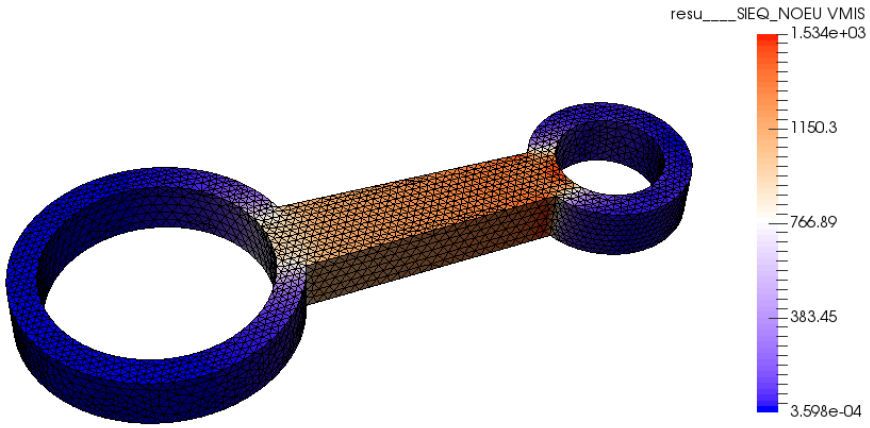
Salome Meca & Code_Aster

Static Non Linear analysis



1 Salome Meca....2
2 Geometry module....2
2.1 Importing the CAD-file....2
2.2 Modifying geometry....2
2.3 Partitioning the geometry....5
2.4 Creating groups....6
3 Mesh module....8
3.1 Meshing....8
3.2 Mesh Groups (node / element set)....11
4 Code_Aster....12
4.1 Adding a new stage to the case....12
4.2 Reading Mesh....12
4.3 Defining and assigning material....13
4.3.1 Creating the plastic material....13
4.3.2 Defining Material....14
4.3.3 Assigning the material....14
4.4 Other functions....15
4.4.1 Displacement....15
4.4.2 Time Step....16
4.5 Boundary....17
4.5.1 Fixed boundary....17
4.5.2 Moving the B-surf....17
4.6 Static non linear....18
4.7 Selecting what to be calculated Post Processing....18
4.7.1 Fields....19
4.8 Output....19
4.9 Running the analysis....20
5 Viewing results (Post processing)....21
5.1 Opening the result....21
5.2 Viewing results....21

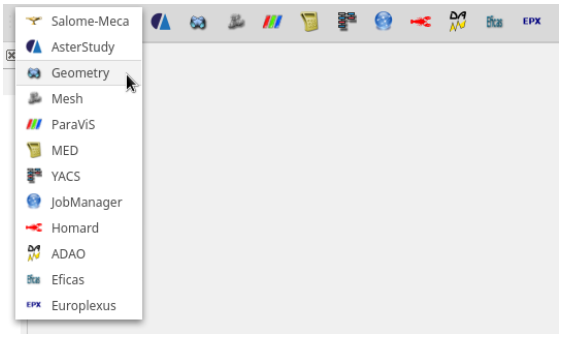
1 Salome Meca

1. Open Salome Meca
2. Create new document
3. Save study as Study1 in a folder

2 Geometry module

2.1 Importing the CAD-file

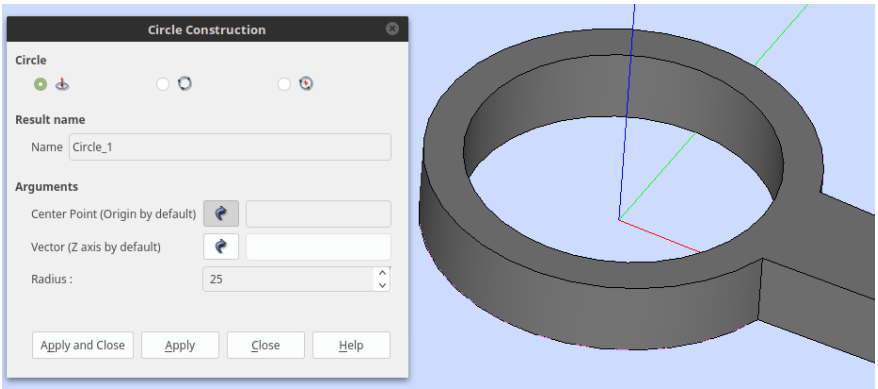
1. Open the geometry module.



2. Import the CAD.step (File→ Import → STEP ...)

2.2 Modifying geometry

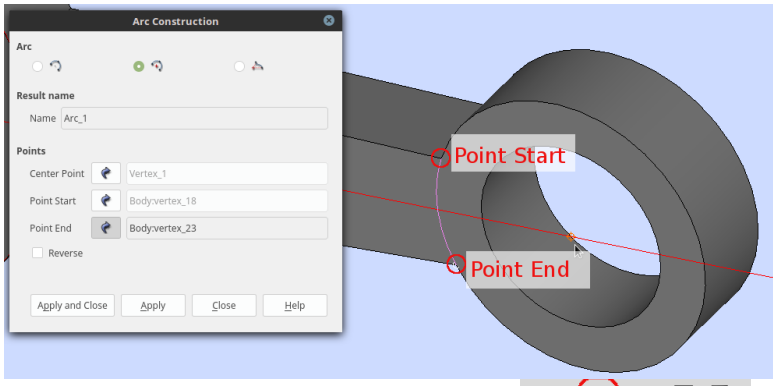
1. Create a circle with Radius = 25



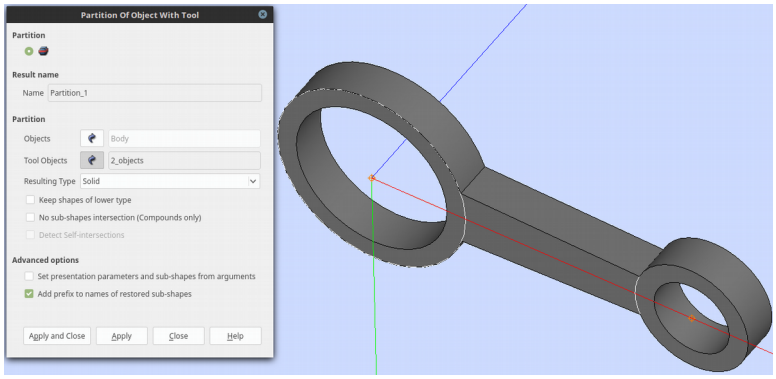
If not specified, the GUI will assume origin as center point and z-axis as vector.

