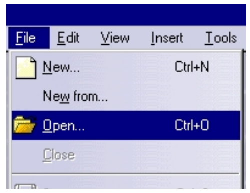
FEM Surface

Detailed Steps

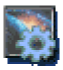
Version 5 Release 8
March 2002
EDU-CAT-E-FMS-FS-V5R8

Table of Contents

Master Exercise

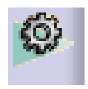


Step (1): The Analysis Document

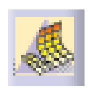
 **Load the Catia document “CATFMSParametric_Pillar.CATPart”**

The Parametric_Pillar part is opened in the Part Design Workbench.

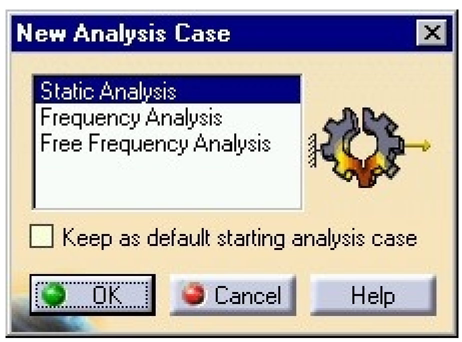
1. Open the **FMS Workbench** and create a static analysis case.
   a. Click on shortcut icon.



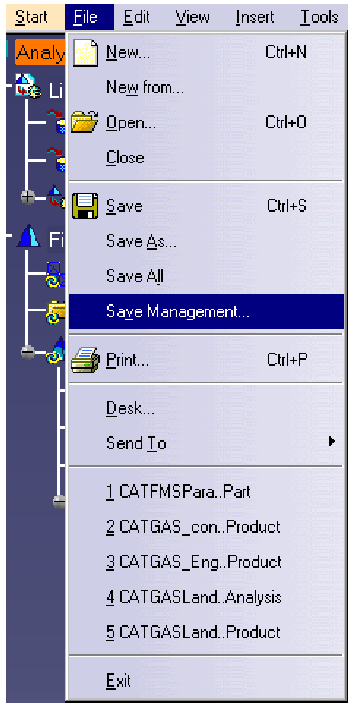
   b. Click on **Advanced Meshing Tools** button.

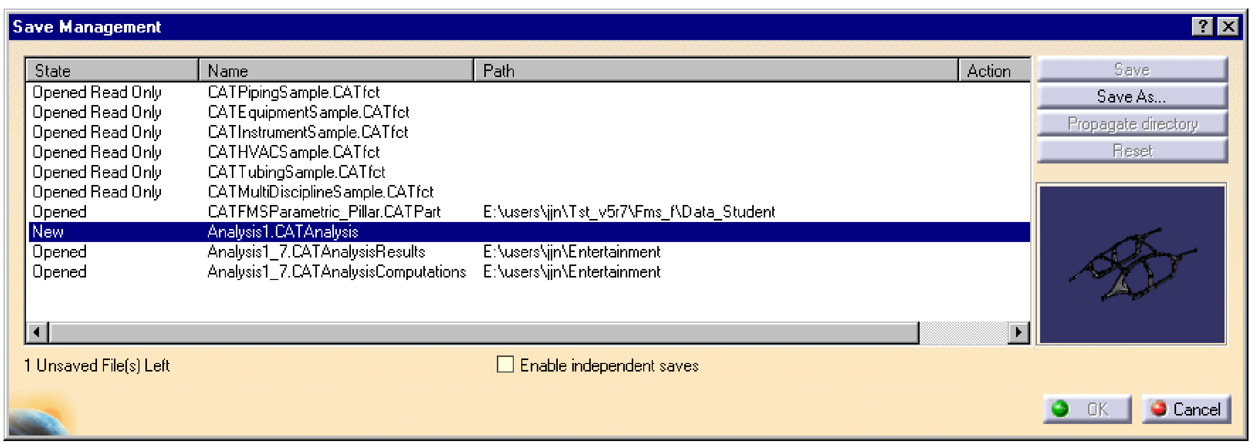


   c. Select **Static Analysis**



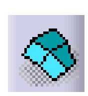
2. Save the analysis document under the name “CATFMSParametric_Pillar.CATAnalysis”.
   d. Click on **Save Management…** in the File menu.
   a. Save Part and Analysis in a diferent folder.



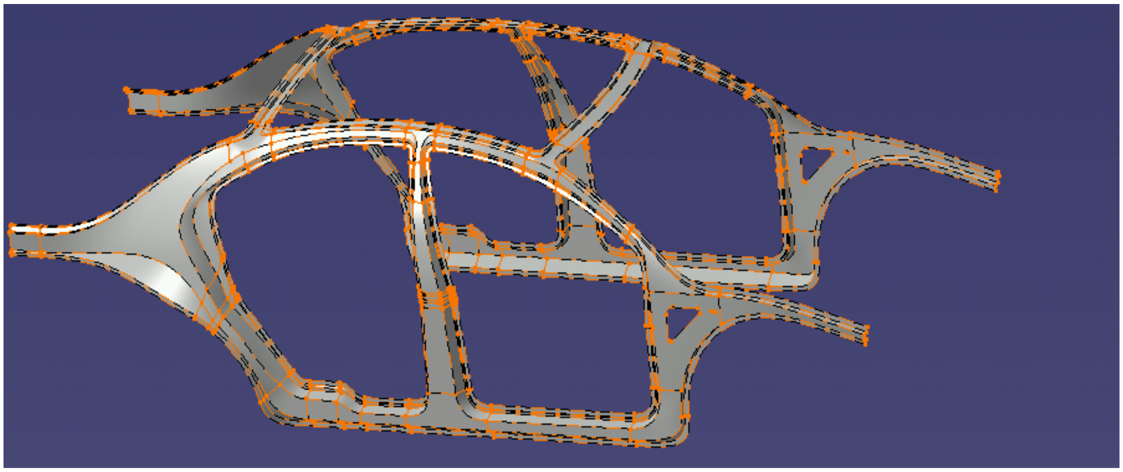


Step (2): Create a Mesh according to Specifications

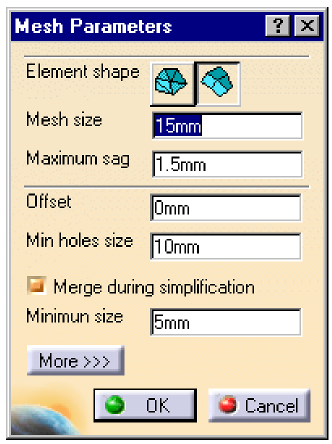
3. Set Surfacic Mesh Global Parameters.
4.
5.
    a. Click on Surface Mesher icon in Meshing Methods toolbar.



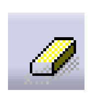
    b. Select the geometry.



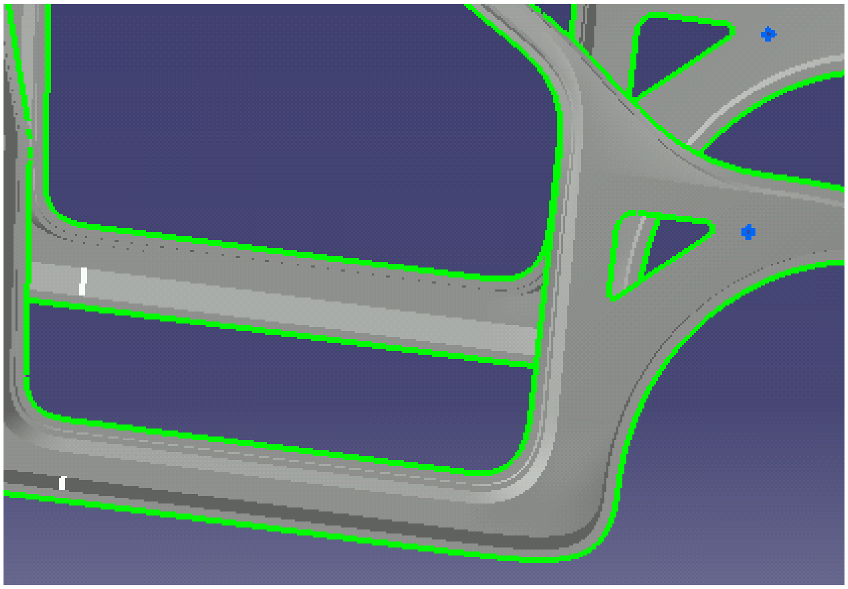
c. Set Global Parameters.

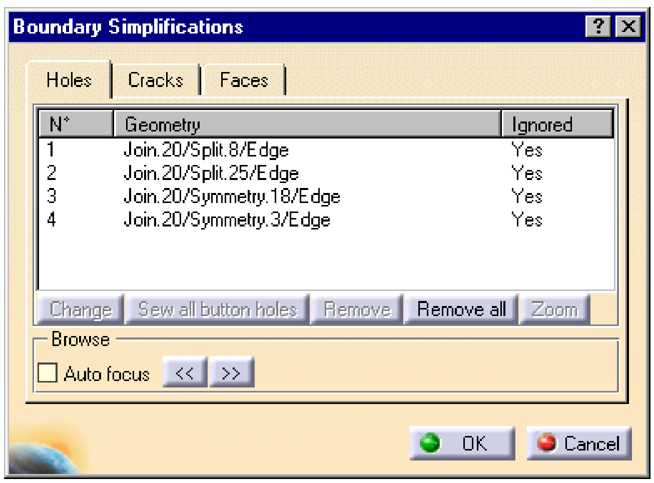


d. Clean the geometry.
e. Click on **Boundary Simplifications** icon in **Specification Tools** toolbar.

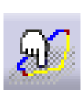


f. Select the two holes and gaps.
g. Click OK

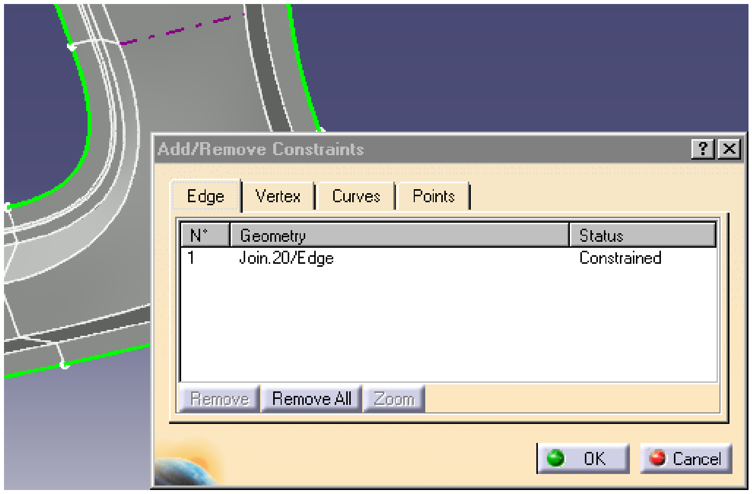




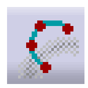
6. Add an edge constrain and define a node distribution.
    a. Click on Add/remove Contraints icon in Specifications Tools toolbar.



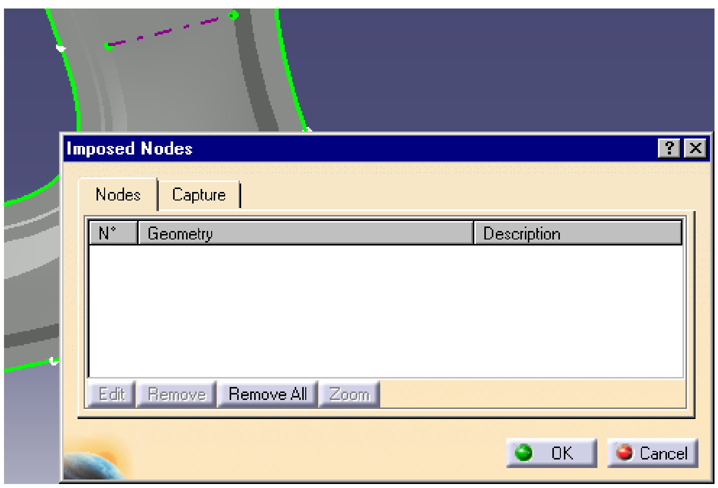
    b. Select an edge.

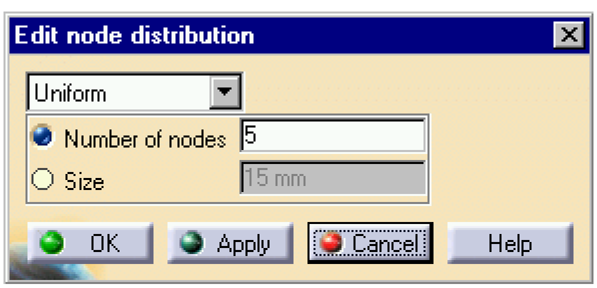


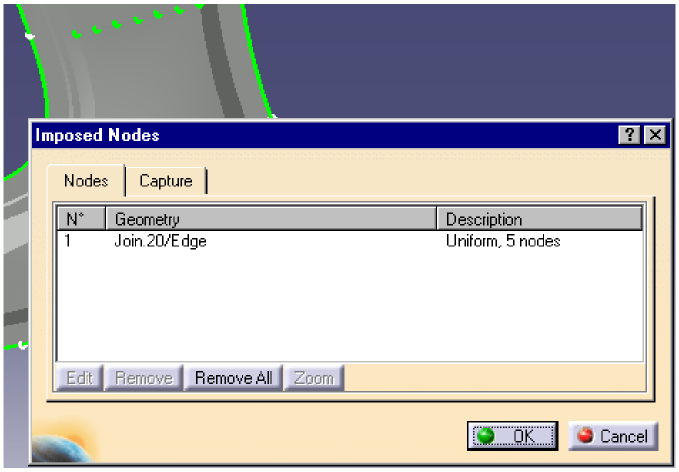
c. Click on **Imposed Nodes** icon in **Specifications Tools** toolbar.



d. Select an edge and set the nodes distribution parameters.

