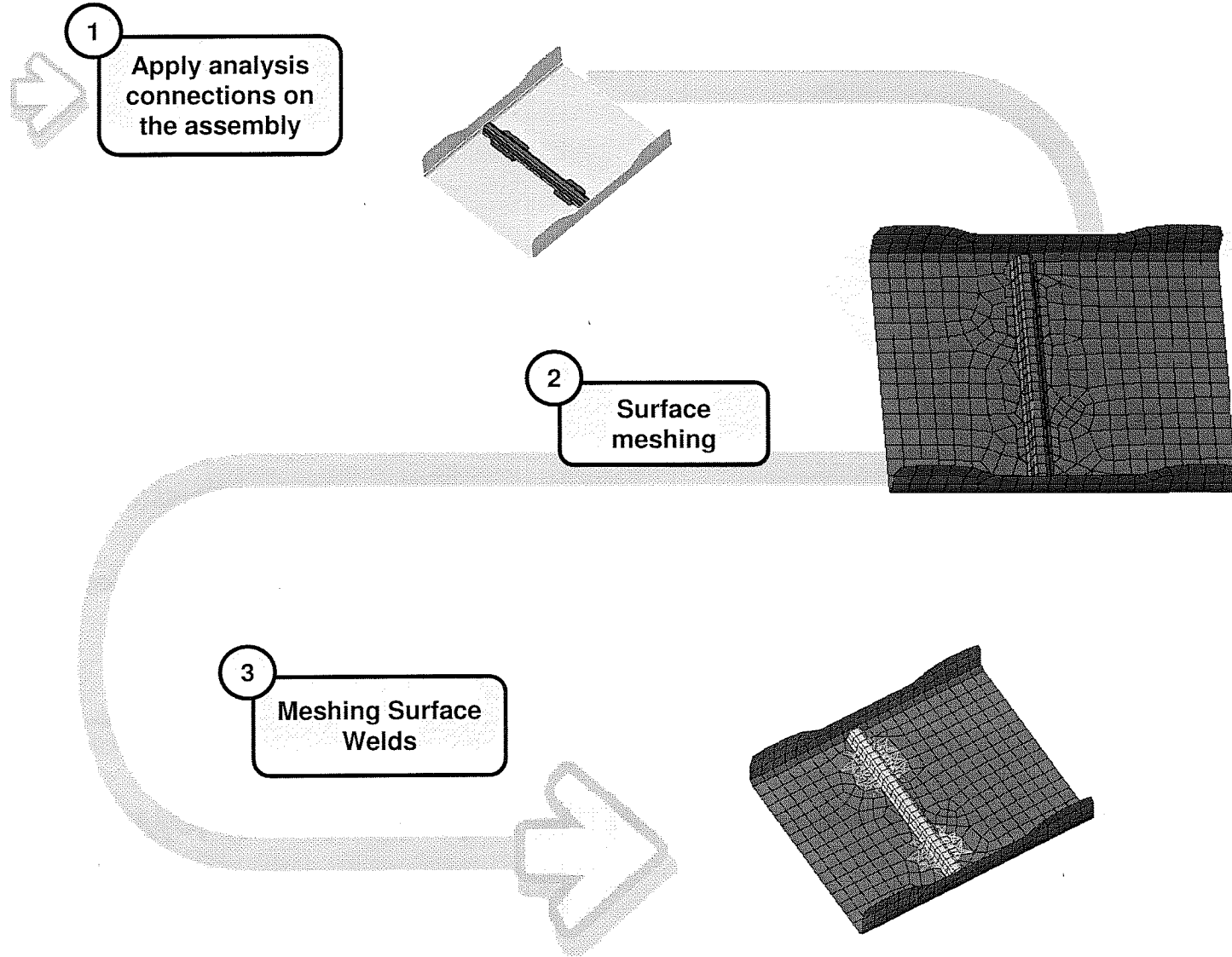
- Create analysis welding surface connections.
- Mesh all the structure with FMS tools.
- Mesh surface welding connections.



 Load: Surface\_Welding.CATProduct

Student Notes:

## Design Process : Surface Welding Analysis



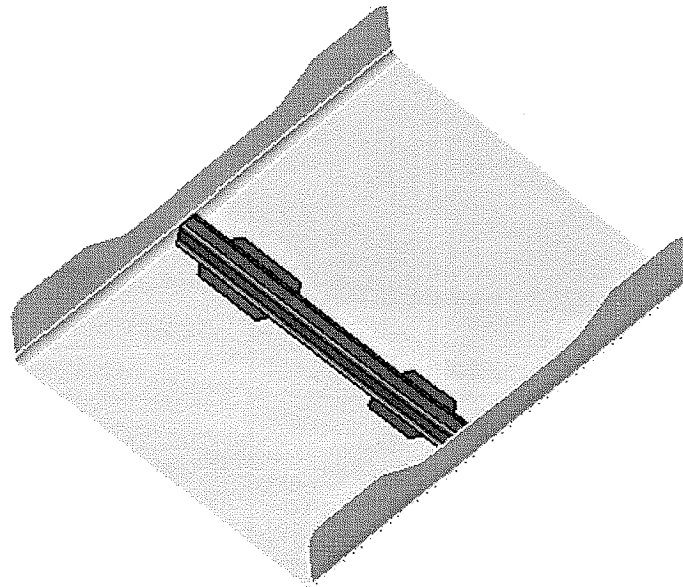
# Additional Exercise

## Surface Welding Analysis : Surface welding Analysis connections



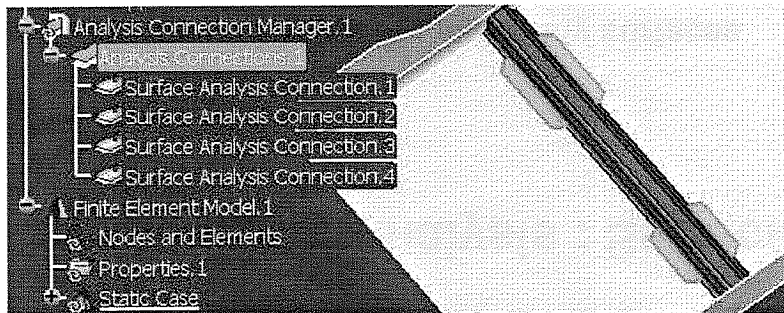
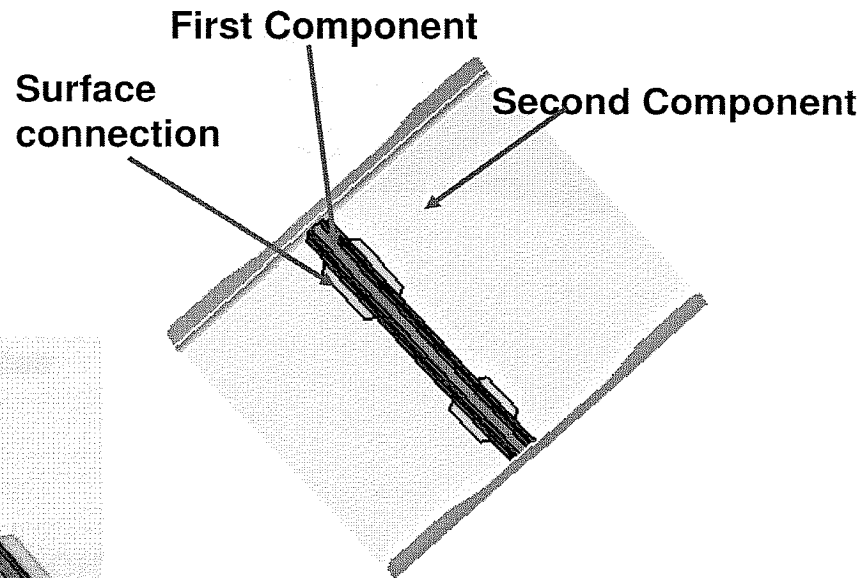
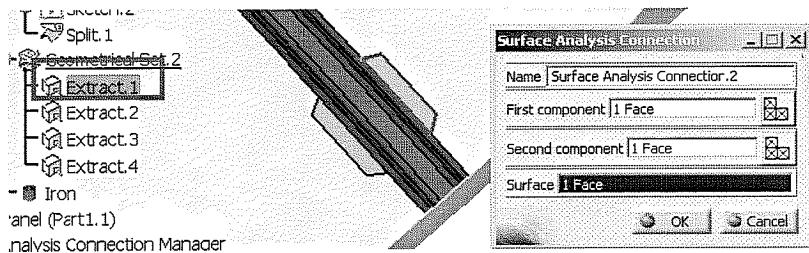
In this step you will :

- ◆ Apply analysis welding Surface connections



## Do It Yourself

- ❖ Enter in *Advanced Meshing Tools* workbench : Static analysis
- ❖ Choose *Surface Analysis Connection* icon. 
- ❖ Select Stiffener geometry as first component.
- ❖ Select Panel geometry as second component.
- ❖ Select connecting surface: blue face as shown: Extract.1 in the tree
- ❖ Idem to define surface connections on the 3 others surfaces



❖ Load: Surface\_Welding.CATProduct



Student Notes:

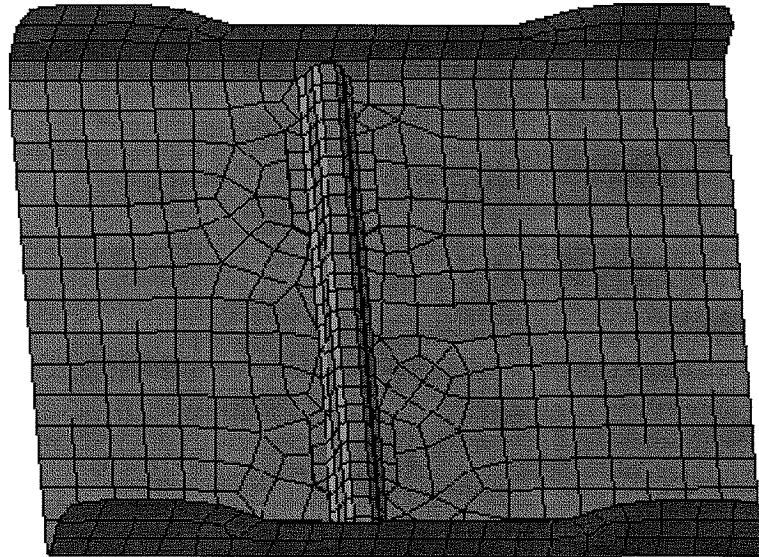
# Additional Exercise

## Surface Welding Analysis : Surfacic meshing



15 min.

In this step you will Mesh the assembly surfaces



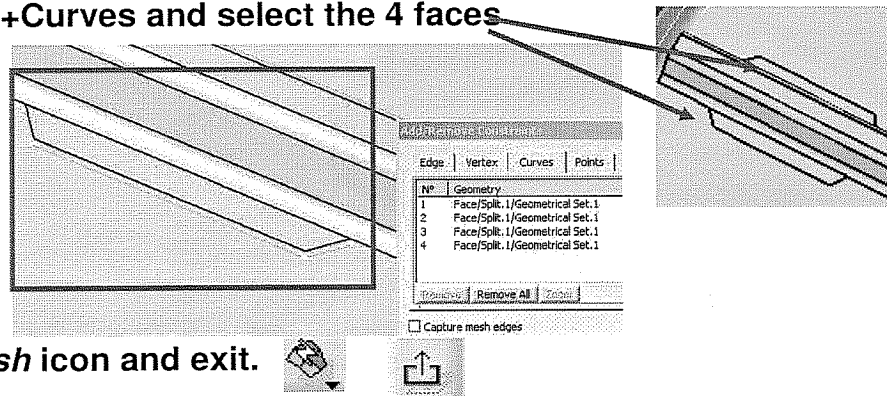
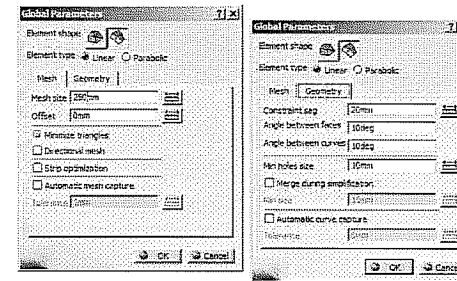
## Student Notes:

## Do It Yourself

## Hide Geometrical Set.2

## Mesh Panel :

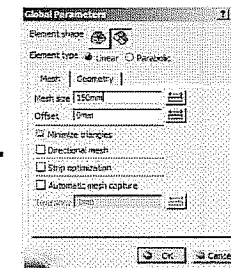
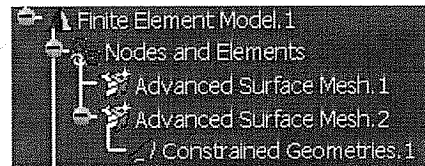
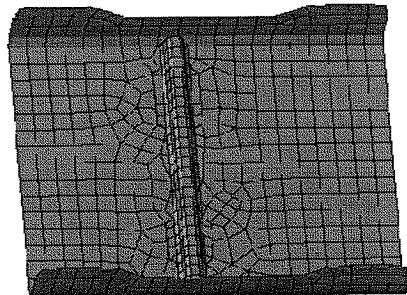
- Select Advanced *Surface Mesher* icon.
- Select the Panel geometry, quadrangle element shape.
- Enter a 250 mm Mesh size and a 20 mm Constraint sag. Validate.
- Constrain area of the 4 connecting surfaces from stiffner geometry: use Add/Remove Constraint+Curves and select the 4 faces



- Select *mesh* icon and exit.

## Mesh Stiffner :

- Repeat the sequence for the *Stiffener* geometry, but enter a 150 mm Mesh size.



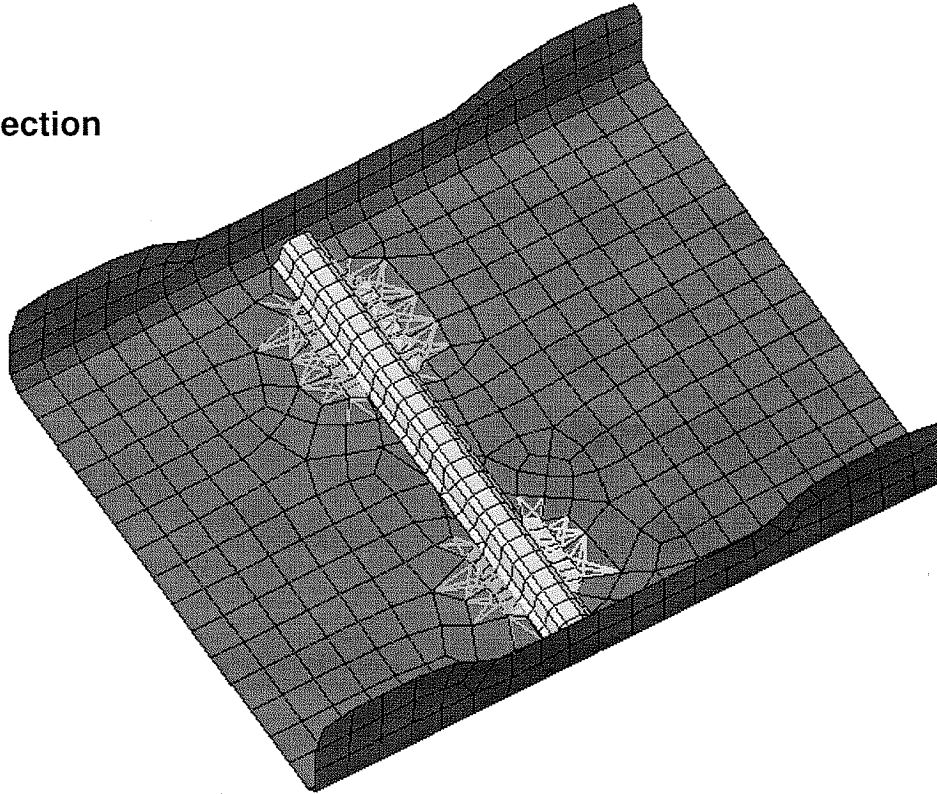
# Additional Exercise

## Surface Welding Analysis : Surface welds meshing



15 min.

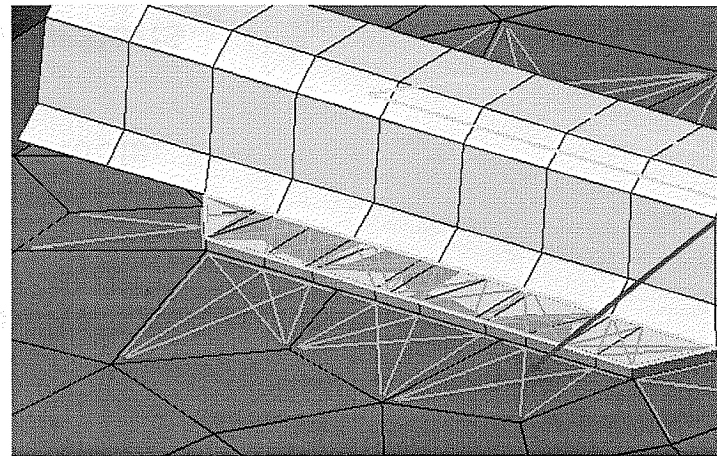
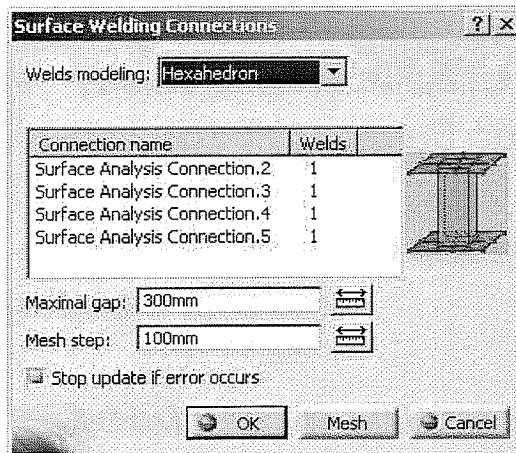
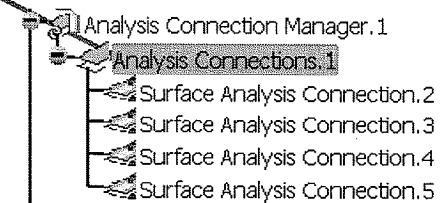
In this step  
you will Mesh the surface Welds Connection



## Do It Yourself

### Mesh Surface welds:

- Click on the Surface Welding connection icon, and select Analysis connections.1 branch in the tree: 4 surface connections are selected
- Enter parameters:  
300 mm as maximal gap for connection  
100 mm as step between elements,
- Click on *Mesh*



Notice that incompatible meshes are handle thanks to join type elements

- Thanks to *Nodes and Elements* contextual menu, color all meshparts

## Fundamental steps

- **Analysis connections**
  - ◆ Apply Surface Welding Analysis connections.
- **Surfacic meshing**
  - ◆ Use of FMS mesh definition tools.
- **Mesh Surface Welding connections**
  - ◆ Use of Surface analysis connection
  - ◆ Define type of elements and mesh parameter.

# Additional Exercise

## 7. Compatible Seam Welds

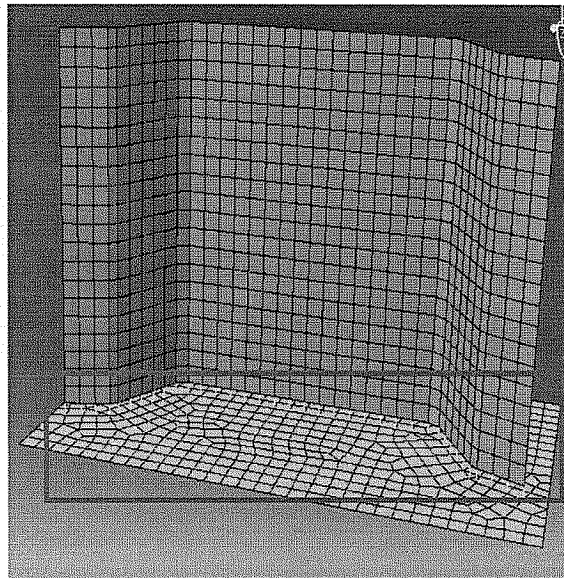


15 min.

In this exercise you will Create a Compatible Seam Weld Mesh.

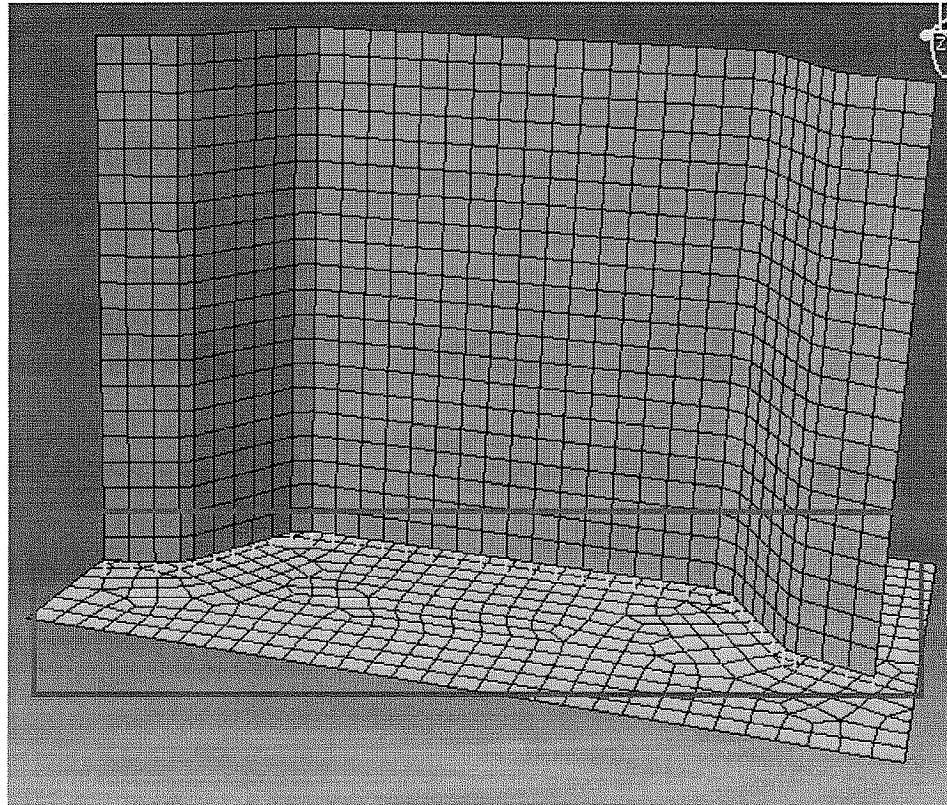
You will use :

- ◆ Line Analysis Connection in FMS
- ◆ Add/Remove Constraint Tool in FMS
- ◆ Compatible Seam Welding Connections



## Design Intent: Compatible Seam Welds

- Create *Lines Analysis Connection* in *Advanced Meshing Tools* workbench .
- Constraint These Lines
- Create Compatible Seam Weld Mesh



⚙ Load: seam\_welds\_compatible.CATAnalysis

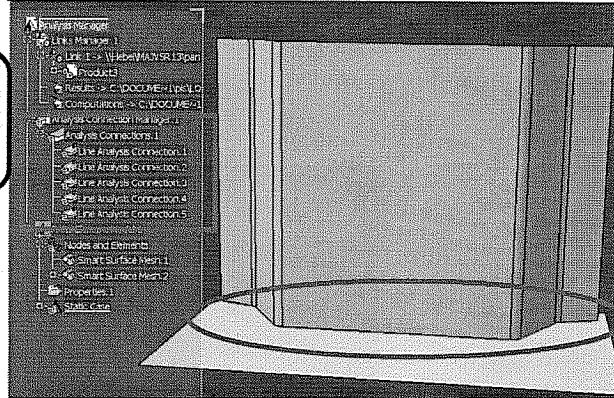


Student Notes:

## Design Process : Compatible Seam Welds

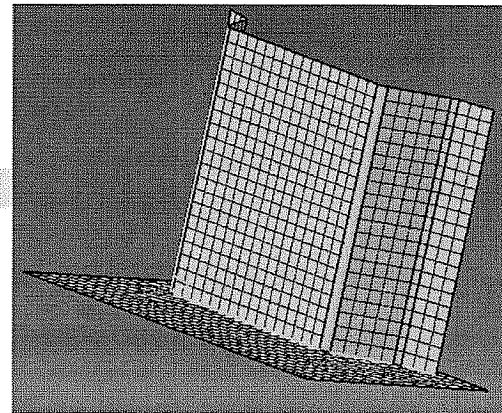
1

Create Lines Analysis  
Connection in FMS



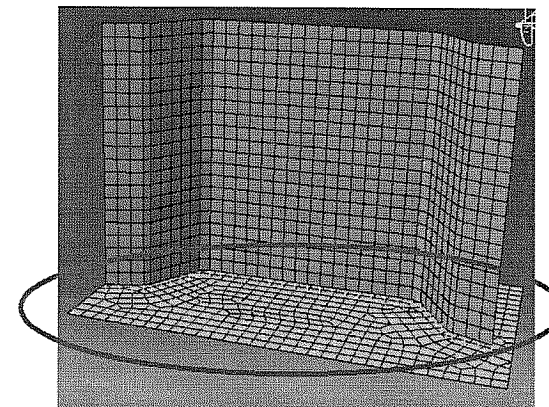
2

Constrain Lines in  
FMS



3

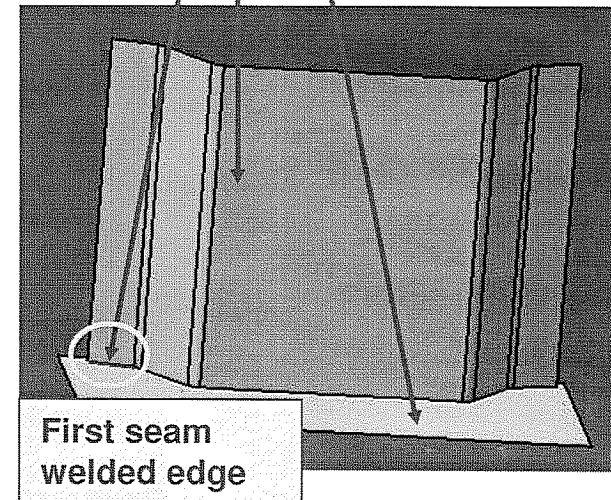
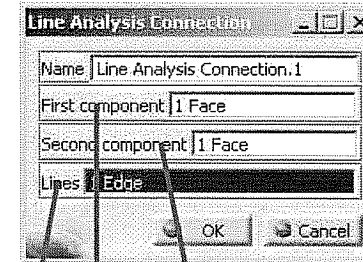
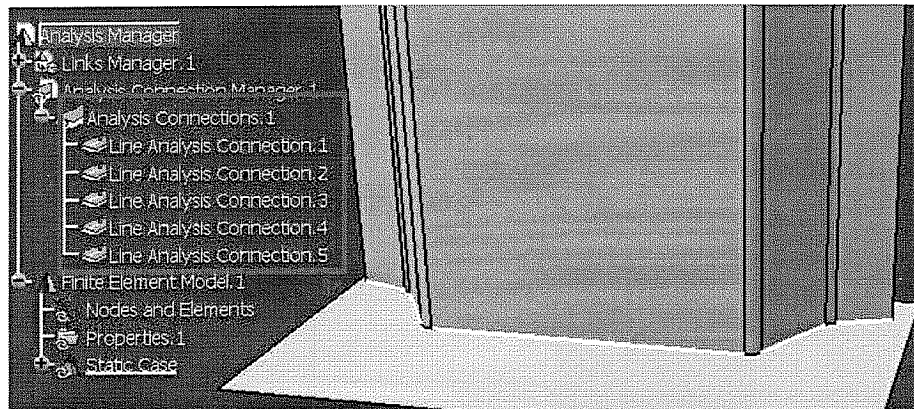
Create Compatible Seam  
Weld Mesh



## Do It Yourself (1/3)

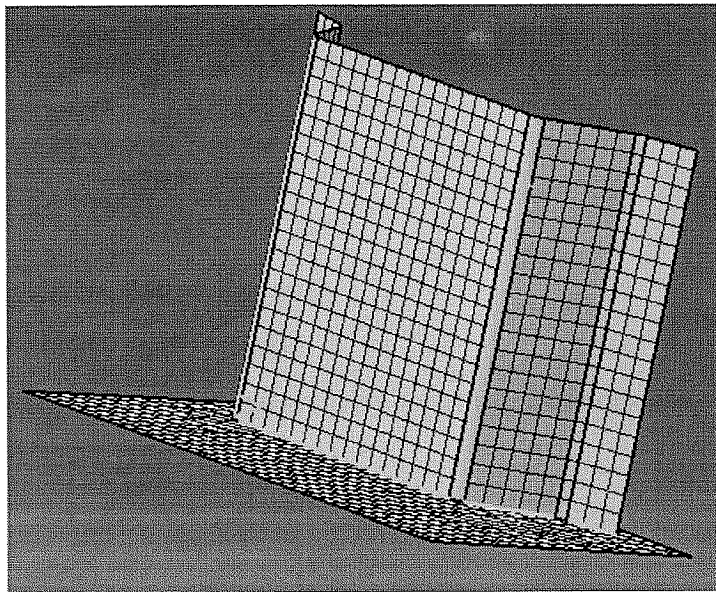
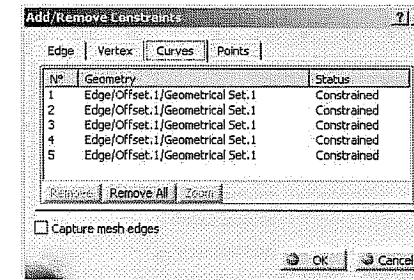
- Enter in *Advanced Meshing Tools* workbench : Static analysis
- Choose *Line Analysis Connection* icon. 
  - Select the Face of Part1 as first component.
  - Select the Face of Part2 as second component.
  - Select an Edge to be welded as shown
- Idem to define 4 others Lines Connection,

You obtain :



## Do It Yourself (2/3)

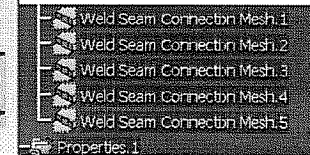
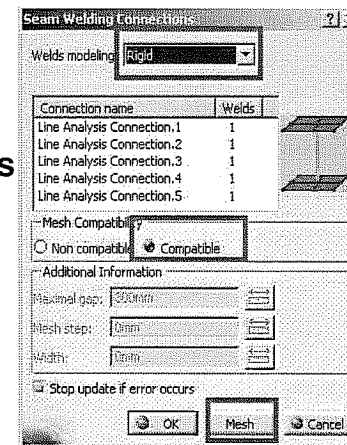
- Mesh each parts: size=6mm, sag=0.6mm
- Bottom part mesh: Constrain external welded edges :
  - Using Add/remove Constraint icon, Curves tab,
  - Select the 5 top part edges  
(same edges selected in Lines Connection definition)



## Do It Yourself (3/3)

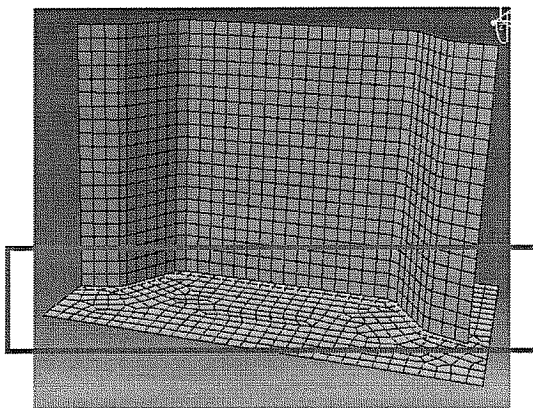
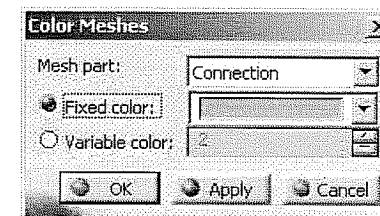
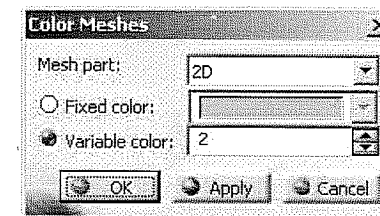
### Seam welding connections mesh:

- Select Analysis connection to select all 5 line connections
- Choose Rigid as welds modelling.
- Choose Compatible as compatibility



### Color meshparts:

- Using Nodes and Elements contextual menu, select *Color Meshes*
- In *Meshpart* list, choose *2D*, activate *Variable color* and choose 2 colors, click on *Apply*
- In *Meshpart* list, choose *Connection*, activate *Fixed color* and choose one color, click on *OK*



# Additional Exercise

## 8. Spot Welding With Hemming

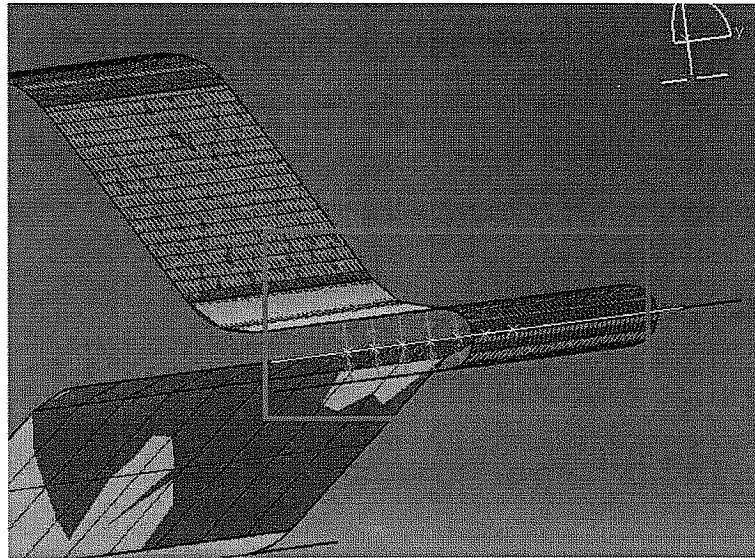


15 min.

In this exercise you will Create Spot Weld Mesh in a Hemming area.

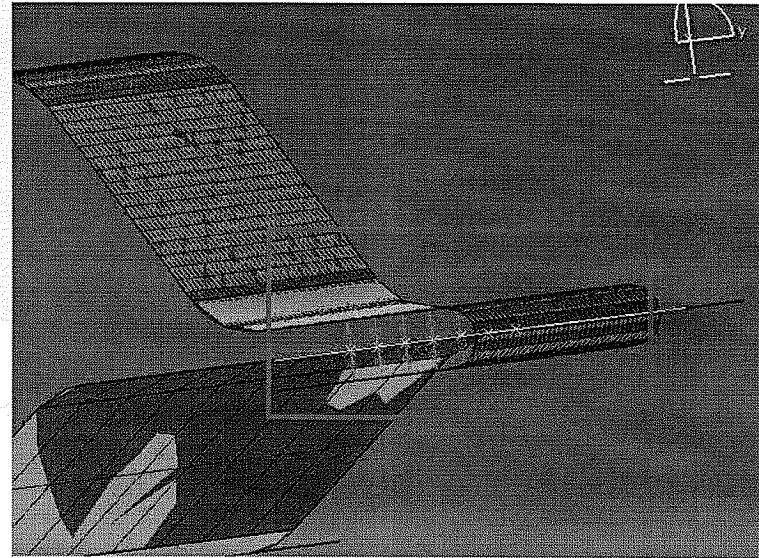
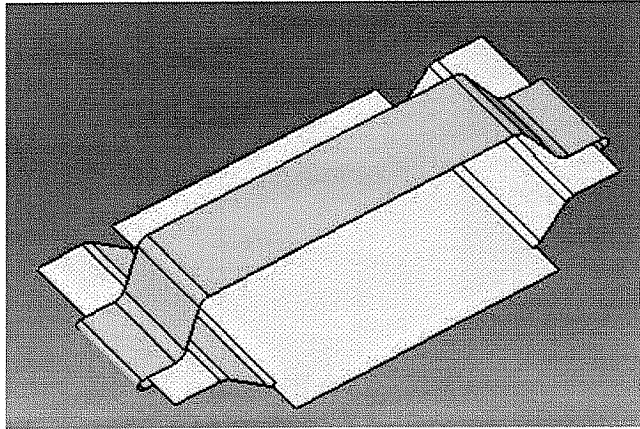
You will use :

- ◆ Hemming option in ABF
- ◆ Mesh Definition Tools



## Design Intent: Spot Welding With Hemming

- Define Hemming in ABF Product.
- Mesh all the structure with FMS tools.
- Create Spot Welds in Hemming Zone, using Non-Compatible Elements.