2. Create a point with coordinates x=85, y=0, z=0

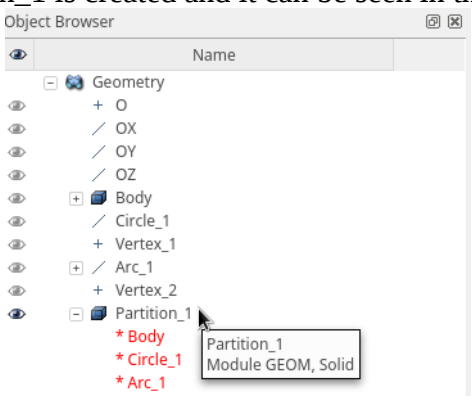
3. Create an arc with Vertex_1, created previously, as center point and point start and point end as shown in figure below.



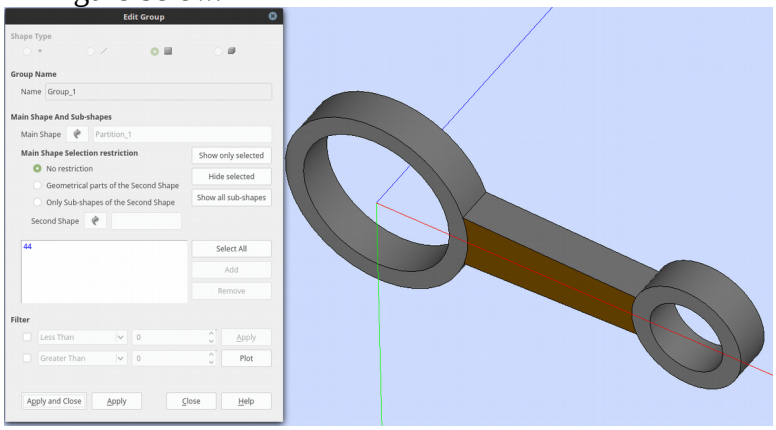
4. Partition the body with circle and arc created previously . Resulting type should be Solid.



A new object called Partition_1 is created and it can be seen in the Object Browser

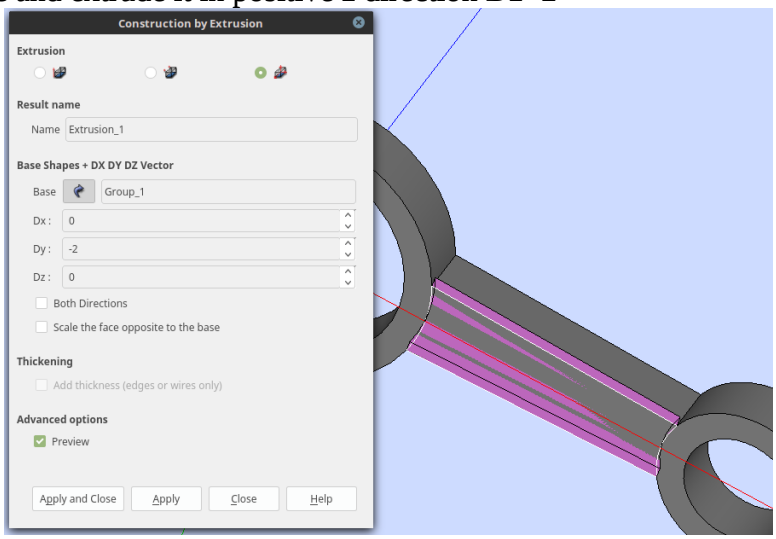


5. Create a new group by selecting New Entity → Group → Create Group and select the surface as shown in figure below.

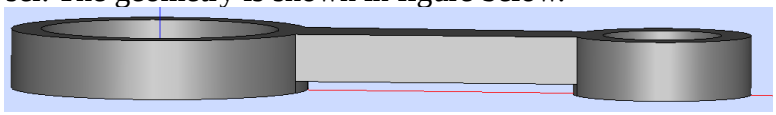


Select the Partition_1 as main shape, then select the face and press Add. Apply and Close

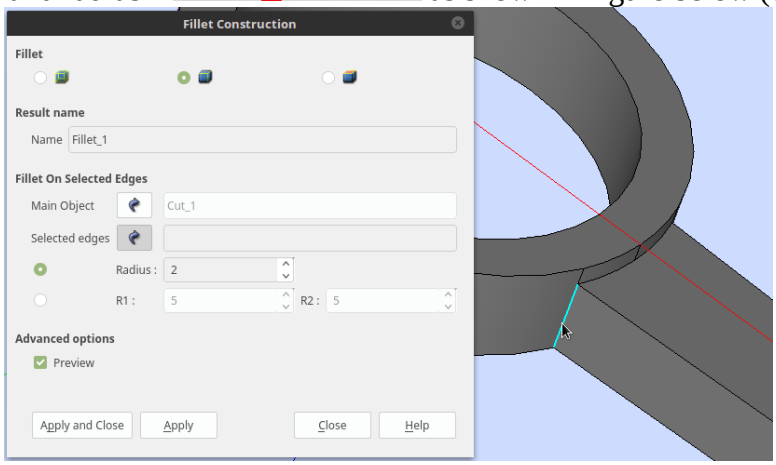
6. Extrude the surface as shown in figure below . Select the surface or Group_1 as base and extrude it in positive z direction Dz=2



7. Cut the main shape (Partition_1) with the new extrusion . Main Object = Partition, Tool Objects = Extrusion_1. A new object, Cut_1, is created and can bee seen in the Object Browser. The geometry is shown in figure below.