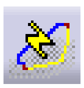


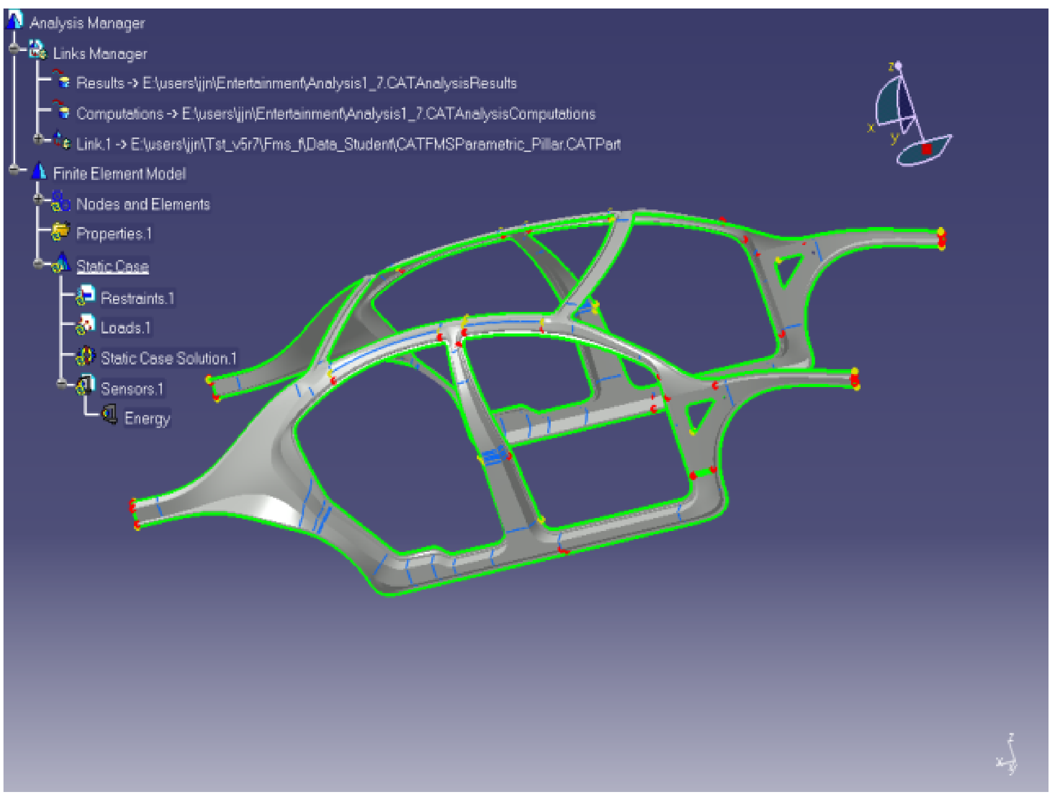




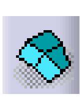
e. Click OK.

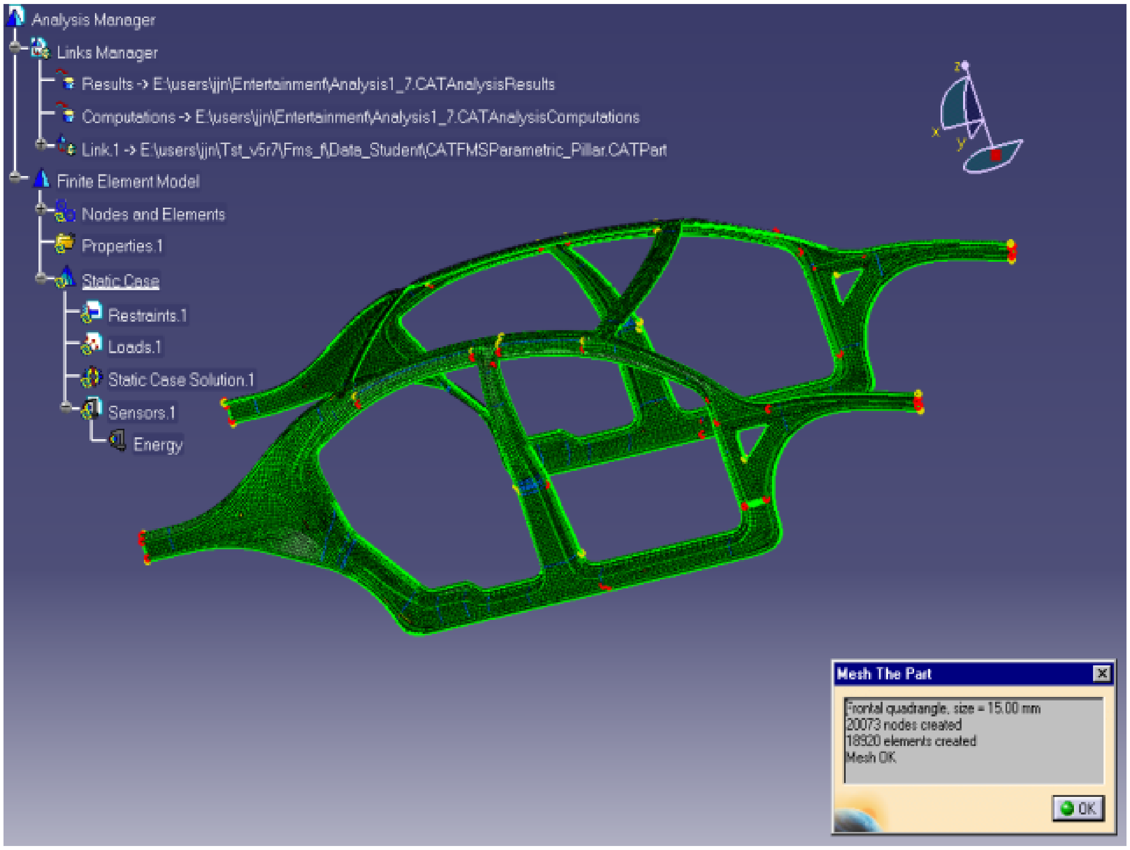
7. Geometry Simplification.
   a. Click on Geometry Simplification icon in Specifications Tools toolbar.





8. Create the mesh.
   a. Click on Mesh the Part icon in Specifications Tools toolbar.





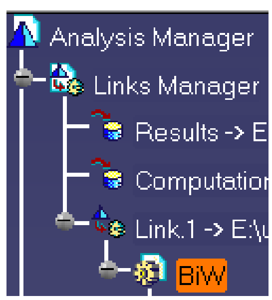
Step (3): modify and update an existing mesh

9. Modify pillar section and update geometry.

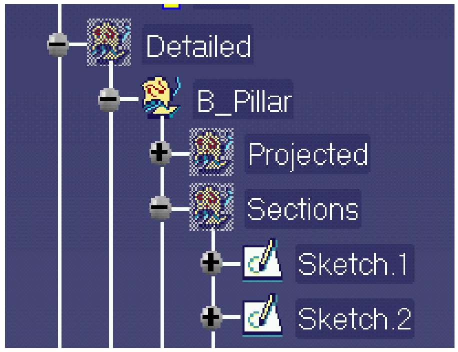
10.

11.

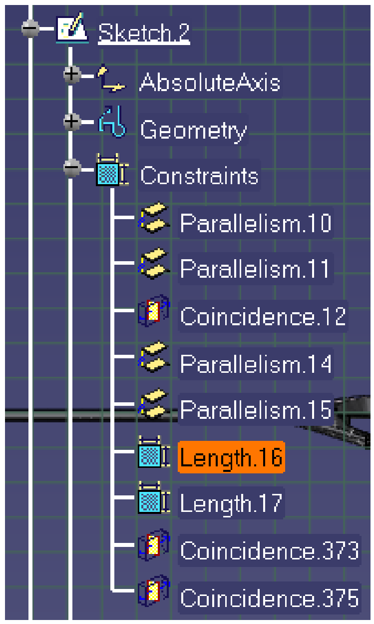
   a. Double click on BiW part.

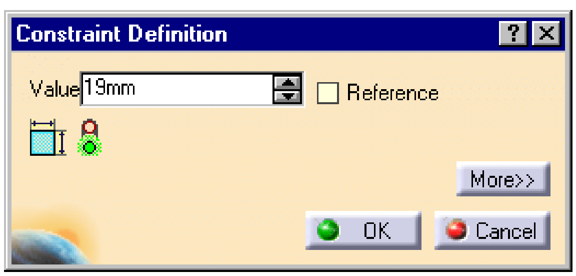


   b. Double click on Biw / Detailed / B_Pilar / Sections / Sketch.2.

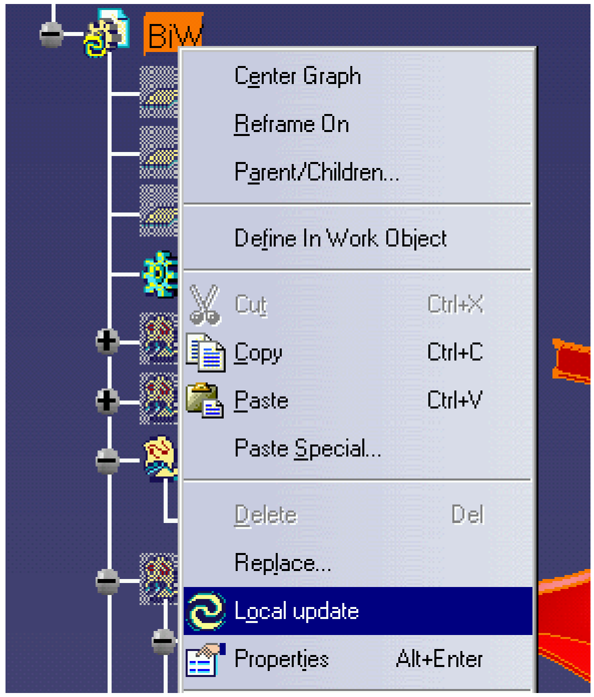


c. Double click on **Length.16**. Set its value to 19 mm.

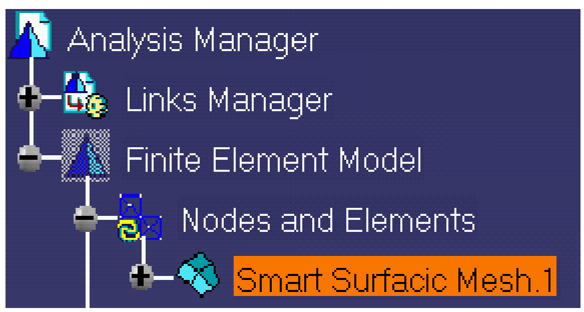




d. Double click on BIW.
e. Update geometry.

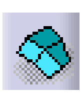


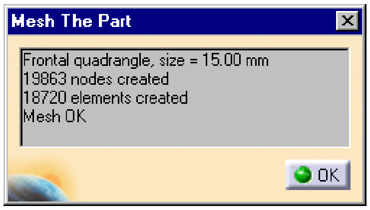
12. Update mesh.
    a. Double click on Smart Surfacic Mesh.1.



    b. Click Yes in the warning panel.