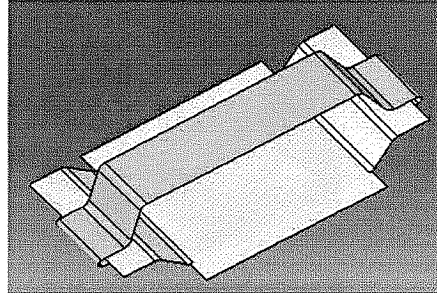
• Load: Hemming.CATProduct



## Design Process : Spot Welding With Hemming

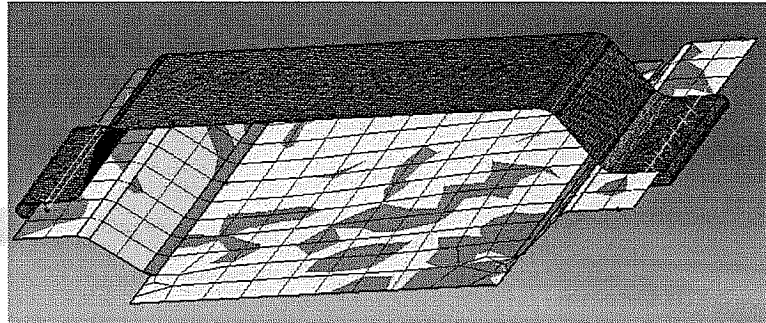
1

Define Hemming in  
ABF Product



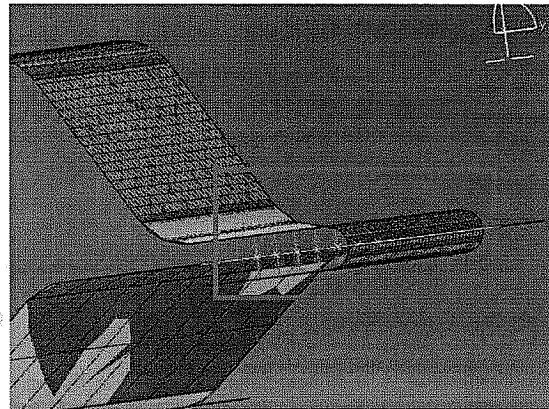
2

Surface Meshing



3

Create Spot Welds in FMS  
using Non-Compatible  
Elements





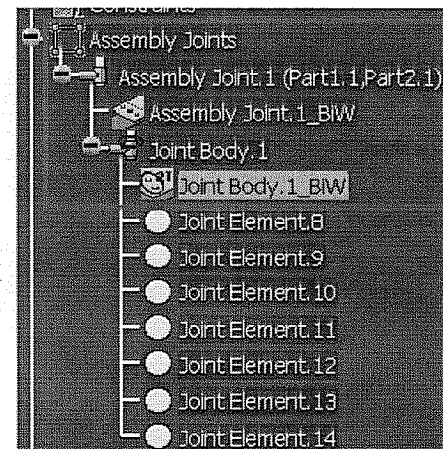
## Do It Yourself (1/2)

## OPTIONNAL STEP

- Enter in *Automotive BiW Fastening* workbench.



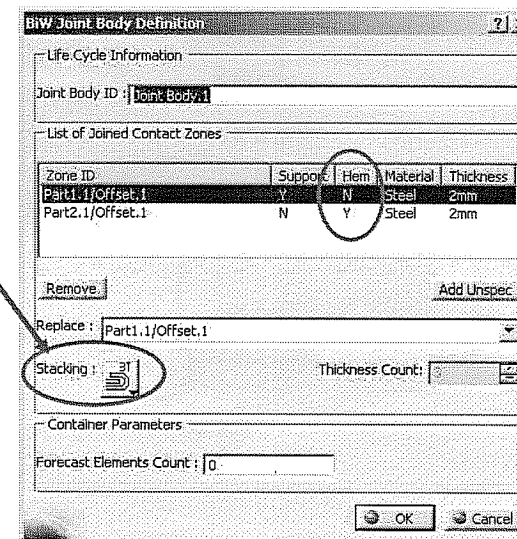
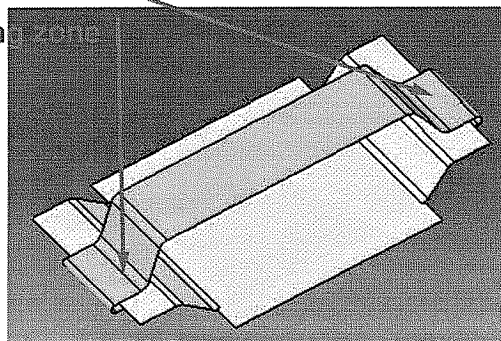
- Double Click, in the Specification Tree, on Joint Body.1\_BiW (under Assembly Joints).



- Define Hemming, in BiW Joint Body Definition Dialog Box (Yes/No), using Stacking option.

- Update Your Product .

Spot Welds will be defined in  
hemming zone



- Load: Hemming.CATProduct

## Do It Yourself (2/2)

- Enter in *Advanced Meshing Tools* workbench.



- Surfacic Meshes Part1 and Part2



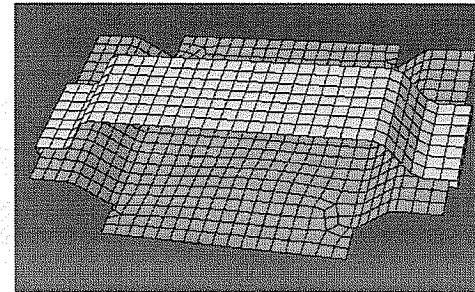
- Mesh size=6mm

- Mesh Sag=0.6mm

- Select *Mesh the Part* icon and exit.



- Color the 2 meshes



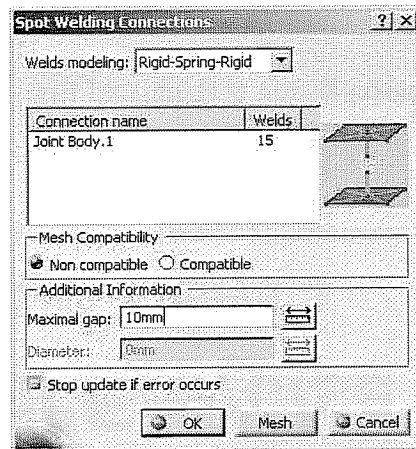
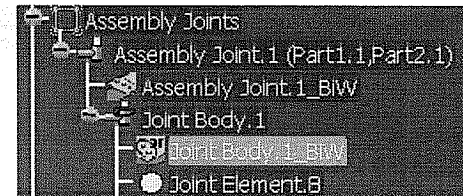
- Spot Welding Connections.



- Select Joint Body.1 in Assembly Joint

- Choose *Rigid-Spring-Rigid* as modeling

- Choose *Non Compatible* and a *maximal gap* = 10mm



# Additional Exercise

## 9. Wing Structure Analysis

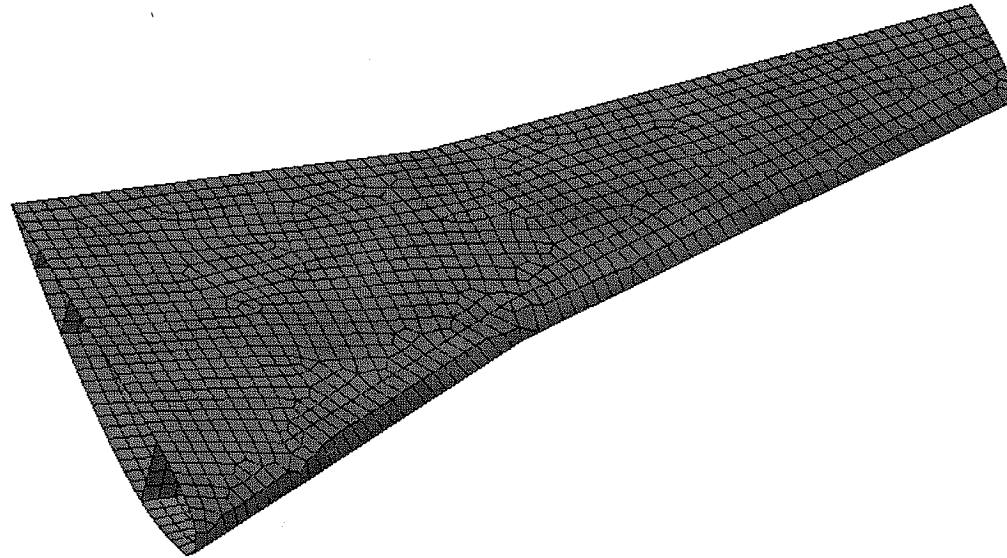


60 min.

In this exercise you will perform a Wing Structure meshing and a static analysis.

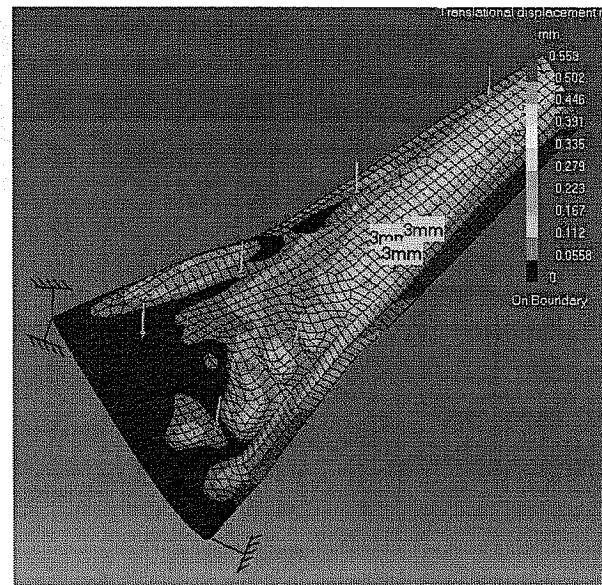
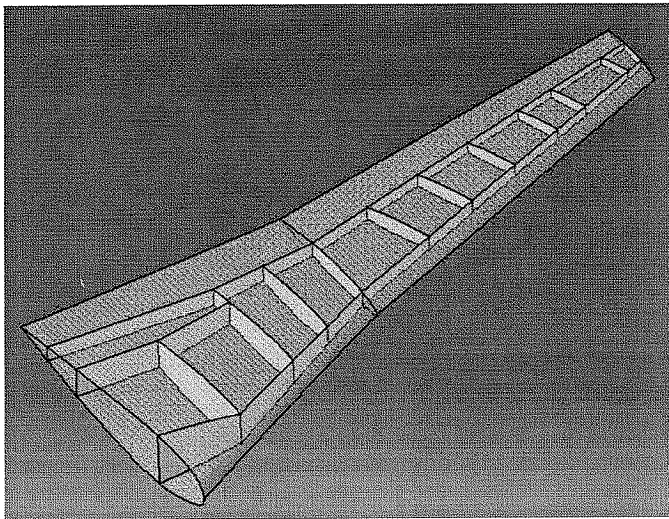
◆ You will use :

- ◆ Mesh definition
- ◆ Mesh specifications
- ◆ Pressure with Data Mapping
- ◆ Local thickness
- ◆ Static Analysis
- ◆ Post-processing



## Design Intent: Wing Structure Analysis

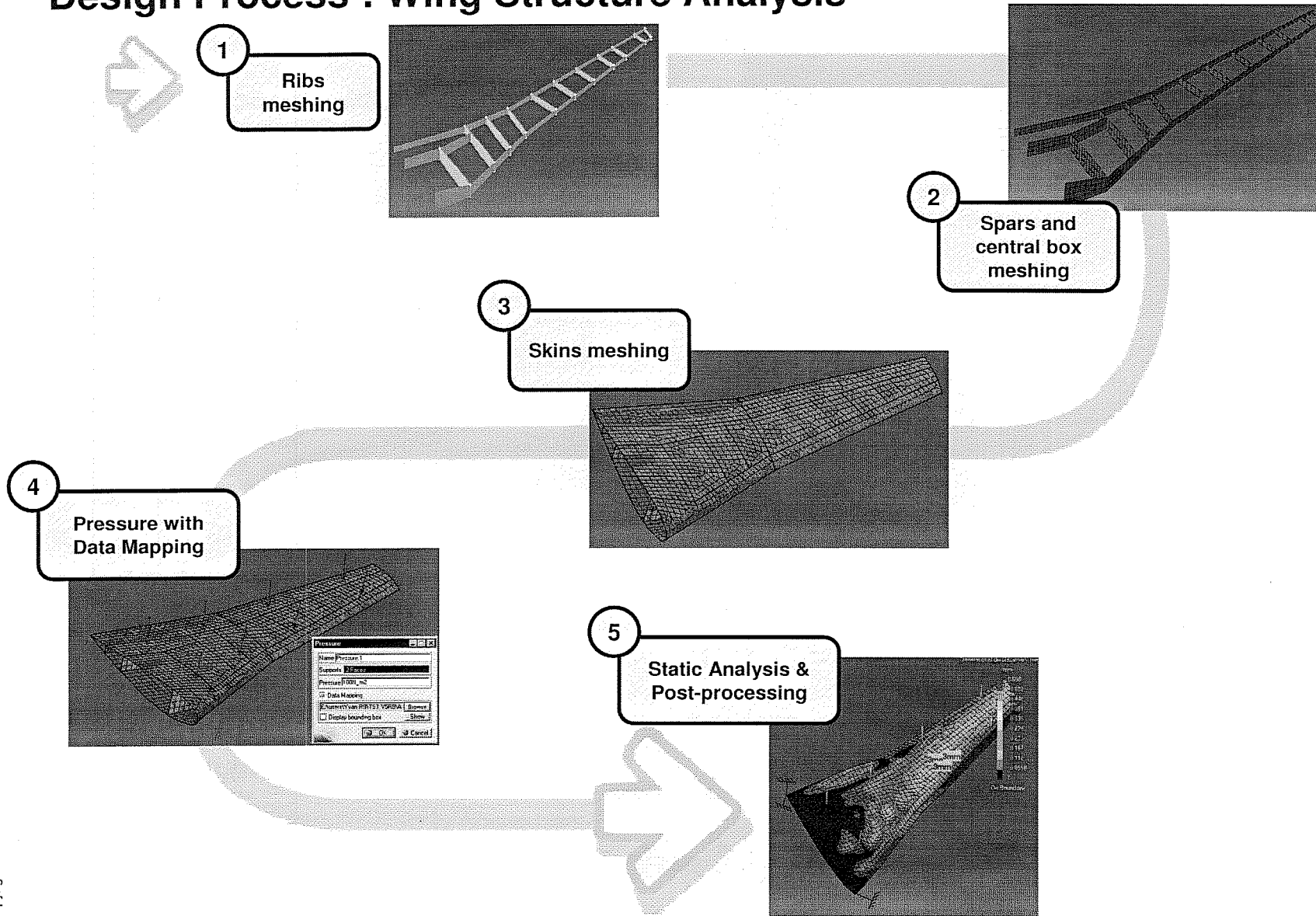
- Ribs meshing.
- Spars and central box meshing.
- Skins meshing.
- Local thickness.
- Apply Pressure with Data mapping.
- Static Analysis and Post-Processing.



⚙ Load: Wing\_structure.CATPart

Student Notes:

## Design Process : Wing Structure Analysis



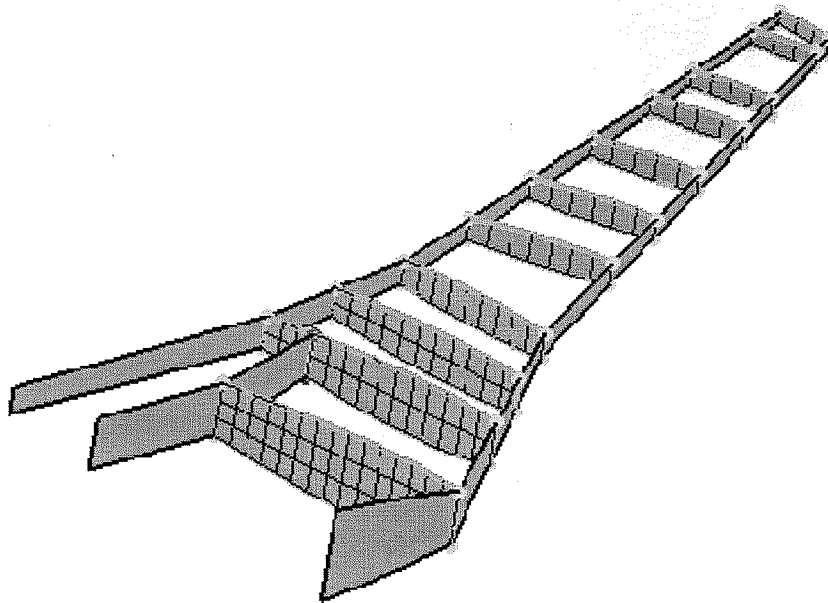
# Additional Exercise

## Wing Structure Analysis : Ribs Meshing



15 min.

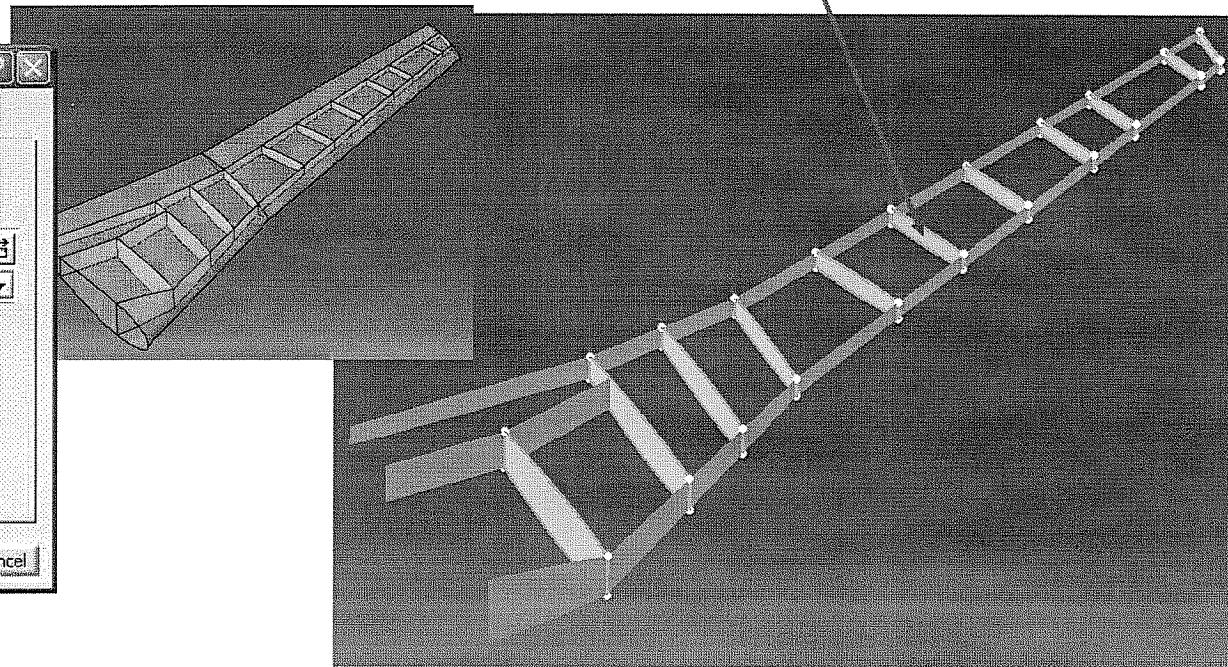
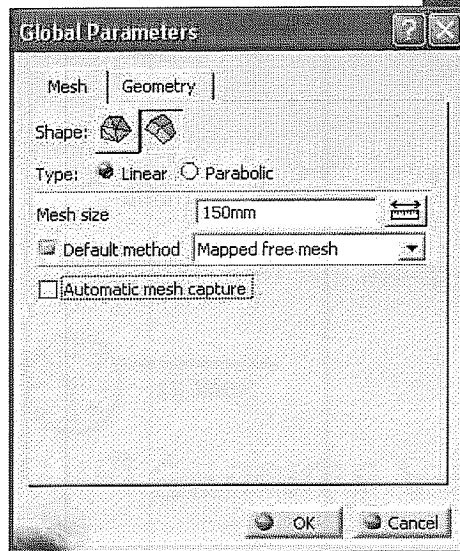
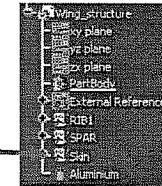
In this step you will mesh the ribs parts.





## Do It Yourself (1/3)

- Enter the *Advanced Meshing Tools* workbench: Static Analysis
- Put the Skin in *No Show* mode.
- Select *Surface Mesher* icon : 
  - Select the ribs surfaces and choose quadrangle elements shape.
  - Enter a 150mm Mesh size, and a 15mm Constraint sag.
  - Define *Mapped Free mesh* as *Default Method*

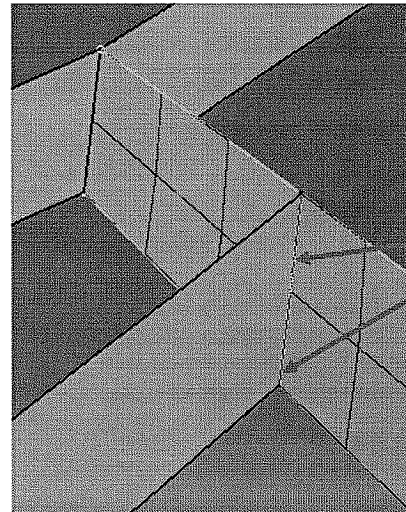
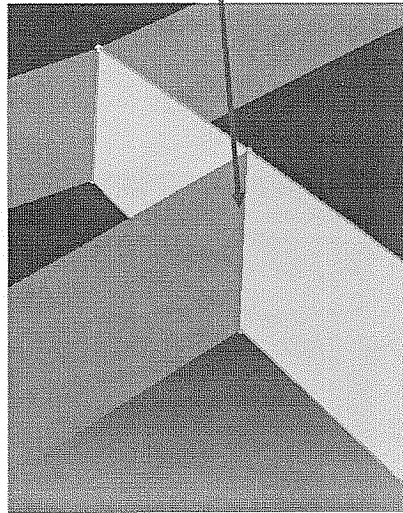
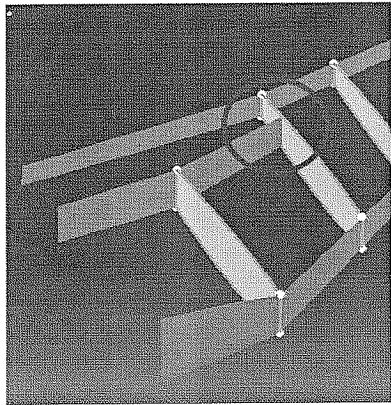




Student Notes:

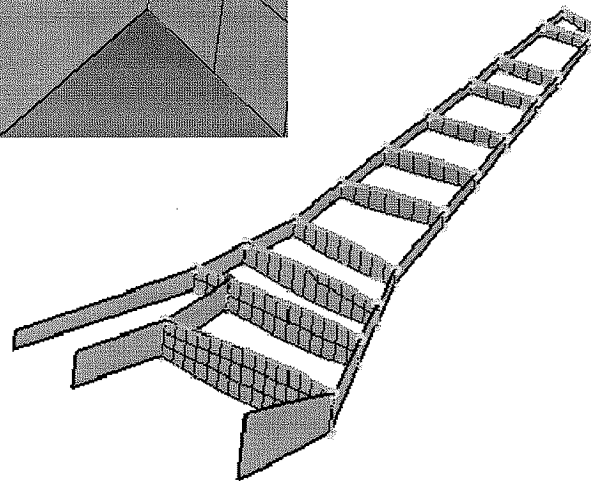
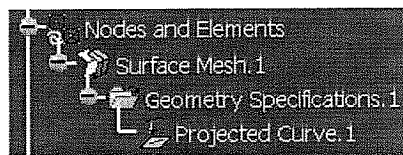
## Do It Yourself (2/3)

- Select *Project External Curve* icon
- Select the connected edge spar.
- Tolerance = 15 mm



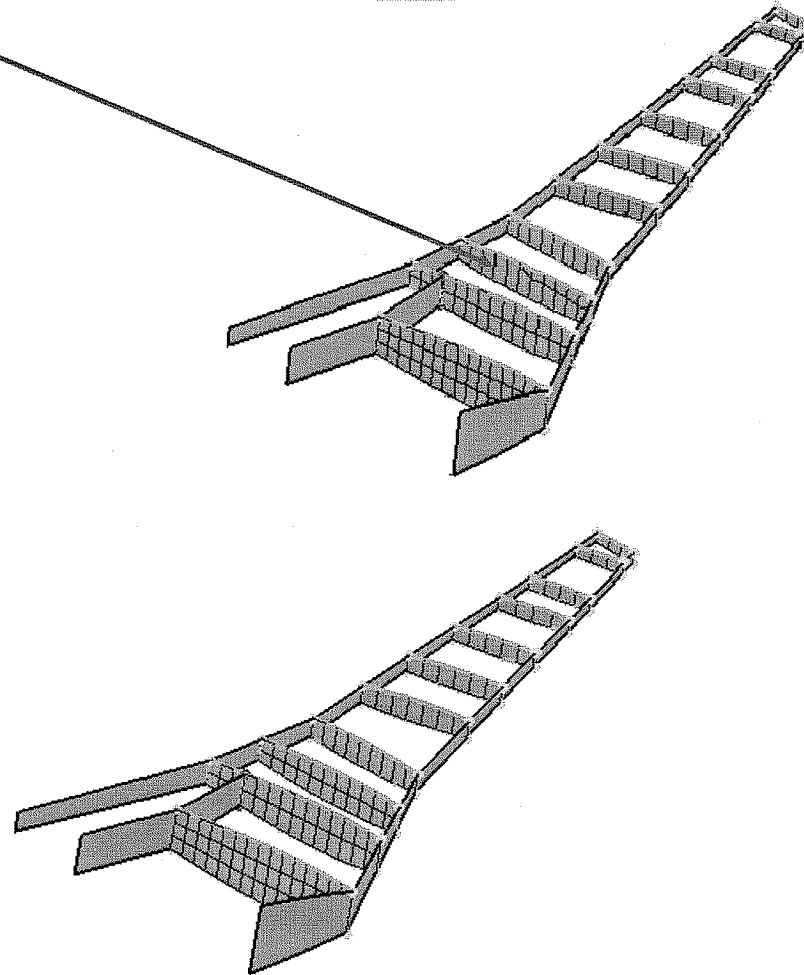
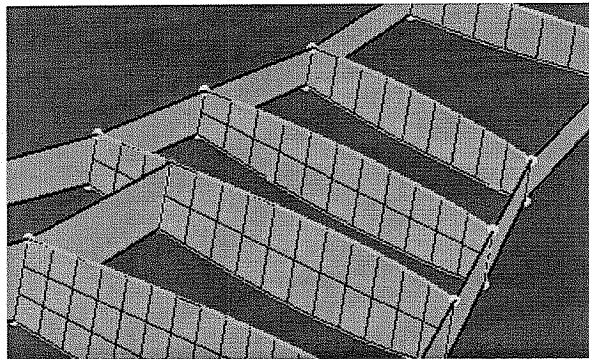
There are edges elements on the spar edge.

- Select *Mesh the part* icon

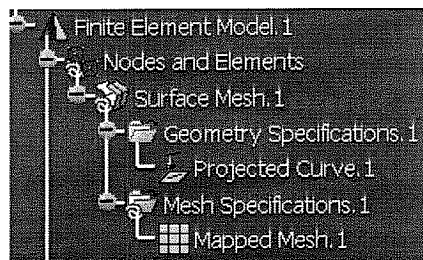


## Do It Yourself (3/3)

- Select *Remesh domain with mapped method* icon
- Select the not mapped mesh.
- Mesh size = 150 mm



● Exit



# Additional Exercise

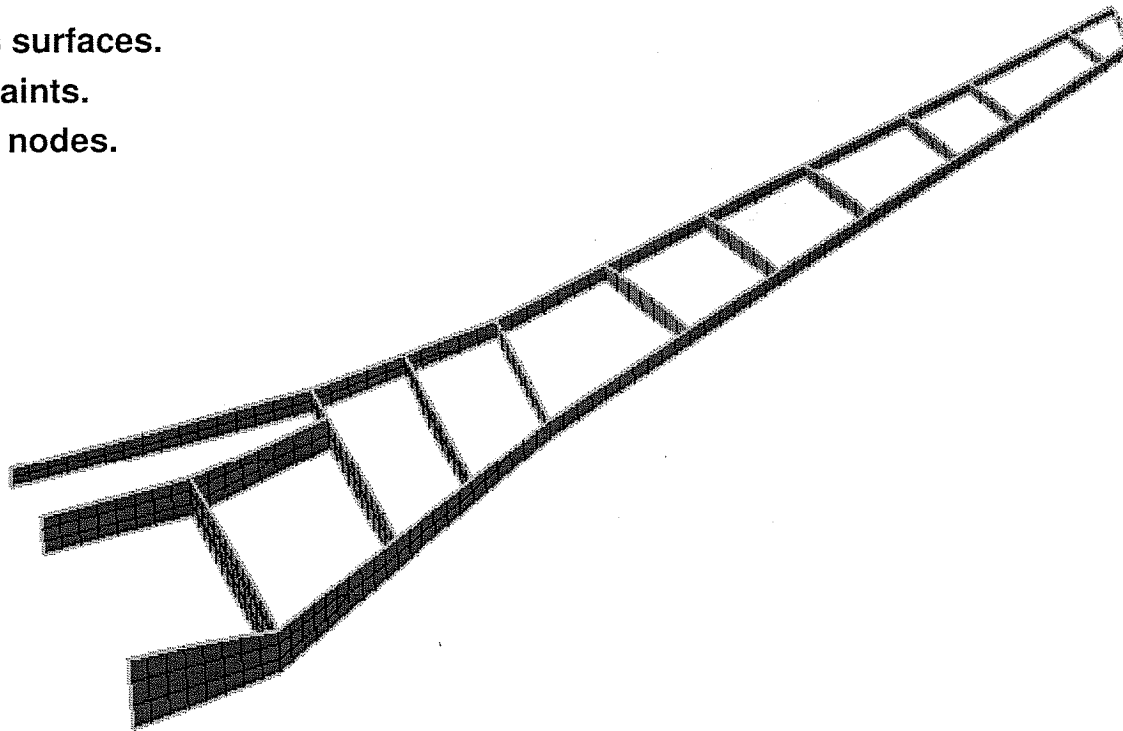
## Wing Structure Analysis : Spars and central box Meshing



15 min.

In this step you will :

- ◆ Mesh the spars surfaces.
- ◆ Remove constraints.
- ◆ Apply imposed nodes.