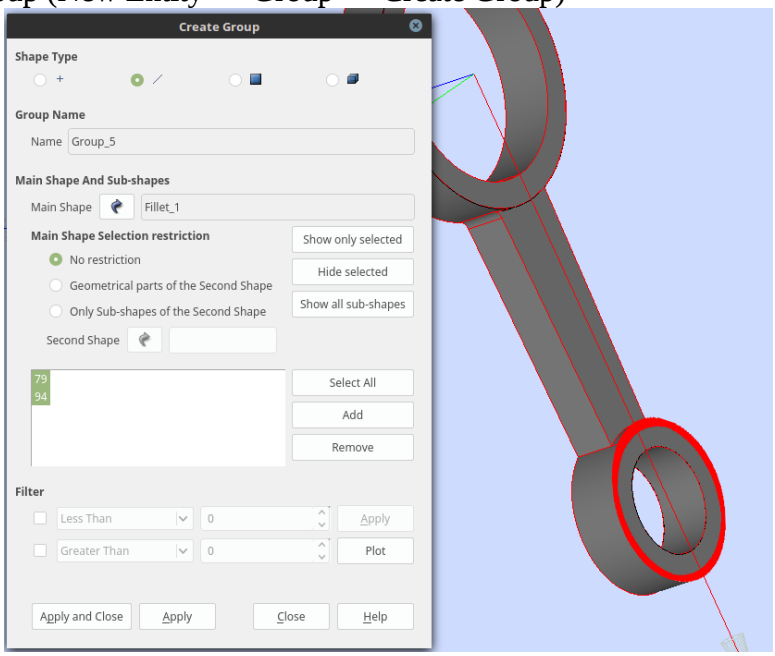


8. Create a fillet with a radius 2 as shown in figure below (on each side)

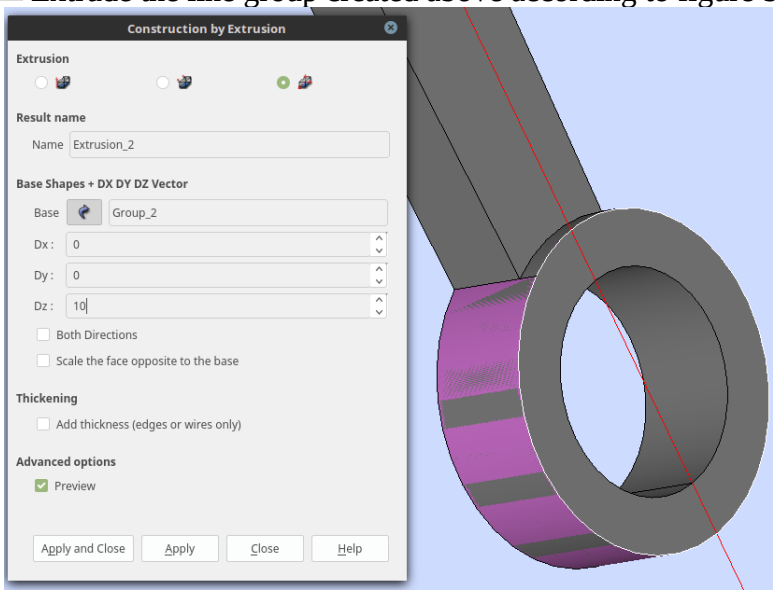


2.3 Partitioning the geometry

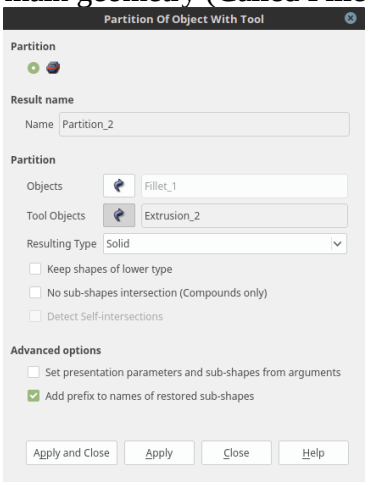
1. Create a line group (New Entity → Group → Create Group)