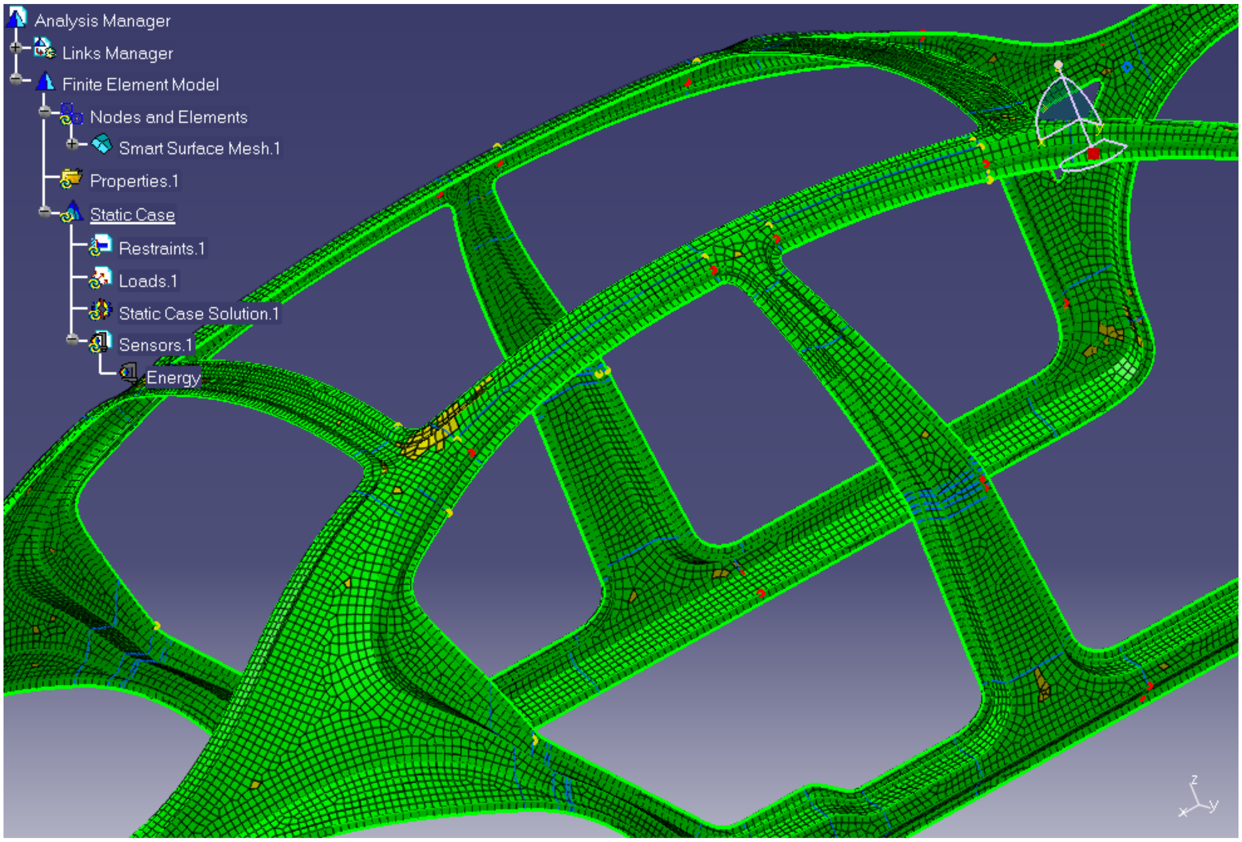
    c. Click on **Mesh the Part** icon in **Specifications Tools** toolbar.



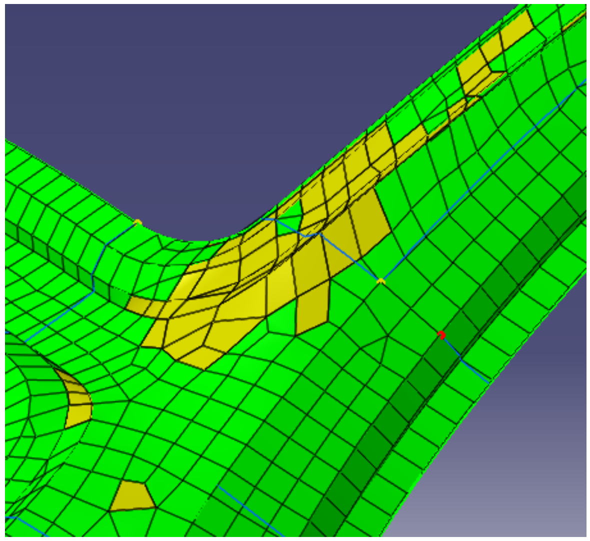


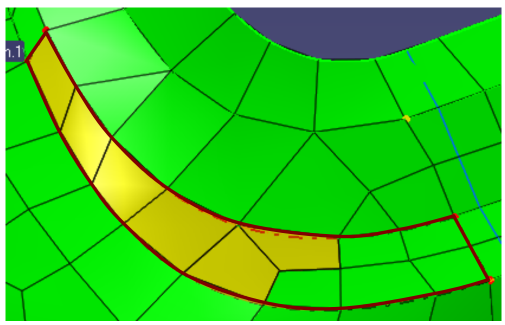
d. Click OK.

13. Remesh a domain.

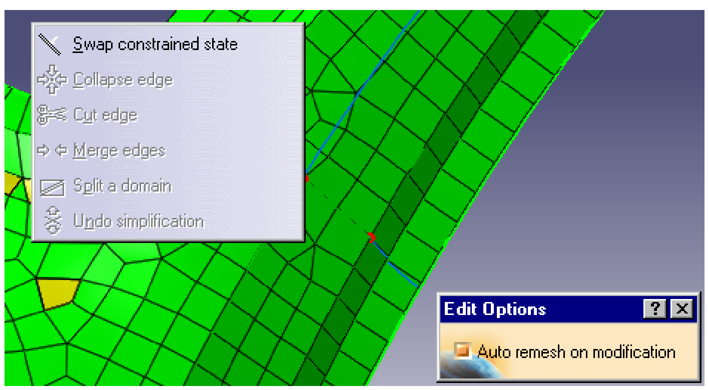


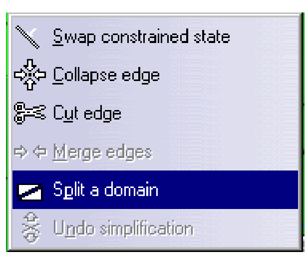
a. Click on Manual Simplifications in Modification Tools toolbar.
b. Add constraints on four edges to create an area.



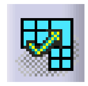


c. Click right on edges to use the menu.

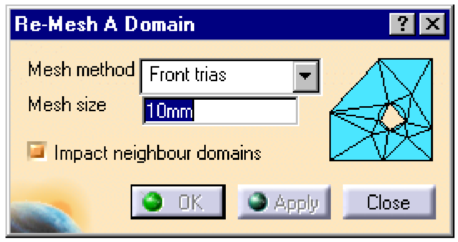




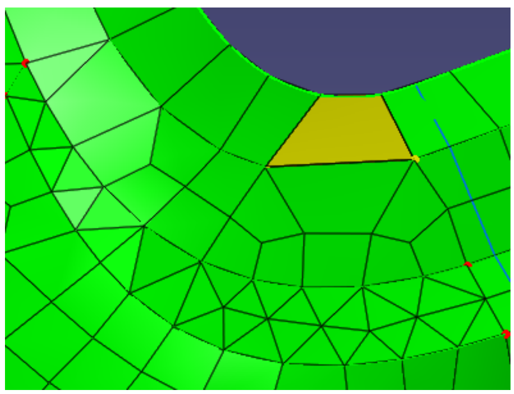
d. Click on **Re-Mesh a Domain** icon in **Modification Tools** toolbar.



e. Select the domain you have defined in previous step.
f. Set new mesh parameters

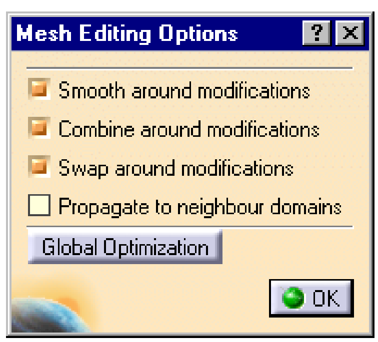


g. Click OK.

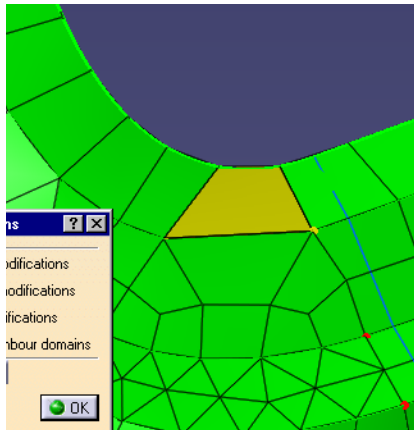


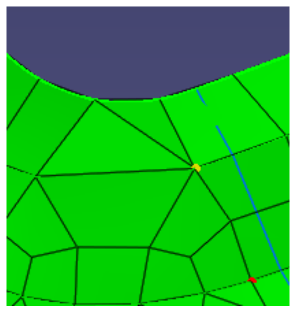
h. Click on **Edit Mesh** icon in **Modification Tools** toolbar

i. Select the options



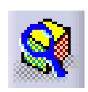
j. Split the biggest red or orange element in two.





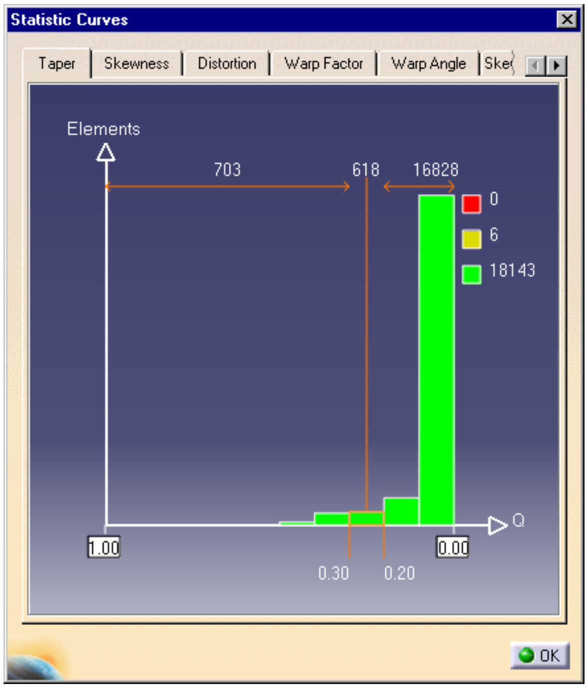
Step (4): analyse mesh quality

1. Shape factor analyse.
   k. Click **Quality analysis** icon.

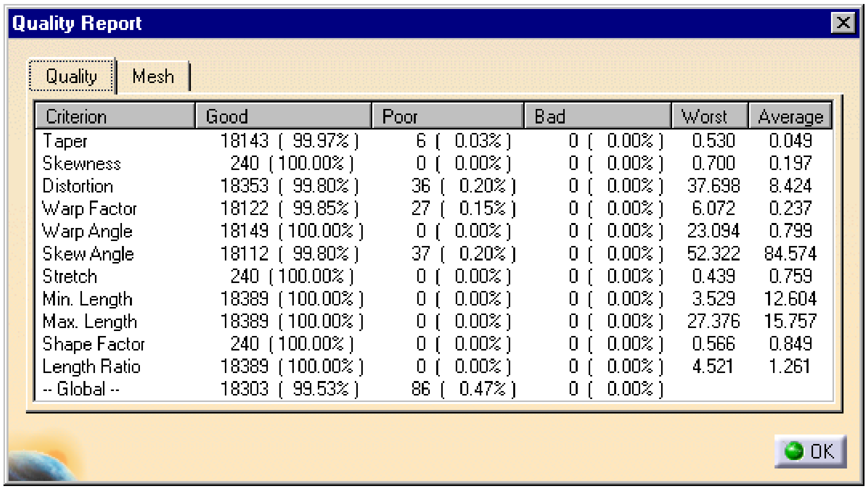


   a. Click **Statistic curves** icon in Quality Analysis panel.

b. Click **Shape Factor** icon in **Statistic Curves panel**.