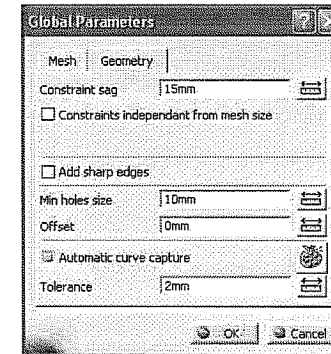
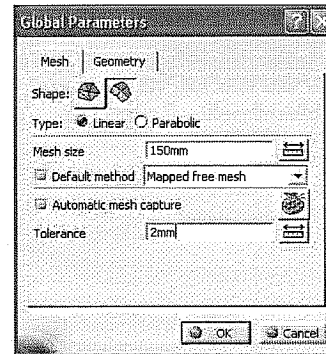


Student Notes:

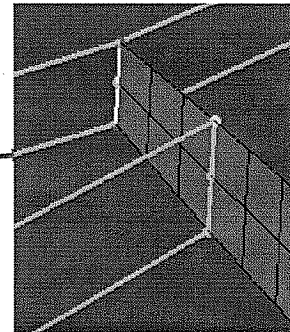
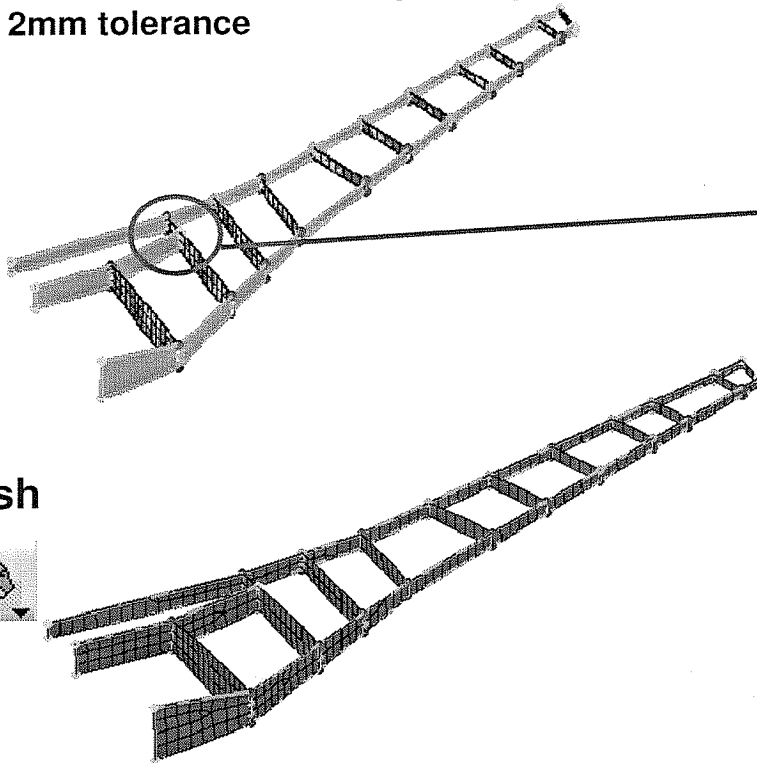
## Do It Yourself (1/3)

Select *Surface Mesher* icon 

- Select the spars surfaces and choose quadrangle elements shape.
- Enter a 150mm Mesh and 15mm as constraint sag
- Activate Automatic mesh capture option, with a 2mm tolerance.
- Activate Automatic curve capture option, with a 2mm tolerance



Curves are projected and  
Nodes are captured

Mesh 

Student Notes:

## Do It Yourself (2/3)

- Remesh locally using Mapped method



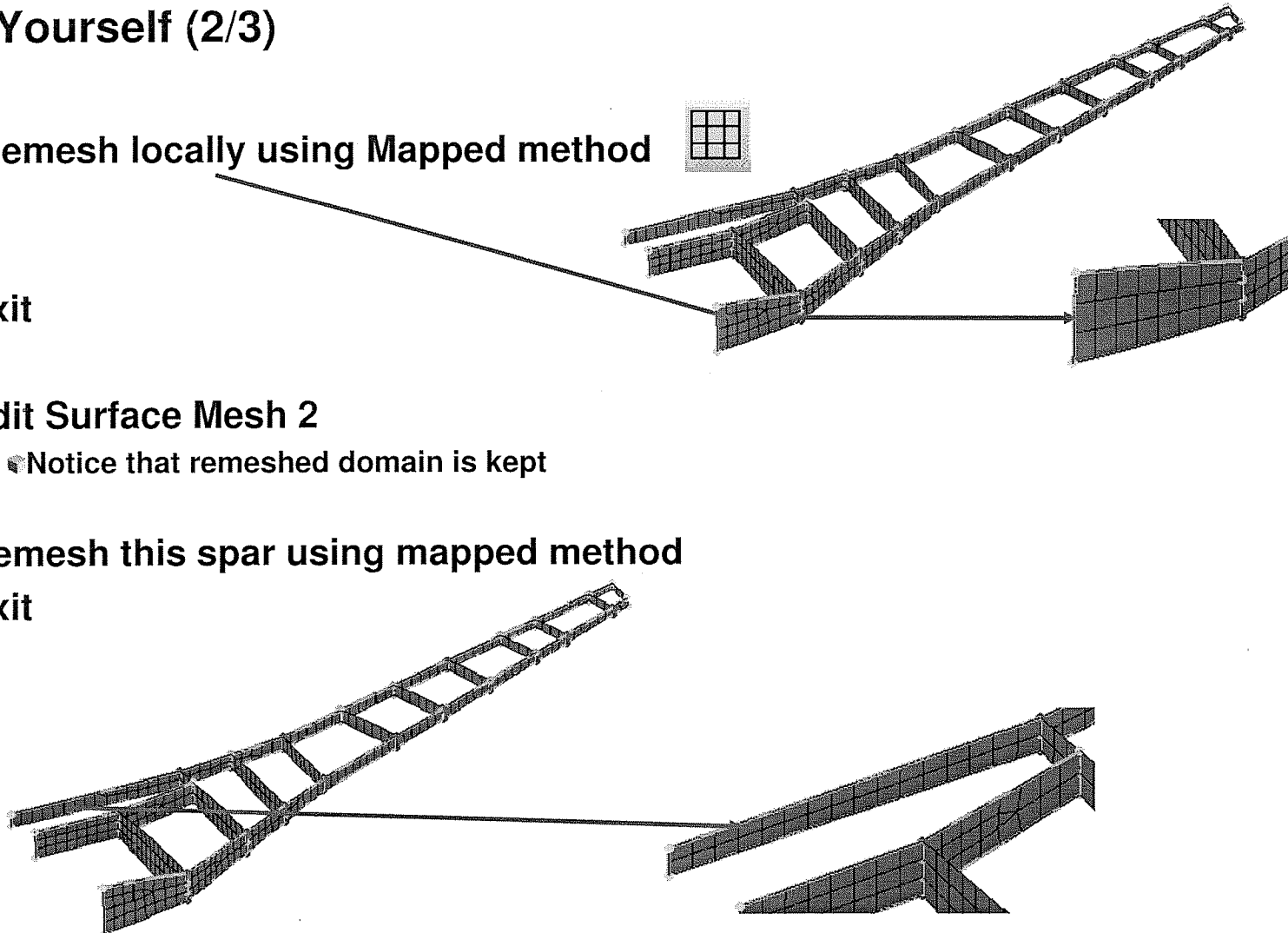
- Exit

- Edit Surface Mesh 2

- Notice that remeshed domain is kept

- Remesh this spar using mapped method

- Exit



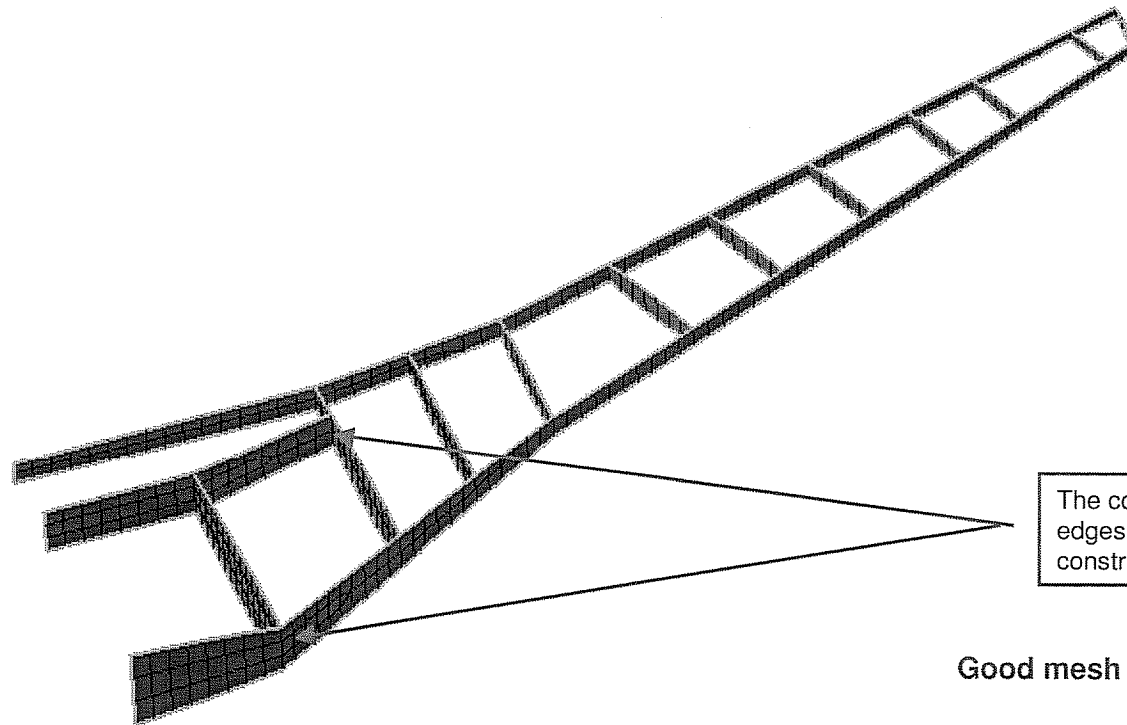
Student Notes:

## Do It Yourself (3/3)

☛ Check complete mesh compatibility with Free edges visualization tool.

☛ Click Free edges icon.

You can see that mesh is consistent



Good mesh compatibility

# Additional Exercise

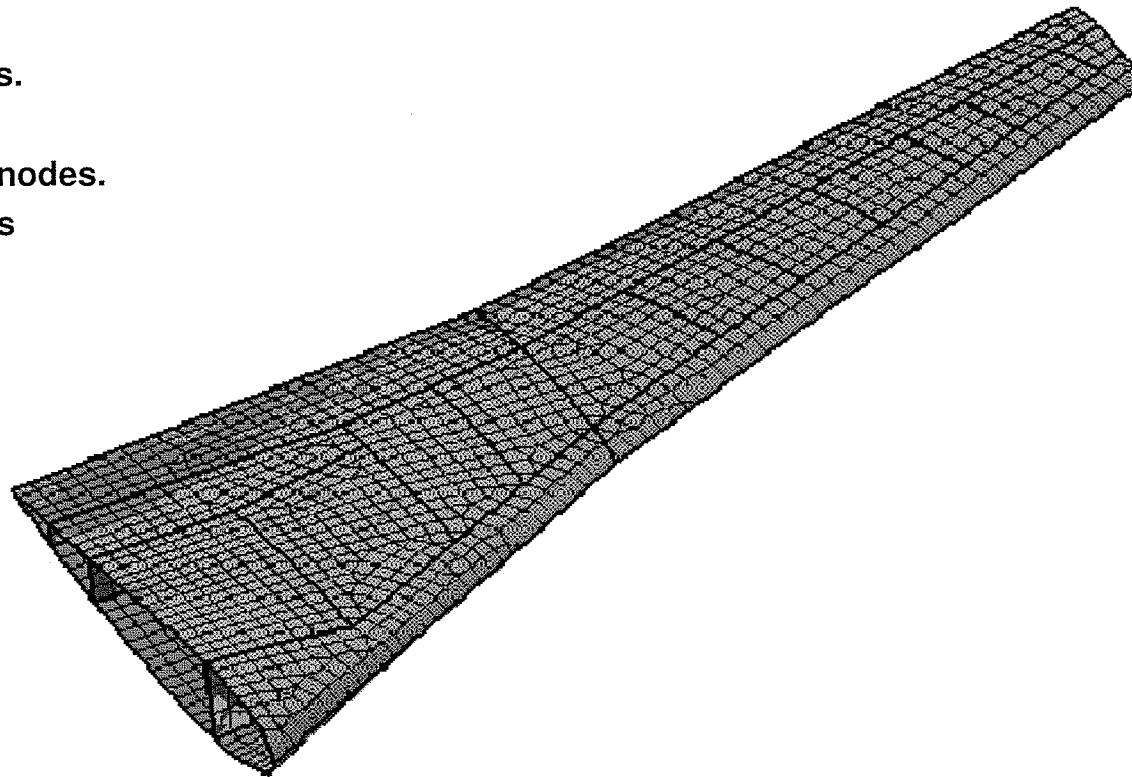
## Wing Structure Analysis : Skins Mesh



15 min.

In this step you will :

- ◆ Mesh the skins surfaces.
- ◆ Add constraints.
- ◆ Capture and condense nodes.
- ◆ Mesh quality Algorithms
- ◆ Mesh quality Analysis
- ◆ Apply properties.



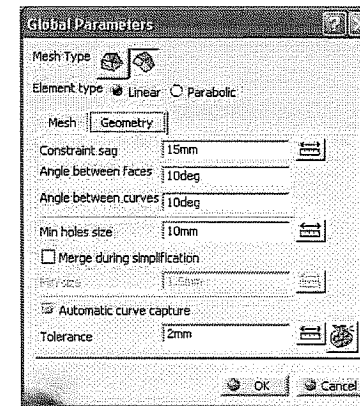
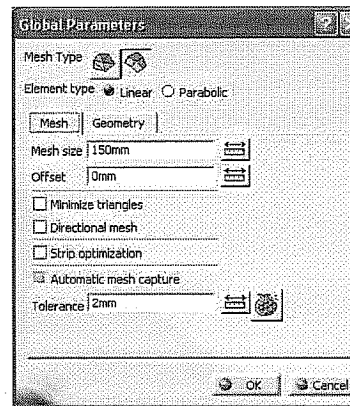


## Do It Yourself (1/3)

### Put Skin in Show mode

### Skin Meshing

- Click on Advanced Surface Mesher
- Select the Skin
- Frontal quadrangle method
- Enter 150 mm Mesh size, and 15 mm constraint sag
- Put Automatic Mesh Capture Option on, with a 2 mm tolerance
- Put Automatic Curve Capture on, with a 2 mm tolerance

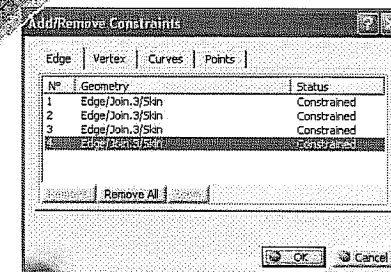
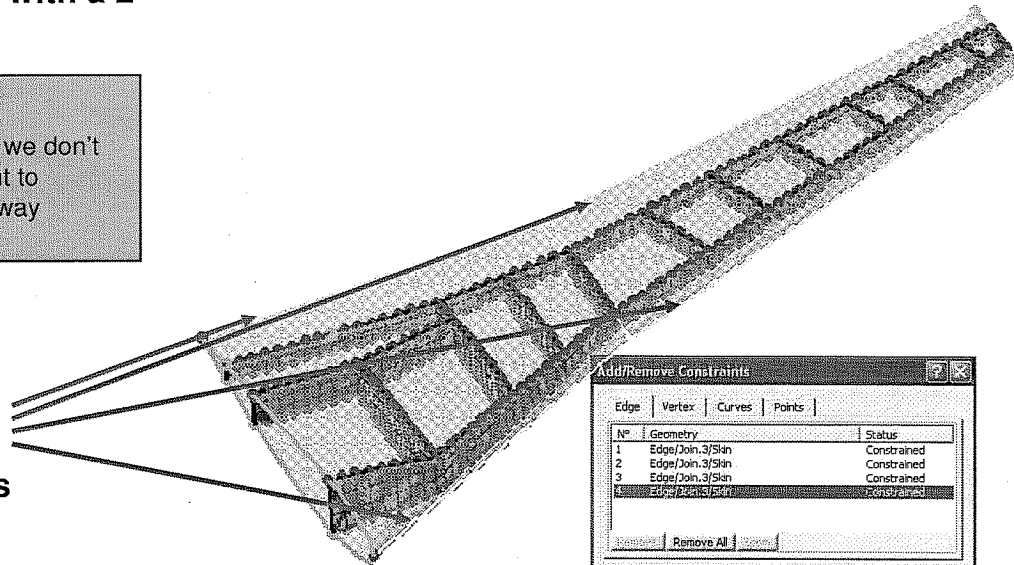


#### Note:

With Automatic Curve Capture option, we don't need anymore to select edges we want to constrain. The Mesh operation is that way shorter and easier.

### Constrain the skin edges

- Add Remove Constraints/ Edges
- Select 4 edges as shown



## Do It Yourself (2/3)

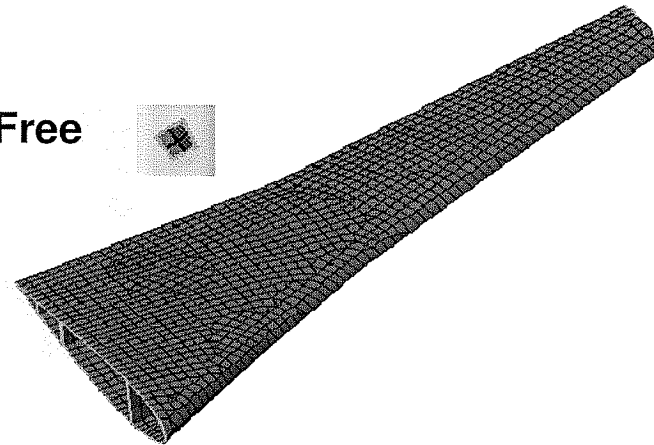
### Mesh and Exit



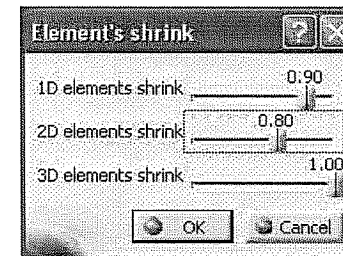
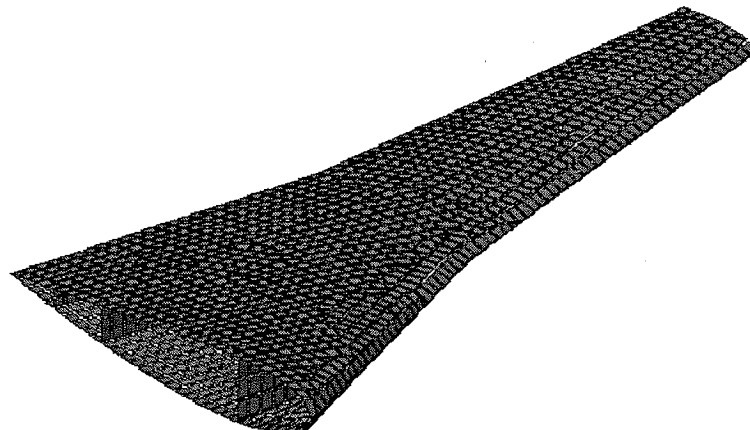
### Check complete mesh compatibility with Free edges visualization tool.

Click Free edges icon.

You can see that mesh is consistent



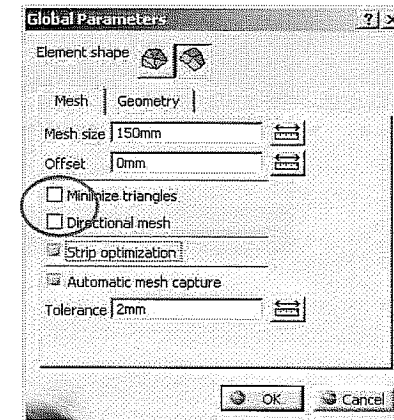
### Decrease 2D elements shrink coefficient to 0.8



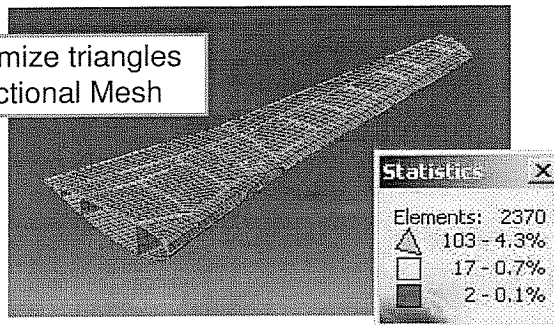
## Mesh Quality Enhancement

During the Mesh operation, you can choose between several combination of mesh algorithms

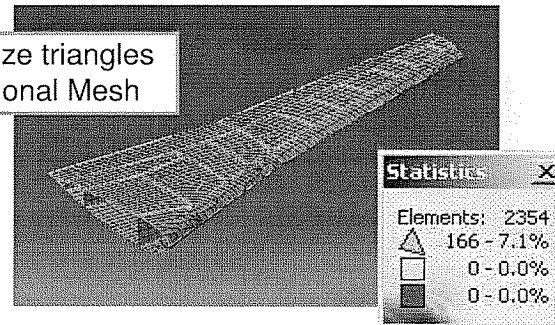
- ◆ produce more regular mesh flow patterns
- ◆ Nodes degree optimization in order to reduce nodes connected to 3 quads
- ◆ Enhanced optimization tools to reduce number of poor/bad elements



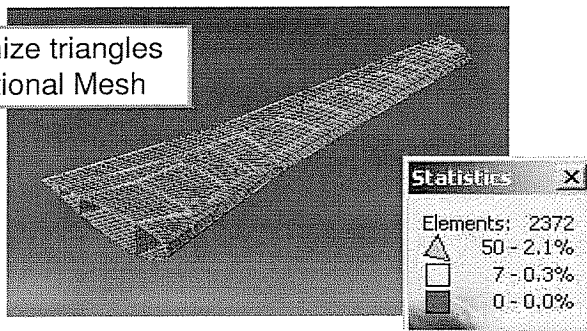
Minimize triangles  
Directional Mesh



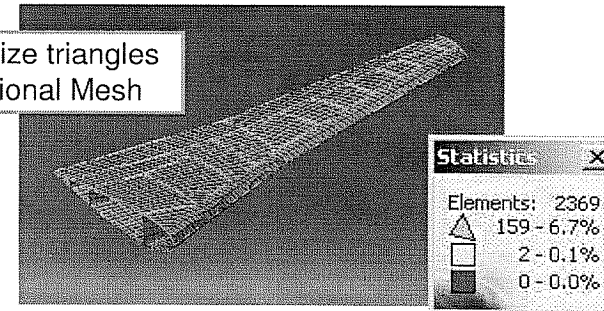
Minimize triangles  
Directional Mesh



Minimize triangles  
Directional Mesh

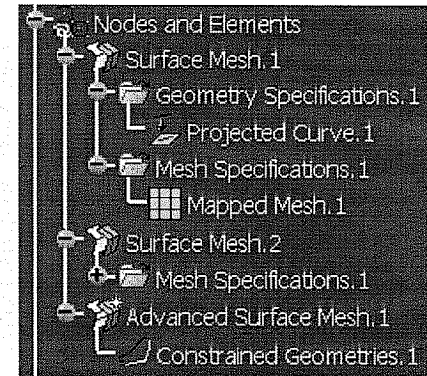
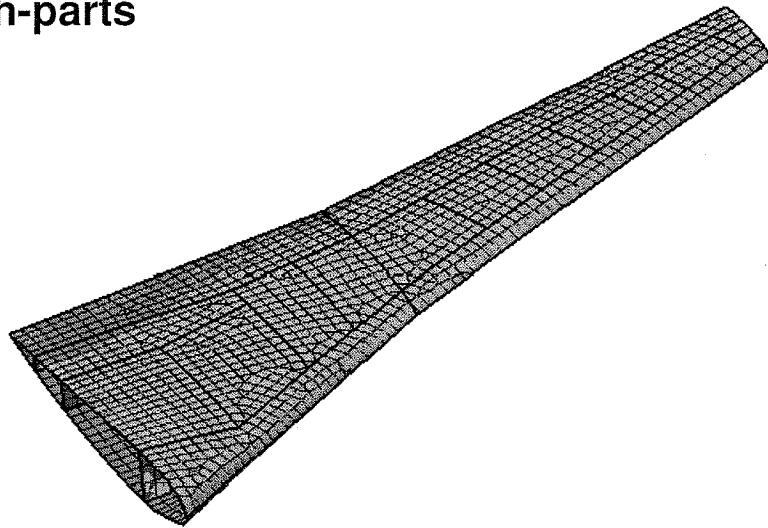


Minimize triangles  
Directional Mesh



## Do It Yourself (3/3)

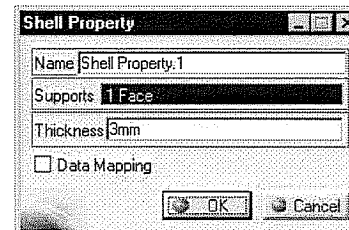
- Let see in the tree : there are one single part and three mesh-parts



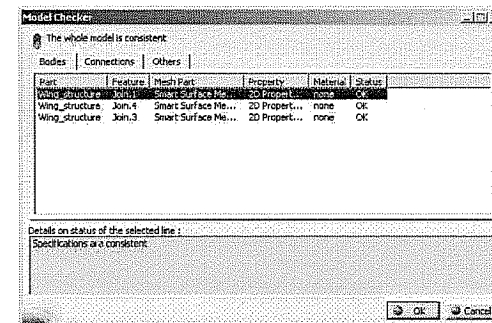
- Enter Generative Structural Analysis

- Apply properties :

- Select each mesh-part and enter a thickness of 3mm.



- Check the consistency of the model



# Additional Exercise

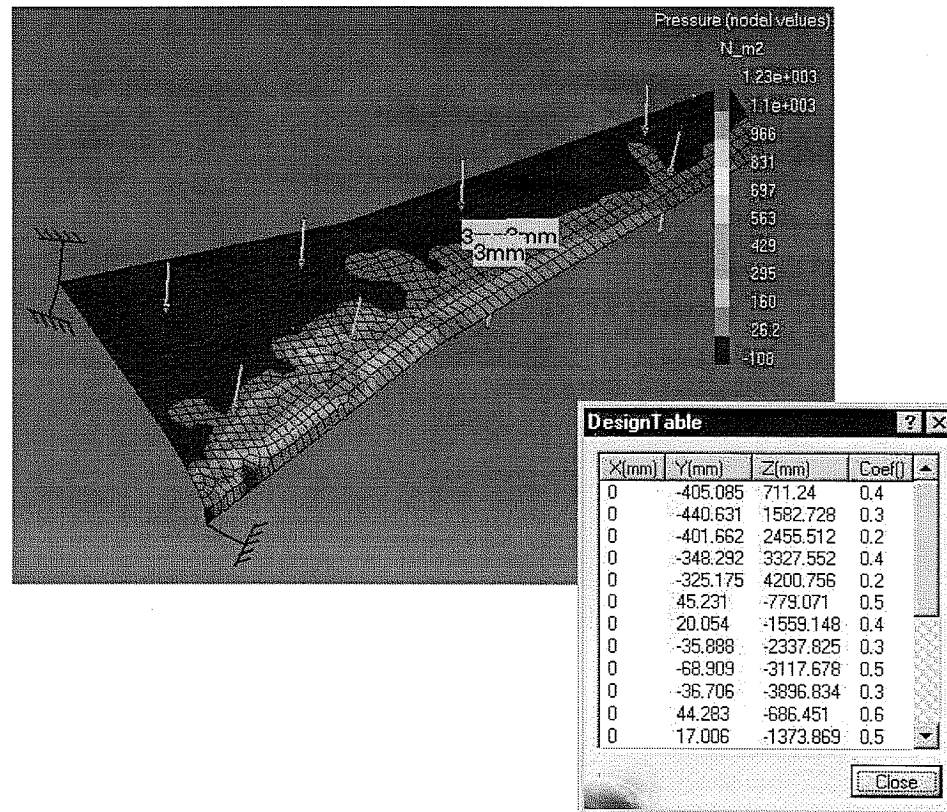
## Wing Structure Analysis : Pressure with Data Mapping



15 min.

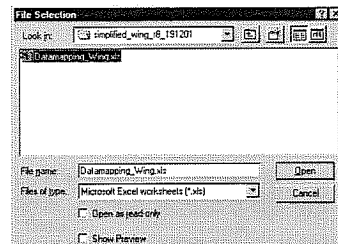
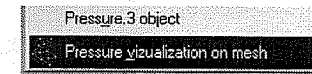
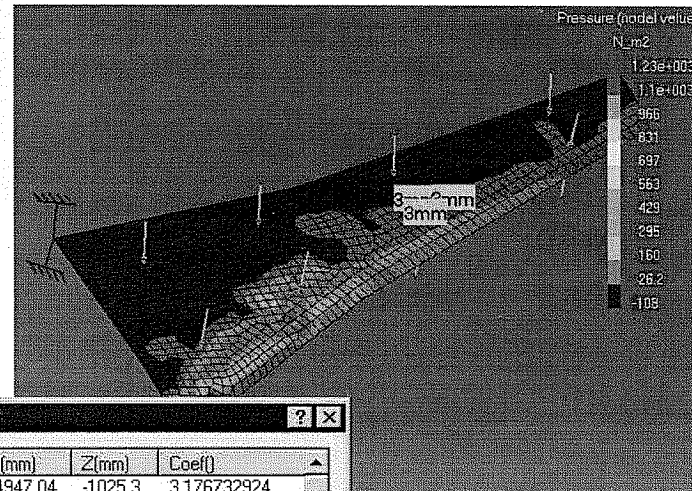
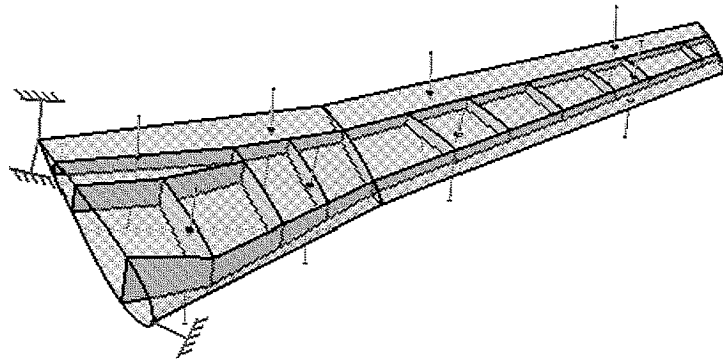
In this step you will :

- ◆ Apply clamps.
- ◆ Apply pressure with Data Mapping.
- ◆ Visualize pressure on skins.



## Do It Yourself

- Put clamps on fuselage connection edges.
- Apply and visualize a 100 Pa pressure on the skin with *Data mapping*.
  - Choose the *Cp.xls* file.
  - After computation, Select *Pressure Visualization on mesh* with contextual menu on *Pressure.1* icon.
- Apply an opposite 35 Pa pressure on the second skin.



With Data Mapping, we are able to apply an un-uniform loading, closer to reality.

| X(mm)   | Y(mm)    | Z(mm)    | Coef()      |
|---------|----------|----------|-------------|
| 10011.9 | -4947.04 | -1025.3  | 3.176732924 |
| 10389.1 | -5662.36 | -1029.07 | 2.916649109 |
| 10766.4 | -6378.1  | -1032.84 | 2.635655344 |
| 10957.2 | -6739.93 | -1034.74 | 2.484424294 |
| 11147.1 | -7100    | -1036.63 | 2.327343222 |
| 11338.6 | -7463.24 | -1038.55 | 2.182218768 |
| 11531.7 | -7829.6  | -1040.47 | 1.988315009 |
| 11717.6 | -8182.41 | -1042.33 | 1.812878871 |
| 11893   | -8514.72 | -1044.18 | 1.640001537 |
| 10554.6 | -5127.94 | -912.391 | 1.410937967 |
| 10737.3 | -5481.54 | -916.294 | 1.264314053 |

Space coordinates of pressure experimental points.

Amplification coefficient : at this point, the pressure value is  $0.3 \times 100$  Pa.

# Additional Exercise

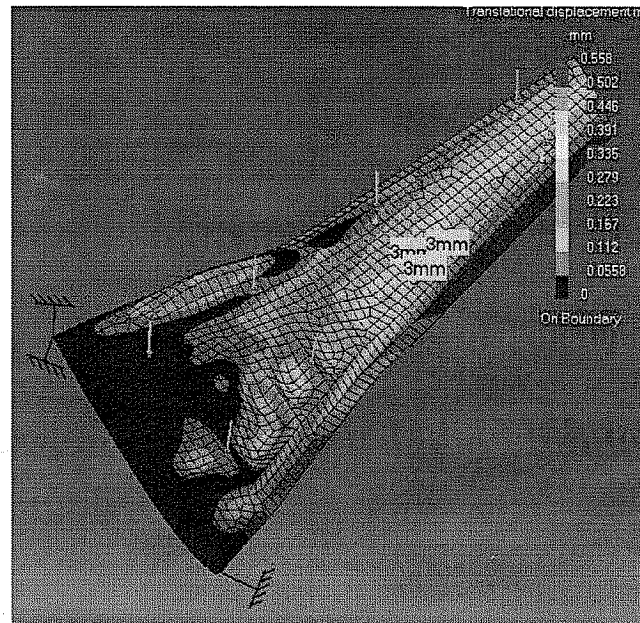
## Wing Structure Analysis : Static Analysis and Post-Processing



15 min.

In this step you will :

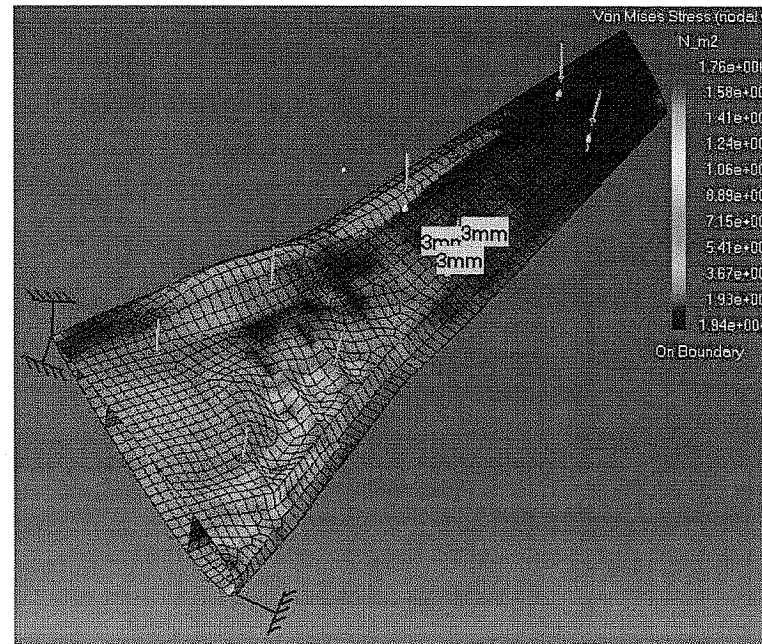
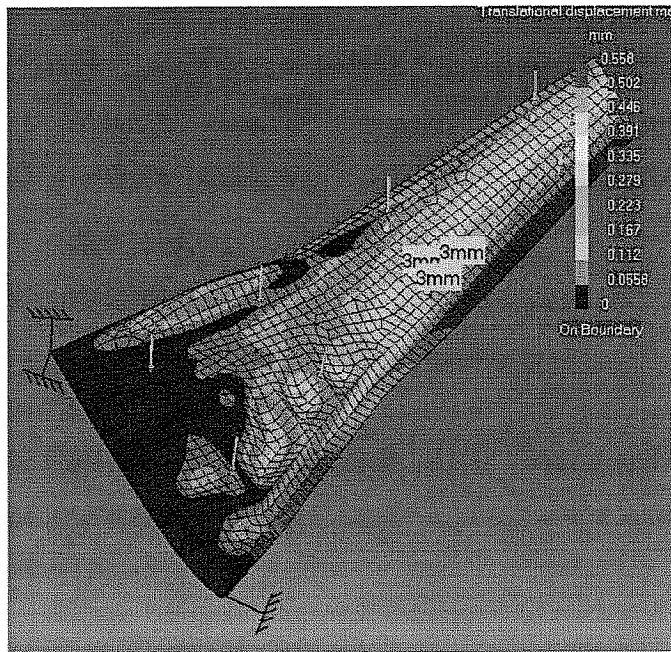
- ◆ Compute a static analysis.
- ◆ Visualize displacements and Von Mises stresses.
- ◆ Animate results.





## Do It Yourself

- Compute a static analysis.
- Animate deformations : check that ribs, spars and skins meshes are well connected.
- Show displacements results.
- Show Von Mises stresses.
- Animate results.



## Fundamental steps

### ■ Meshing

- ◆ Surface mesh.
- ◆ Automatic capture with Add/Remove constraint tool.
- ◆ Mesh a part.
- ◆ Free edges visualization.
- ◆ Mesh analysis.
- ◆ Mesh-part statistics.
- ◆ Apply shell properties and check model.

### ■ Apply Pressure with Data Mapping

- ◆ Apply clamps.
- ◆ Apply a pressure.
- ◆ Enter Data Mapping.

### ■ Static Analysis and Post-Processing

- ◆ Compute a static analysis.
- ◆ Visualize displacements and animation.
- ◆ Visualize Von Mises stresses.

# Additional Exercise

## 10. Meshing of Parametric Conceptual Body

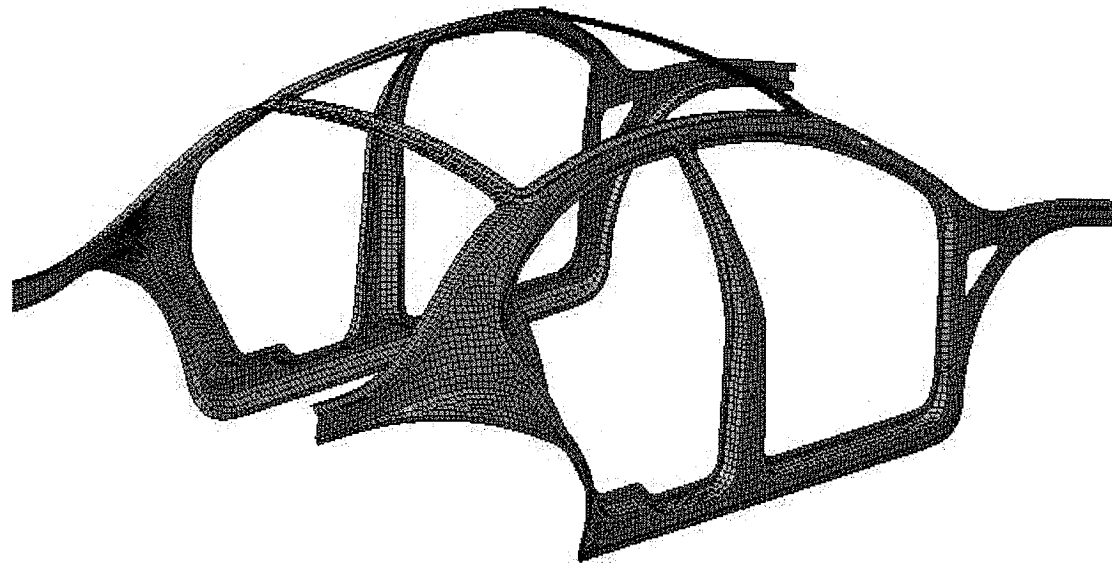


60 min.

In this exercise you will perform a shell meshing on a conceptual body, update it after design modification and realize a quality analysis.