2. Extrude the line group created above according to figure below

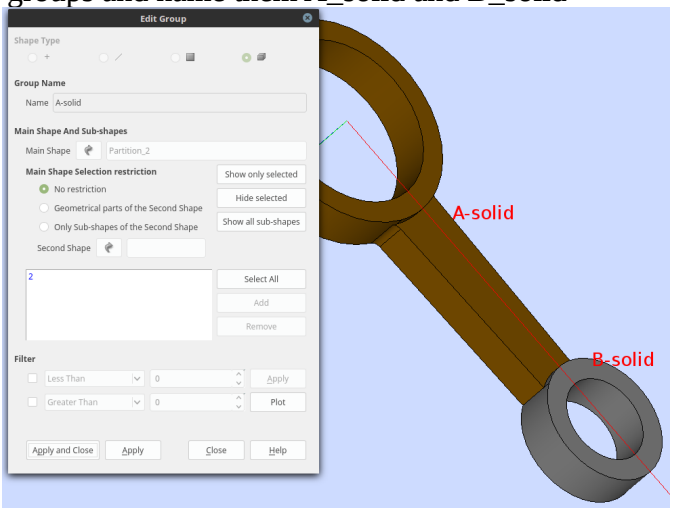


3. Partition the main geometry (Called Fillet_1) with the extruded surface.

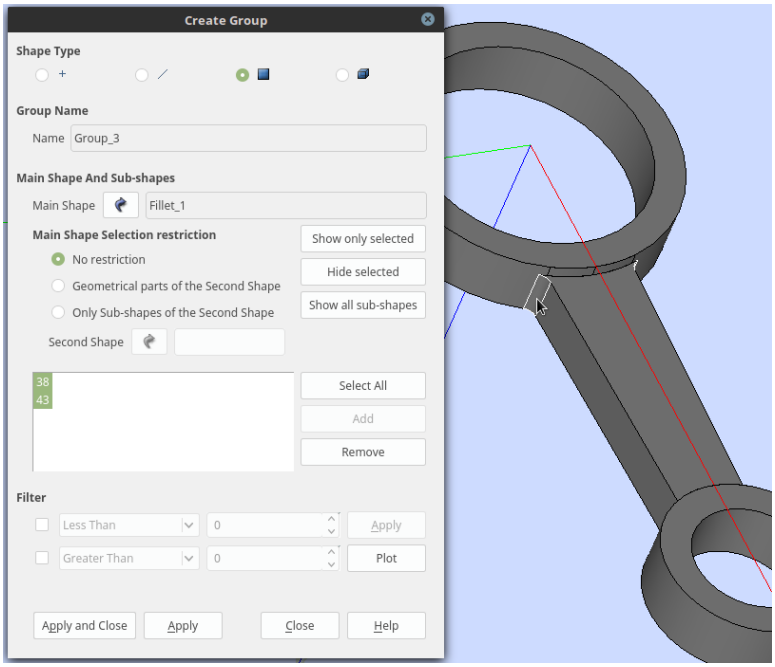


2.4 Creating groups

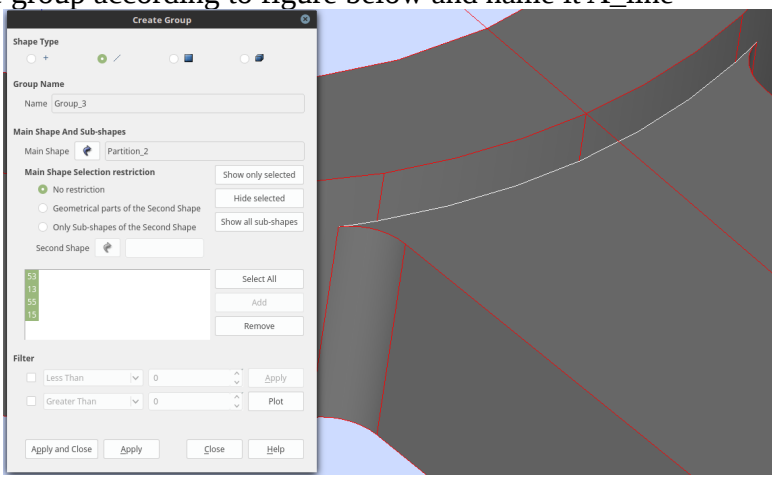
1. Create two solid groups and name them A_solid and B_solid



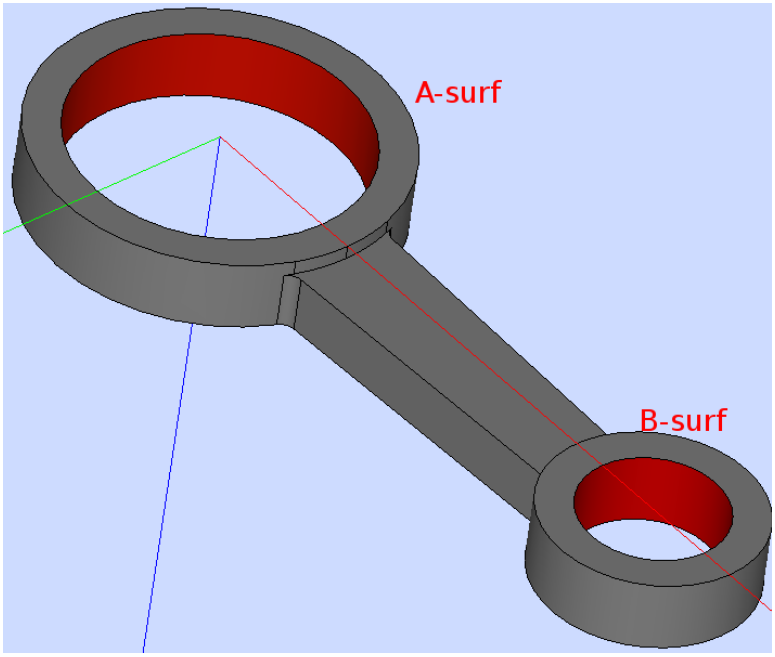
2. Create one surface groups of the two fillet surfaces and name the group Fillet_surf



3. Create a line group according to figure below and name it A_line



4. Create two surface groups according to figure below.

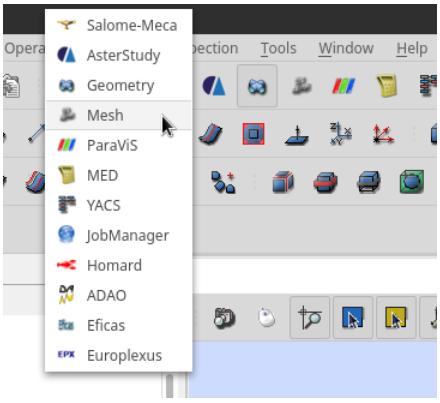


5. Rename the main geometry, Partition_2, to Connecting_rod

3 Mesh module

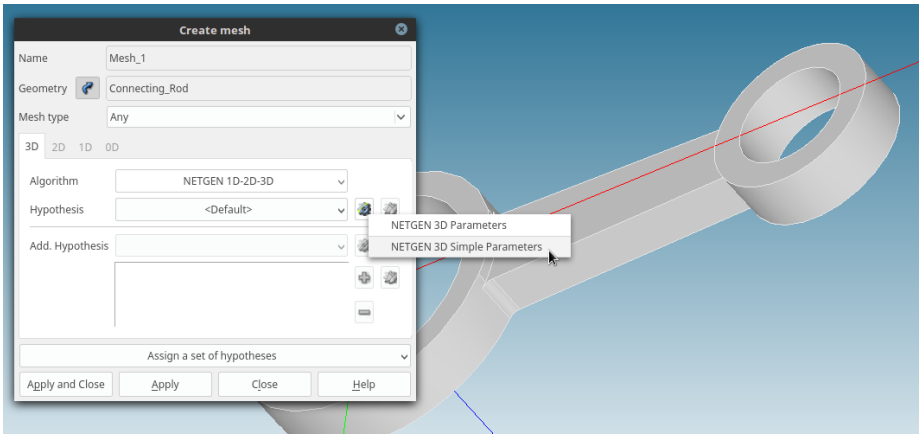
3.1 Meshing

1. Open the mesh module

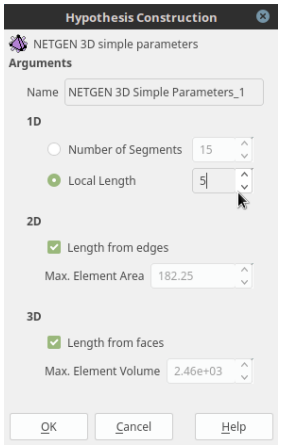


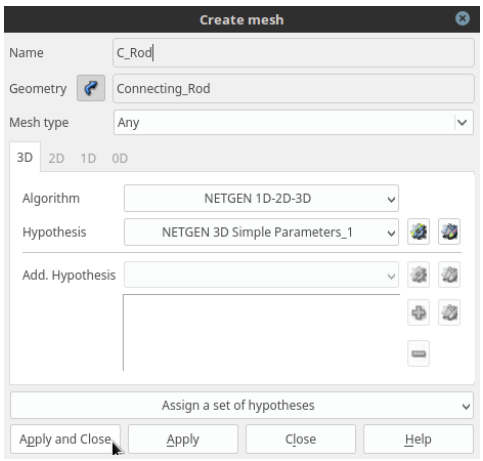
Notice that no geometry is visible in the main window. The geometry can be displayed by making them visible in the Object Browser.

2. Create a mesh selecting the Connecting_Rod as Geometry. The Algorithm should be NETGEN 1D-2D-3D. Under Hypothesis select NETGEN 3D Simple Parameters.