2. Quality report.
   c. Click **Quality Report** icon in **Statistic Curves panel**.



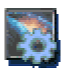
Quality Report

Quality | Mesh

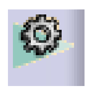
Criterion	Good	Poor	Bad	Worst	Average
Taper	18143 (99.97%)	6 (0.03%)	0 (0.00%)	0.530	0.049
Skewness	240 (100.00%)	0 (0.00%)	0 (0.00%)	0.700	0.197
Distortion	18353 (99.80%)	36 (0.20%)	0 (0.00%)	37.698	8.424
Warp Factor	18122 (99.85%)	27 (0.15%)	0 (0.00%)	6.072	0.237
Warp Angle	18149 (100.00%)	0 (0.00%)	0 (0.00%)	23.094	0.799
Skew Angle	18112 (99.80%)	37 (0.20%)	0 (0.00%)	52.322	84.574
Stretch	240 (100.00%)	0 (0.00%)	0 (0.00%)	0.439	0.759
Min. Length	18389 (100.00%)	0 (0.00%)	0 (0.00%)	3.529	12.604
Max. Length	18389 (100.00%)	0 (0.00%)	0 (0.00%)	27.376	15.757
Shape Factor	240 (100.00%)	0 (0.00%)	0 (0.00%)	0.566	0.849
Length Ratio	18389 (100.00%)	0 (0.00%)	0 (0.00%)	4.521	1.261
-- Global --	18303 (99.53%)	86 (0.47%)	0 (0.00%)		

OK

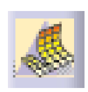
Step (4): torsional stiffness analysis

 **Load the Catia document “CATFMSParametric_Pillar-Holes.CATPart”**

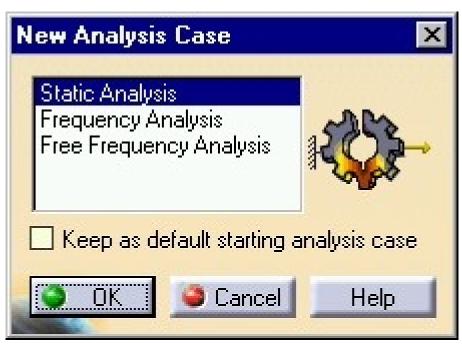
1. Surface mesh creating.
   a. Click on shortcut icon



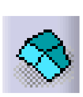
   b. Click on **Advanced Meshing Tools** button



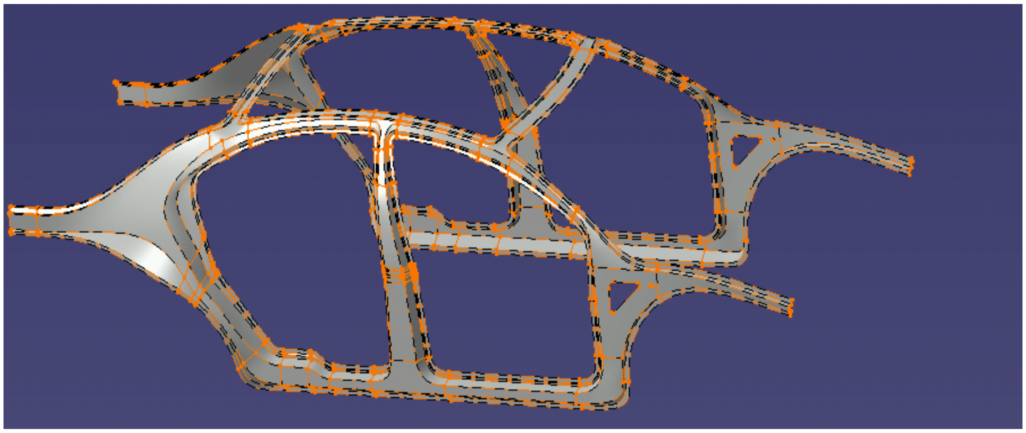
   c. Select **Static Analysis**



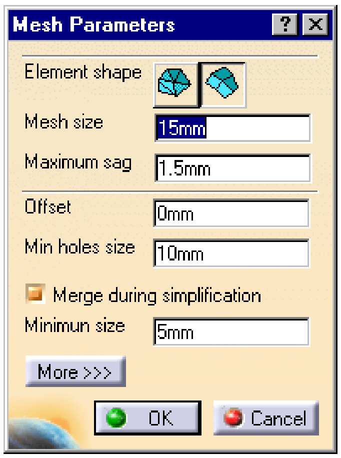
   d. Click on **Surface Mesher** icon in **Meshing Methods** toolbar.



   e. Select the geometry.

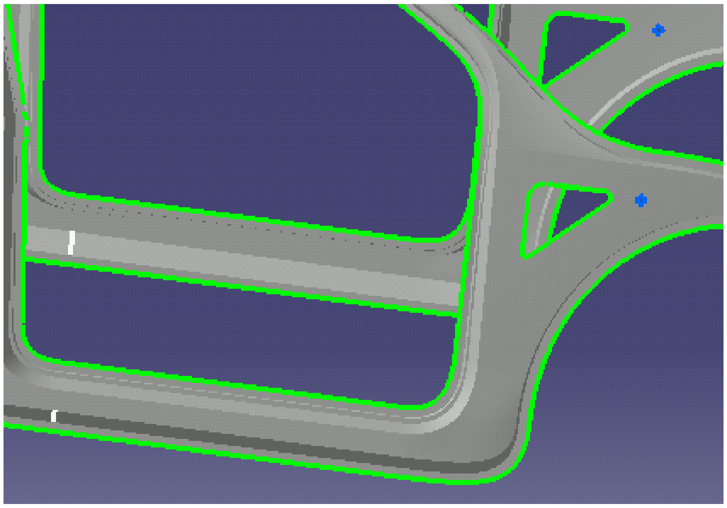


f. Set Global Parameters.

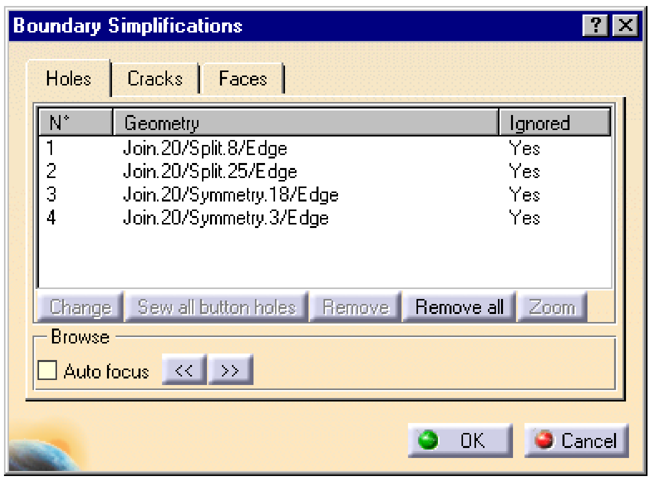


g. Click on **Boundary Simplifications** icon in **Specifications Tools** toolbar.

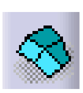
h. Select the two holes and gaps.

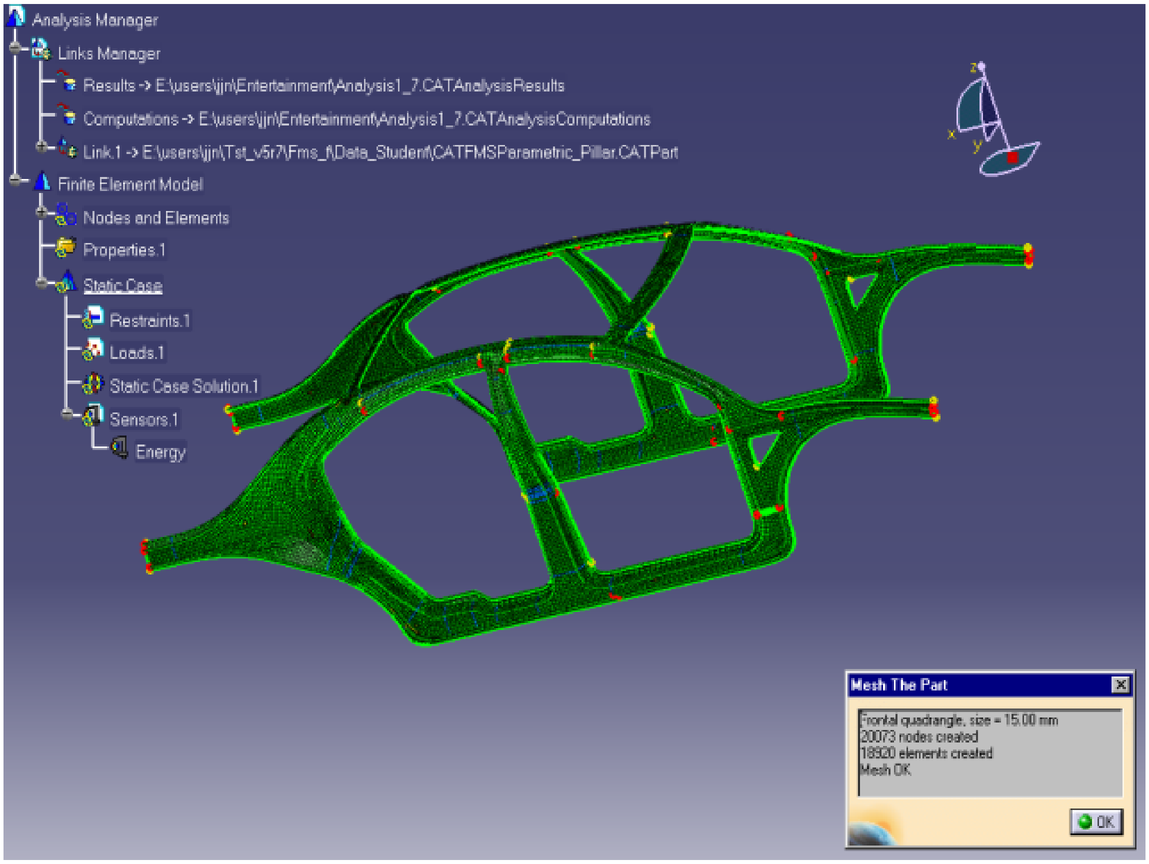


i. Click OK.



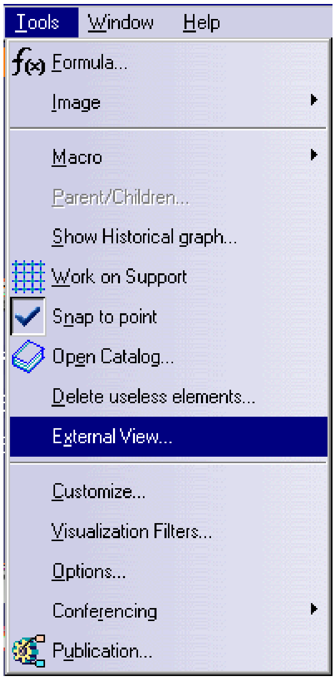
j. Click on **Mesh the Part** icon in **Specifications Tools** toolbar.



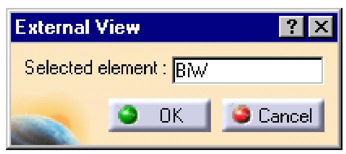


2. Static Analysis performance.
   a. Go to Generative Shape Design.

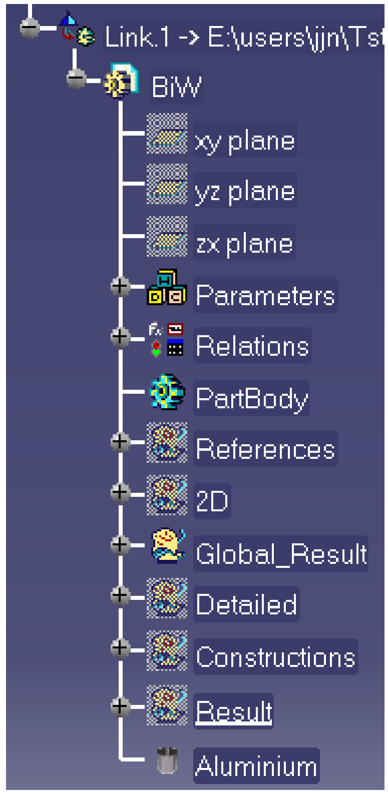
   b. Select **External View** in **Tools** menu.



c. Select **BiW** in order to define the geometrical support for GPS.

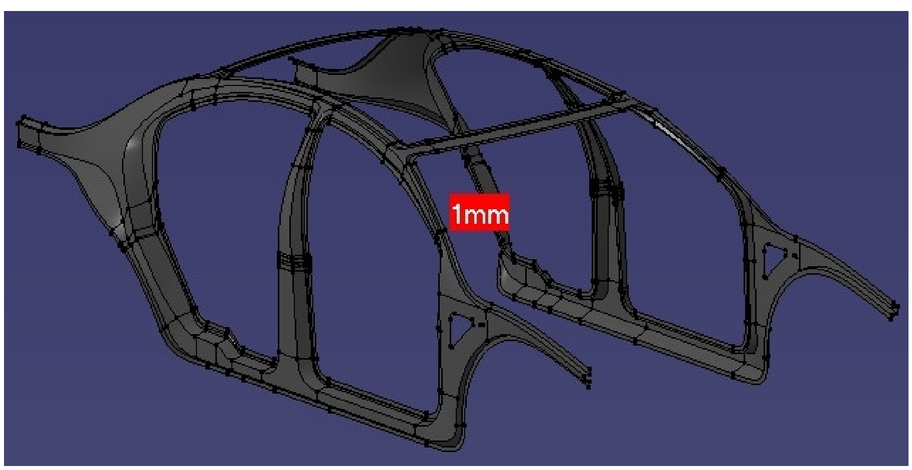


d. Apply **Aluminium** Material to the BiW Part

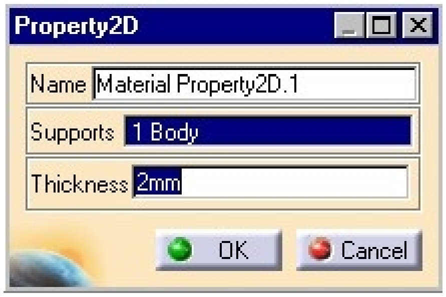


e. Click on **Generative Structural Analysis** in Start menu.