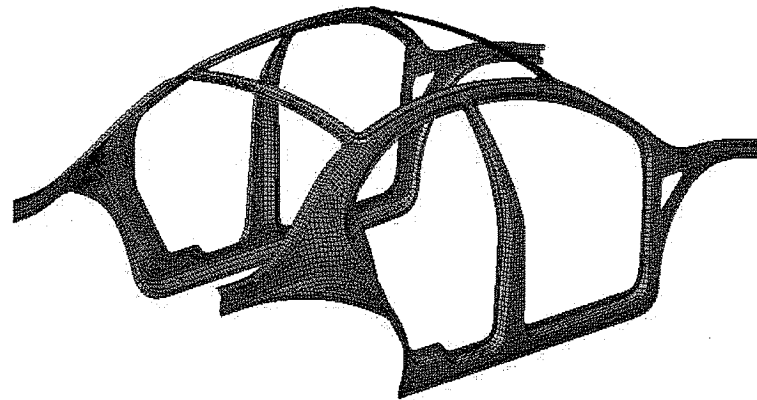
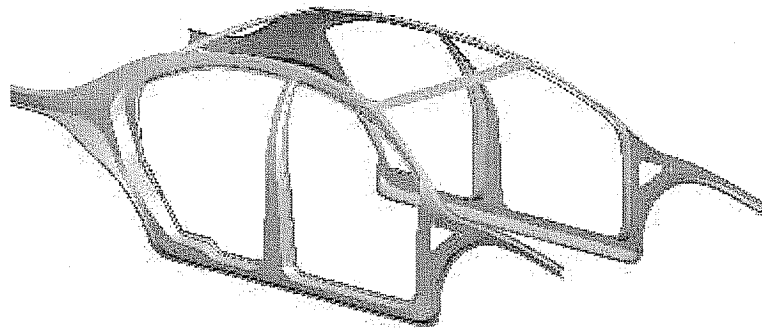
You will use :

- ◆ Mesh Definition Tools
- ◆ Mesh Specification Tools
- ◆ Mesh Modification Tools
- ◆ Quality Analysis Tools
- ◆ Pre-processing Analysis



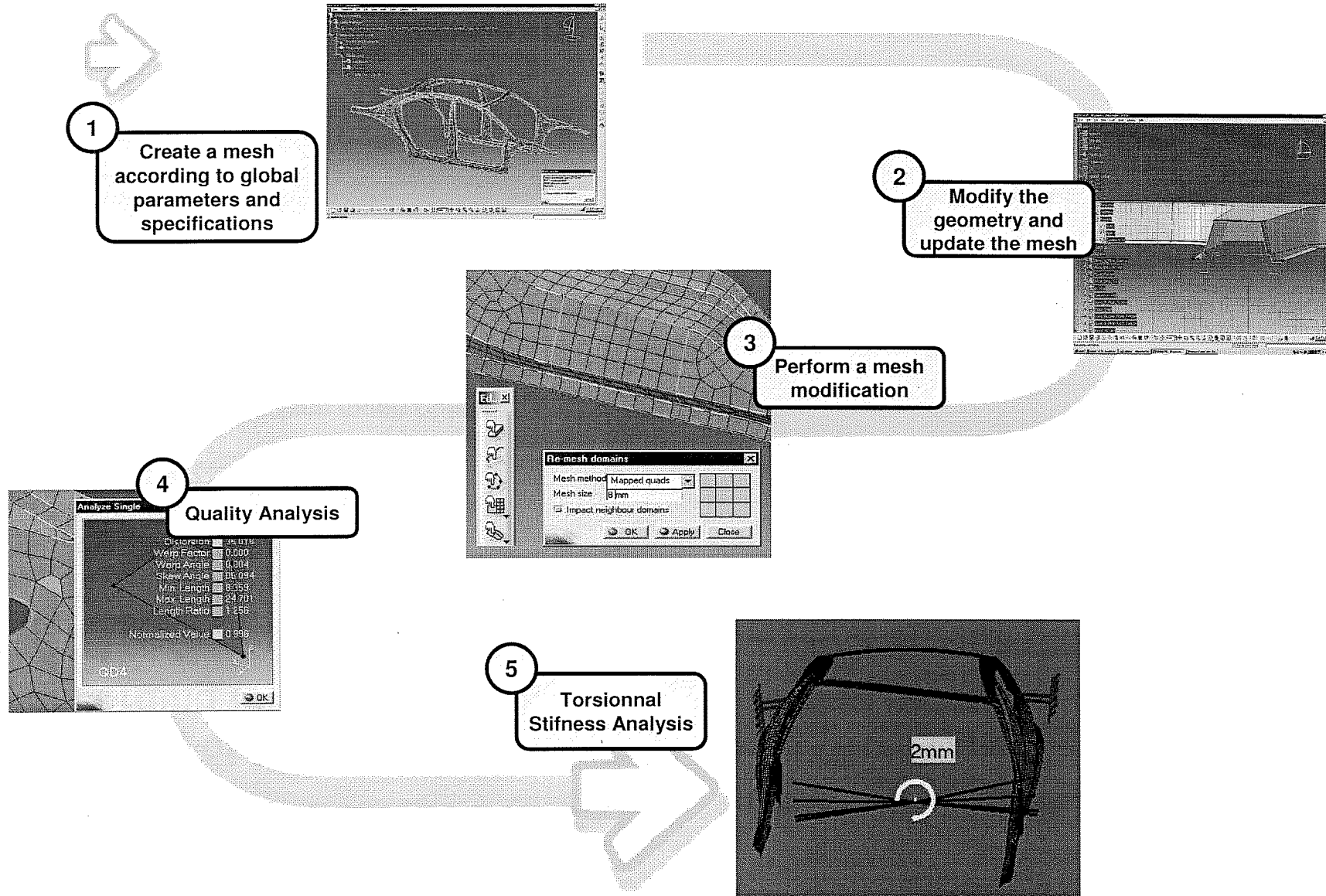
## Design Intent: Meshing of Parametric Conceptual Body

- Visualize Static Pre-Processing Elements on Mesh
- Create and Name Static Analysis
- Define meshing specifications
- Mesh the part
- Perform geometric modifications and update the mesh
- Analyse elements quality
- Modify the mesh
- Perform a torsional stiffness analysis



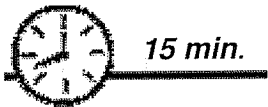
Student Notes:

## Design Process : Meshing of Parametric Conceptual Body



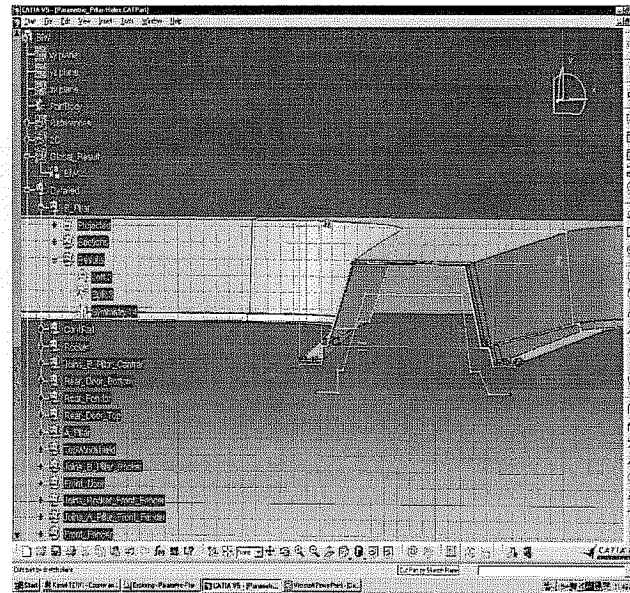
## ***Additional Exercise***

## Meshing of Parametric Conceptual Body: Creating a mesh according to specifications



**In this step you will :**

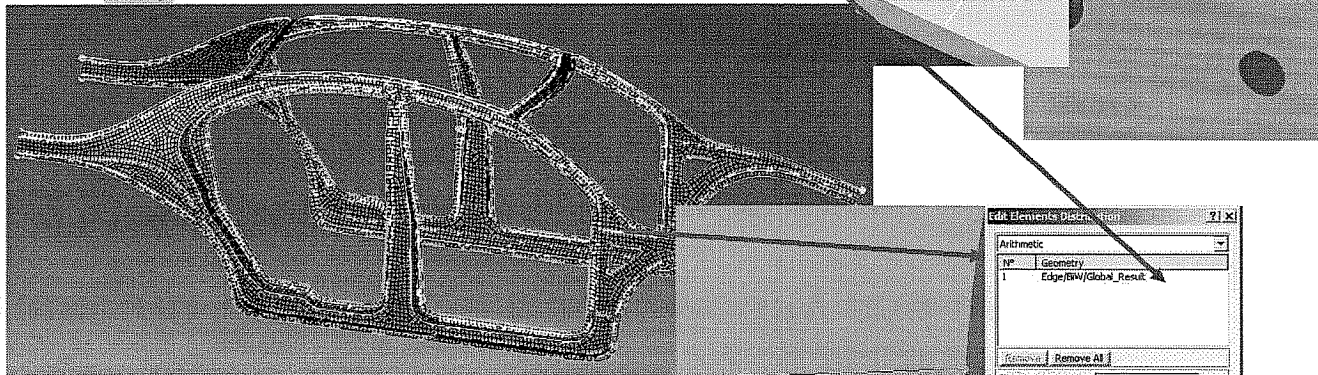
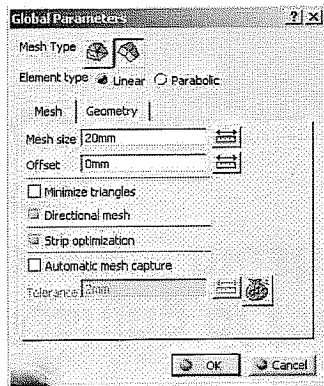
- ◆ create a new surface mesh object
- ◆ save the analysis document



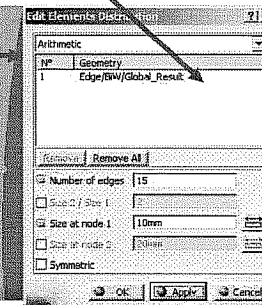
**Student Notes:**

## Do It Yourself

- Visualize how the restraints defined on the assembly are translated onto the mesh
- Creating a mesh according to specifications :
  - Enter in the “Advanced Meshing Tools” workbench. Create a new static case
  - Create a new surface mesh (use quadrangle frontal method, mesh size = 20mm, maximum sag = 1.5mm and minimum holes size = 5mm)
  - Remove the two smallest holes and the two gaps
  - See topology with geometry simplification, and go back to previous toolbar.
  - Define edge constraints and node distributions arithmetic distrib, 15 as nbs of edge, 10mm size at node1
  - Create the mesh . Exit.



Load: CATFMSParametric\_Pillar.CATPart





## Additional Exercise

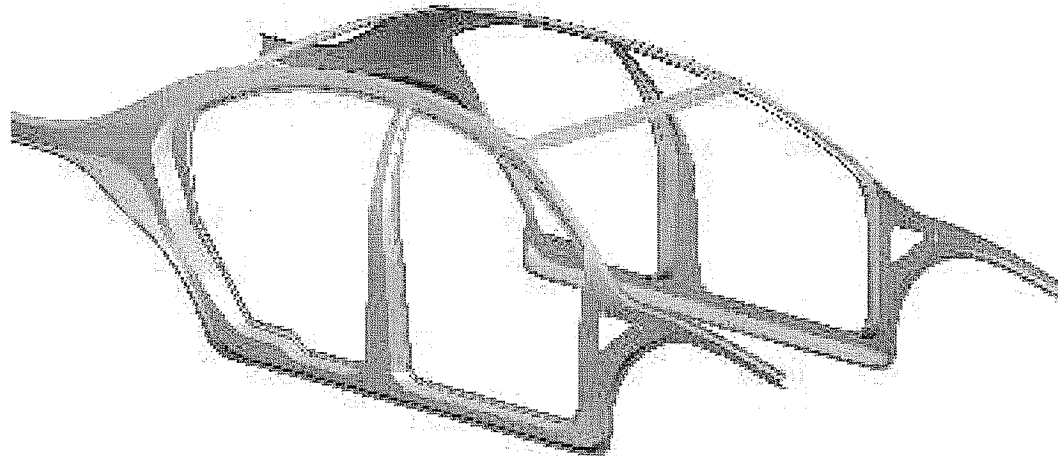
**Meshing of Parametric Conceptual Body: Modify and update an existing mesh**



15 min.

In this step you will :

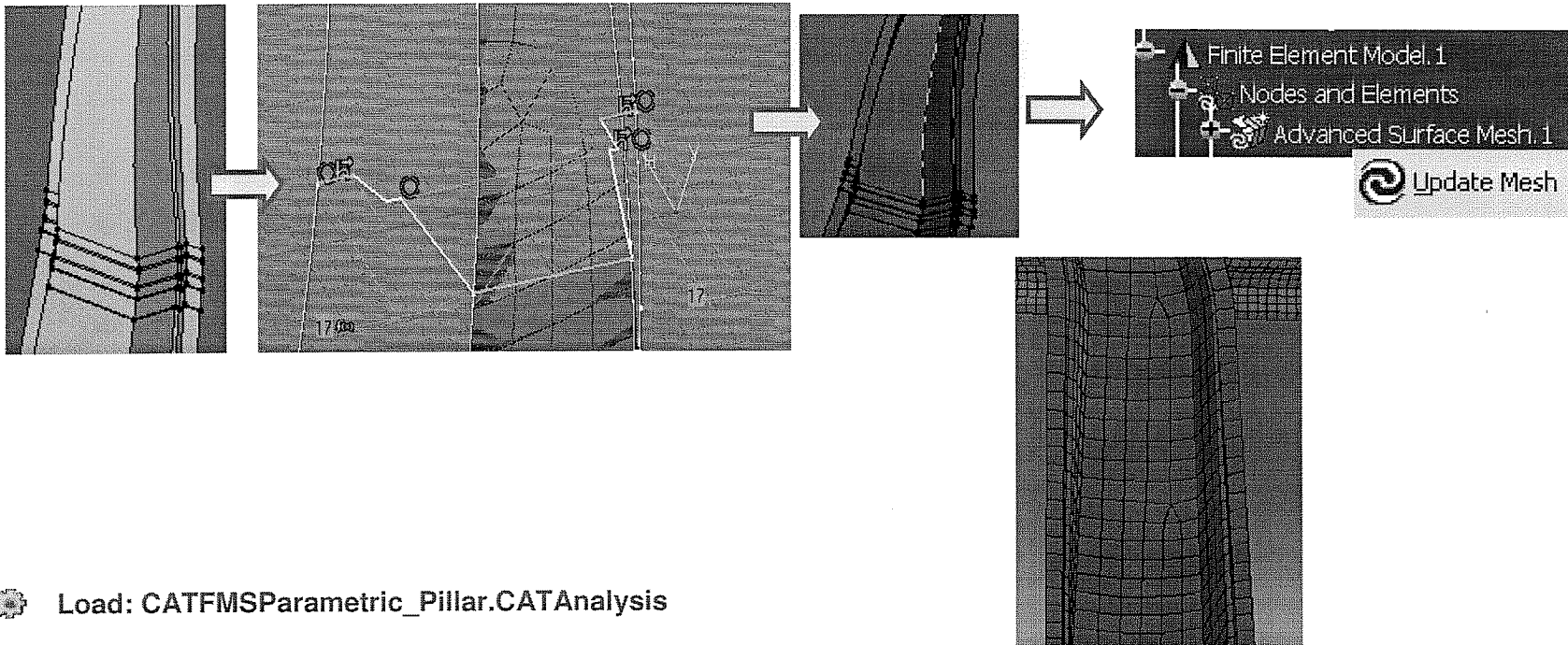
- ◆ modify the geometry
- ◆ update the mesh
- ◆ directly modify it again.



## Do It Yourself (1/2)

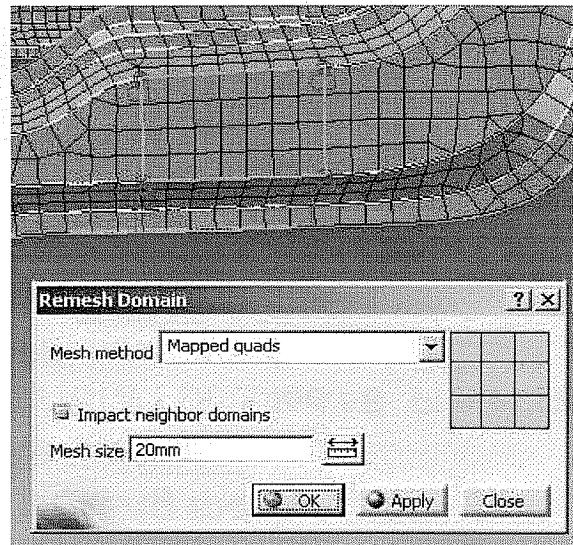
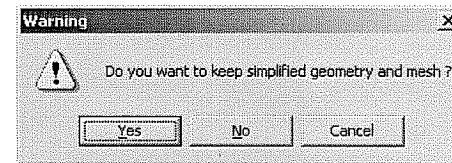
### ◆ Modify and update an existing mesh :

- ◆ Double click on BiW in the tree.
- ◆ Modify a pillar section (BiW / Detailed / B\_Pillar / Section / Sketch.2) : replace dimension 17 mm by 19 mm (put sketch in Show).
- ◆ Double click on Finite Element Model to go back to Advanced Meshing Tools workbench
- ◆ Using its contextual menu, update the mesh



## Do It Yourself (2/2)

- Edit Advanced Surface Mesh.1 and answer YES
- Define a Mapped Mesh domain:
  - ◆ Add constraint to create a not mapped domain with 4 edges
  - ◆ Click on the *Re-mesh a domain* icon
  - ◆ select this domain.
  - ◆ Choose *Mapped quads* method.
  - ◆ And 20 mm as size



# Additional Exercise

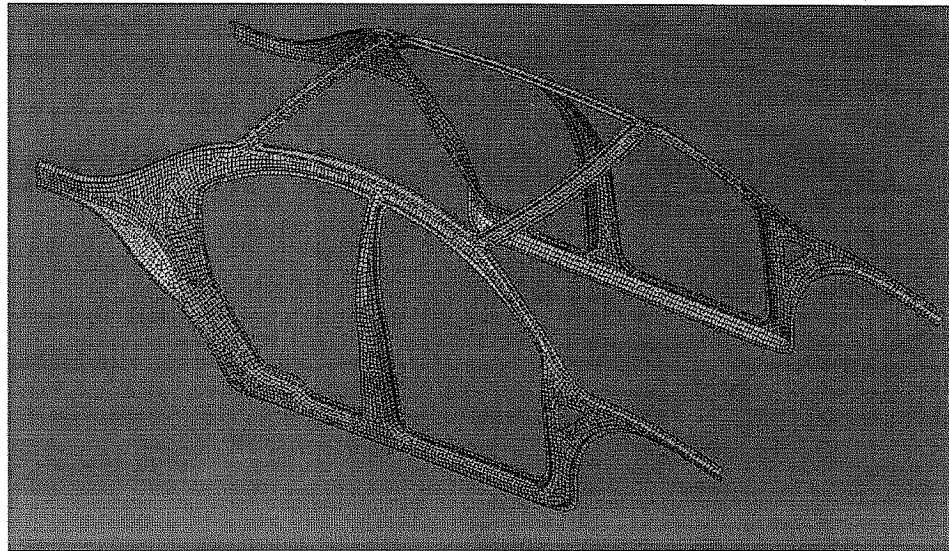
## Meshing of Parametric Conceptual Body : Analyse Mesh Quality



15 min.

In this step you will :

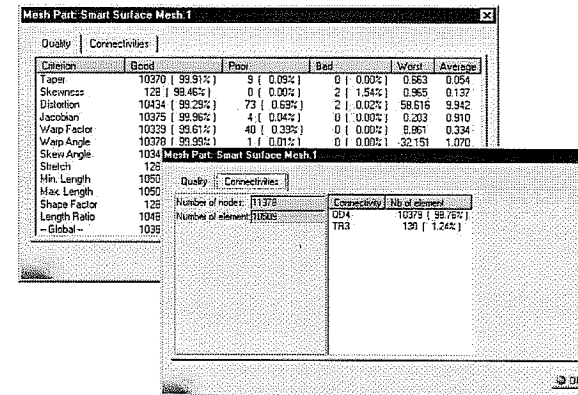
- ◆ analyse the mesh quality
- ◆ edit an analysis report.
- ◆ modify the mesh to improve quality



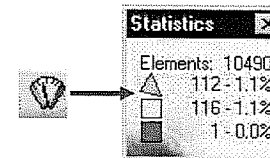
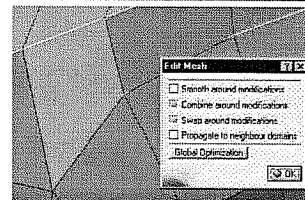
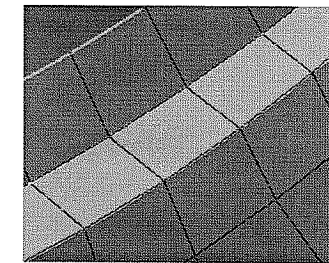
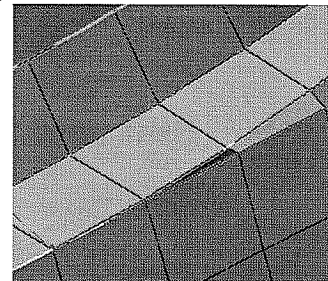
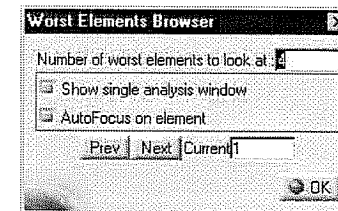
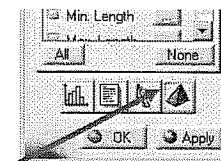
## Do It Yourself

### ■ Analyse Mesh Quality :

- ◆ Click on the *Quality visualization* icon. 
- ◆ Hide geometry.
- ◆ Edit an analysis report : click right on the mesh-part and select *Analyse*.
  - There are some bad elements to correct.
  - There are few triangle elements (5-10 % max in theory).



- ◆ Modify the mesh in order to increase its quality :
  - First, check free edges (change your visualization before). 
  - Then, click the *Quality analysis* icon. 
  - Choose *Worst Elements Browser* function.
  - Double click on the mesh-part : choose YES to keep specifications and mesh.
  - Click on the icon *Edit Simplification* .
    - Select the yellow constrained curve.
    - Repeat for the other side.
    - Correct the last one with *Edit Mesh*.
- ◆ Check for intersections. 
  - Enter a 3,5 mm tolerance.



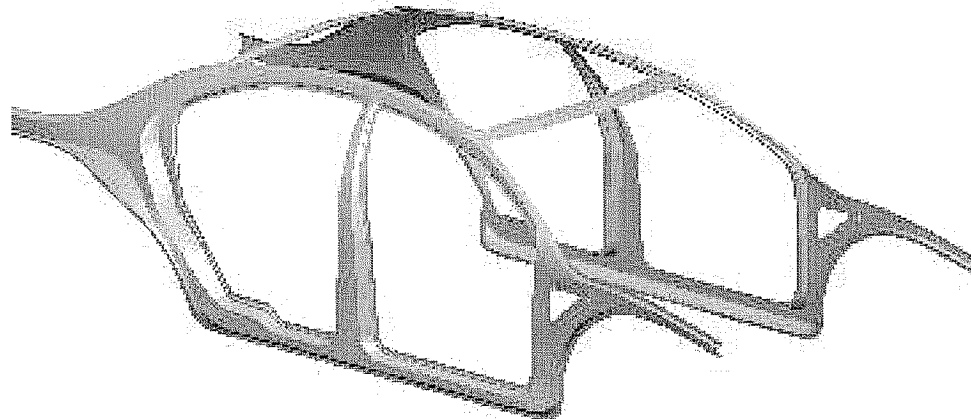
## Additional Exercise

### Meshing of Parametric Conceptual Body : Static Analysis performance



15 min.

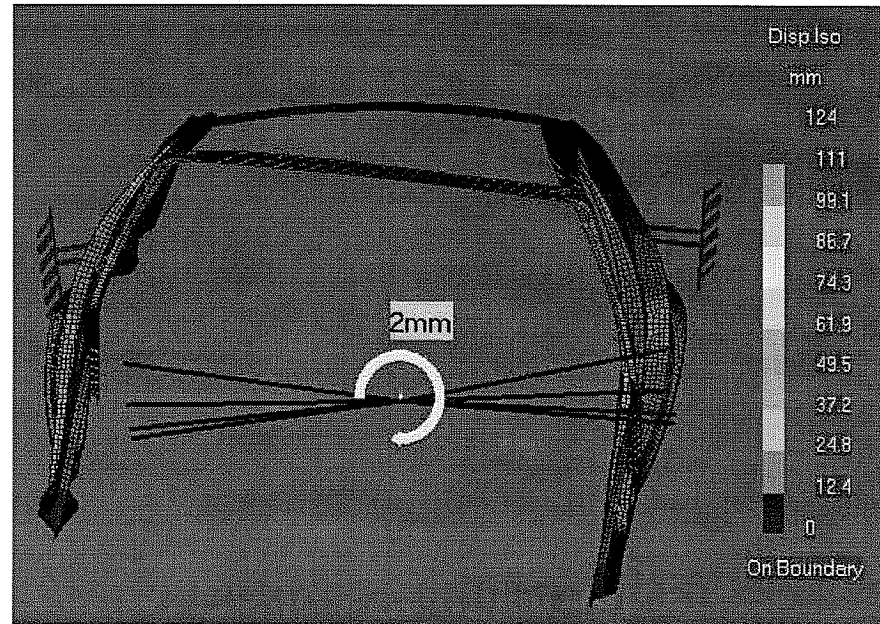
In this step you will perform a torsional stiffness analysis.



## Do It Yourself

### ◆ Static Analysis Performance :

- ◆ Perform a torsional stiffness analysis by applying Clamps, rigid virtual Part and a moment of 500 N.m around X axis on the rigid virtual Part. Enter a thickness of 2 mm for shell elements.
- ◆ Compute and display the displacements.





## Fundamental steps

- **Creating a mesh according to specifications**
  - ◆ Create a new surface mesh object.
  - ◆ Remove holes and gaps.
  - ◆ Define edge constraints and node distributions.
  - ◆ Create the mesh.
  
- **Modify and update an existing mesh**
  - ◆ Modify the geometry.
  - ◆ Update the geometry and the mesh.
  - ◆ Re-mesh a domain.
  
- **Analyze Mesh Quality**
  - ◆ Analyse the mesh quality.
  - ◆ Edit an analysis report.
  - ◆ Modify the mesh to increase quality.
  
- **Static Analysis Performance**
  - ◆ Perform a torsional stiffness analysis.
  - ◆ Compute and display the displacements.

# Additional Exercise

## 11. Meshing of a Fuselage



60 min.

In this exercise you will learn to mesh a cross-member with FEM Surface tools.

You will use :

Mesh definition

- ◆ Mesh specifications
- ◆ Mesh modifications
- ◆ Mesh quality

## Fundamental steps

- Create first fuselage mesh using minimal method
  - ◆ Define global parameters
  - ◆ Project external curves
  
- Extract 1D Coating Elements
  - ◆ Choose extraction type
  
- Change elements visualisation
  - ◆ Color Meshparts
  - ◆ Shrink elements
  
- Create second fuselage mesh using mapped free method
  - ◆ Define global parameters.
  - ◆ Capture neighbourhood mesh .
  - ◆ Propagate nodes distribution.
  
- Extract 1D Coating Elements

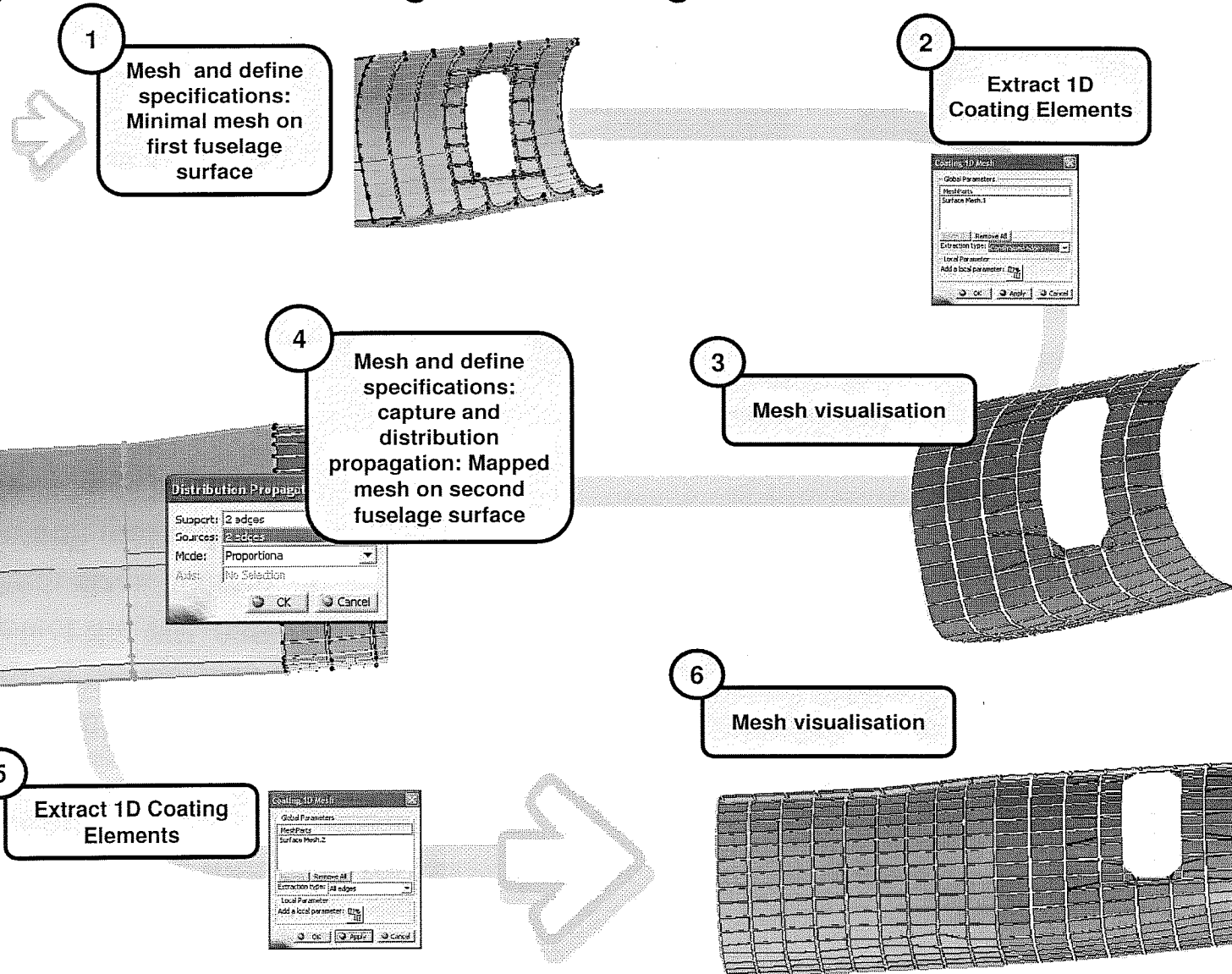
## Design Intent : Meshing of a Fuselage

- Surface mesh using minimal and mapped free method
- Distribution propagation
- Extract coating elements
- Color Meshparts
- Shrink elements

■ Load : Fuselage.CATPart

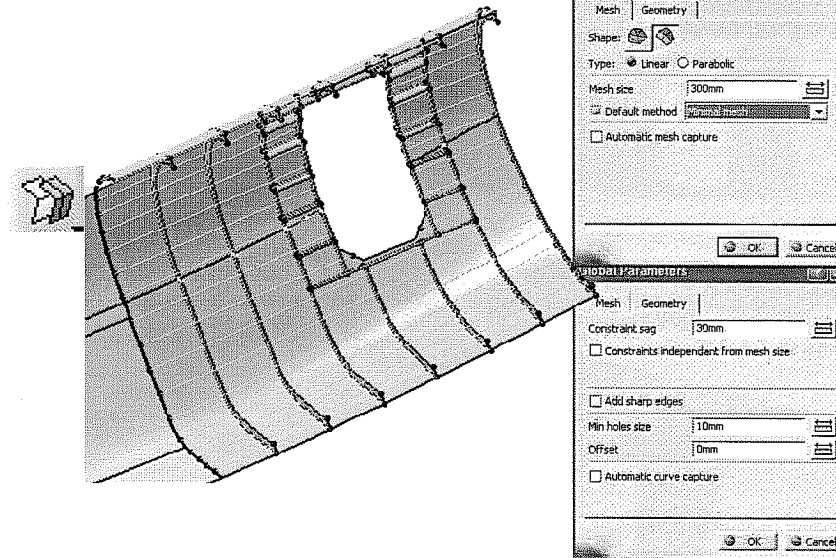
Student Notes:

## Design Process : Meshing of a Fuselage



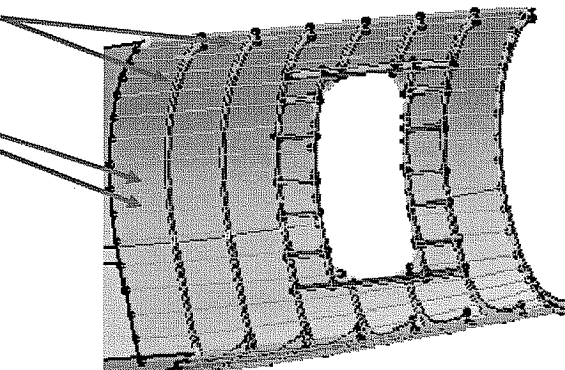
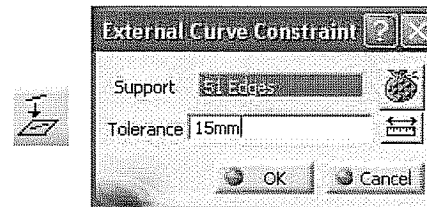
## Do It Yourself (1/7)

- Go to Advanced meshing tools workbench
- Mesh Fuselage surface
  - Select the Surface Mesher icon
  - Select the WP\_03S Fuselage surface
  - Mesh size= 300mm
  - Default method: Minimal Mesh
  - Mesh sag=30mm



## Constrain external curves

- Click on Project external curves
- Select all grey lines and light blue lines (total=51 edges)
- Tolerance = 15mm

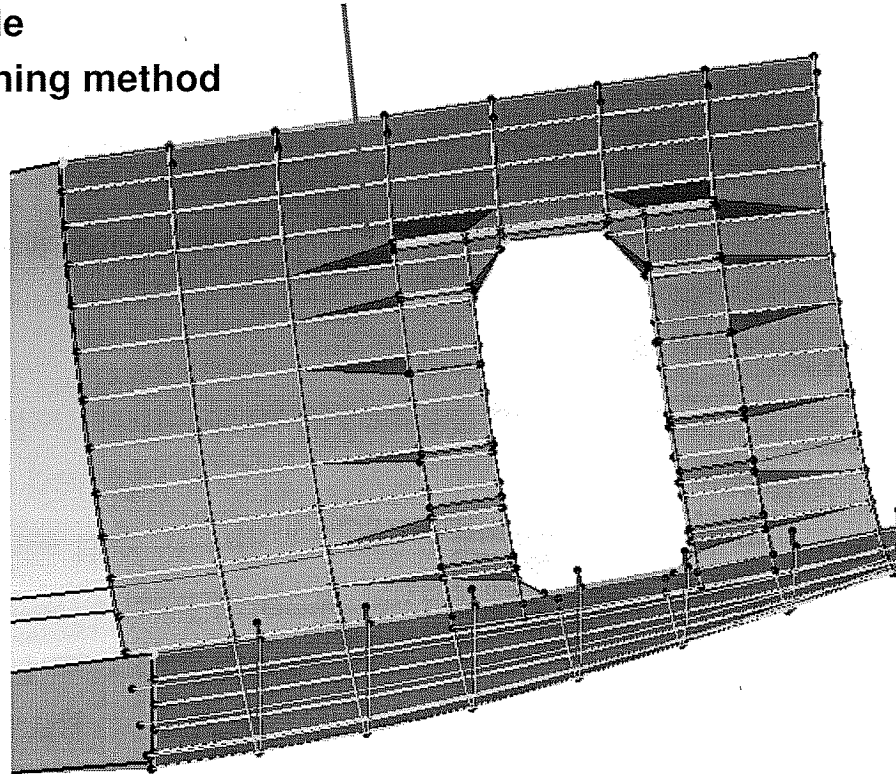


## Do It Yourself (2/7)

### Mesh the part

- One element is generated inside each rectangular domain (limited by constrained projected curves).
- If a domain contains one intermediate points, mesher generates one quadrangle and a triangle
- This is the minimal meshing method