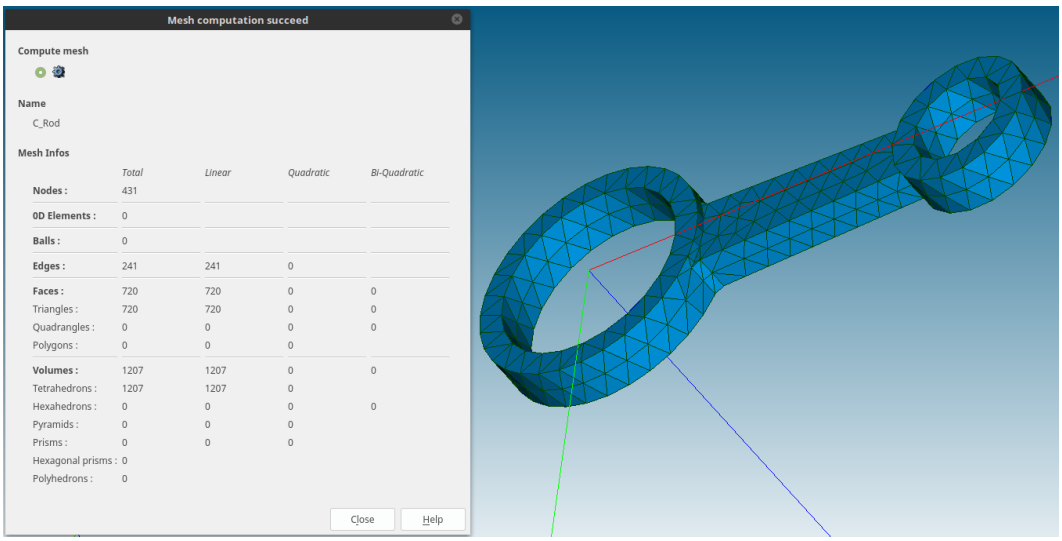


3. In the Hypothesis Construction window change Local Length to 5 and click OK. In the Create mesh window change mesh name to C_Rod and click Apply and Close

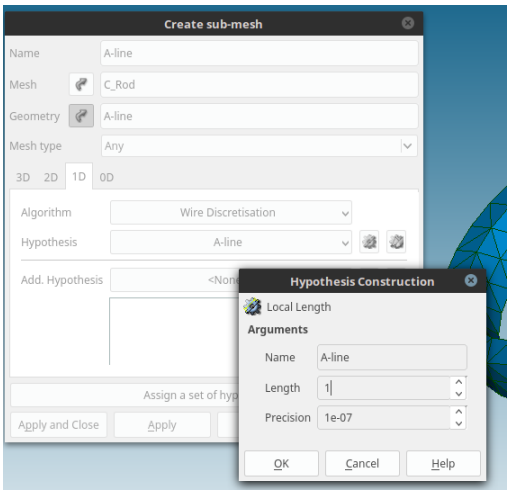


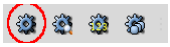


4. Press compute to build the mesh. The result should look like the figure below.

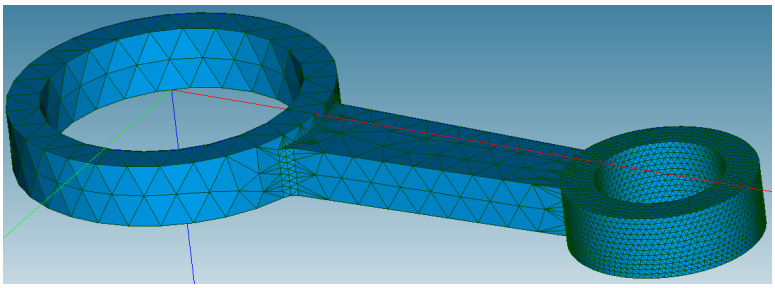


5. Create a sub mesh and call it A-line. Select the Mesh C_Rod and Geometry A_line from the Object Browser. Select Wire Discretisation as Algorithm and choose Local Length for the Hypothesis. Change Length to 1

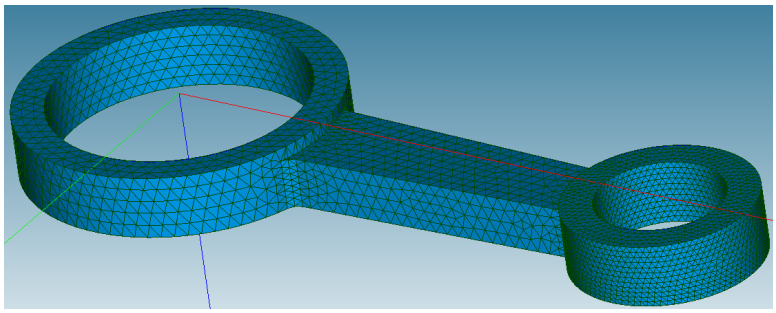


6. Select the C_Rod in the Object Browser and press compute  . The mesh is now finer at the A_line.

7. Create two new sub for the B_solid and Fillet_surf with Local length set to 1. Select C_Rod and press compute in order to build the mesh.

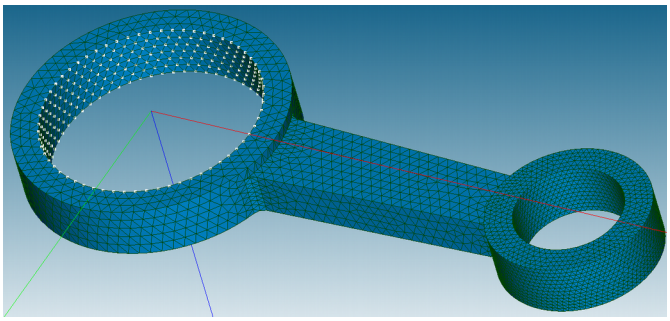


8. In order to change the mesh right click on C_Rod in the Object Browser and select Edit mesh. Edit the Local Length to 2 under Hypothesis. Re-compute the mesh.

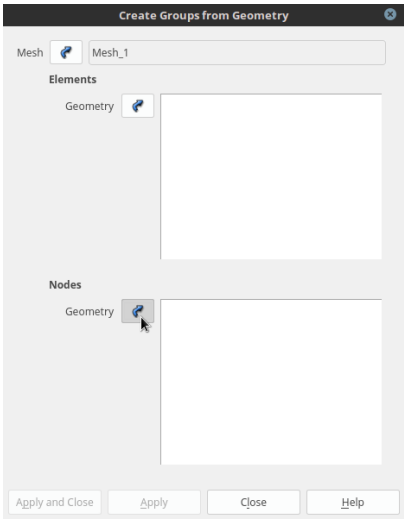


3.2 Mesh Groups (node / element set)

1. Create Node group, (Name = A_surf), by selecting the C_Rod as Mesh, Element Type as Node and Group type as Group on Geometry and select A_surf with the in the Geometrical Object.