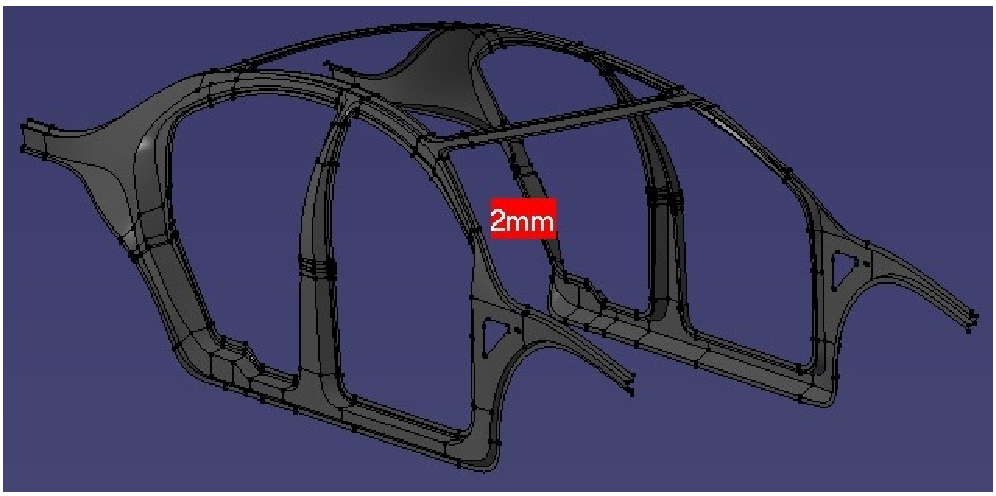
f. Double-click on thickness



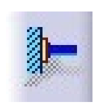
g. Enter a thickness of 2 mm



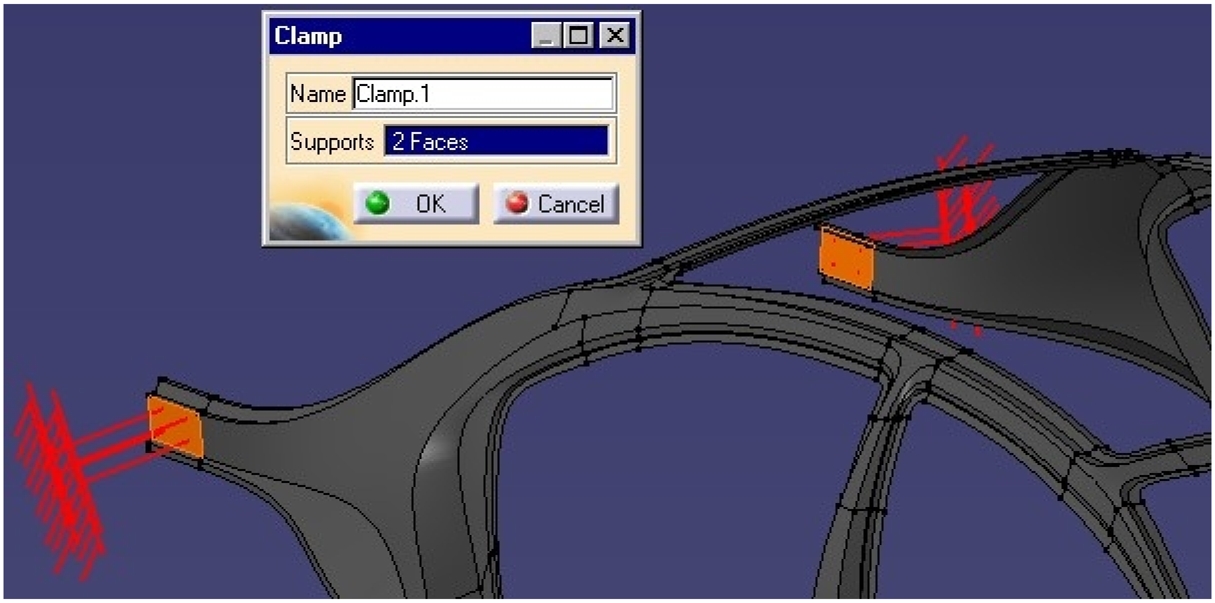
h. Click OK



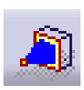
i. Click on Clamp icon



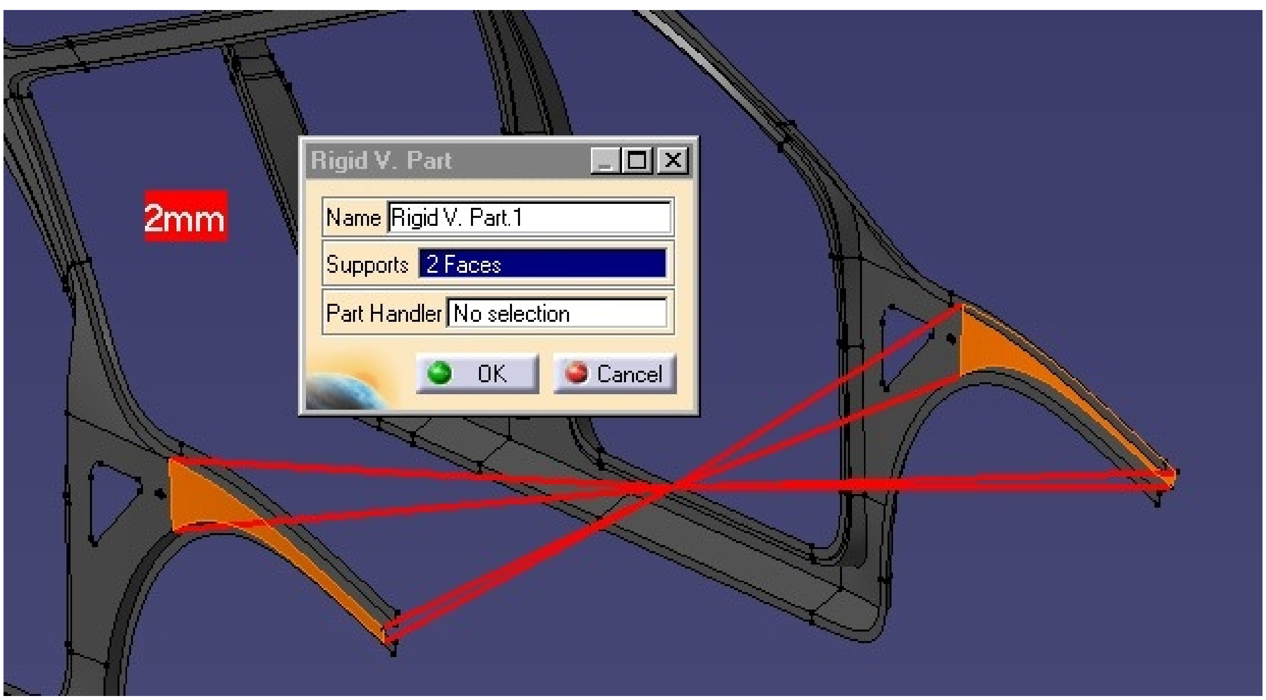
j. Select the both back faces and click OK.



k. Click on **Rigid Virtual Part** icon

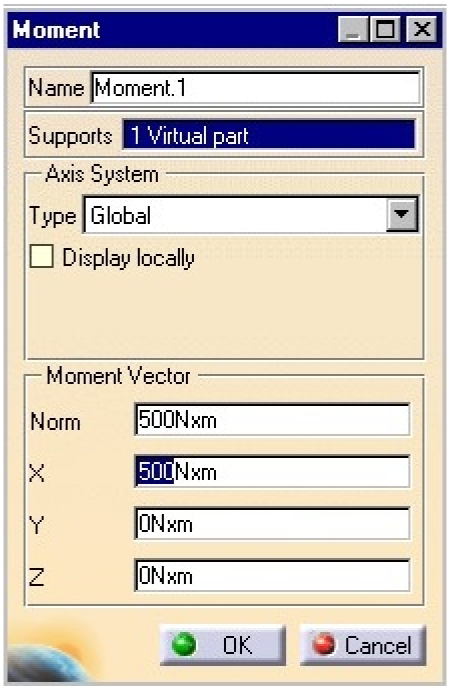


l. Select the both front faces and click OK.

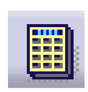


m. Click on **Moment** icon

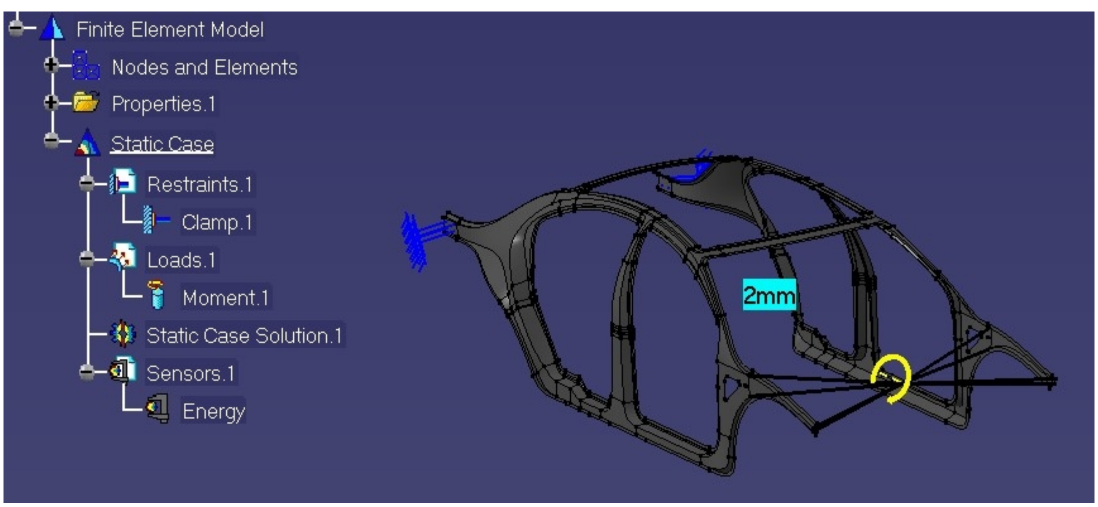
n. Select the rigid Virtual Part like Support, apply a moment of 500 N.m and click OK.



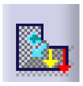
o. Click on **Compute** icon, select All and click OK.



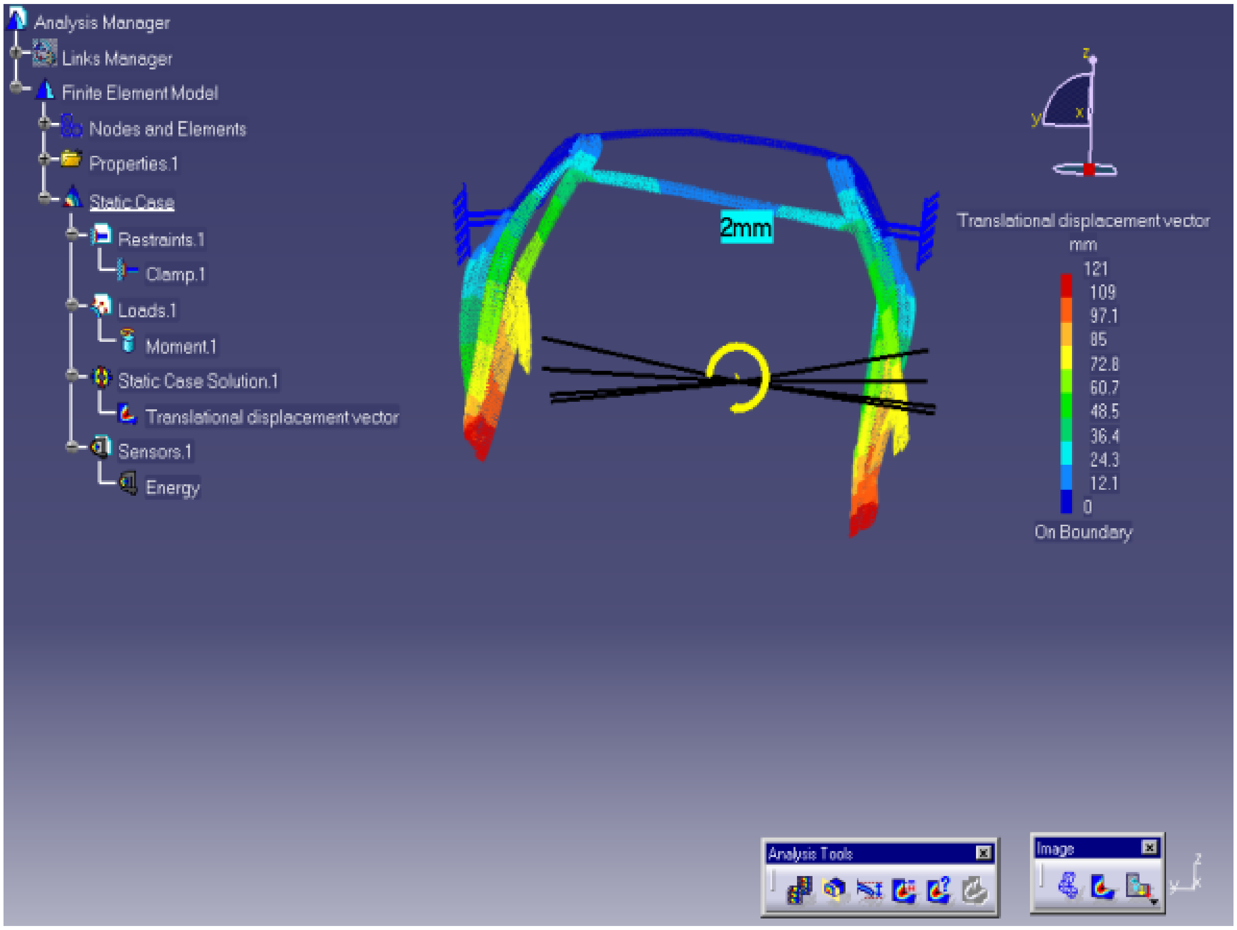
- The result should be like this



3. Post-processing Analysis
   a. Click on **Displacement** icon



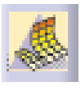
- The result should be like this



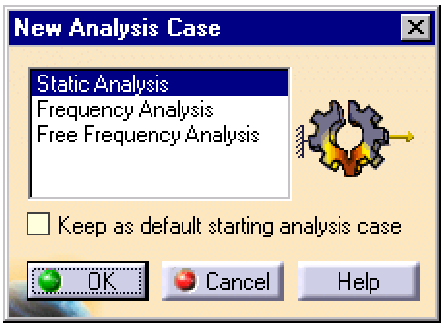
Additional Exercise

Step (1): Floor Mesh

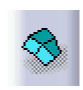
1. Open the “FEM Surface” workbench.
    a. Click Advanced Meshing Tools icon.