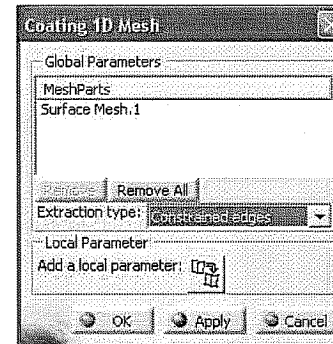
### Exit





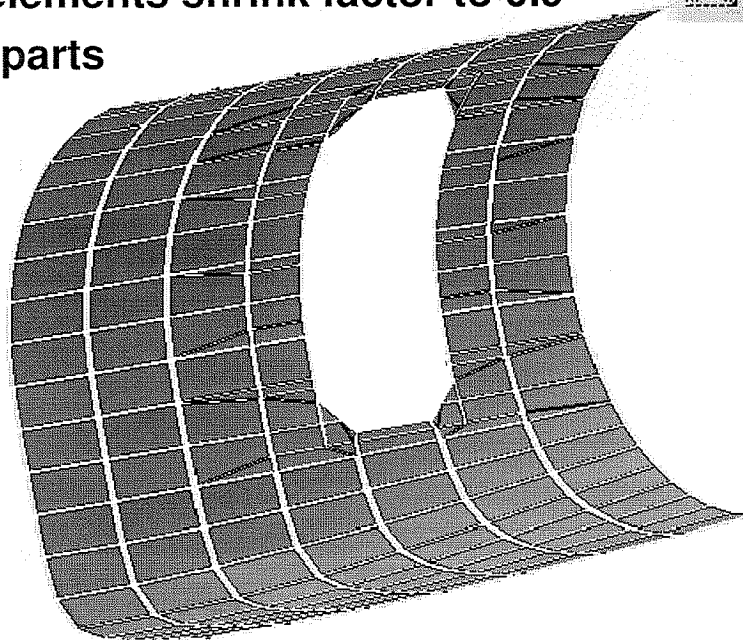
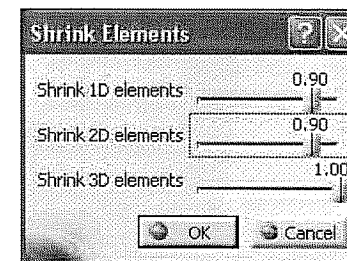
## Do It Yourself (3/7)

- Extract 1D coating elements
  - Select Surface Mesh.1 as meshpart
  - Extraction type is constrained edges
  - Click Apply to generate the mesh
  - Click OK to close the panel



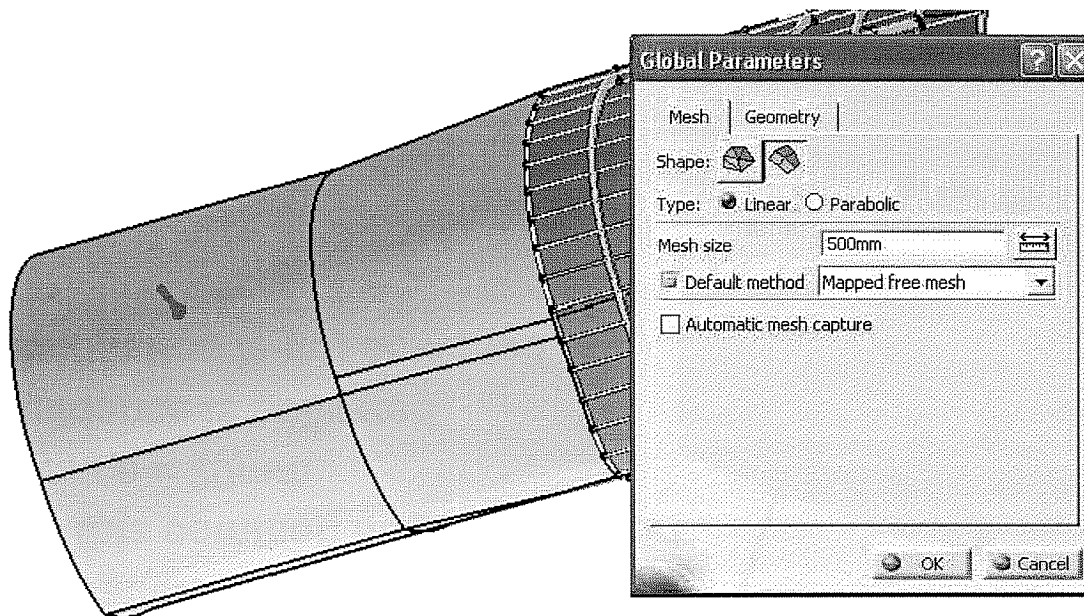
You have extracted 1D elements from 2D mesh on external curves

- Modify 2D elements shrink factor to 0.9
- Color Meshparts



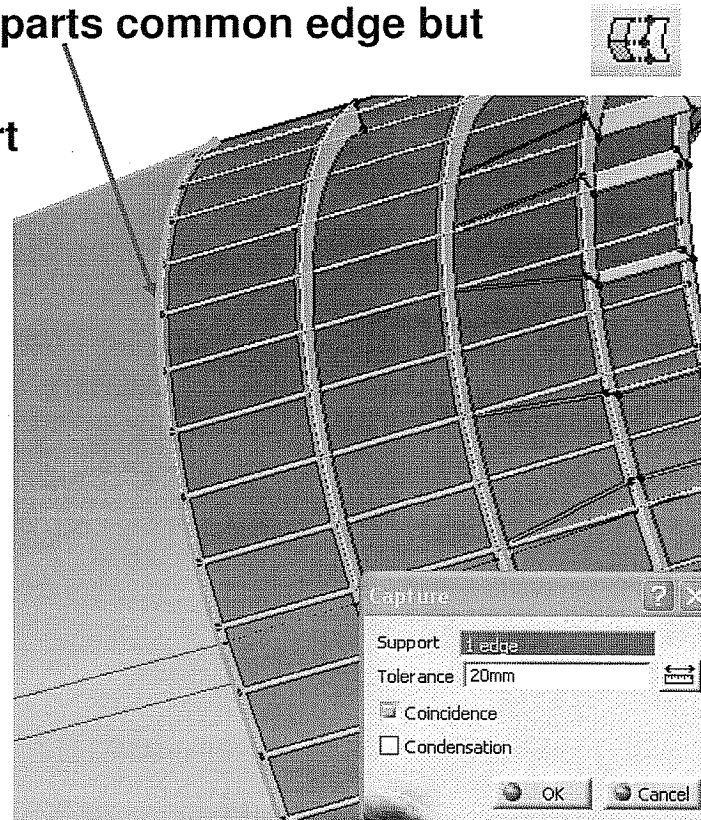
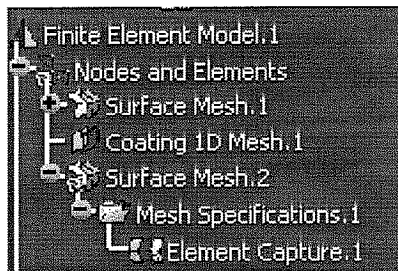
## Do It Yourself (4/7)

- Create a surface mesh on yellow surface, Join.1
  - Choose Mapped free as meshing method
  - Size=500 mm
  - Sag=30mm



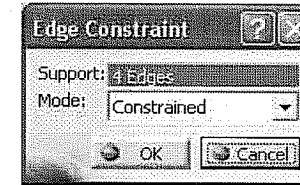
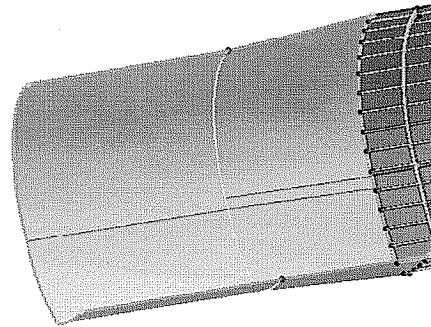
## Do It Yourself (5/7)

- Define coincident nodes on the 2 mesh parts common edge but no condensation
  - Select this common edge as support
  - Tolerance = 20mm
  - Coincidence is activated
  - Condensation is not activated



## Do It Yourself (6/7)

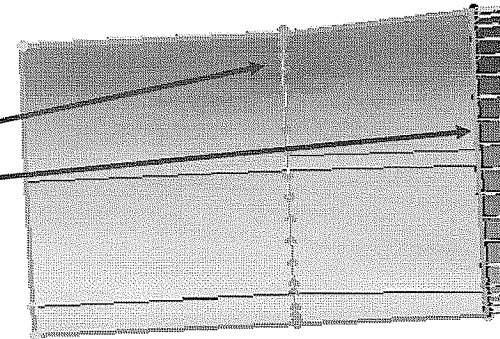
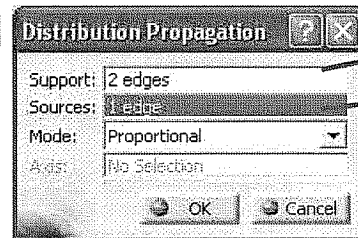
- Constrain intermediate edge



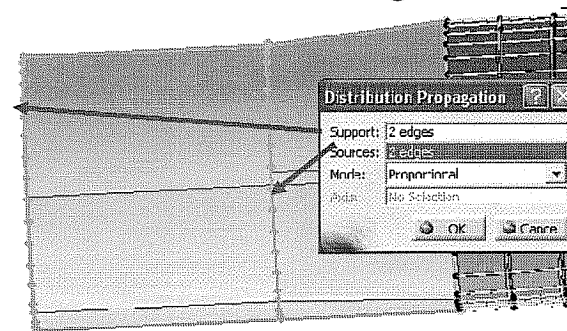
- Propagate nodes distribution to intermediate edge then to opposite edge



- Select this intermediate edge as Support
- Select the common boundary as Sources
- Mode is proportional

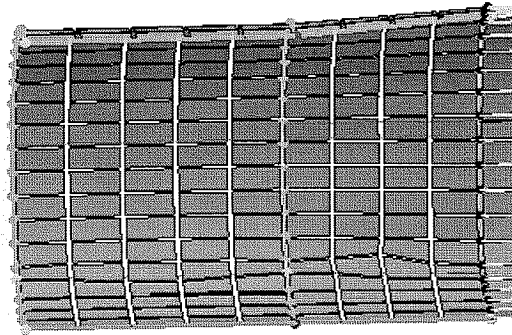


- Iterate to propagate intermediate edge nodes distribution to opposite end edge



## Do It Yourself (7/7)

- Mesh the Part

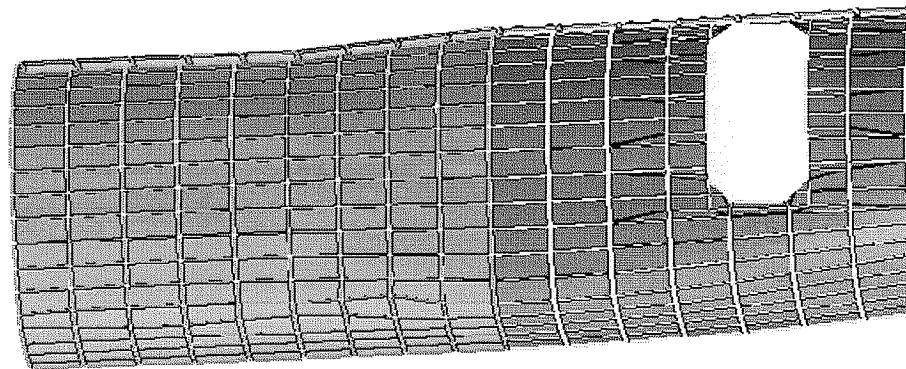
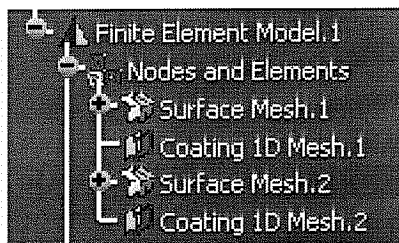


- Exit

- Extract 1D coating element from this 2D meshpart



- Select Surface Mesh.2 as meshpart
  - Extraction type is All edges
  - Click Apply to generate the mesh
  - Click OK to close the panel
- Apply a 2D element shrink of 0.9
- Color Meshparts



# Additional Exercise

## 12. Fuselage Connection



60 min.

In this exercise you will learn to connect a fuselage using Points to Points connection and point interface.

You will use :

Connection

- ◆ Points to Points Analysis Connection
- ◆ Point Analysis Interface
- ◆ Nodes to Nodes Connection Mesh
- ◆ Node Interface Mesh

## Fundamental steps

- Create a Points to Points connection between 2 components
  - ◆ Connect meshes
  - ◆ On a row of points
  
- Create a Point Interface
  - ◆ Interface of a mesh along a row of points.
  
- Mesh connections.
  - ◆ Generate a Nodes to Nodes mesh based on a points to points analysis connection
  - ◆ Generate a Node Interface mesh based on a points analysis interface
  
- Modify connection property
  - ◆ Edit Nodes to Nodes property
  - ◆ Inactive some degrees of freedom



## Design Intent : Connecting of a Fuselage

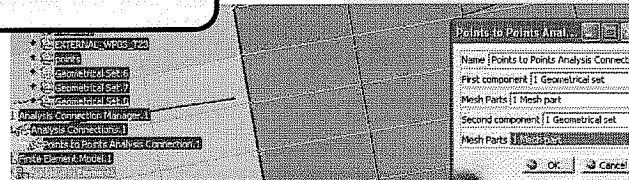
- Points to Points connection creation based on meshes and geometry
- Point analysis Interface based on a mesh and geometry
- Mesh connections as nodes to nodes mesh and node interface mesh
- Modify connection property

⚙ Load : Fuselage connection\_START.CATAnalysis

# Design Process : Connecting of a Fuselage

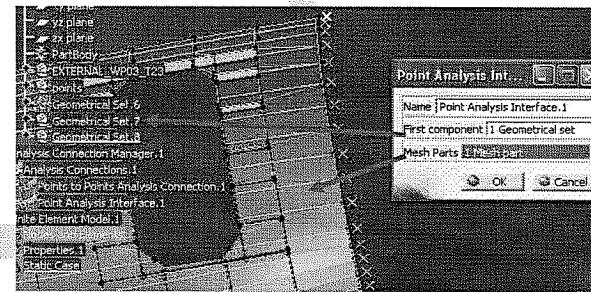
1

Points to Points  
Connection



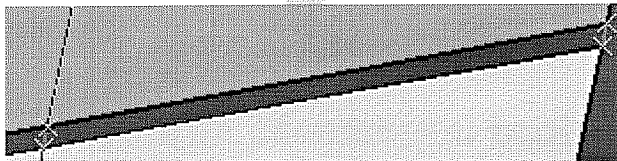
2

Point Interface



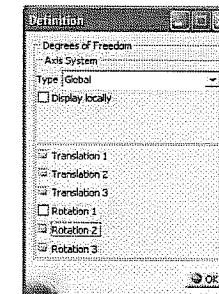
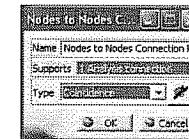
3

Mesh connections



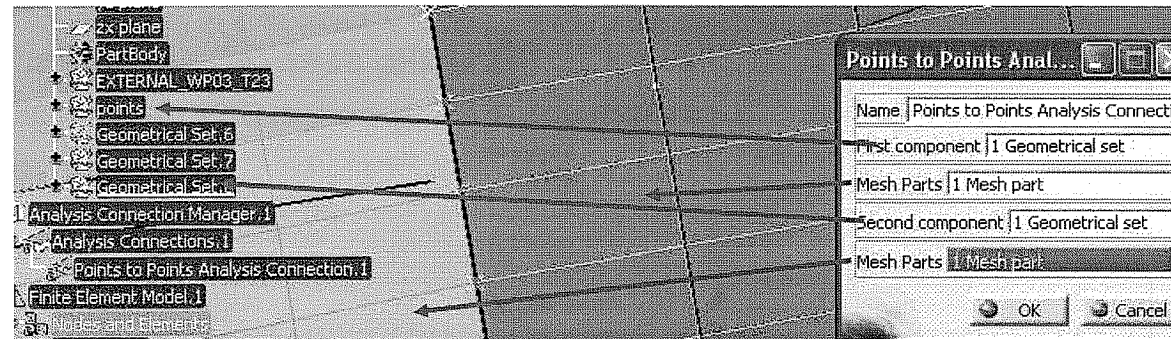
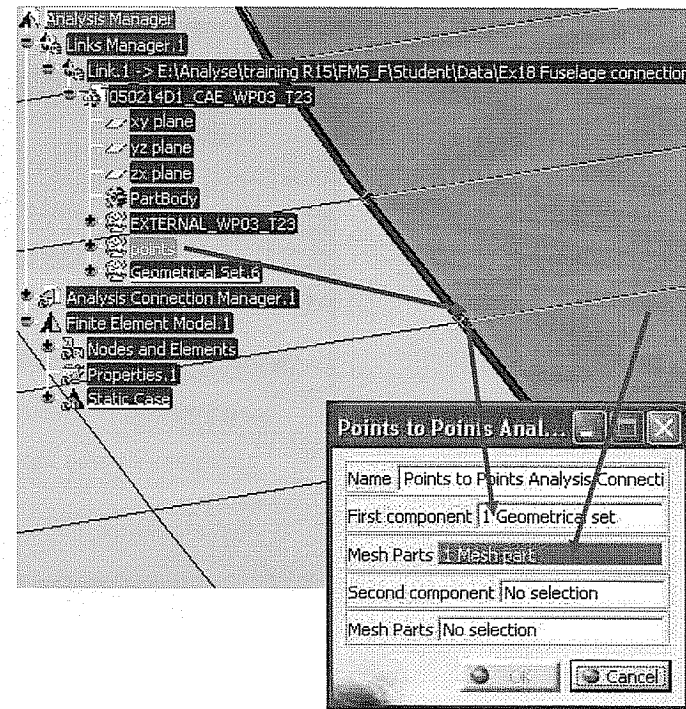
4

Edit connection  
property



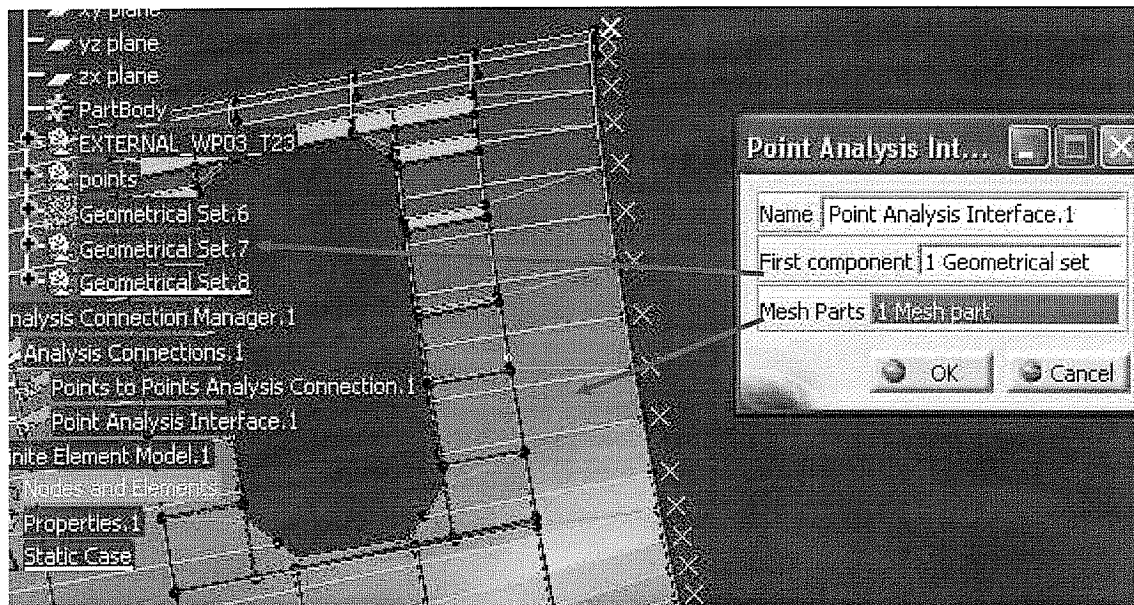
## Do It Yourself (1/4)

- Surface meshes are already created
- Select Points to Points Analysis connection icon
  - As first component select one row of points (click on Points geometrical set in the tree)
  - Select corresponding meshpart (Surface Mesh.1)
  - As second component select all points contained in geometrical set 8
  - Select corresponding meshpart (Surface Mesh.2)
  - Click OK to close the panel



## Do It Yourself (2/4)

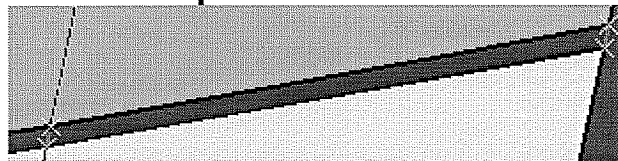
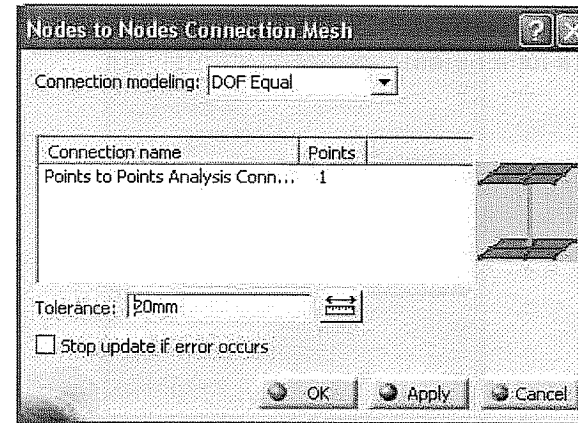
- Select Points Analysis Interface icon 
- As component select all points contained in geometrical set 7
- Select corresponding meshpart (Surface Mesh.1)
- Click OK to close the panel



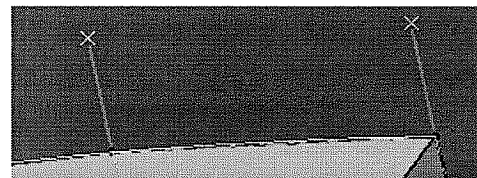
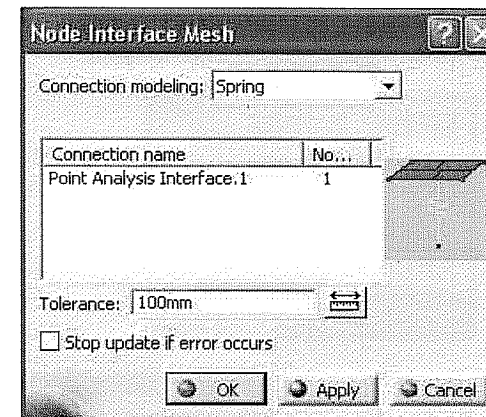
## Do It Yourself (3/4)

### Mesh connections

- Select Nodes to Nodes Connection Mesh icon
  - Select points to points analysis connection
  - Connection modeling: DOF equals
  - Tolerance= 20 mm
  - Select Apply to generate the mesh
  - Click OK to close the panel

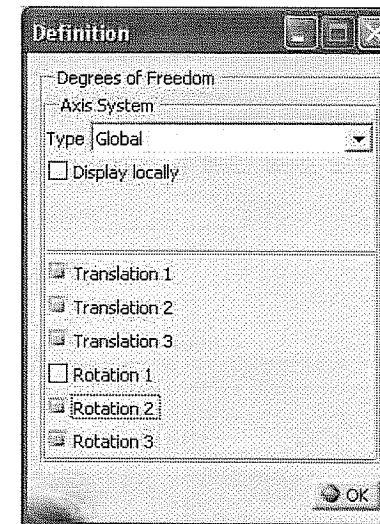
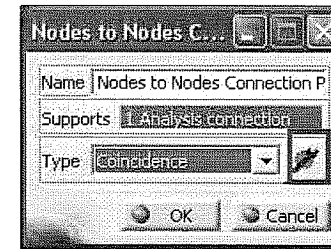
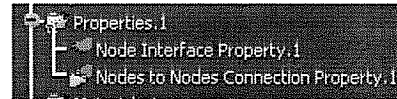


- Select Node Interface Mesh icon
  - Select points analysis Interface
  - Connection modeling: Spring
  - Tolerance= 20 mm
  - Select Apply to generate the mesh
  - Click OK to close the panel



## Do It Yourself (4/4)

- Switch to Generative Structural analysis wb
- Edit nodes to nodes connection Property.1
- Click on Component Edition
- Inactivate Rotation.1



# Additional Exercise

## 13. Pan Frt Floor Mesh

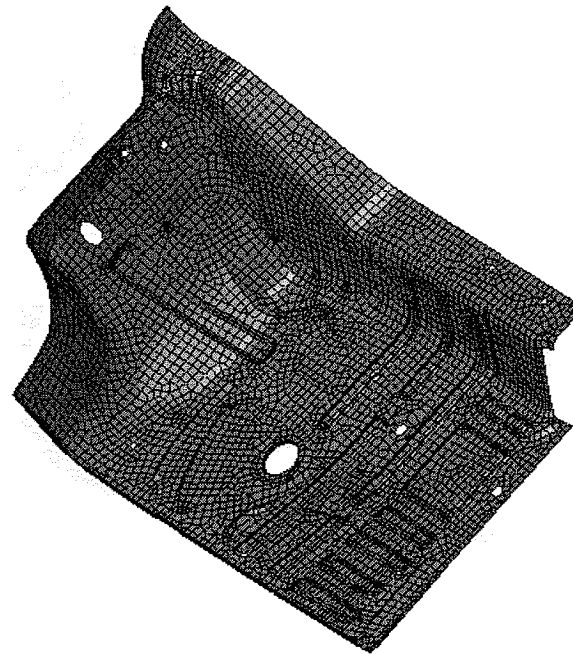


15 min.

In this exercise you will use advanced meshing tools to generate a good quality surfacic mesh of the floor part.

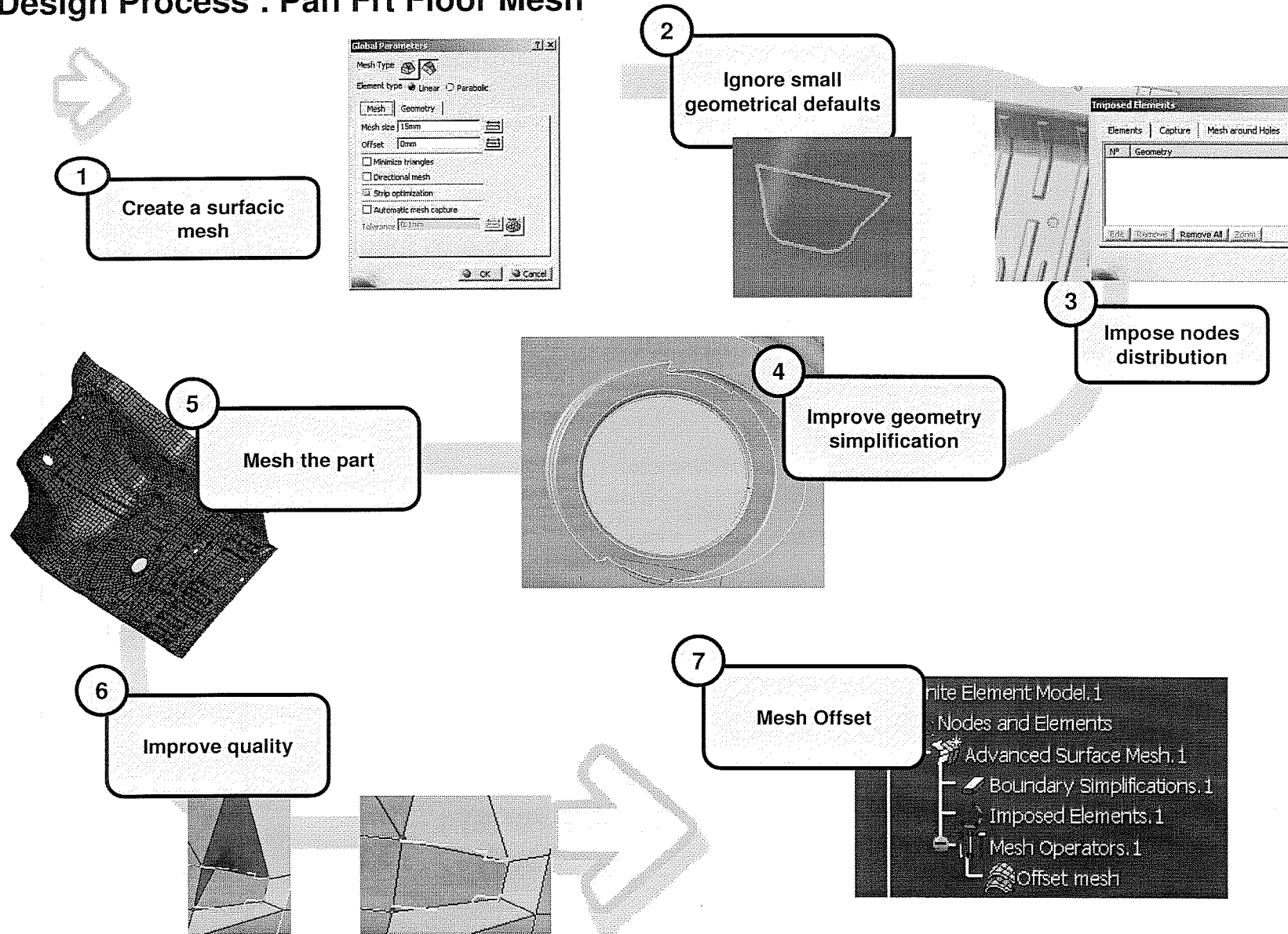
You will use :

- ◆ Advanced surfacic mesh
- ◆ Remove geometrical defaults
- ◆ Mesh around holes
- ◆ Non uniform Nodes distribution
- ◆ Analyse and improve Element Quality





## Design Process : Pan Frt Floor Mesh



## Do It Yourself (1/6)

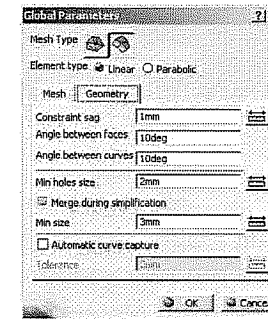
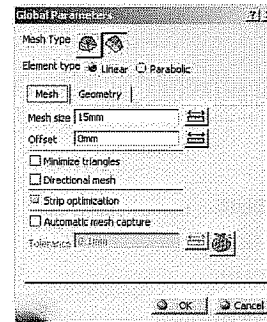
- Enter in *Advanced Meshing Tools* workbench



- Create a 2D mesh on the surface

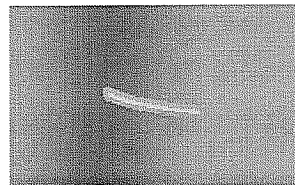
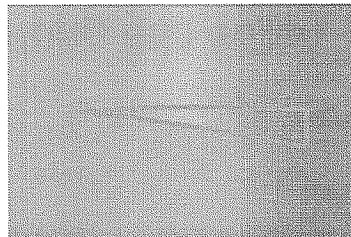
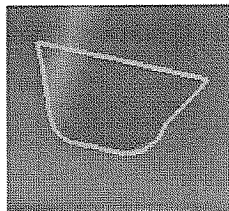
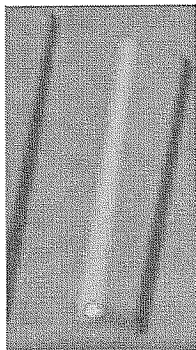
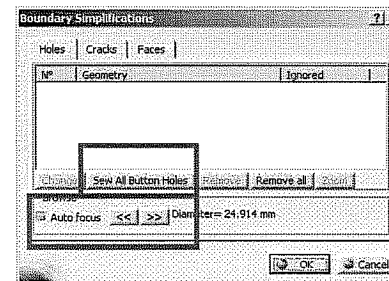


- Size=15 mm
- Sag= 1mm
- Merge During simplification, min size = 2mm



- Remove geometrical defaults :

- Click *Boundary Simplification* icon
- In Holes Tab click on *Sew all Button Holes*
- Using *Auto Focus*, remove the 4 last geometrical defaults



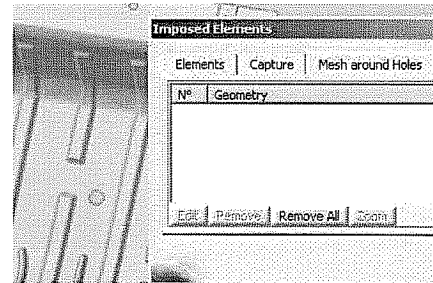
- Load: Pan Frt Floor.CATPart

Student Notes:

## Do It Yourself (2/6)

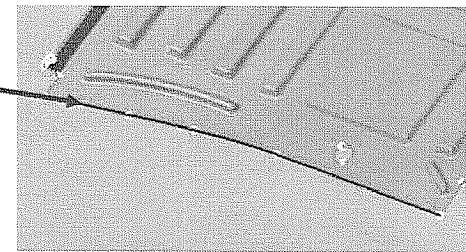
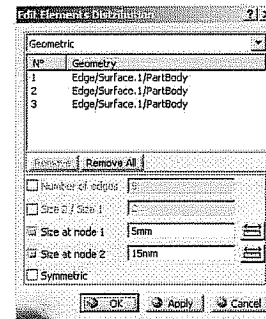
### Imposed Element around this hole:

- 6 elements and 2 rows of elements



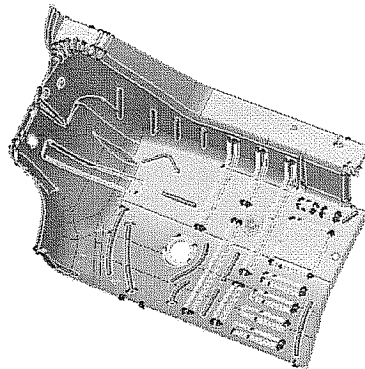
### Define a nodes distribution along this edge:

- Impose Elements
- Geometric distribution
- Size at node1= 5mm
- Size at node 2 = 15mm



Nota: to propagate the selection use Shift key: Shift key pressed, highlight first edge, check that the arrow has the right direction, select first edge, and click on last edge