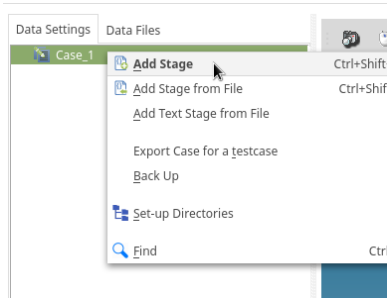
2. Create a Node group for the B_surf by selecting Create Groups by Geometry. . Select the Geometry button for Nodes, as shown below, and select the B_Surf group created in the geometry module in the Object Browser.



4 Code_Aster

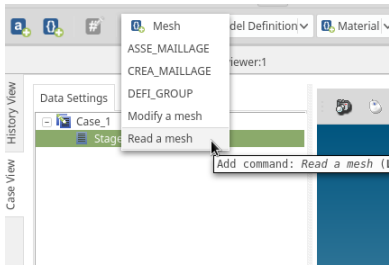
4.1 Adding a new stage to the case

1. Open the Code_Aster module
2. Right click at the Case_1 under Data Settings and select Add Stage.

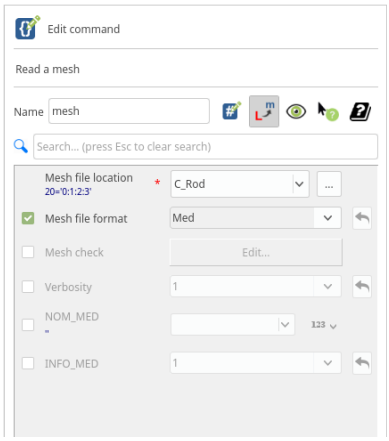


4.2 Reading Mesh

1. Select Stage_1 and click on the category Mesh and select Read Mesh. A Edit command window will show up to the right of the main window.



2. In the Edit command window select C_Rod in the Mesh file location and set Med ash Mesh file format. The mesh will appear in the main window.



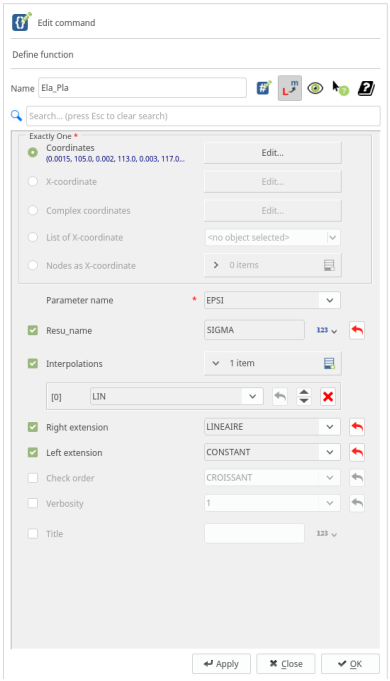
3. Assign the mesh to a model by selecting Assign finite element in the Model Definition category. In the command editor select finite element assignment and add a new item . Press the Edit button and select Everywhere = Yes. Select Phenomenon as Mechanic. Under Modelisation add a item and select 3D. In the information window the code should be:

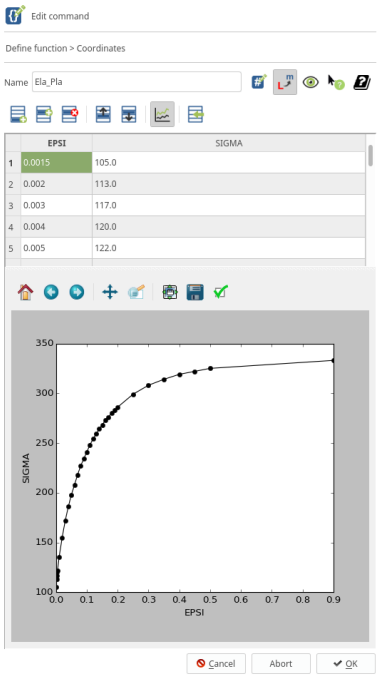
```
model = AFFE_MODELE(
  AFFE=_F(
    MODELISATION=('3D', ),
    PHENOMENE='MECANIQUE',
    TOUT='OUI'
  ),
  MAILLAGE=mesh
)
```

4.3 Defining and assigning material

4.3.1 Creating the plastic material.

1. Under category Functions ans Lists select Define function.
2. In the command editor change the name to Ela_Pla. Set Parameter name to EPSI. Change the Resu_name to SIGMA. Select the Interpolations and add a new item and set it to LIN. Change the Right extension to LINEAIRE and Left extension to CONSTANT.
3. Select Coordinates and press the Edit button. In the new window select import table and find the material.txt file. The EPSI and SIGMA values are now loaded in the table. It is also possible to write the values directly in the table.
4. Plot the values in order to inspect the curve .





4.3.2 Defining Material

1 Define a new material by clicking on the Material button.

2 In the command editor change the name to Alu.

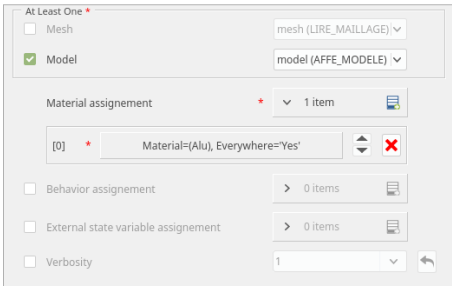
3 Select the Linear isotropic elasticity and press Edit

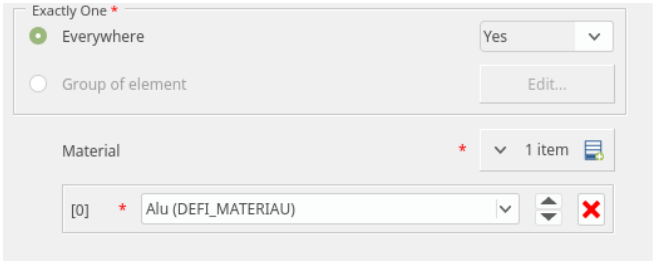
3.1 Set Young's modulus to 66000 MPa and Poisson's ratio to 0.35.