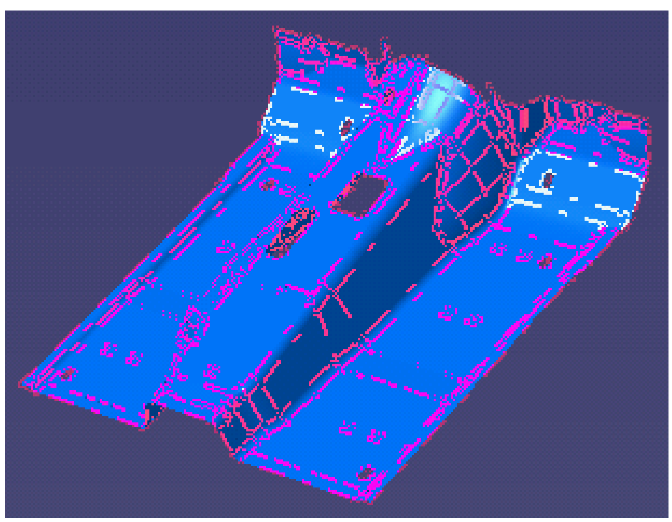
    b. Select **Static Analysis**.



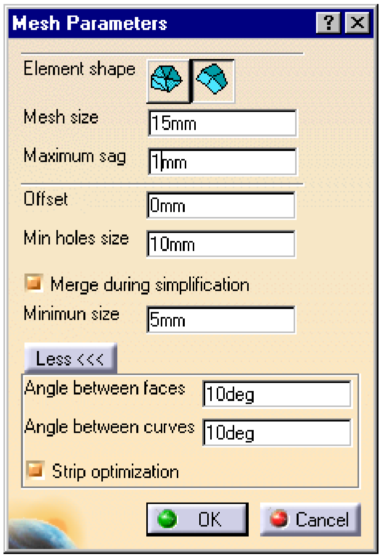
2. Set global mesh parameters values.
    a. Click Surface Mesher icon.



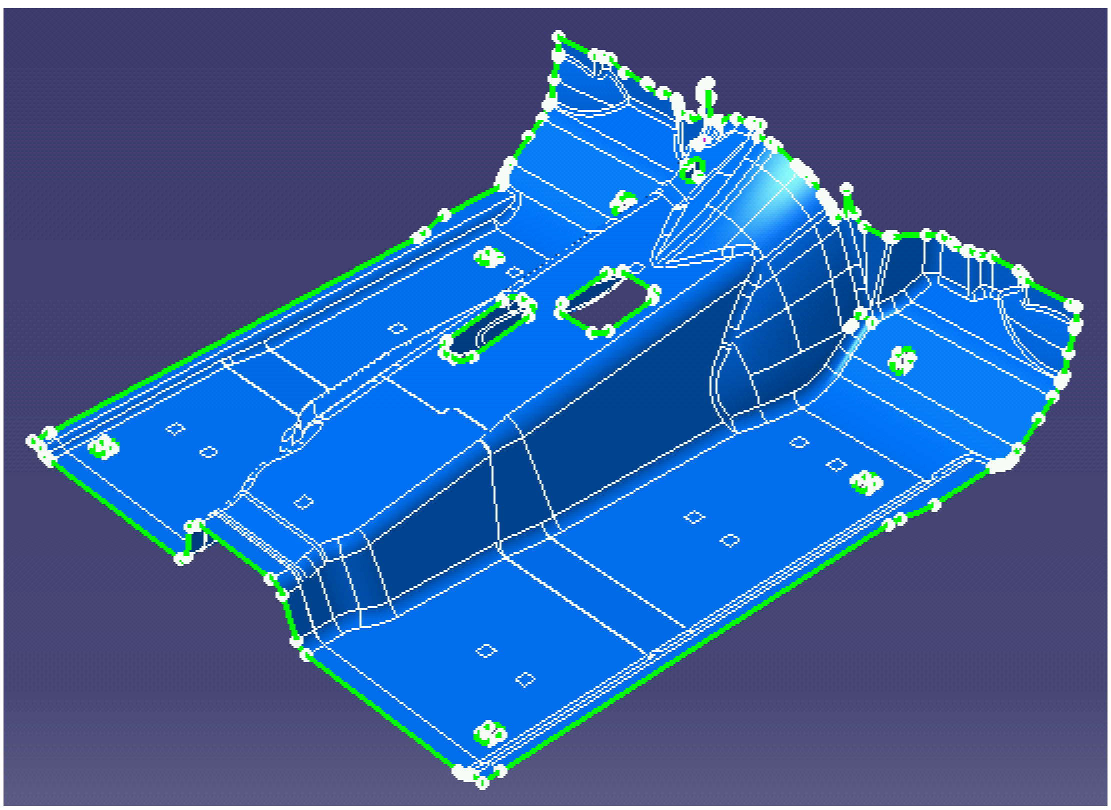
    b. Select geometry.



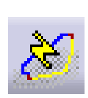
    c. Set global mesh parameters values.



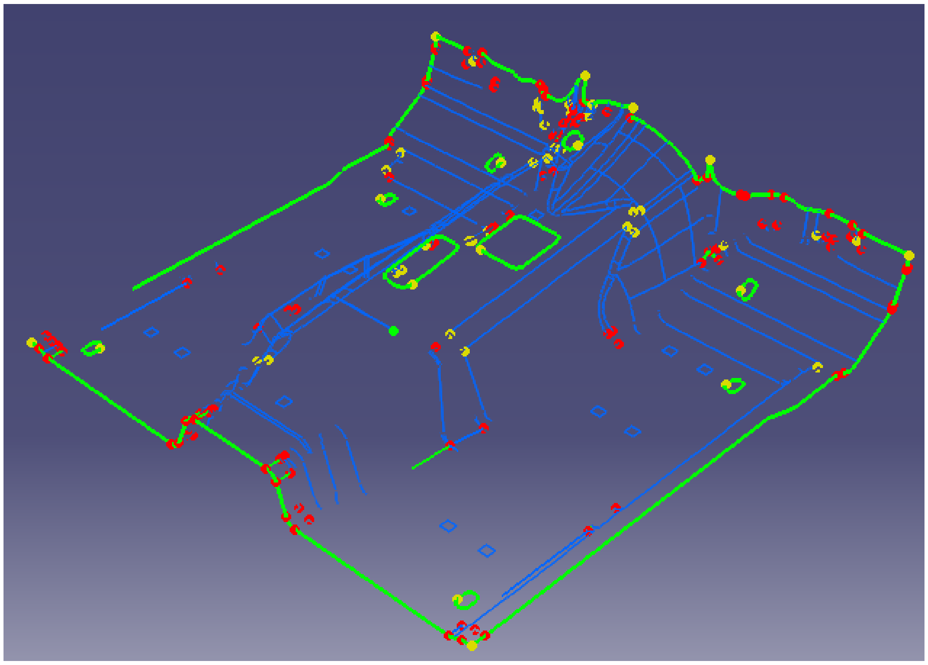
d. Click OK.



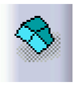
3. Geometry Simplification and mesh.
   a. Click Geometry Simplification icon.

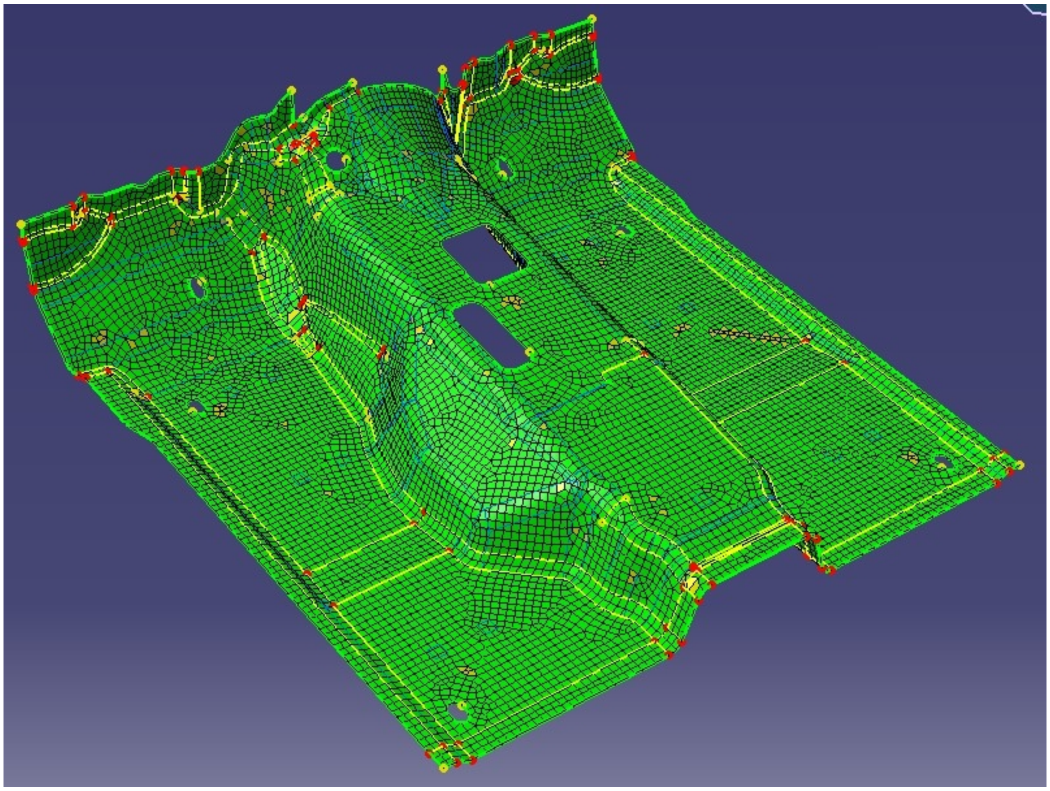


b. Simplification is made.

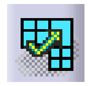


c. Click Mesh the Part icon.

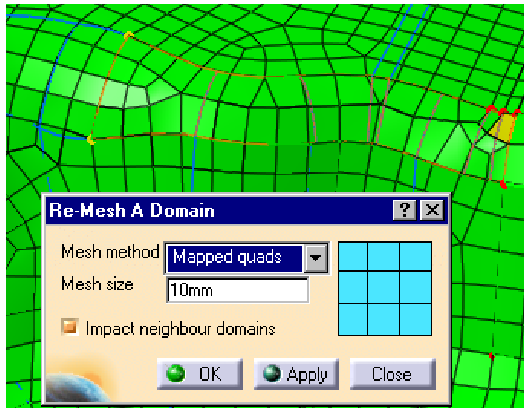




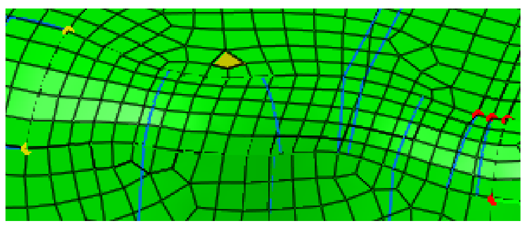
4. Mesh failure problem, solution 1: mapped quads local mesh
   a. Click Re-Mesh a Domain icon



   b. Set new mesh parameters (mapped quads method, 10 mm mesh size)

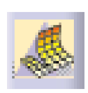


   c. Click “OK”.

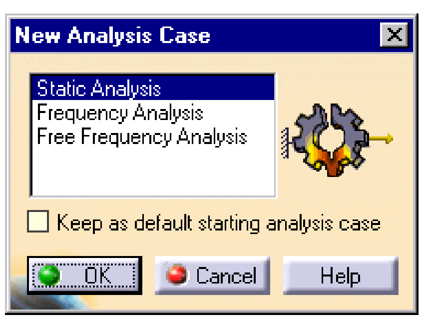


Step (2): Fuselage Door Static Analysis

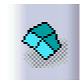
1. Open the “FEM Surface” workbench.
   a. Click Advanced Meshing Tools icon.