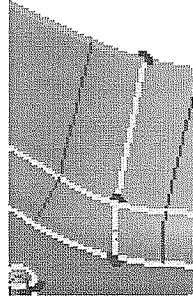
### Check Geometry simplification



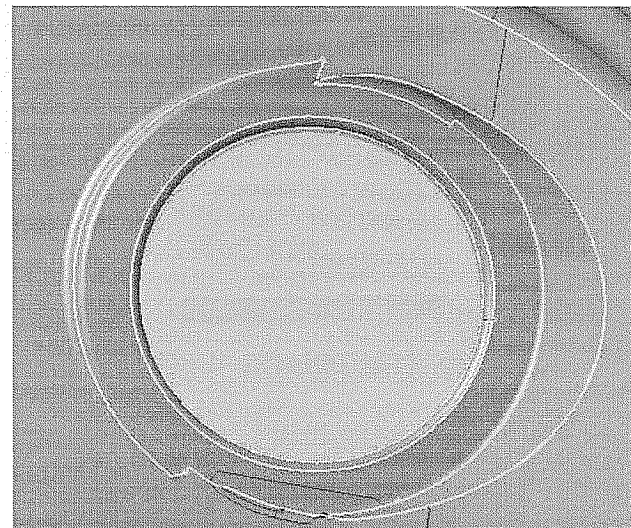
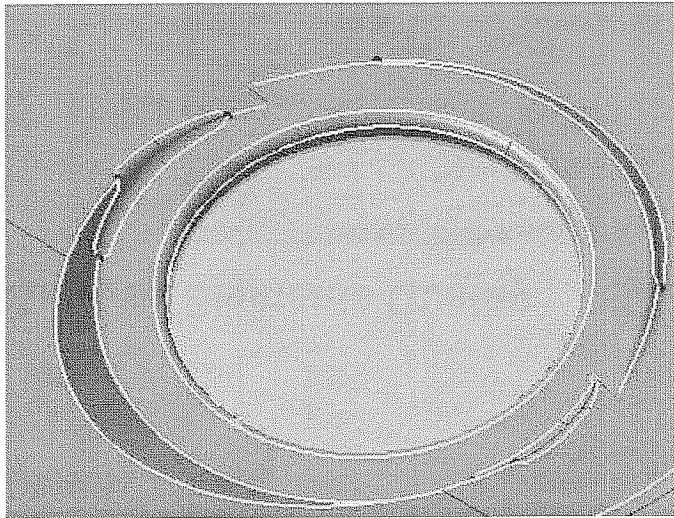
## Do It Yourself (3/6)

### ■ Add/Remove Constraint:

- By splitting



- Modifying simplification



## Do It Yourself (4/6)

### Mesh:



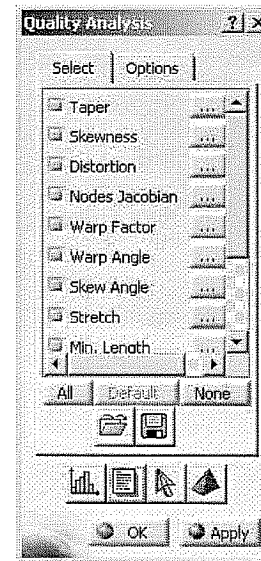
### Analyse Elements Quality



Quality Report

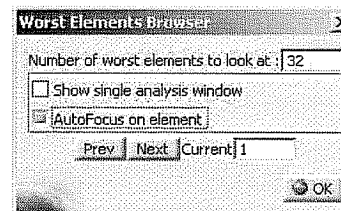
| Criterion      | Good             | Poor          | Bad          | Worst   | Average |
|----------------|------------------|---------------|--------------|---------|---------|
| Taper          | 6115 ( 99.11% )  | 50 ( 0.81% )  | 5 ( 0.08% )  | 0.976   | 0.130   |
| Skewness       | 300 ( 94.34% )   | 18 ( 5.66% )  | 0 ( 0.00% )  | 0.897   | 0.257   |
| Distortion     | 6242 ( 96.21% )  | 218 ( 3.36% ) | 28 ( 0.43% ) | 241.543 | 14.641  |
| Nodes Jacobian | 6160 ( 99.84% )  | 10 ( 0.16% )  | 0 ( 0.00% )  | 0.012   | 0.795   |
| Warp Factor    | 6098 ( 98.83% )  | 69 ( 1.12% )  | 3 ( 0.05% )  | 171.951 | 0.440   |
| Warp Angle     | 6157 ( 99.79% )  | 12 ( 0.19% )  | 1 ( 0.02% )  | 139.566 | 1.264   |
| Skew Angle     | 6130 ( 99.35% )  | 40 ( 0.65% )  | 0 ( 0.00% )  | 42.074  | 83.638  |
| Stretch        | 312 ( 98.11% )   | 6 ( 1.89% )   | 0 ( 0.00% )  | 0.132   | 0.716   |
| Min. Length    | 6488 ( 100.00% ) | 0 ( 0.00% )   | 0 ( 0.00% )  | 0.130   | 9.952   |
| Max. Length    | 6488 ( 100.00% ) | 0 ( 0.00% )   | 0 ( 0.00% )  | 39.997  | 14.763  |
| Gauss Jacobian | 6170 ( 100.00% ) | 0 ( 0.00% )   | 0 ( 0.00% )  | 0.279   | 0.871   |
| Shape Factor   | 315 ( 99.06% )   | 3 ( 0.94% )   | 0 ( 0.00% )  | 0.139   | 0.817   |
| Length Ratio   | 6431 ( 99.12% )  | 54 ( 0.83% )  | 3 ( 0.05% )  | 12.278  | 1.503   |
| -- Global --   | 6168 ( 95.07% )  | 288 ( 4.44% ) | 32 ( 0.49% ) |         |         |

OK



### Improve Quality

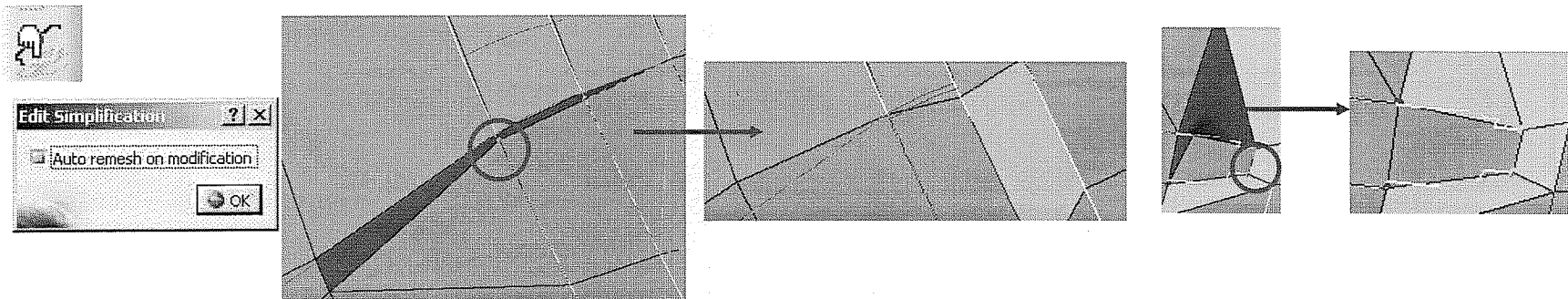
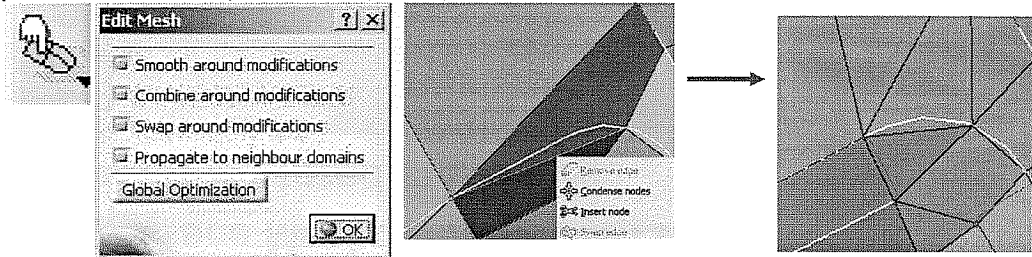
- Check all bad element



## Do It Yourself (5/6)

### ● Improve Quality

- Improve them by editing simplification (auto remesh is activated) or by editing mesh

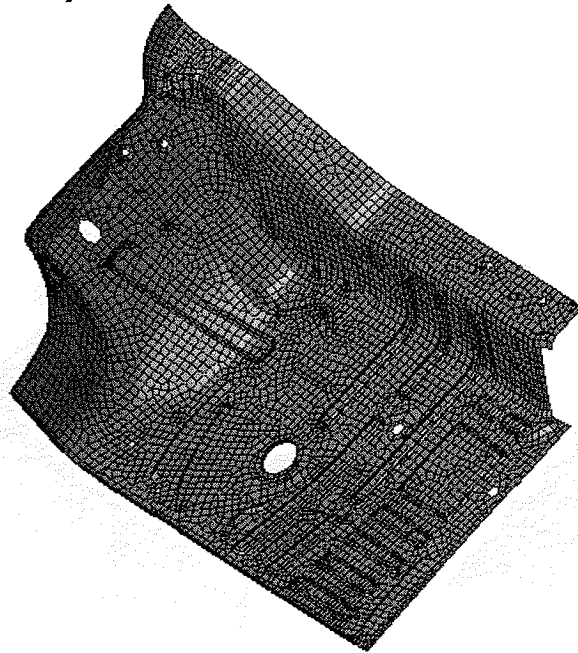


Quality Report

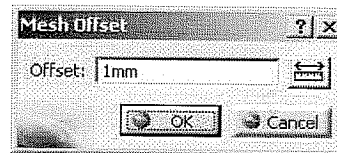
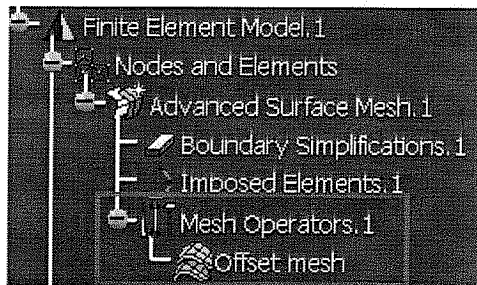
| Quality        | Connectivities   |               |             |        |         |
|----------------|------------------|---------------|-------------|--------|---------|
| Criterion      | Good             | Poor          | Bad         | Worst  | Average |
| Taper          | 6099 ( 99.53% )  | 29 ( 0.47% )  | 0 ( 0.00% ) | 0.671  | 0.128   |
| Skewness       | 343 ( 97.44% )   | 9 ( 2.56% )   | 0 ( 0.00% ) | 0.871  | 0.236   |
| Distortion     | 6276 ( 96.85% )  | 204 ( 3.15% ) | 0 ( 0.00% ) | 44.343 | 14.554  |
| Nodes Jacobian | 6127 ( 99.98% )  | 1 ( 0.02% )   | 0 ( 0.00% ) | 0.197  | 0.795   |
| Warp Factor    | 6072 ( 99.09% )  | 56 ( 0.91% )  | 0 ( 0.00% ) | 9.977  | 0.396   |
| Warp Angle     | 6118 ( 99.84% )  | 10 ( 0.16% )  | 0 ( 0.00% ) | 40.393 | 1.174   |
| Skew Angle     | 6094 ( 99.45% )  | 34 ( 0.55% )  | 0 ( 0.00% ) | 49.211 | 83.646  |
| Stretch        | 352 ( 100.00% )  | 0 ( 0.00% )   | 0 ( 0.00% ) | 0.305  | 0.732   |
| Min. Length    | 6480 ( 100.00% ) | 0 ( 0.00% )   | 0 ( 0.00% ) | 1.254  | 9.967   |
| Max. Length    | 6480 ( 100.00% ) | 0 ( 0.00% )   | 0 ( 0.00% ) | 39.997 | 14.748  |
| Gauss Jacobian | 6128 ( 100.00% ) | 0 ( 0.00% )   | 0 ( 0.00% ) | 0.441  | 0.871   |
| Shape Factor   | 352 ( 100.00% )  | 0 ( 0.00% )   | 0 ( 0.00% ) | 0.393  | 0.835   |
| Length Ratio   | 6429 ( 99.21% )  | 51 ( 0.79% )  | 0 ( 0.00% ) | 7.464  | 1.488   |
| -- Global --   | 6208 ( 95.80% )  | 272 ( 4.20% ) | 0 ( 0.00% ) |        |         |

## Do It Yourself (6/6)

Exit



- Offset this Surface Mesh: distance = 1 mm  
Offset operator is stored in the tree





# Additional Exercise

## 14. Octree mesh on Block cylinder

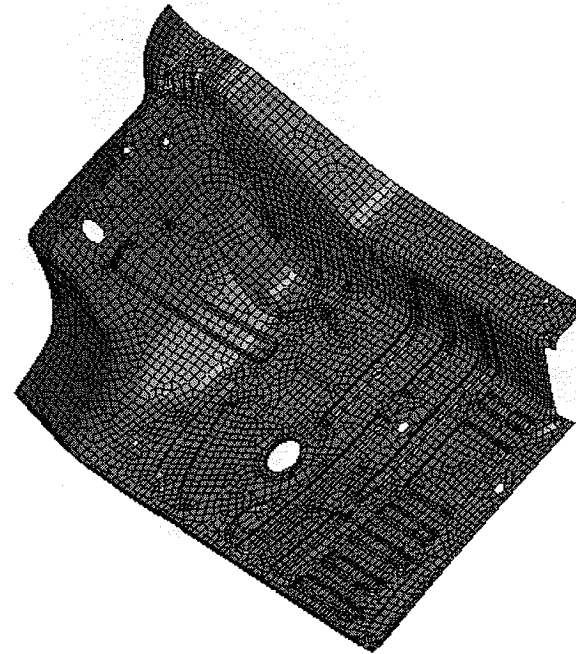


15 min.

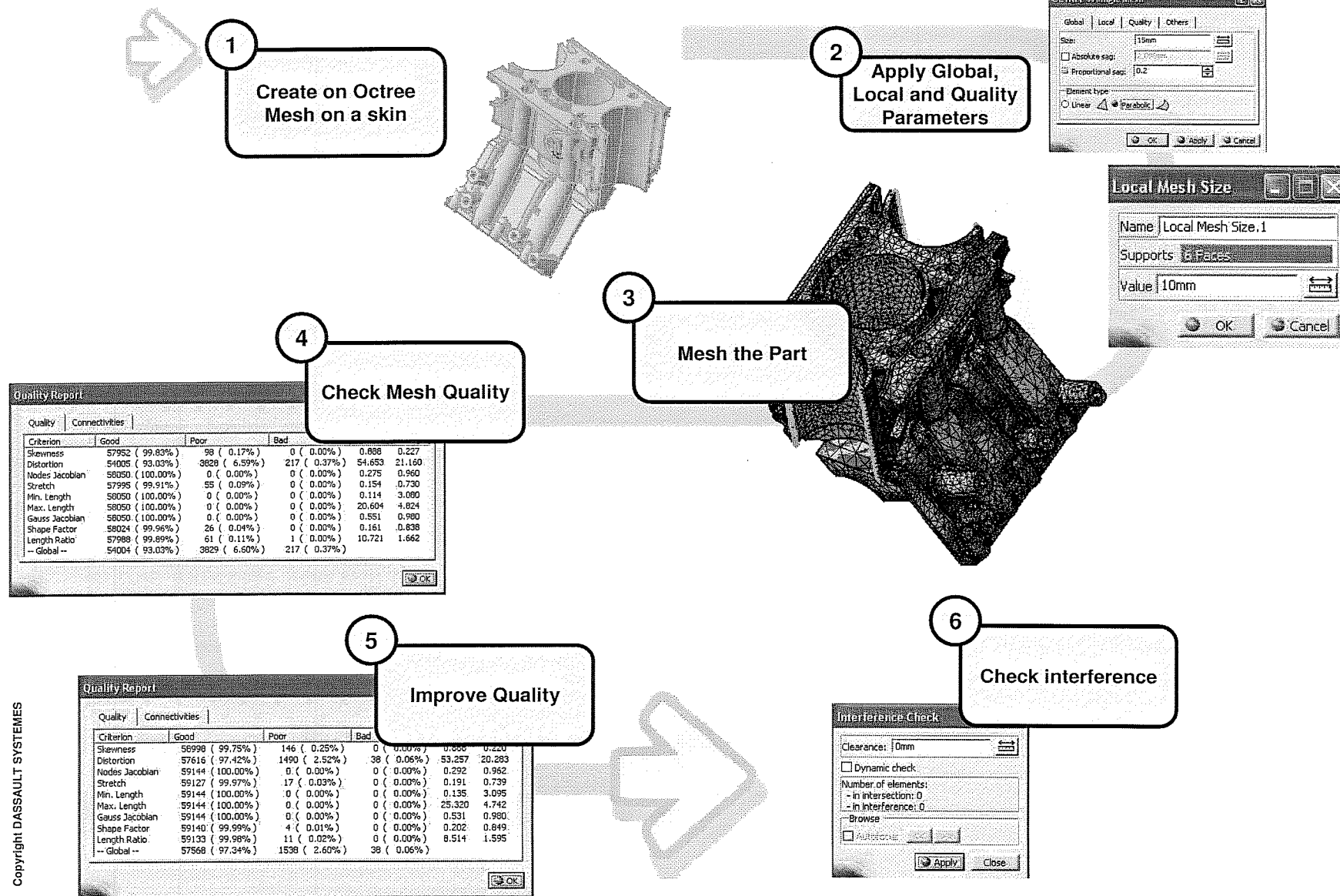
In this exercise you will use advanced meshing tools to generate a good quality octree mesh of a block cylinder

You will use :

- ◆ Octree mesher
- ◆ Global specification\
- ◆ Local specification
- ◆ Quality improvement
- ◆ Other specification



# Design Process : Octree mesh on Block cylinder



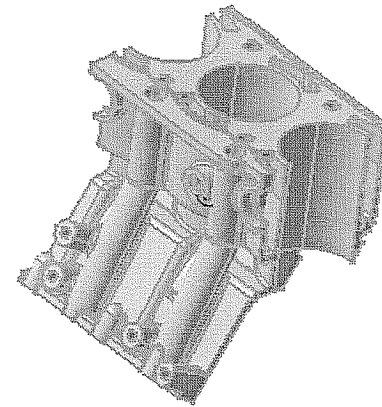
## Do It Yourself (1/7)

- Enter in *Advanced Meshing Tools* workbench 

- Create an Octree mesh on the skin surface of the part 

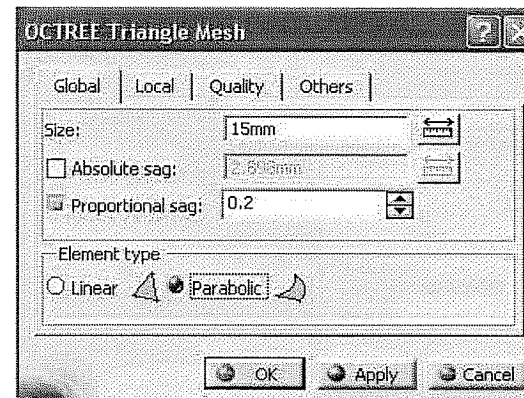
- Click on Octree Mesher icon
- Select the cylinder block

The skin is selected and will be meshed



- Enter Global parameters :

- Size = 15mm
- Absolute sag is deactivated
- Proportional sag is activated and equals 0.2
- Choose Parabolic as element Type



- Load Slice\_Cylinder\_Block\_Half.CATPart

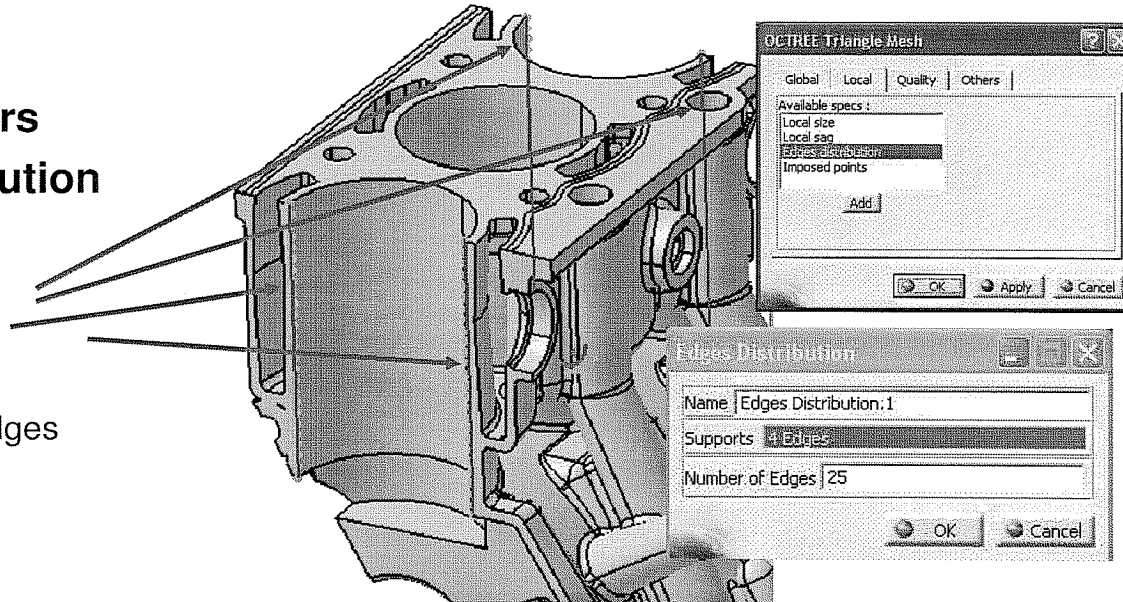
Student Notes:

## Do It Yourself (2/7)

### ■ Create Local Parameters

### ■ Apply elements distribution

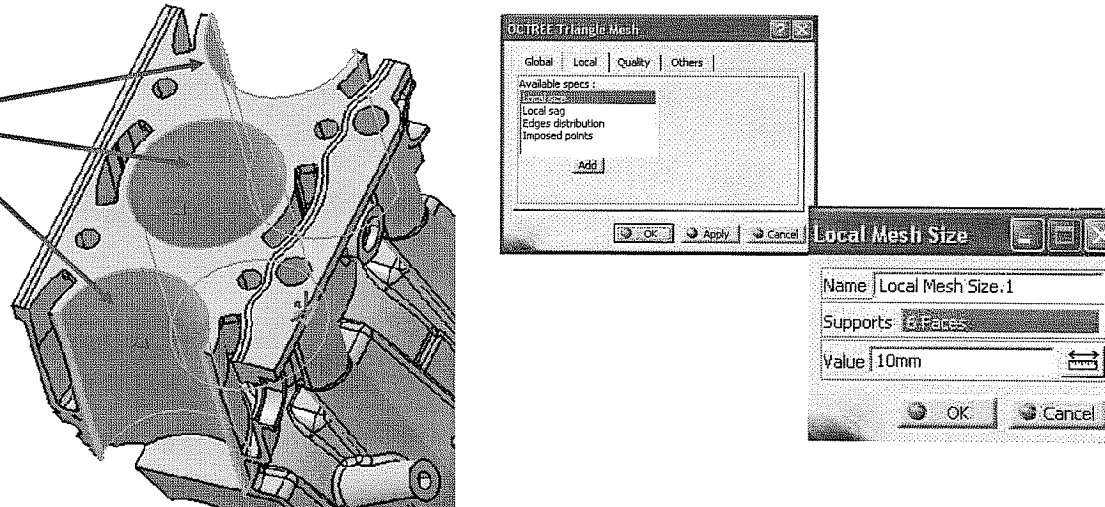
- Select Local Tab
- Select Edges distribution
- Click on Add
- Select these four edges
- Enter 25 as number of edges



### ■ Apply local size

Inside cylinder and the two half cylinder apply a 10mm elements size

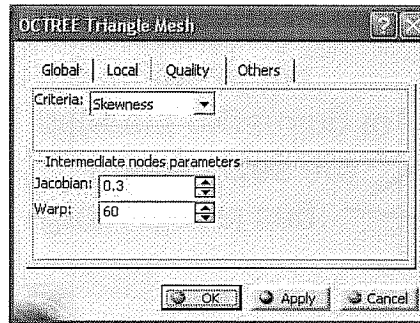
- ▶ Select Local size
- ▶ Click on Add
- ▶ Select the 6 faces
- ▶ Enter 10mm as size



## Do It Yourself (3/7)

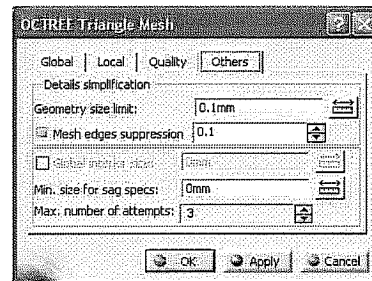
### ■ Create Quality parameter:

- Select Quality Tab
- Choose Skewness as criteria



### ■ Check Others parameter

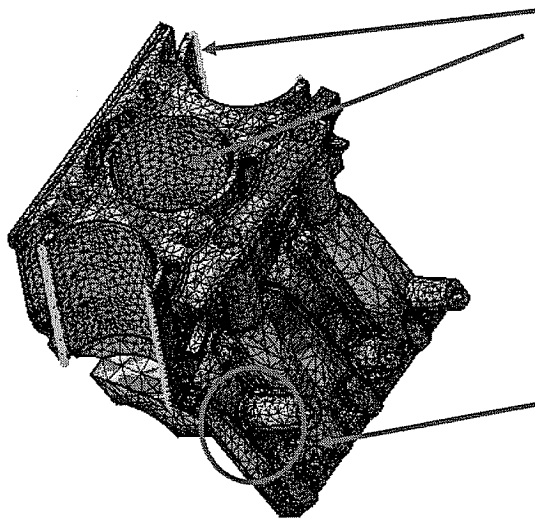
You should have these default values



## Do It Yourself (4/7)

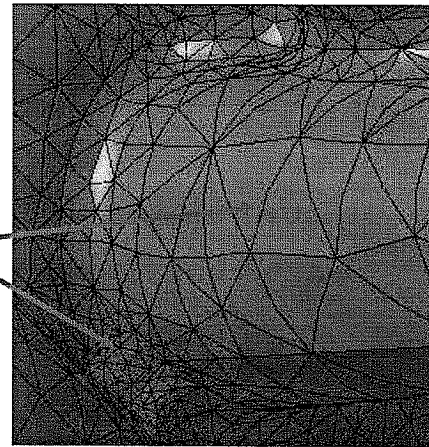
### Mesh the Part:

- Click on Apply to mesh the skin
- Click OK to close the panel



Check elements distribution And local size

Proportional sag:  
elements sizes are  
different regarding to  
the shape of the  
geometry



## Do It Yourself (5/7)

### ■ Analyse Elements Quality

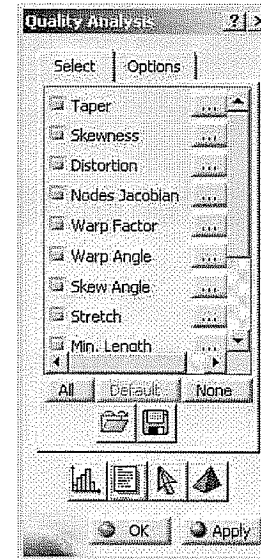
- Edit a Report

**Quality Report**

Quality | Connectivities

| Criterion      | Good              | Poor           | Bad           | Worst  | Average |
|----------------|-------------------|----------------|---------------|--------|---------|
| Skewness       | 57952 ( 99.83% )  | 98 ( 0.17% )   | 0 ( 0.00% )   | 0.888  | 0.227   |
| Distortion     | 54005 ( 93.03% )  | 3828 ( 6.59% ) | 217 ( 0.37% ) | 54.653 | 21.160  |
| Nodes Jacobian | 58050 ( 100.00% ) | 0 ( 0.00% )    | 0 ( 0.00% )   | 0.275  | 0.960   |
| Stretch        | 57995 ( 99.91% )  | 55 ( 0.09% )   | 0 ( 0.00% )   | 0.154  | 0.730   |
| Min. Length    | 58050 ( 100.00% ) | 0 ( 0.00% )    | 0 ( 0.00% )   | 0.114  | 3.080   |
| Max. Length    | 58050 ( 100.00% ) | 0 ( 0.00% )    | 0 ( 0.00% )   | 20.604 | 4.824   |
| Gauss Jacobian | 58050 ( 100.00% ) | 0 ( 0.00% )    | 0 ( 0.00% )   | 0.551  | 0.980   |
| Shape Factor   | 58024 ( 99.96% )  | 26 ( 0.04% )   | 0 ( 0.00% )   | 0.161  | 0.838   |
| Length Ratio   | 57988 ( 99.89% )  | 61 ( 0.11% )   | 1 ( 0.00% )   | 10.721 | 1.662   |
| -- Global --   | 54004 ( 93.03% )  | 3829 ( 6.60% ) | 217 ( 0.37% ) |        |         |

OK

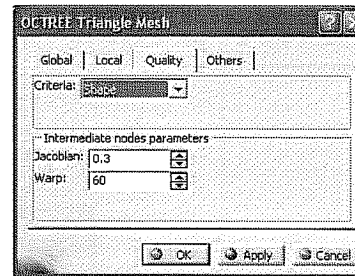




## Do It Yourself (6/7)

### Improve Quality

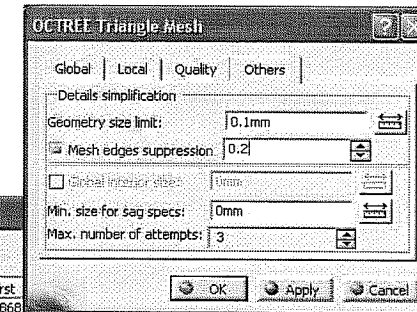
- Edit Octree Triangle mesh by double clicking on it in the tree
- In Quality Tab, change criteria to Shape
- Click on Apply to Mesh the Part
- Check Mesh Quality Report



Quality Report

| Criterion      | Good              | Poor           | Bad          | Worst  | Average |
|----------------|-------------------|----------------|--------------|--------|---------|
| Skewness       | 59058 ( 99.55% )  | 264 ( 0.44% )  | 4 ( 0.01% )  | 0.937  | 0.226   |
| Distortion     | 57119 ( 96.28% )  | 2126 ( 3.58% ) | 81 ( 0.14% ) | 52.943 | 20.550  |
| Nodes Jacobian | 59326 ( 100.00% ) | 0 ( 0.00% )    | 0 ( 0.00% )  | 0.222  | 0.962   |
| Stretch        | 59303 ( 99.96% )  | 23 ( 0.04% )   | 0 ( 0.00% )  | 0.176  | 0.735   |
| Min. Length    | 59326 ( 100.00% ) | 0 ( 0.00% )    | 0 ( 0.00% )  | 0.121  | 3.080   |
| Max. Length    | 59326 ( 100.00% ) | 0 ( 0.00% )    | 0 ( 0.00% )  | 21.421 | 4.744   |
| Gauss Jacobian | 59326 ( 100.00% ) | 0 ( 0.00% )    | 0 ( 0.00% )  | 0.511  | 0.960   |
| Shape Factor   | 59320 ( 99.99% )  | 6 ( 0.01% )    | 0 ( 0.00% )  | 0.205  | 0.845   |
| Length Ratio   | 59310 ( 99.97% )  | 16 ( 0.03% )   | 0 ( 0.00% )  | 7.527  | 1.611   |
| -- Global --   | 57038 ( 95.14% )  | 2206 ( 3.72% ) | 82 ( 0.14% ) |        |         |

- Edit Octree Triangle mesh by double clicking on it in the tree
- In Others Tab, change mesh edge suppression to 0.2
- Click on Apply to Mesh the Part
- Check Mesh Quality Report



Quality Report

| Criterion      | Good              | Poor           | Bad          | Worst  | Average |
|----------------|-------------------|----------------|--------------|--------|---------|
| Skewness       | 58998 ( 99.75% )  | 146 ( 0.25% )  | 0 ( 0.00% )  | 0.668  | 0.226   |
| Distortion     | 57616 ( 97.42% )  | 1490 ( 2.52% ) | 38 ( 0.06% ) | 53.257 | 20.283  |
| Nodes Jacobian | 59144 ( 100.00% ) | 0 ( 0.00% )    | 0 ( 0.00% )  | 0.292  | 0.962   |
| Stretch        | 59127 ( 99.97% )  | 17 ( 0.03% )   | 0 ( 0.00% )  | 0.191  | 0.739   |
| Min. Length    | 59144 ( 100.00% ) | 0 ( 0.00% )    | 0 ( 0.00% )  | 0.135  | 3.095   |
| Max. Length    | 59144 ( 100.00% ) | 0 ( 0.00% )    | 0 ( 0.00% )  | 25.320 | 4.742   |
| Gauss Jacobian | 59144 ( 100.00% ) | 0 ( 0.00% )    | 0 ( 0.00% )  | 0.531  | 0.960   |
| Shape Factor   | 59140 ( 99.99% )  | 4 ( 0.01% )    | 0 ( 0.00% )  | 0.202  | 0.849   |
| Length Ratio   | 59133 ( 99.98% )  | 11 ( 0.02% )   | 0 ( 0.00% )  | 8.514  | 1.595   |
| -- Global --   | 57568 ( 97.34% )  | 1538 ( 2.60% ) | 38 ( 0.06% ) |        |         |

## Do It Yourself (7/7)

### ■ Check Interference

- Clearance = 0mm
- Click on Apply to launch the check

No interference or intersection are detected, this skin mesh is ready for tetrafiller 3D mesher (available in FMD)

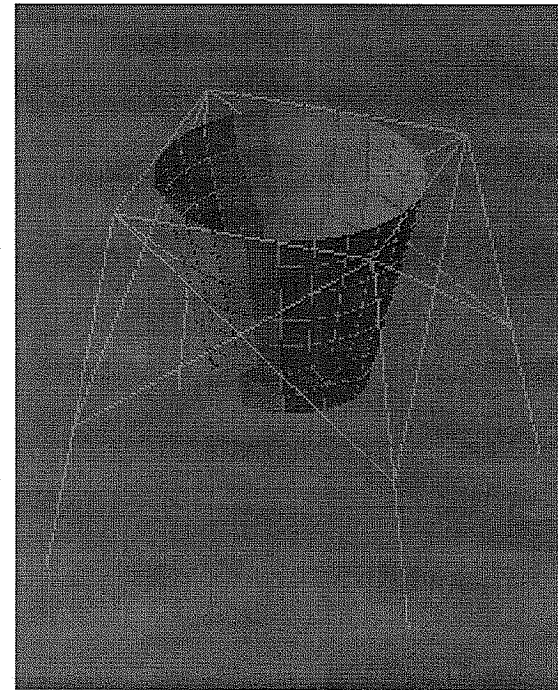


# Additional Exercise

## 15. Hybrid Mesh: 1D and 2D

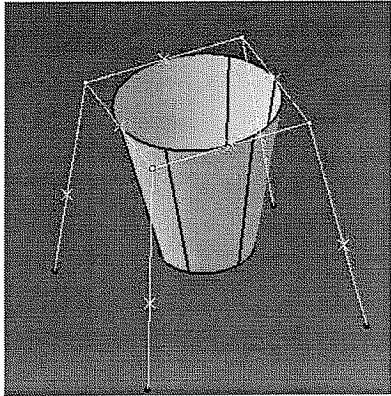
In this step you will :

- ◆ generate a 1D mesh
- ◆ Use capture nodes between 1D meshes and 1D-2D meshes
- ◆ Create and analyse a static case
- ◆ Modify/add and connect 1D meshes