4 Find TRACTION in the list or search in the search bar. Select TRACTION and press Edit. Select the Ela_Pla funcion

4.3.3 Assigning the material

1. Select category Material and click on the Assign a material.
2. Select Model and chose the model created earlier.
3. In Material assignment add a new item and press Edit.
4. Select Everywhere = Yes
5. Add a new item for the Material and select the Alu material created earlier.

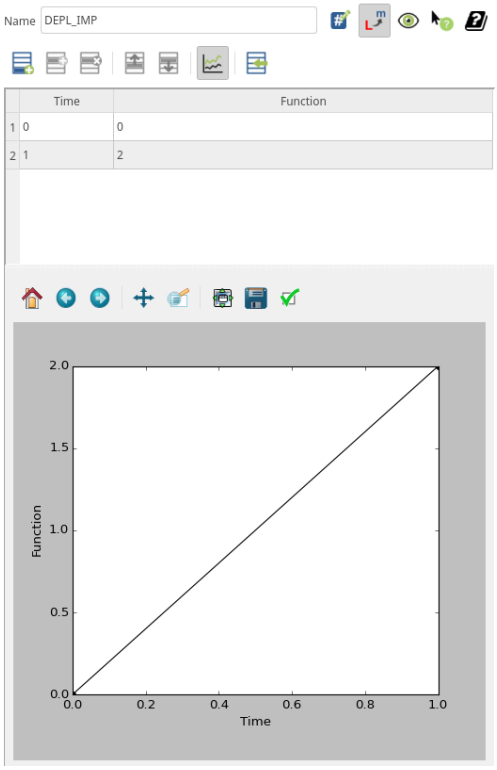

At Least One *
Mesh
mesh (LIRE_MAILLAGE)
Model
model (AFFE_MODELE)
Material assignement
*
1 item
[0]
*
Material=(Alu), Everywhere='Yes'
Behavior assignement
0 items
External state variable assignement
0 items
Verbosity
1


Exactly One *
Everywhere
Yes
Group of element
Edit...
Material
*
1 item
[0]
*
Alu (DEFI_MATERIAU)

4.4 Other functions

4.4.1 Displacement

1. Define a new function by selecting Define function in the Functions and Lists category.
2. Change the name to DEPL_IMP
3. Set Parameter name to Time (NOM_PARA=’INST’)
4. Select Coordinates and press the Edit button. Fill the table according to figure below.
5. *Select the Interpolations and add a new item* *and set it to LIN.*
6. *Change the Right extension to LINEAIRE and Left extension to CONSTANT.*



4.4.2 Time Step

1. Select DEFI_LIST_REEL from the Functions and Lists category.
2. Change the name to timestep
3. Select DEBUT and set value 0.0
4. Add a new item to the INTERVALLE and press edit.
5. Select PAS and set value 0.1 and set value 1.0 for JUSQU_A

4.5 Boundary

4.5.1 Fixed boundary

1. Select Assign mechanical load from the BC and Load category.
2. Change name to Fixed
3. Select Enforce DOF, add a new item , end press the Edit button.
4. Select Group of node and press the Edit button.
5. Select the A_surf from the Nodes list.
6. Set 0.0 to the DX, DY, and DZ. Press OK
7. Be sure that Model is selected.

4.5.2 Moving the B-surf

1. Select AFFE_CHAR_MECA_F form the BC and Load category.
2. Change the name to Move
3. Select Enforce DOF, add a new item , end press the Edit button.
4. Select Group of element and press the Edit button.
5. Select the B_surf from the Nodes list.
6. Select DX and from the drop down menu select DEPL_IMP
7. Be sure that Model is selected.

4.6 Static non linear

1. Select STAT_NON_LINE from the Analysis category
2. Change the name to **Result**
3. The Model should be model(AFFE_MODELE) and Material field fieldmat(AFFE_MATERIAU).
4. Add two new items to the Loads . And add the two boundaries created earlier.
5. Edit the Timestepping and select the timestep for the Time step list.

4.7 Selecting what to be calculated Post Processing

1. Change the name to **Result (Same name as above)**
2. Result should be Result(STAT_NON_LINE), Model = model(AFFE_MODELE), Material field = fieldman(AFFE_MATERIAU), TOUT_ORDE = Yes
3. Select CONTRAINTE and add following items;
   - SIGM_ELGA
   - SIGM_ELNO
   - SIGM_NOEU
   - SIEF_ELNO
   - SIEF_NOEU
4. Select DEFORMATION and add following items;
   - EPSI_ELGA
   - EPSI_ELNO
   - EPSI_NOEU
5. Select CRITERES and add following items;
   - SIEQ_ELGA
   - SIEQ_ELNO
   - SIEQ_NOEU

4.7.1 Fields

SIEF → Stress or force/moment [SIgma or EFfort]

SIGM → Principal stresses (stress in x,y,z,xy,yz,zx)

SIEQ → Equivalent stresses (vonMises, Tresca …)

ELGA → Field at gauss points

ELNO → Field at elements

NOEU → Field at nodes

DEPL: displacement

4.8 Output

1. Select Set output results from the Output category.
2. Create a new result file by pressing at the Result file location. Change name to results.rmed. Notice the ‘r’ in rmed.
3. Set Med as the Format
4. Select Results, add a new item and press the Edit button.
5. Select following; MAILLAGE = mesh(LIRE_MAILLAGE), RESULTAT = Result(CALC_CHAMP).
6. Select NOM_CHAM and add following items;

   [0] SIGM_ELGA

   [1] SIGM_ELNO

   [2] SIGM_NOEU

   [3] SIEF_ELGA

   [4] SIEF_ELNO

   [5] SIEF_NOEU

   [6] SIEQ_ELGA

   [7] SIEQ_ELNO

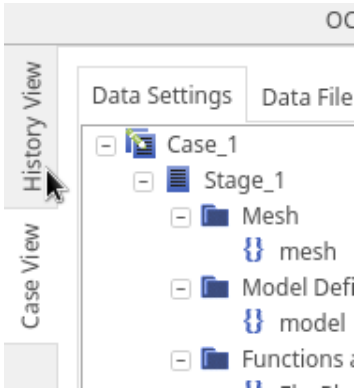
   [8] SIEQ_NOEU

   [9] FORC_NODA

   [10] DEPL

4.9 Running the analysis

1. Select the History View tab



2. Press the red button next to Stage_1.

3. Press the Run button.

And Run the analysis

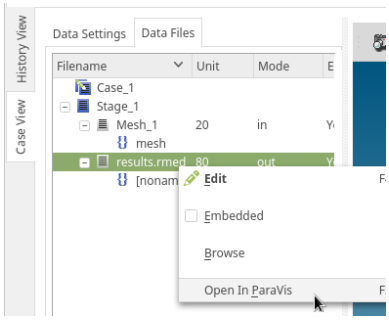
4. A case will be created and the simulation will run.