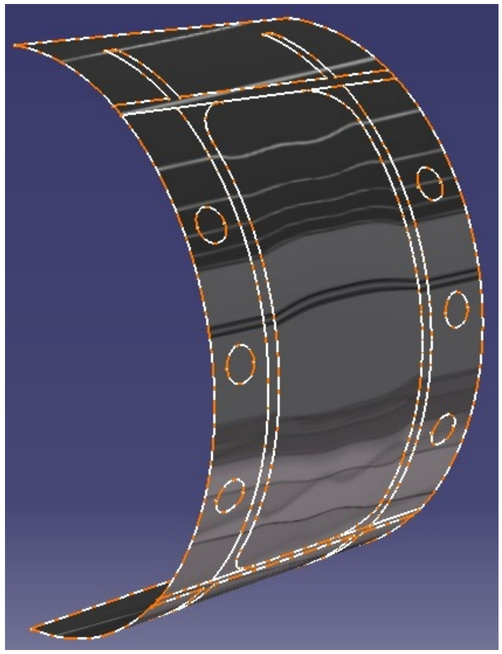
   b. Select **Static Analysis**.



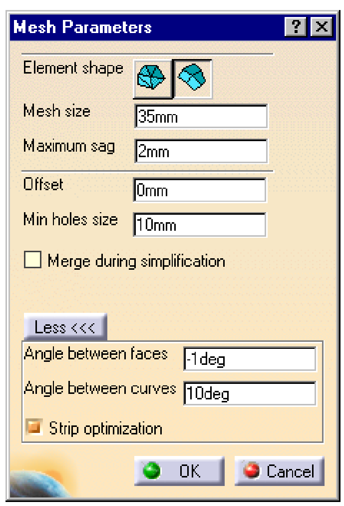
2. Set global mesh parameters values.
   a. Click Surface Mesher icon.



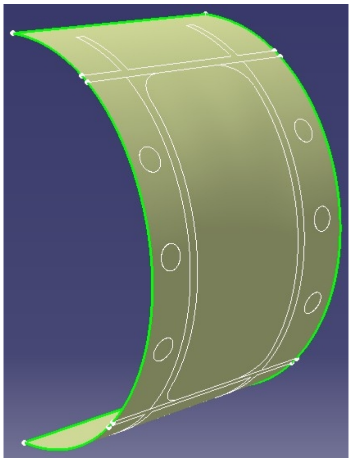
b. Select geometry.



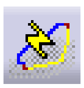
c. Set global mesh parameters values.

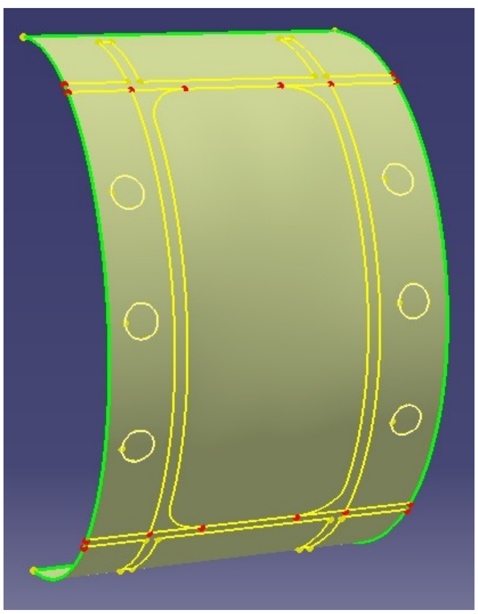


d. Click OK.

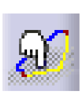


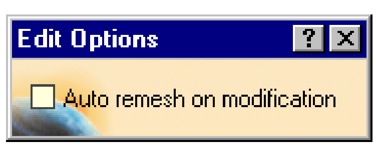
3. Mesh creating
   a. Click Geometry Simplification icon.



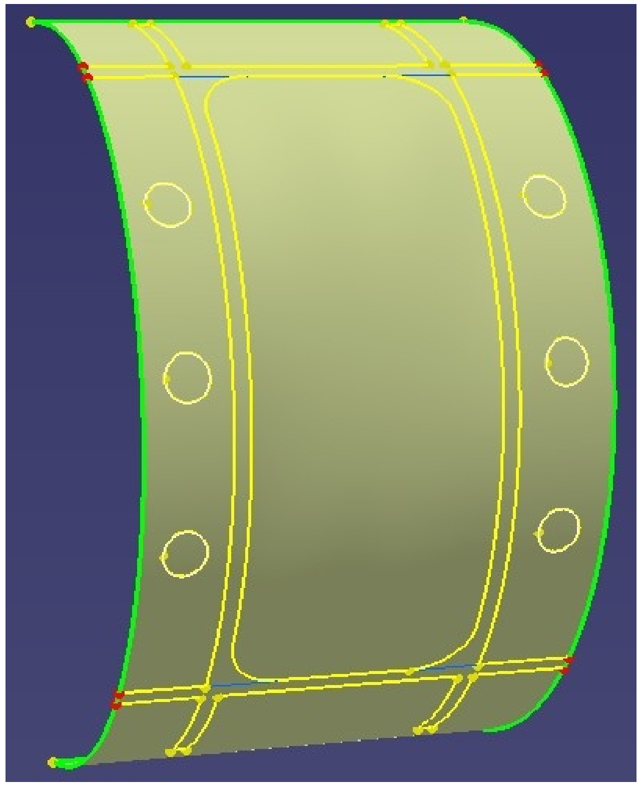


   b. Click on **Manual Simplifications** icon

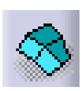




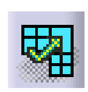
   c. Select the edges to remove



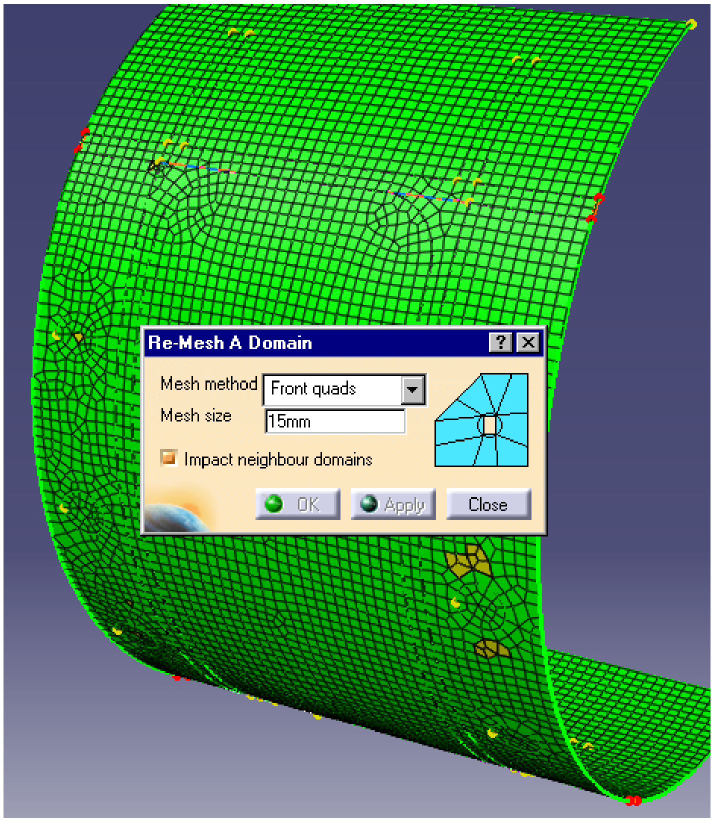
d. Click **Mesh the Part** icon



e. Click **Re-Mesh a Domain** icon.

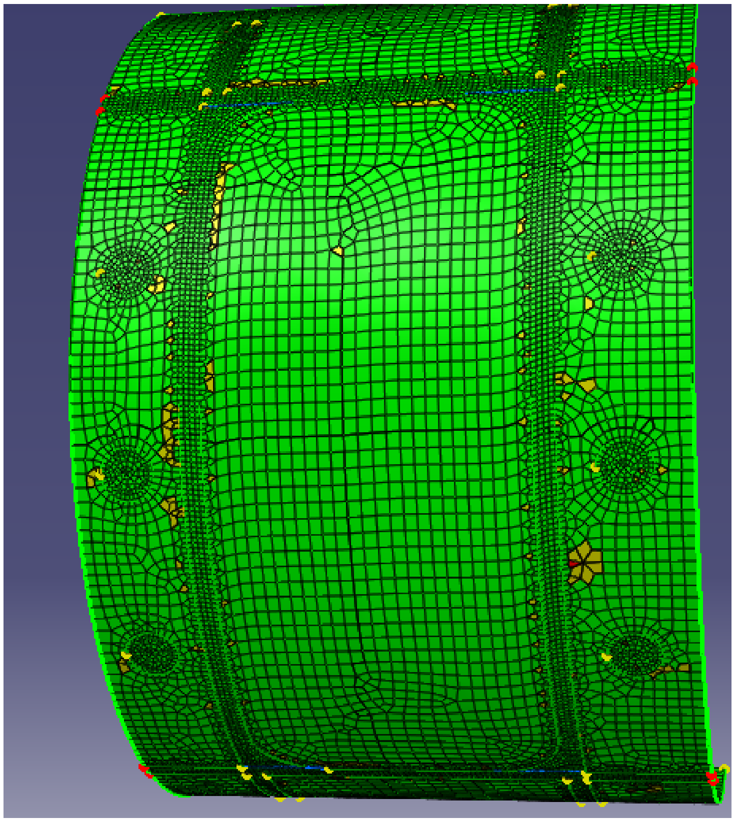


f. Enter the new mesh size



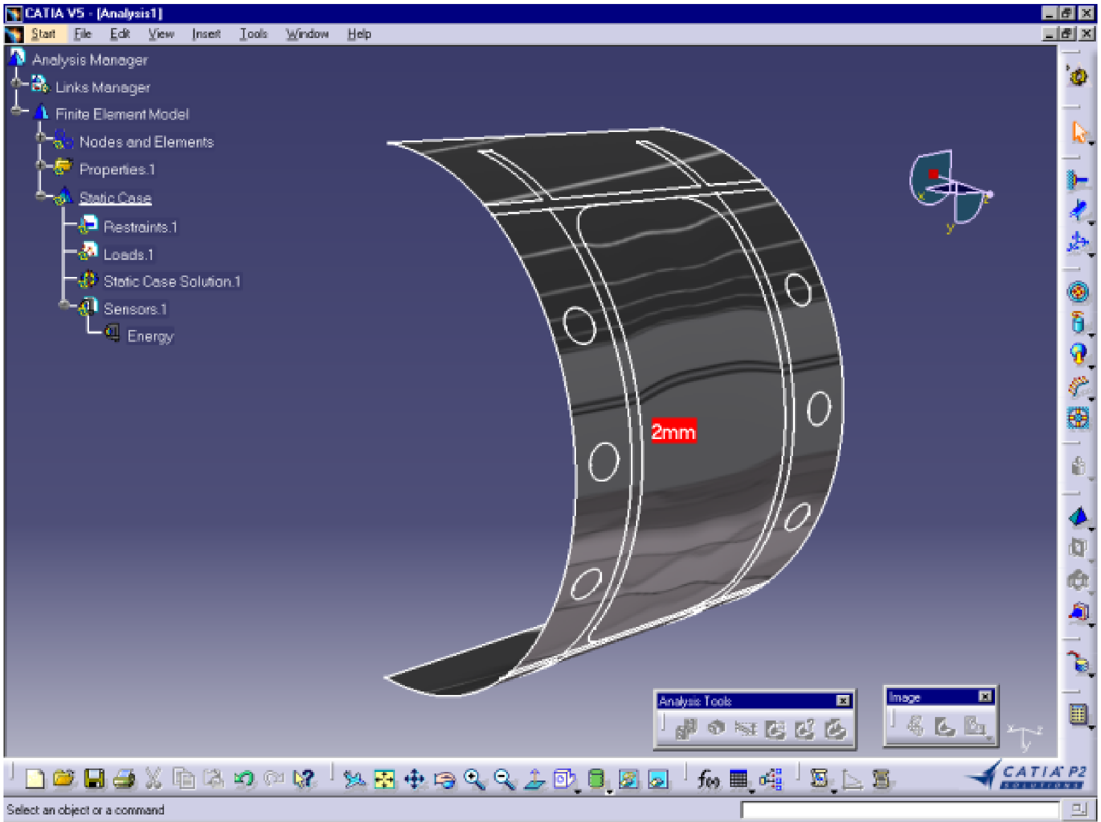
g. Select the surface and click Apply
h. Do the same thing with each hole