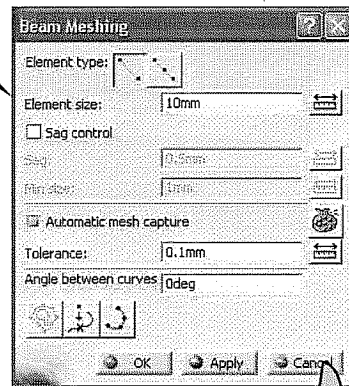


 Load: silo.CATPart

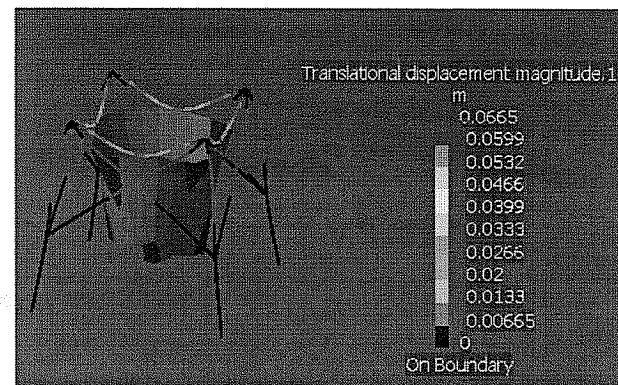
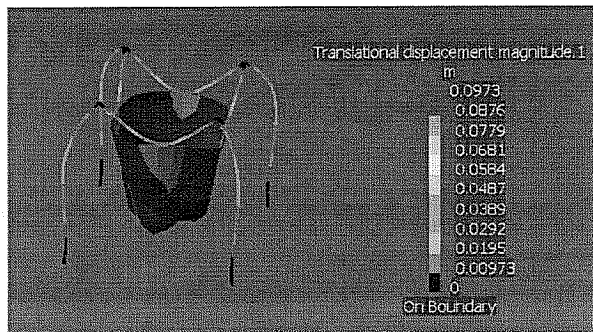
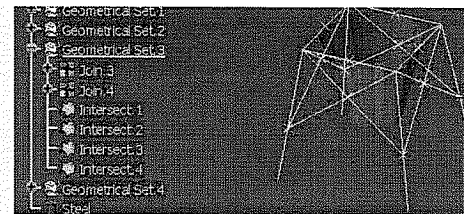
## Design Process : 1D Mesh and hybrid mesh



From an hybrid geometry, define 1D and 2D meshes, connections are coincident and condensed nodes



Add beams to minimize displacements and modify the whole structure behavior



## Do It Yourself (1/6)

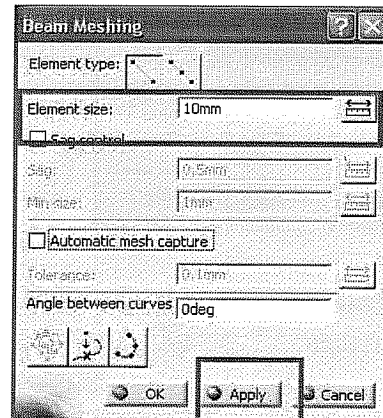
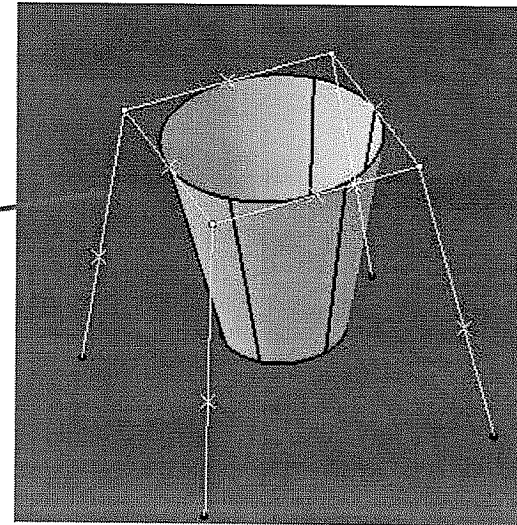
- Tools+Options, General +Parameters and Measures/ Units: Change Length unit to meter
- Go to Advanced Meshing Tools wb
- Generate the 1D Meshes

Select *Beam Mesher* icon



Select the support lines

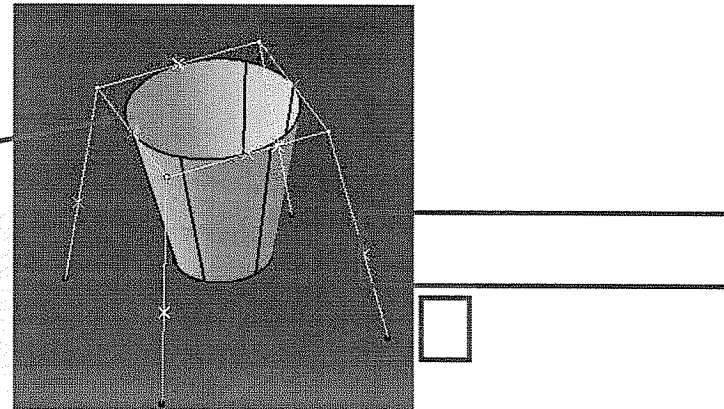
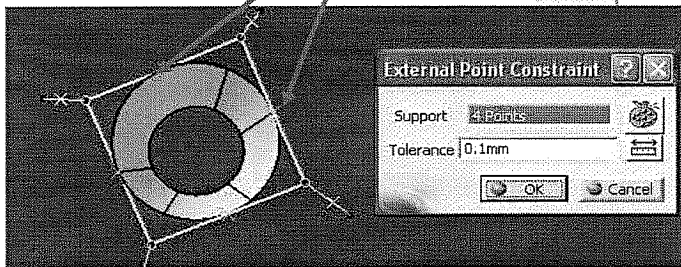
- 10m as size, no sag
- Click Apply to mesh and OK to close



Student Notes:

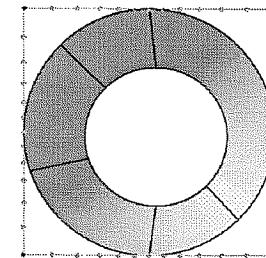
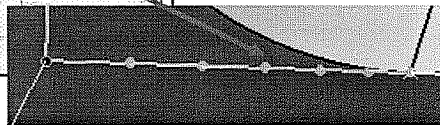
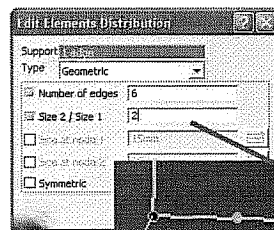
## Do It Yourself (2/6)

- Select *Beam Mesher* icon 
- Select the top structure
  - ◆ 10m as size
  - ◆ Activate automatic mesh capture, Tolerance = 0.1m
- Imposed points: each middle point\* 
  - ◆ Tolerance = 0.1 mm



- Apply a Nodes Distribution to increase nodes density around middle point 

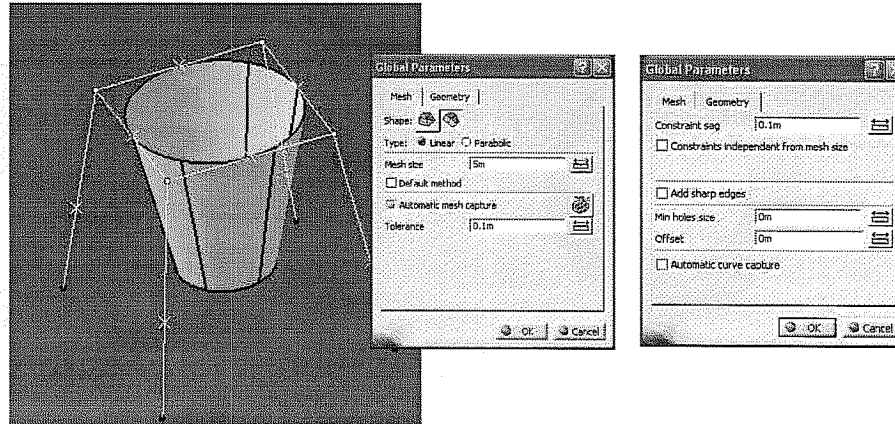
- ◆ Geometric distribution
- ◆ Nb of elements= 6
- ◆ Size 2 / Size 1 = 2 for left edges
- ◆ Size 2/ Size 1 = 0.5 for right edges



- Click Apply to mesh

## Do It Yourself (3/6)

- **Generate a Surface Mesh**
- **Select the surface**
  - ◆ 5m as size and 1m as sag
  - ◆ Automatic mesh capture activated  
Tolerance = 0.1m

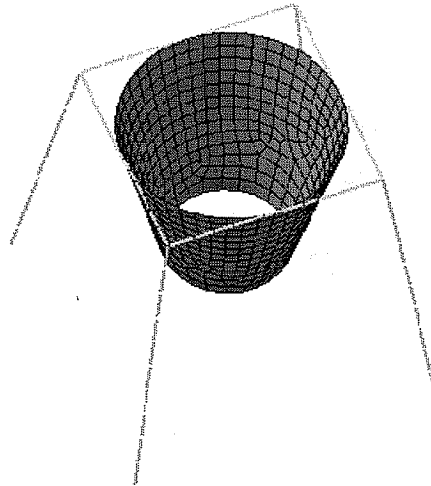
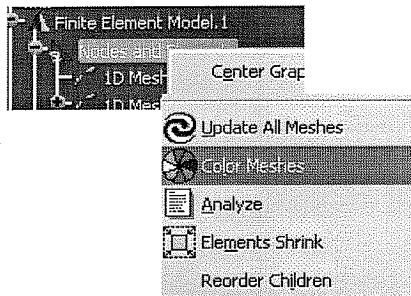


Nota: Nodes from 1D meshes are automatically detected and captured

- **Mesh and Exit**



- **Color All Meshparts**





Student Notes:

## Do It Yourself (4/6)

### Go To Generative Structural analysis

### Apply properties

### Beam properties

- ◆ Click on *1D Property* icon
- ◆ Select 1D Mesh.1 (or geometry)
- ◆ Choose Cylindrical beam as type
- ◆ Click on Component Edition
- ◆ Enter 0.5m as radius
- ◆ Idem 1D Mesh.2

### Surfacic properties

- ◆ Thickness = 0.05m

### Defining the case

### Clamp the 4 bottom points

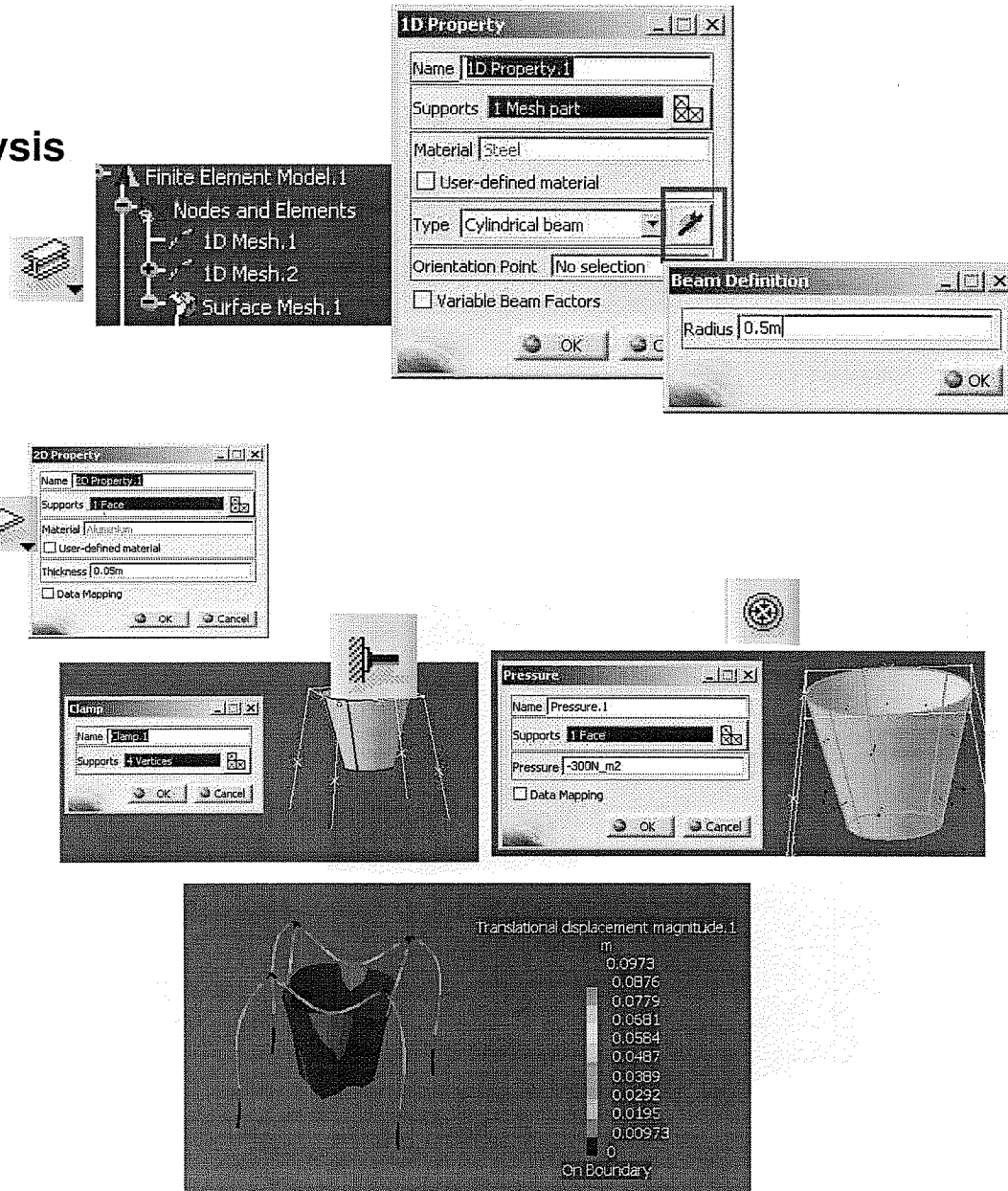
### Pressure inside Sillo

- ◆ Value = -300N/m2

### Compute

### Visualise Displacements image

- ◆ Average Iso visualisation



## Do It Yourself (5/6)

To decrease Displacements and the deformation, we add beams to the structure: add 1D meshes

- In sillo.catpart, show Geometrical set.3
- In analysis document, go to Advanced Meshing Tools workbench
- Edit 1D mesh.1, and add imposed points:4 intermediate points
- Click Apply to update this mesh

- Mesh one group of new beams (select join.3 Or one beam)

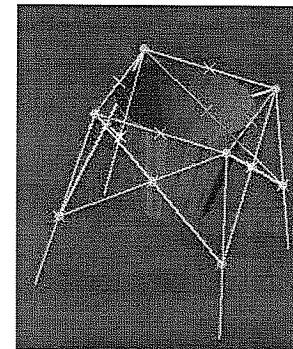
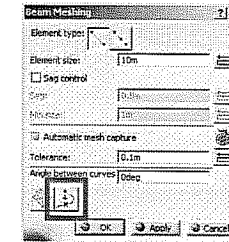
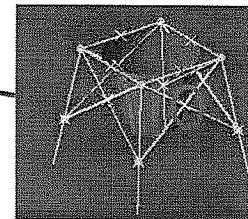
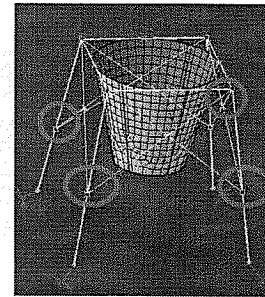
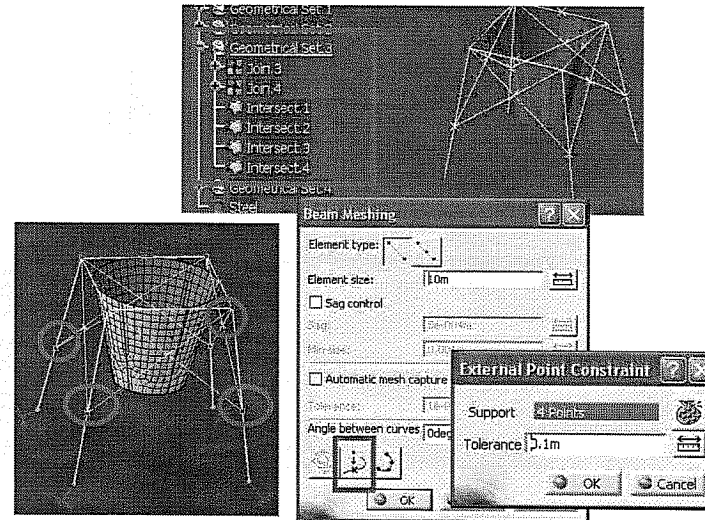
- ◆ Size=10m
- ◆ Automatic capture activated
- ◆ Imposed 4 middle points
- ◆ Click apply to mesh

- Mesh the 2<sup>nd</sup> group: select join.4 or the 2<sup>nd</sup> part of the new beams

- ◆ Size=10m
- ◆ Automatic capture activated

Nota: capture detects the intersection point

- ◆ Click apply to mesh



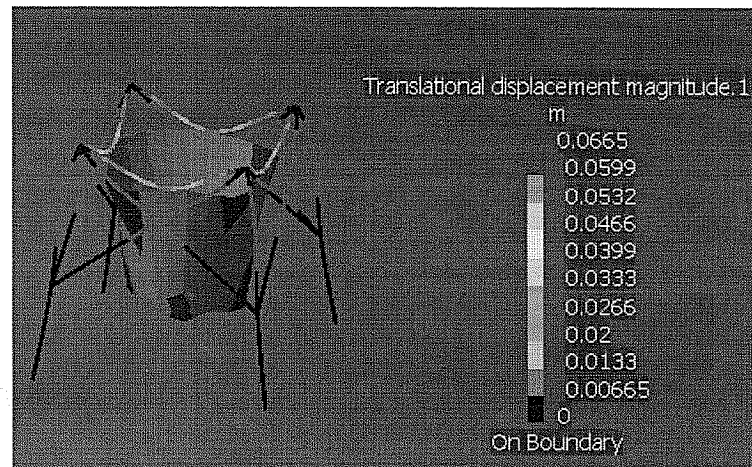
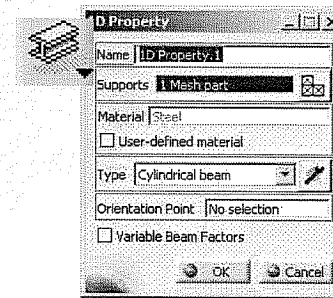
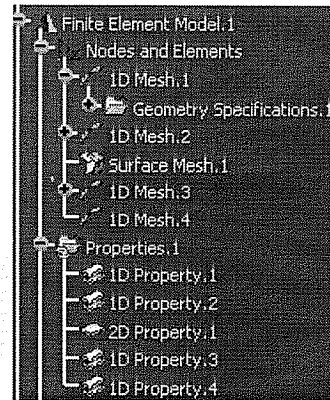
## Do It Yourself (6/6)

Go to Generative Structural Analysis workbench

Apply properties to 2 new 1D Meshes

Re launch computation

Visualise new displacements results



# Additional Exercise

## 16. Extrude 1D element into 2D Mesh

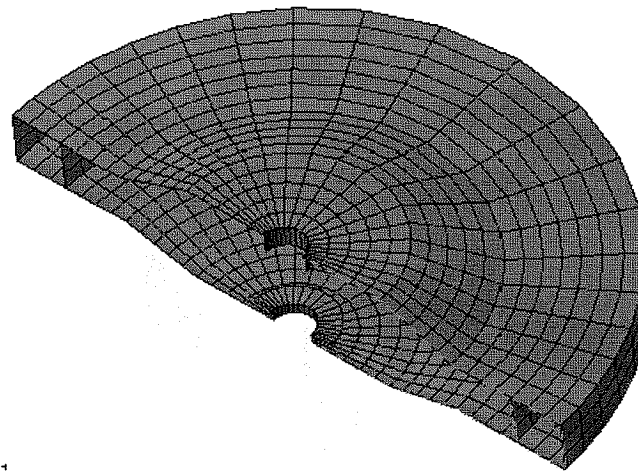


15 min.

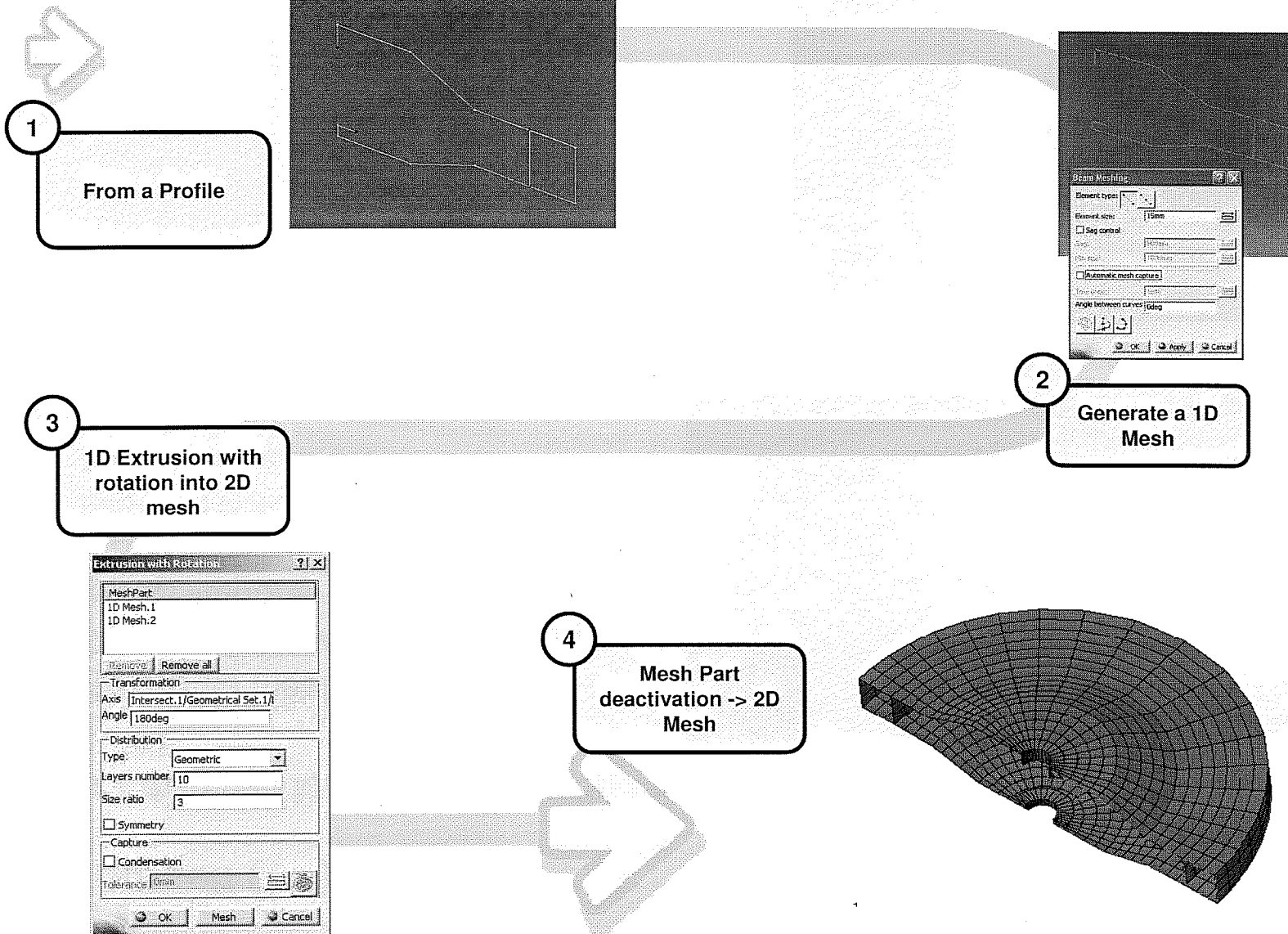
In this exercise you will generate 2D mesh from 1D elements

● You will use :

- ◆ 1D Mesh
- ◆ Elements Extrusion
- ◆ Mesh Part deactivation



## Design Process : Extrude 1D elements into 2D Mesh

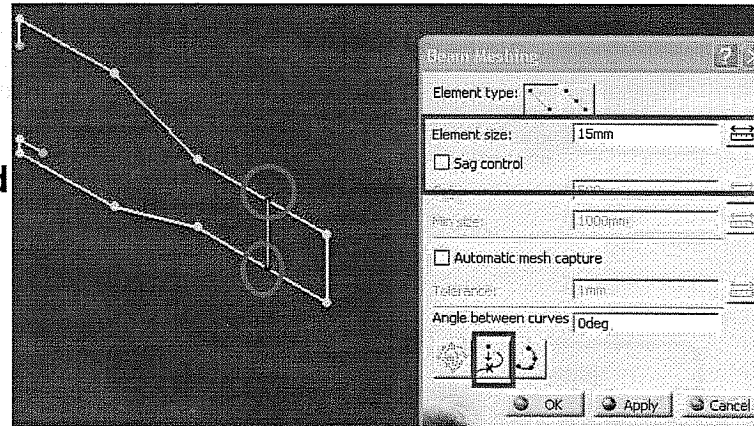


## Do It Yourself (1/4)

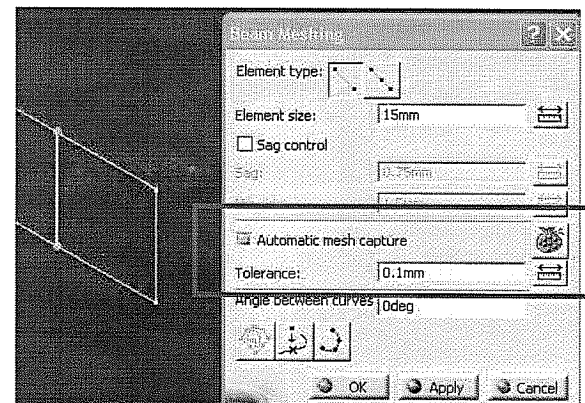
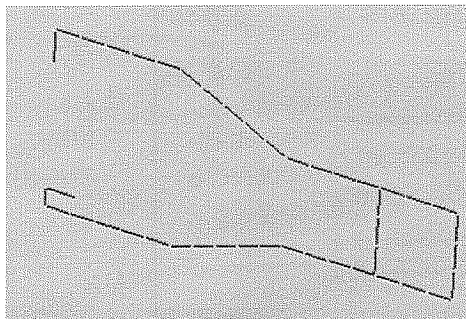
- Enter Advanced Meshing Tools workbench: Static Analysis

- Create profile meshes:

- Click on *Beam Mesher* icon
- select Join.1 as geometry to be meshed
- Size=15mm
- Imposed Points: 2 intersection points
- Click Apply and OK



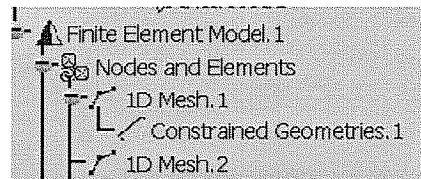
- Idem for Line.11 and capture 1D mesh.1 nodes



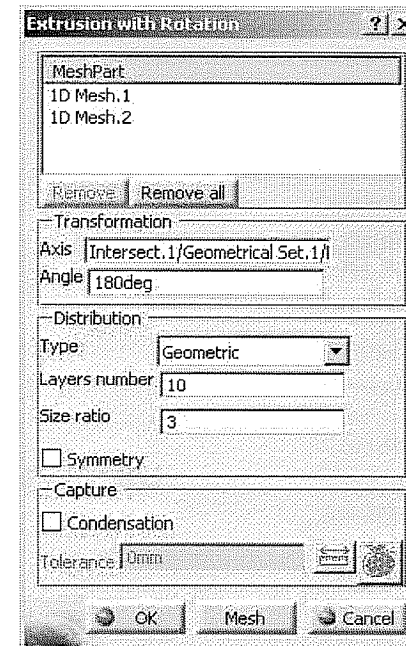
- Load: 1D Extrusion.CATPart

## Do It Yourself (2/4)

- Select the Extrude Mesher with Rotation icon
- Multi select 1D Mesh.1 and 1DMesh.2



- Intersect.1 as axis,
- 180deg as angle,



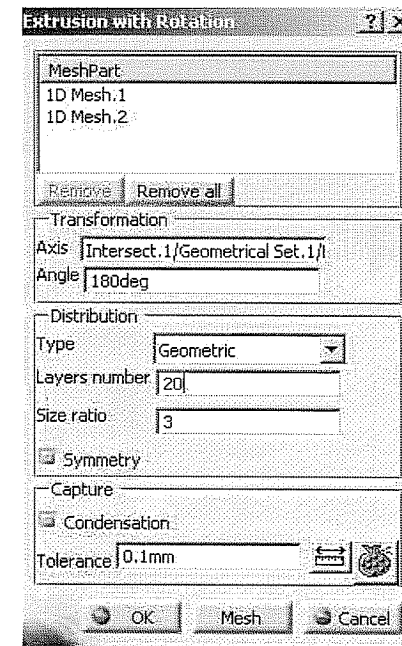
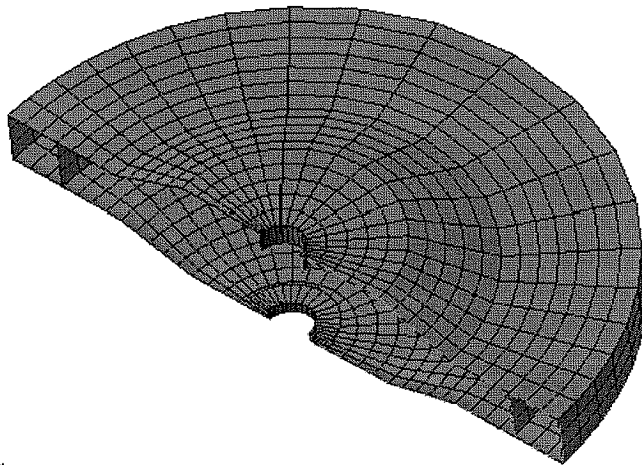
- Load: 1D Extrusion.CATAnalysis



Student Notes:

## Do It Yourself (3/4)

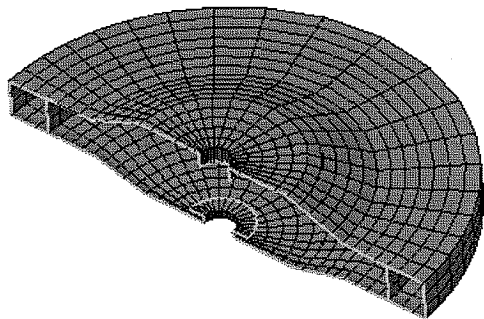
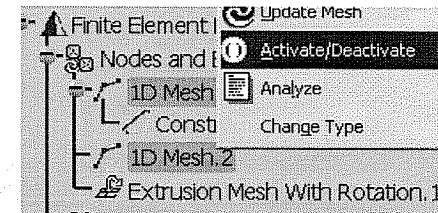
- Choose Geometric Distribution type
- 20 as numbers of layers
- 3 as size ratio
- Activate Symmetry to get a symmetrical distribution
- Click Mesh



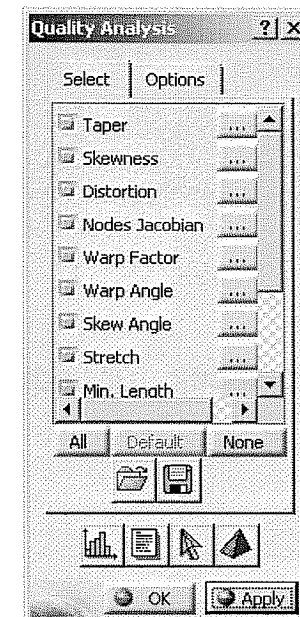
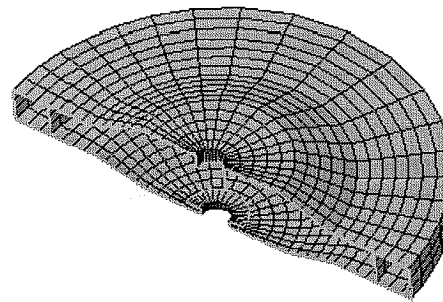
## Do It Yourself (4/4)

Deactivate 1D meshes using their contextual menu.

Check Free Edges



Check Elements Quality



Load: 1D Extrusion End.CATAnalysis

# Additional Exercise

## 17. Mesh Parameters Exposition to Knowledge 1



15 min.

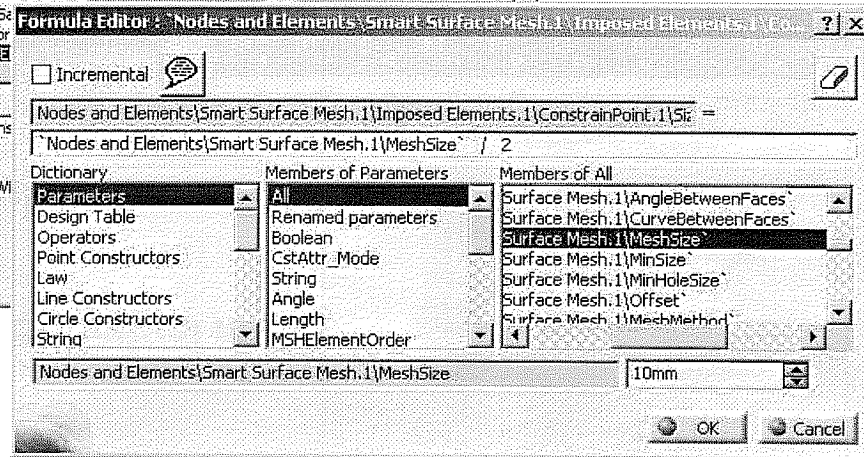
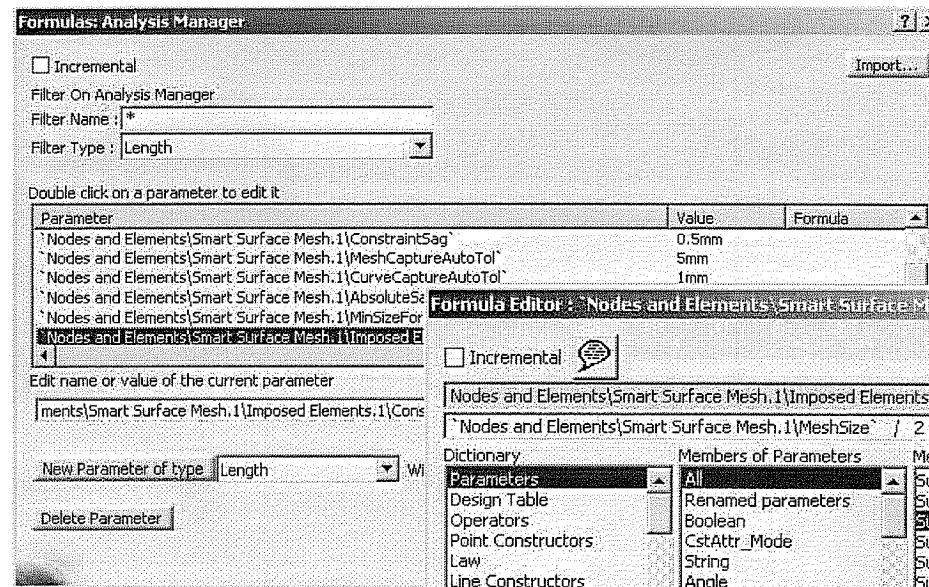
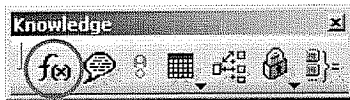
In this exercise you will create a Parameterisation of a FEM Model using rules and formulas and see how the Mesh will be modified, according to the geometry modifications.