5. If successful the case will show

5 Viewing results (Post processing)

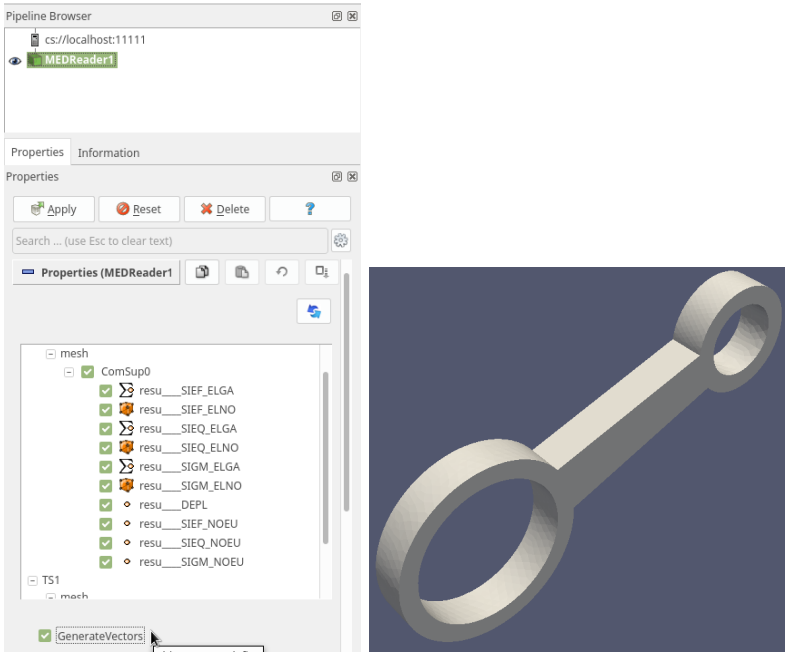
5.1 Opening the result

1. In AsterStudy go to the Case View tab.
2. Press Data Files tab.
3. Locate the results.rmed file and right click. Open In ParaVis

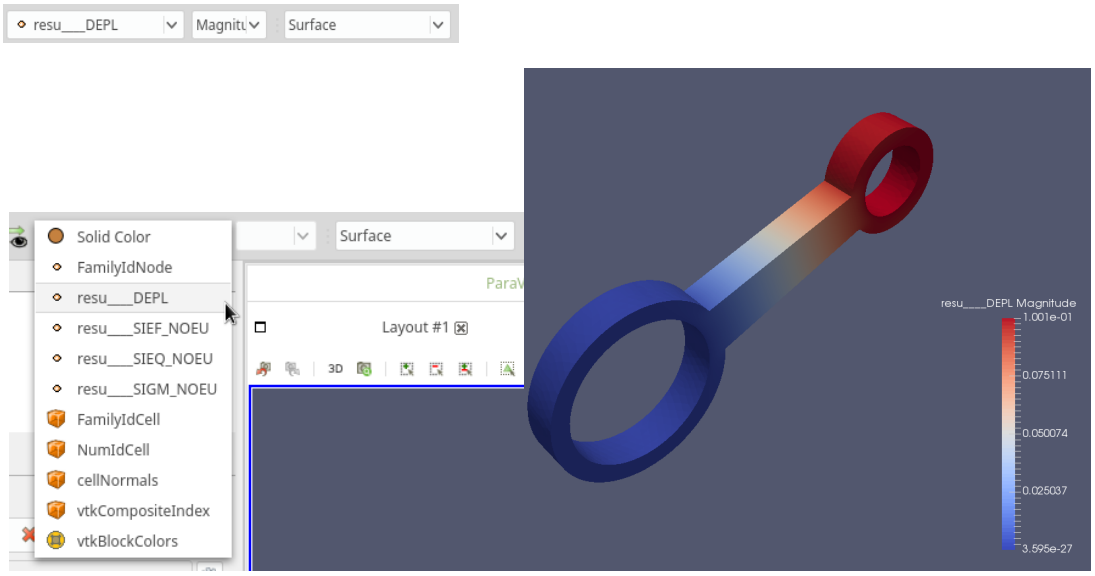


5.2 Viewing results

1. A file named MEDReader1 will appear in the Pipeline Browser. While selected, it is possible to view the fields calculated in the Properties window. Enable GenerateVectors and click on the Apply button.

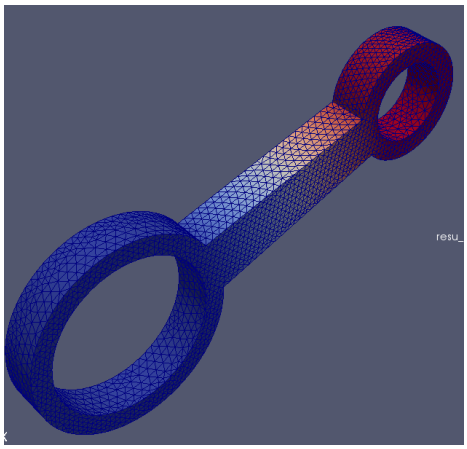


2. The MEDReader1 file will only display the results on nodes. In order to view a contour plot of the displacements select resu_DEPL from the pull down list. The displacements will be represented on the surface of the model with a magnitude.

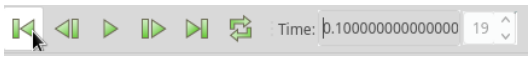


3. Plot the displacements in x direction by selecting Dx from the pull down list.

4. It is possible to view the contour plot with elements. Change Surface to Surface With Edges.



5. In order to view the results for a specific time step it is possible to toggle between the steps. Notice that the model does not show any deformation.



6. To view the model as deformed select the filter Wrap By Vector from Filters→ Common. Apply a desired Scale Factor from the Properties tab and click Apply. A new file called WrapByVector1 is created in the Pipeline Browser.

7. In order to plot stresses on elements select ELNO field to surface from Filters → Mechanics. Change the resu_DEPL to resu_SIEQ_ELNO to view for example the von Mises stresses. Repeat the wrap by vector filter in order to see a deformed model.