- The mesh should be like this

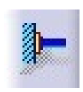


4. Perform a static analysis.
   a. Click Generative Structural Analysis.

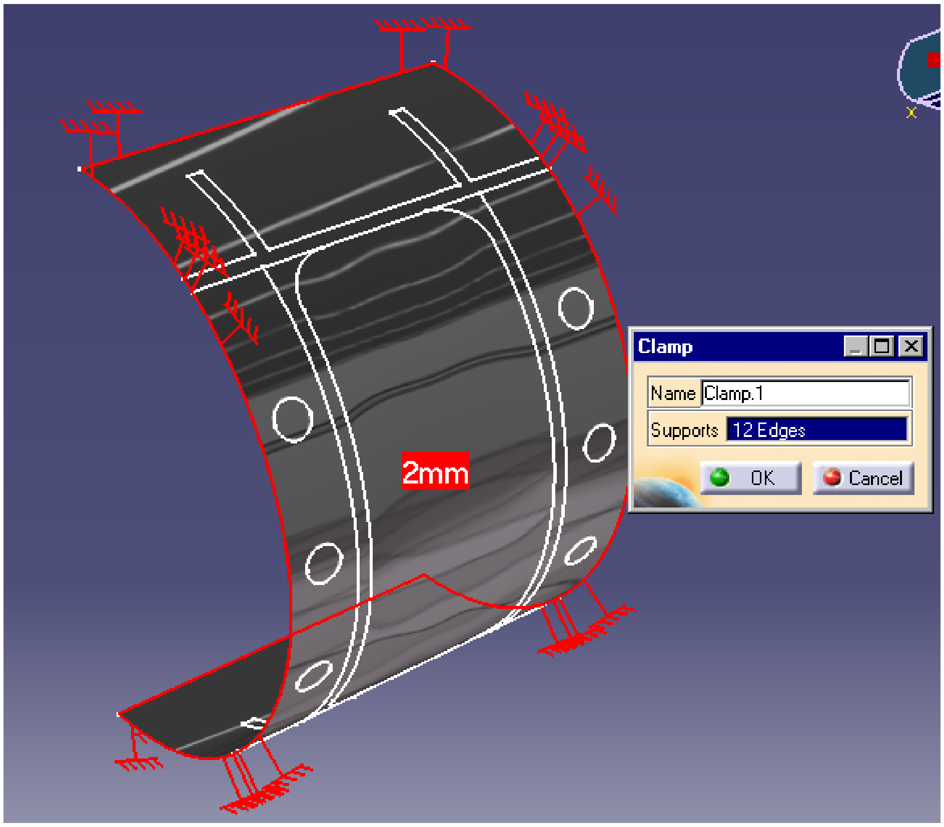
- Notice that the thick appears



b. Click **Clamp** icon

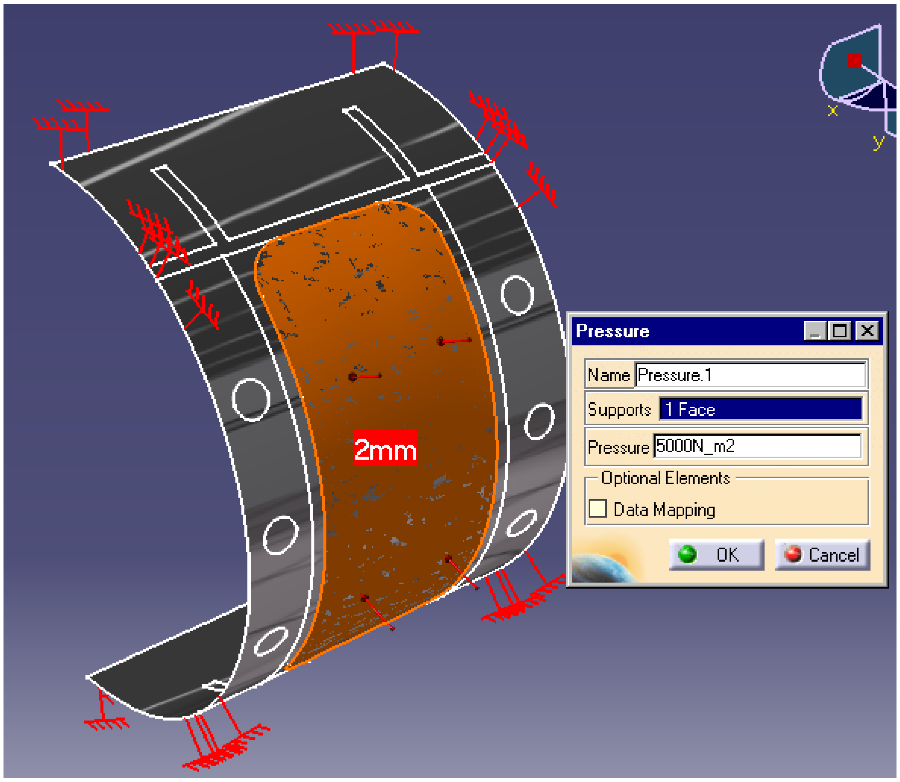


c. Select all the bend face

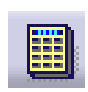


d. Click **Pressure** icon

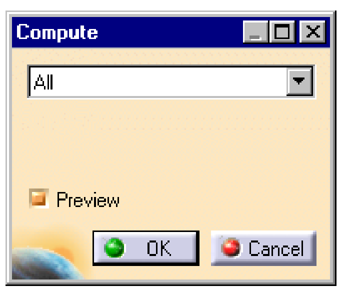
e. Select the door and enter the pressure value



f. Click OK.
g. Click on **Compute** icon

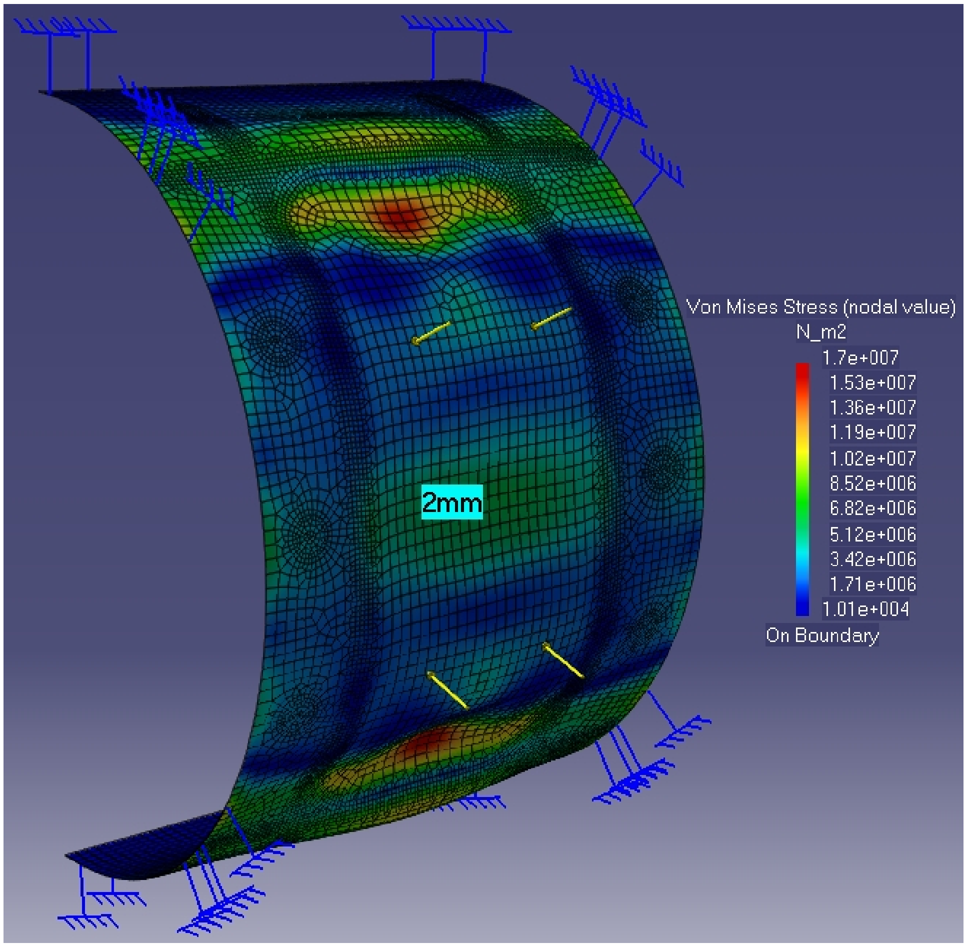


h. Select All and click OK



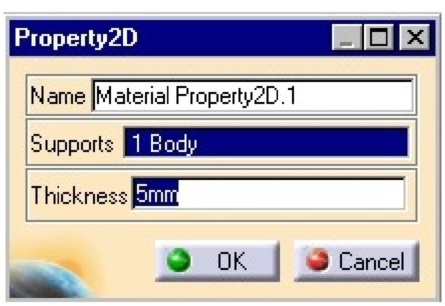
i. Click **Stress Von Mises** icon

- The result should be like this

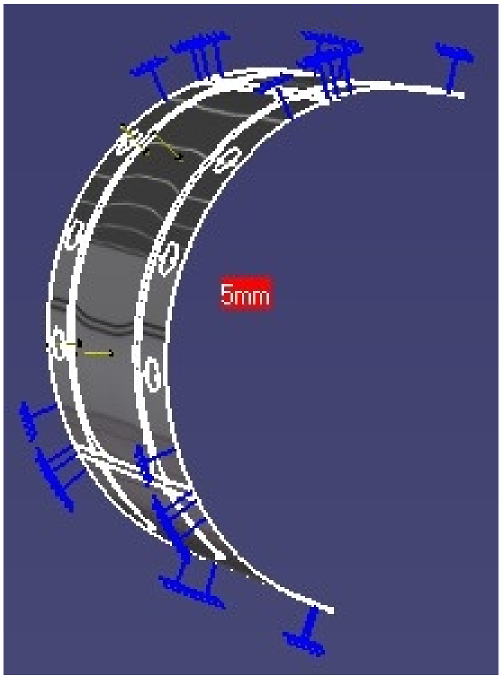


5. Thickness change of elements.

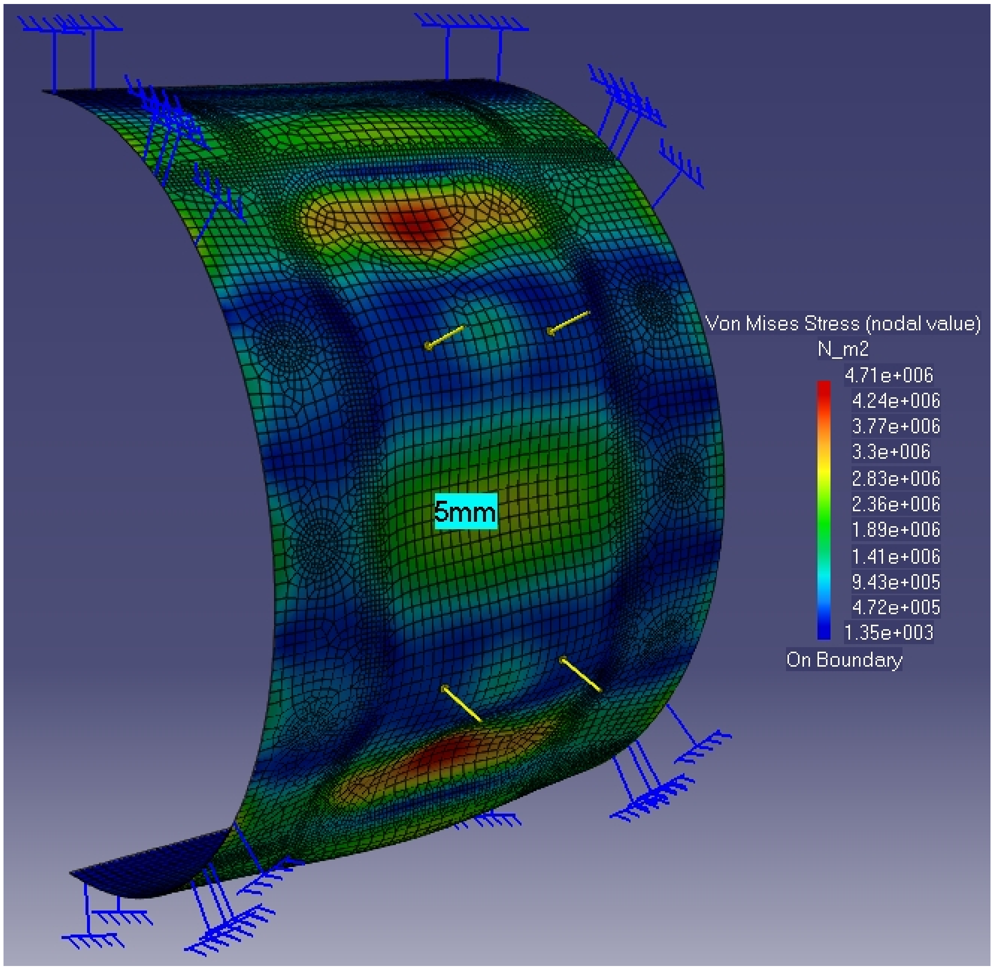
   a. Double-click on thickness and enter the new thickness 5mm



   b. Click OK



c. Recompute the structure.
d. Display the **Stress Von Mises**.


Von Mises Stress (nodal value)
N_m2
4.71e+006
4.24e+006
3.77e+006
3.3e+006
2.83e+006
2.36e+006
1.89e+006
1.41e+006
9.43e+005
4.72e+005
1.35e+003
On Boundary
5mm