You will use :

◆ Knowledge tools:

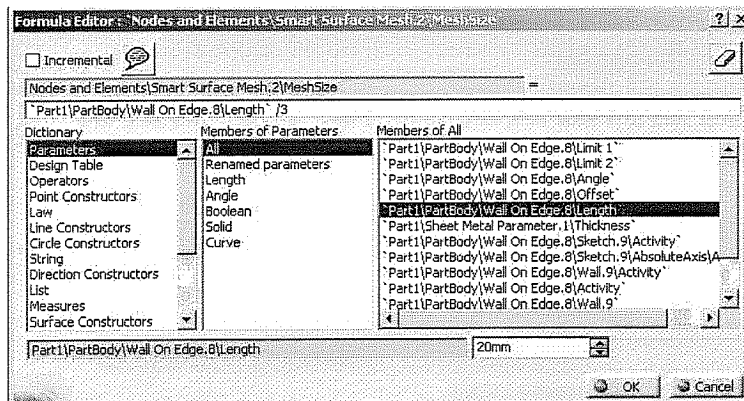
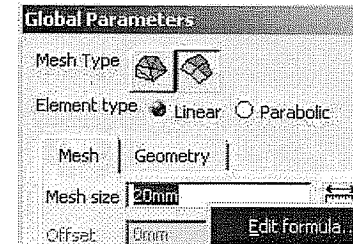
Formula

Rule

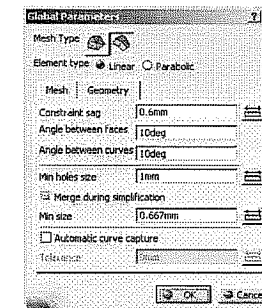
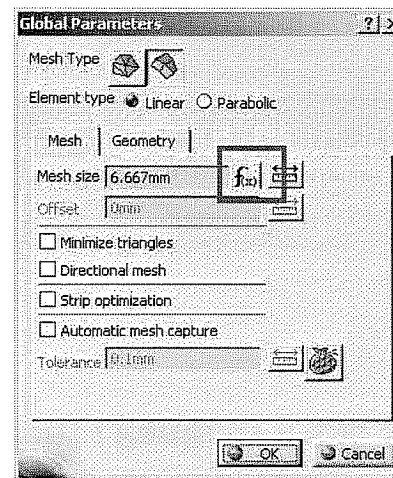


## Do It Yourself (1/6)

- Go to Advanced Meshing Tools workbench
- Define a mesh on the surface
  - In *Mesh Size* field, display the contextual menu
  - Choose *Edit Formula* icon
  - Enter the following formula in the editor (click part1 / PartBody in the tree to filter the parameters)



Mesh Size = "Wall on edge 8" / 3



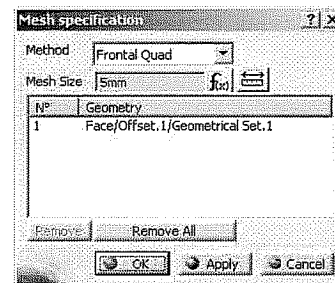
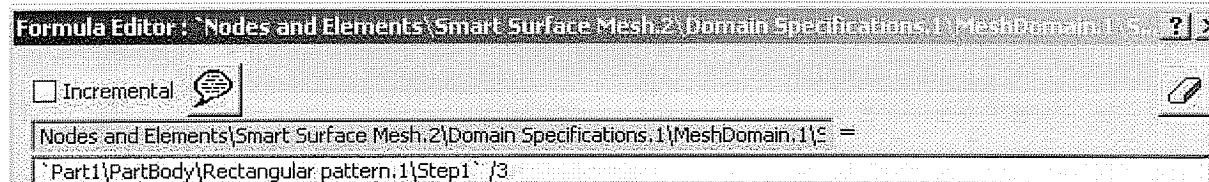
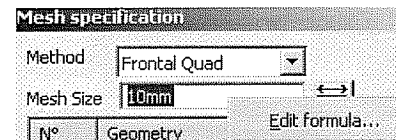
- Constraint sag = 0.6mm
- Minimum holes size=1mm
- Activate *Merge during simplification*

## Do It Yourself (2/6)

### Define *Domain Specifications* on main surface



- Using *Mesh Size* field contextual menu, click on *Edit formula icon*.
- Enter the following formula and validate (click part1 / PartBody in the tree to filter the parameters)



### Mesh and Exit

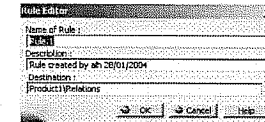


## Do It Yourself (3/6)

- Go to the Knowledge Advisor workbench



- Click on the **Rule** icon and name it **Rule on merge simplification**

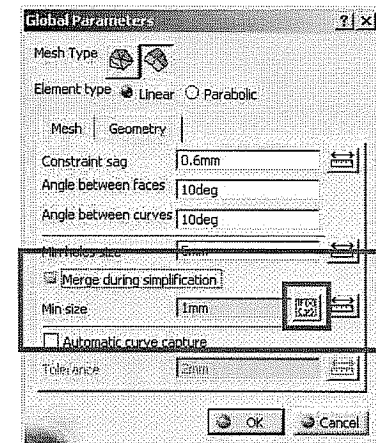
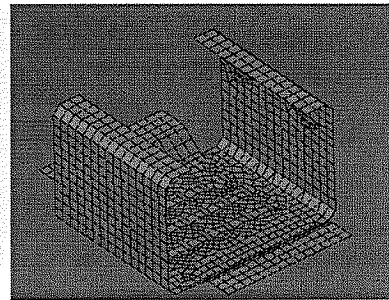
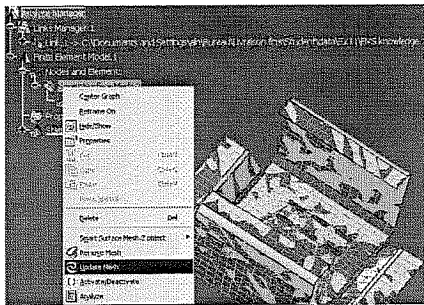


- Enter the following Rule in the Rule Editor and validate :

if `Nodes and Elements\Advanced Surface Mesh.1\Mesh Size` <=10mm  
 `Nodes and Elements\Advanced Surface Mesh.1\Min Size` = 1mm  
 else `Nodes and Elements\Advanced Surface Mesh.1\Min Size` = 8mm

if `Nodes and Elements\Advanced Surface Mesh.1\Mesh Size` <=10mm  
 `Nodes and Elements\Advanced Surface Mesh.1\Min Size` = 1mm  
 else `Nodes and Elements\Advanced Surface Mesh.1\Min Size` = 8mm

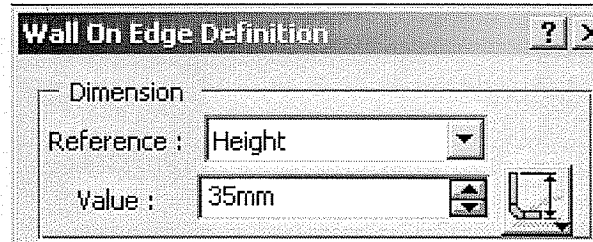
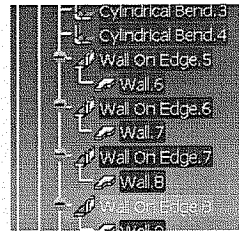
- Update in the Specification Tree the Smart Surface Mesh.1 : Right click on it in the tree and select **Update Mesh**.



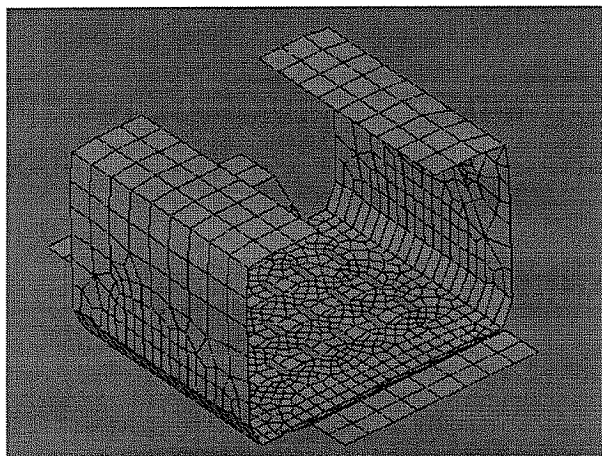
Nota: Tools+Options/Analysis and Simulation, General, activate Show Relations to visualise the rule in the tree

## Do It Yourself (4/6)

- Go to the FMS knowledge 1\_1. CATPart
- Put the PartBody in the show mode and edit the Wall On Edge.8 definition
- Modify the value to 35 mm



- Validate, update the Part if necessary.
- Return to the Analysis document by selecting the Advanced Meshing Tools workbench in the Start menu (or by double clicking on the Finite Element Model.1 in the Tree).
- Right click on Nodes and Elements and select the *Update all meshes* option.



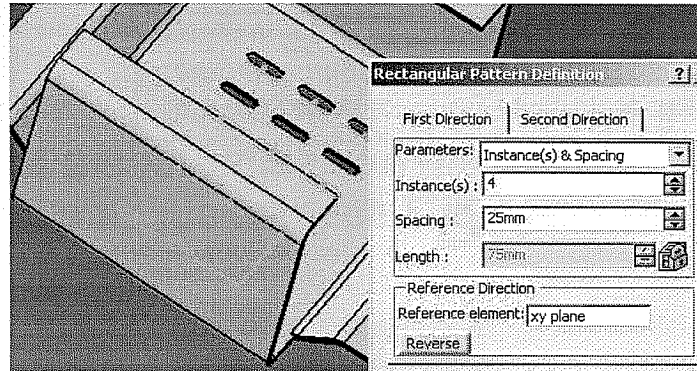
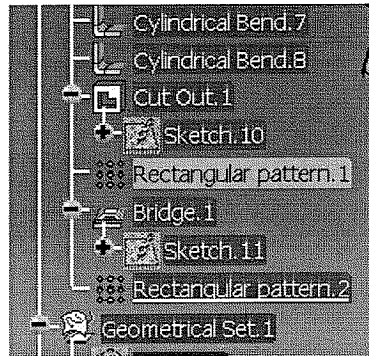
There is a formula and a rule between the length of Wall on Edge.8 and the MeshSize. When the value of this length is modified, the Mesh will be updated consequently.

Edit the Global Parameters on the Smart surface Mesh.2 and note that you have the MinSize value defined in the Rule.

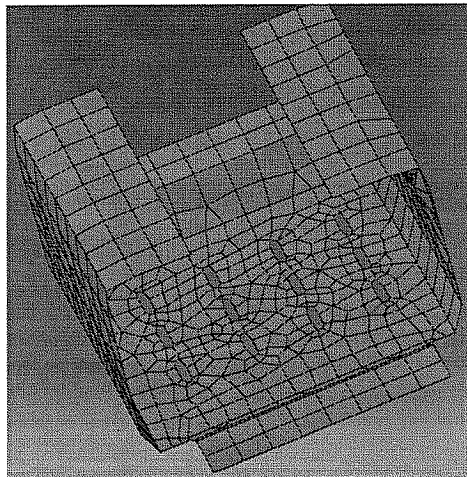


## Do It Yourself (5/6)

- Return to the FMS knowledge 1\_1. CATPart
- Modify the parameters of the Rectangular Pattern.1  
Number of instances : 4, Spacing : 25 mm.



- Hide the PartBody and return to the Analysis file.
- Right click on Nodes and Elements and select the *Update all meshes* option.

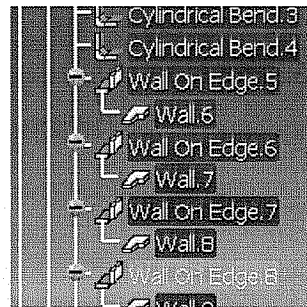
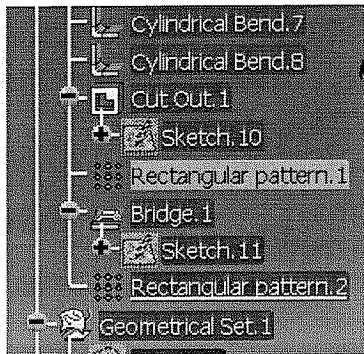


There is a formula between the step1 of the Rectangular Pattern.1 and the Size of the MeshDomain.1. When the value of this step is modified, the Size will be updated consequently.

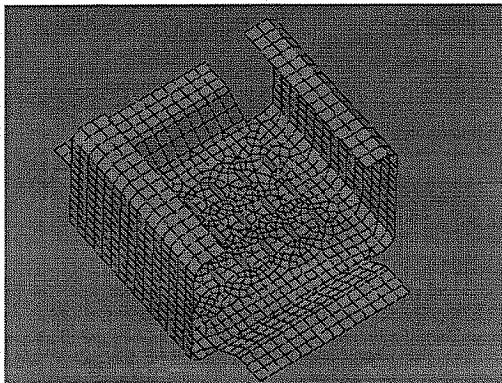
You can edit the Domain Specifications.1 parameters to check that the value is 8,333 mm

## Do It Yourself (6/6)

- In *FMS knowledge 1\_1. CATPart* and enter the initial values of modified parameters
  - 20 mm in the Wall On Edge.8 definition
  - 6 instances and 15 mm spacing in the Rectangular Pattern.1 definition.



- Return to the Analysis file, update all the meshes and check that all the values are updated according to the formulas and the Rule.



This operation is totally reversible, the communication between the part and the analysis file is permanent.

# Additional Exercise

## 18. Mesh Parameters Exposition to Knowledge 2

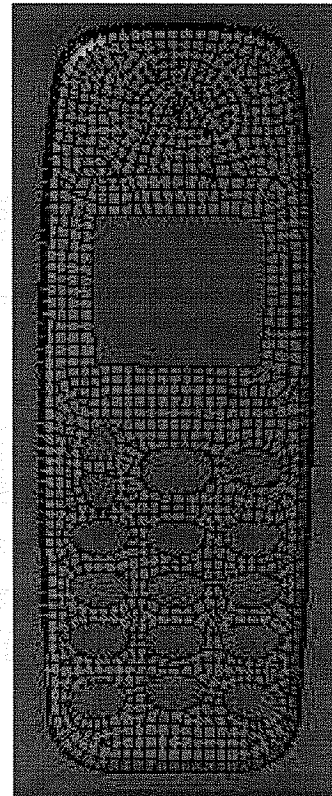
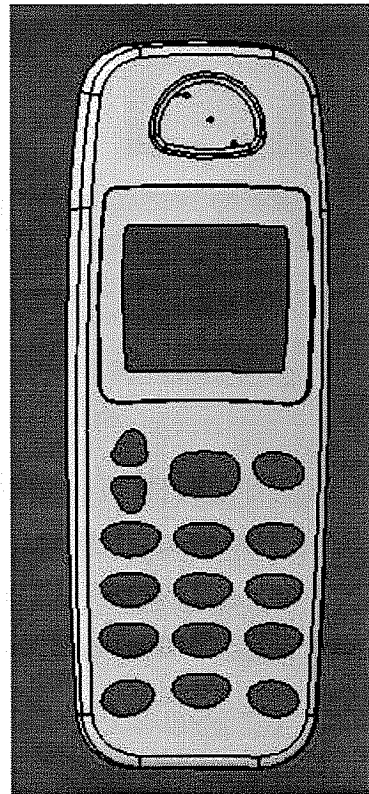


10 min.

In this exercise you will see how the Mesh will be modified, according to the geometry modifications according to the values set in a Design Table.

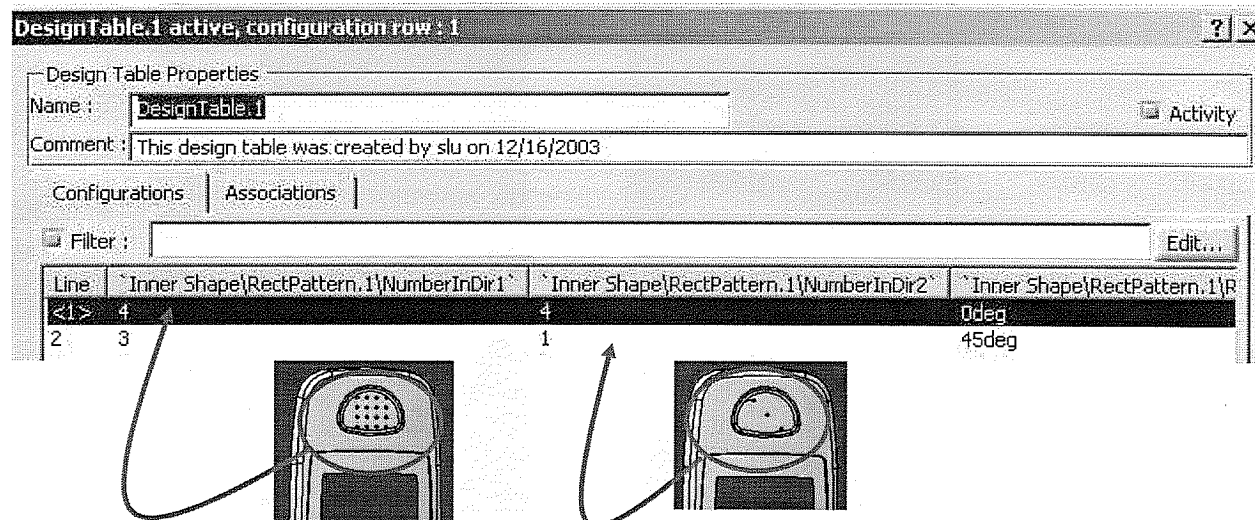
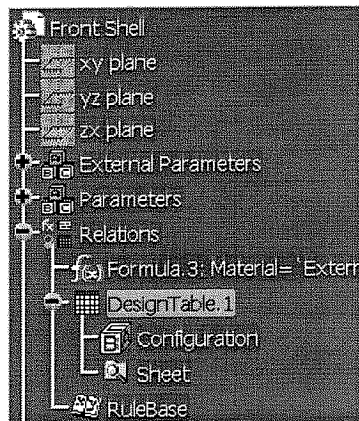
You will use :

- ◆ Rules, design table
- Mesh parameters ruled by geometrical parameters



## Do It Yourself (1/3)

- In the Front Shell.CATPart, note the two configurations available in the Design Table. To do that, edit the Design Table under the relations node and apply the second configuration. You will see that the mobile geometry is modified.



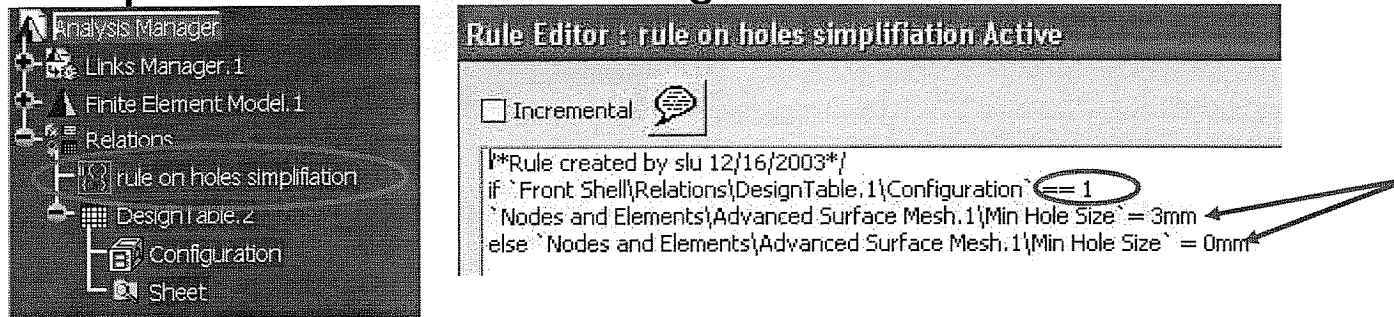
- Keep the first configuration and go to the Analysis file.
- Put the part in the show mode to see the selected configuration on the mobile.

Nota: Tools+Options/Infrastructure and Part Infrastructure, Display, activate Relations

- Load : FMS knowledge Mobile.CATAnalysis
- Load : Front Shell.CATPart

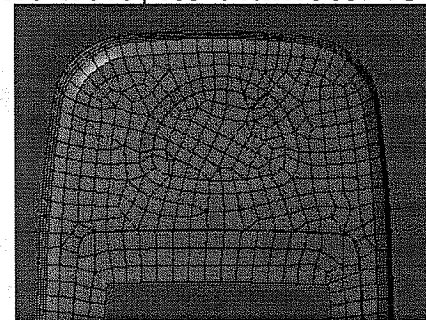
## Do It Yourself (2/3)

- In FMS knowledge Mobile.CATAnalysis, edit the *Rule on holes simplification* located under the Relations node and check that the minimum hole size depends on the selected configuration.



If a hole is lower than min hole size parameter(3mm or 1mm), this hole is ignored in the mesh.

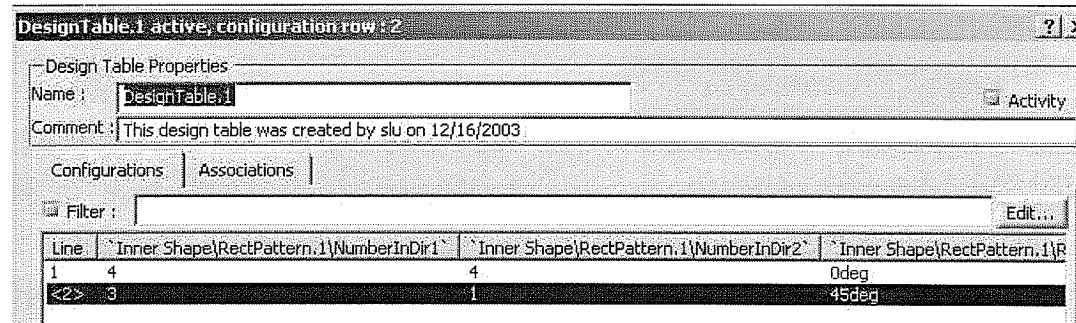
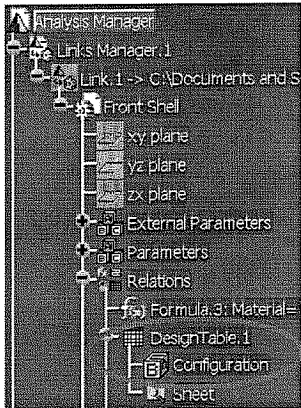
- You can see in the rule that in the configuration 1, the *minimum hole size* parameter is 3 mm. In Front Shell CATPart, measure the size of one hole, its diameter is 1 mm.
- Referring to the rule:  
The holes are ignored in the mesh.



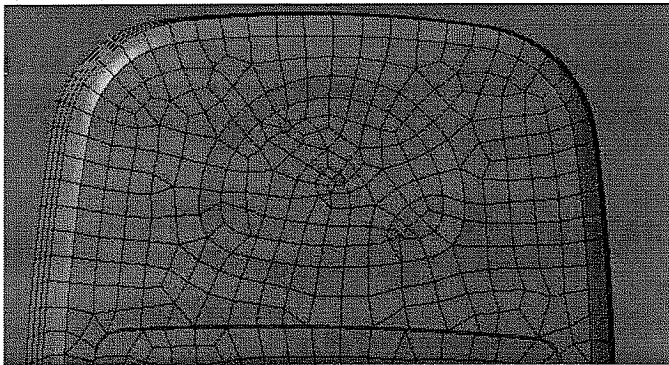
Nota: Tools+Options/Analysis and Simulation, General, activate Show Relations

## Do It Yourself (3/3)

- Change the configuration in the design table : In the Front Shell.CATPart, double-click on the design Table located under the Relations node, choose the configuration 2 and validate.



- In the analysis file, right click on Nodes and Elements and select the *Update all meshes* option.



According to the rule, the value of min size hole is now 0mm. All the holes are taken into account while meshing.



# Additional Exercise

## 19. Mesh Parameters Exposition to Knowledge 3

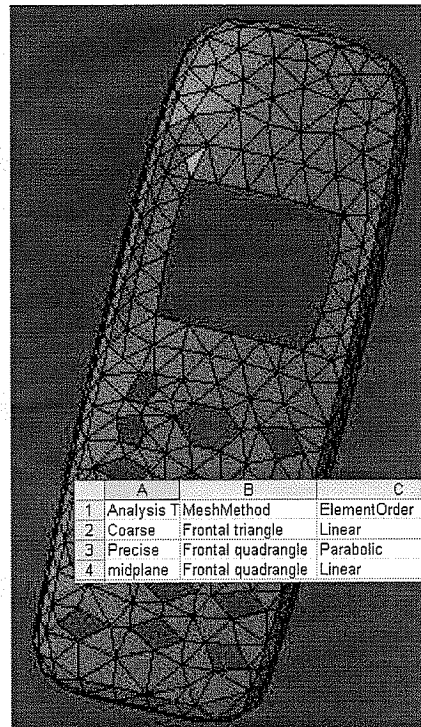
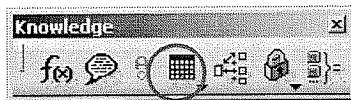


5 min.

In this exercise you will create a Parameterisation of a FEM Model using Design table in the Analysis file.

You will use :

◆ Design Table

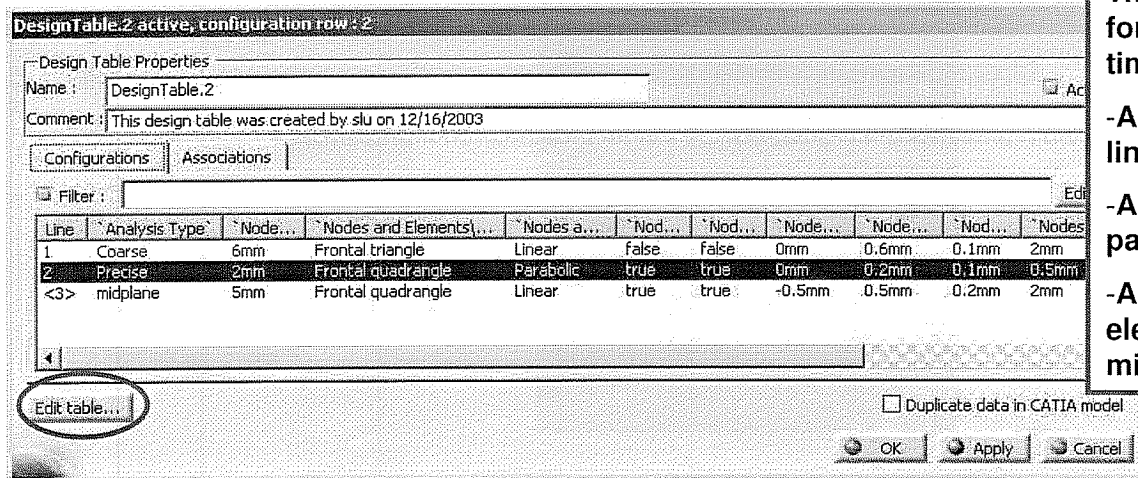


|   | A          | B                  | C            | D             | E            | F               | G            | H              | I          |
|---|------------|--------------------|--------------|---------------|--------------|-----------------|--------------|----------------|------------|
| 1 | Analysis T | MeshMethod         | ElementOrder | MeshSize (mm) | MeshSag (mm) | MeshRelativeSag | MinSize (mm) | OptimizeRegula | MinimizeTi |
| 2 | Coarse     | Frontal triangle   | Linear       | 6             | 0.6          | 0.1             | 2            | FALSE          | false      |
| 3 | Precise    | Frontal quadrangle | Parabolic    | 2             | 0.2          | 0.1             | 0.5          | true           | TRUE       |
| 4 | midplane   | Frontal quadrangle | Linear       | 5             | 0.5          | 0.2             | 2            | true           | TRUE       |



## Do It Yourself (1/4)

- In the FMS knowledge Mobile DT.CATAnalysis, edit the Design Table located under the Relations node.



Thanks to the design table, we can define for example several meshes usable at any time :

-A coarse mesh for quick analysis using linear triangle elements

-A thin mesh for a precise analysis using parabolic elements

-A « midplane » mesh to respect the finite element method hypothesis using the midplane surface to support the mesh

- Click on the edit table... button to see all the specifications.

Nota: Tools+Options/Analysis and Simulation, General, activate Show Relations to visualise the Design Table

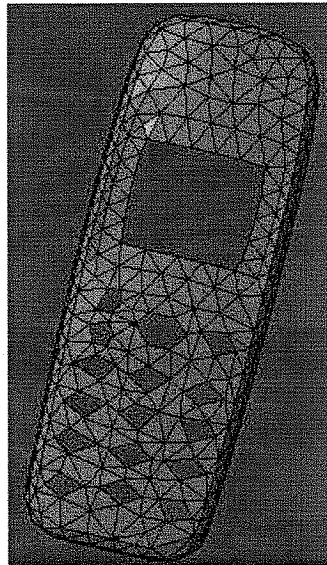
 Load : FMS knowledge Mobile DT.CATAnalysis

## Do It Yourself (2/4)

- An Excel document is loaded and you can see that for all configurations, you have the mesh parameters defined.

|   | A          | B                  | C            | D             | E            | F               | G            | H              | I          |
|---|------------|--------------------|--------------|---------------|--------------|-----------------|--------------|----------------|------------|
| 1 | Analysis T | MeshMethod         | ElementOrder | MeshSize (mm) | MeshSag (mm) | MeshRelativeSag | MinSize (mm) | OptimizeRegula | MinimizeTi |
| 2 | Coarse     | Frontal triangle   | Linear       | 6             | 0.6          | 0.1             | 2            | FALSE          | false      |
| 3 | Precise    | Frontal quadrangle | Parabolic    | 2             | 0.2          | 0.1             | 0.5          | true           | TRUE       |
| 4 | midplane   | Frontal quadrangle | Linear       | 5             | 0.5          | 0.2             | 2            | true           | TRUE       |

- Close without save the excel file and choose the Coarse configuration. Validate and update the mesh (right click on the Advanced Surface Mesh.1 in the specification tree and choose the Update Mesh)

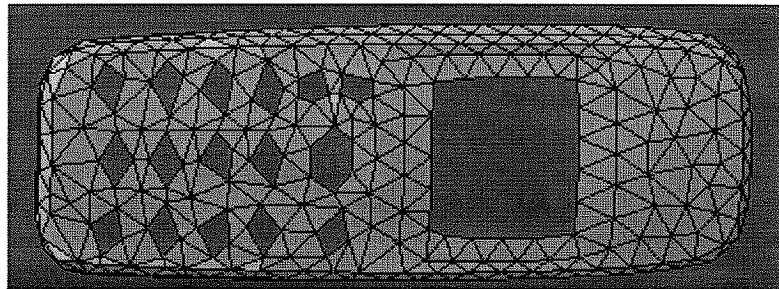
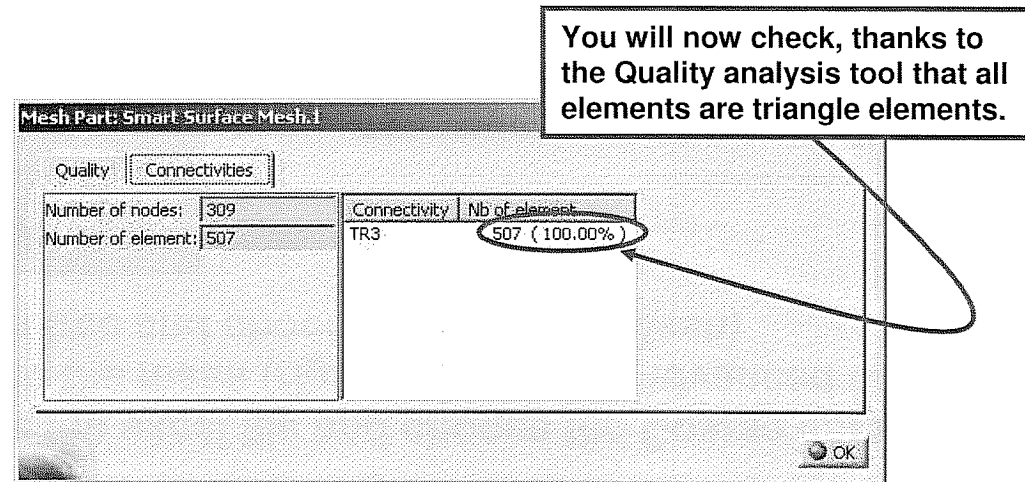


The mesh is modified according to the new mesh parameters.

For the coarse mesh, the chosen elements are linear triangles.

## Do It Yourself (3/4)

- In Advanced Surface Mesh.1 contextual menu, select Analyse 
- In the Connectivities Tab, you can see that all the elements are linear triangles TR3



## Do It Yourself (4/4)

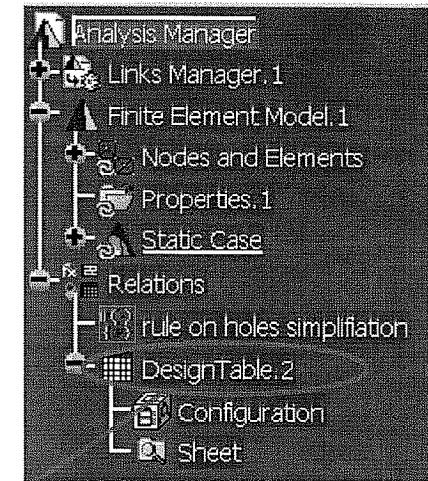
- Edit the Design Table located under the Relations node and choose the second configuration : Precise.

Configurations | Associations

Filter :

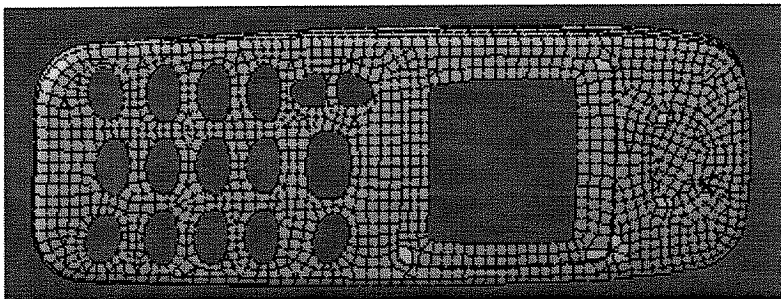
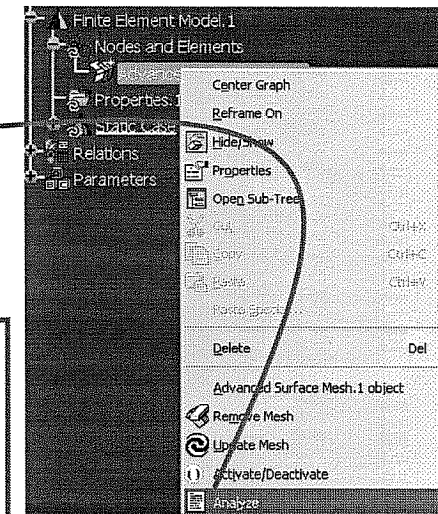
Edit...

| Line | Analysis Type | Node... | Nodes and Elements{... | Nodes a... | Nod... | Nod... | Node... | Node... | Nod... | Nodes and |
|------|---------------|---------|------------------------|------------|--------|--------|---------|---------|--------|-----------|
| 1    | Coarse        | 6mm     | Frontal triangle       | Linear     | false  | false  | 0mm     | 0.6mm   | 0.1mm  | 2mm       |
| 2    | Precise       | 2mm     | Frontal quadrangle     | Parabolic  | true   | true   | 0mm     | 0.2mm   | 0.1mm  | 0.5mm     |
| <3>  | midplane      | 5mm     | Frontal quadrangle     | Linear     | true   | true   | -0.5mm  | 0.5mm   | 0.2mm  | 2mm       |



- Validate and Update the mesh.
- Edit the Quality Analysis : right click on the Advanced Surface Mesh.1 in the specification tree and choose Analyze.

| Mesh Part: Smart Surface Mesh.1 |      |                |                 |
|---------------------------------|------|----------------|-----------------|
| Quality                         |      | Connectivities |                 |
| Number of nodes:                | 5530 | Connectivity   | Nb of element   |
| Number of element:              | 1712 | TR6            | 61 ( 3.56% )    |
|                                 |      | QD8            | 1651 ( 96.44% ) |



The type of elements is modified according to the mesh parameters located in the Design Table. The mesh is precise : the quadrangle elements are thinner and all elements are parabolic.

Student Notes:

## Hints and Tips

- Select “*applies customized view*” to visualize elements edges.