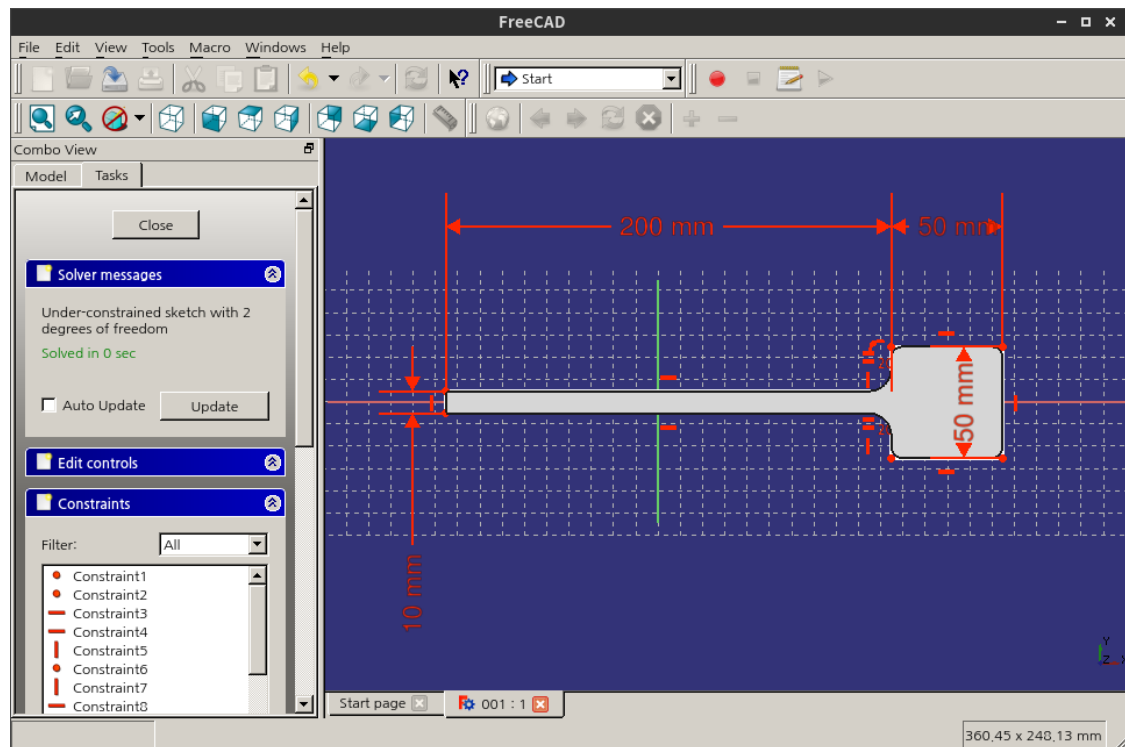


0301 : Gravity Condition

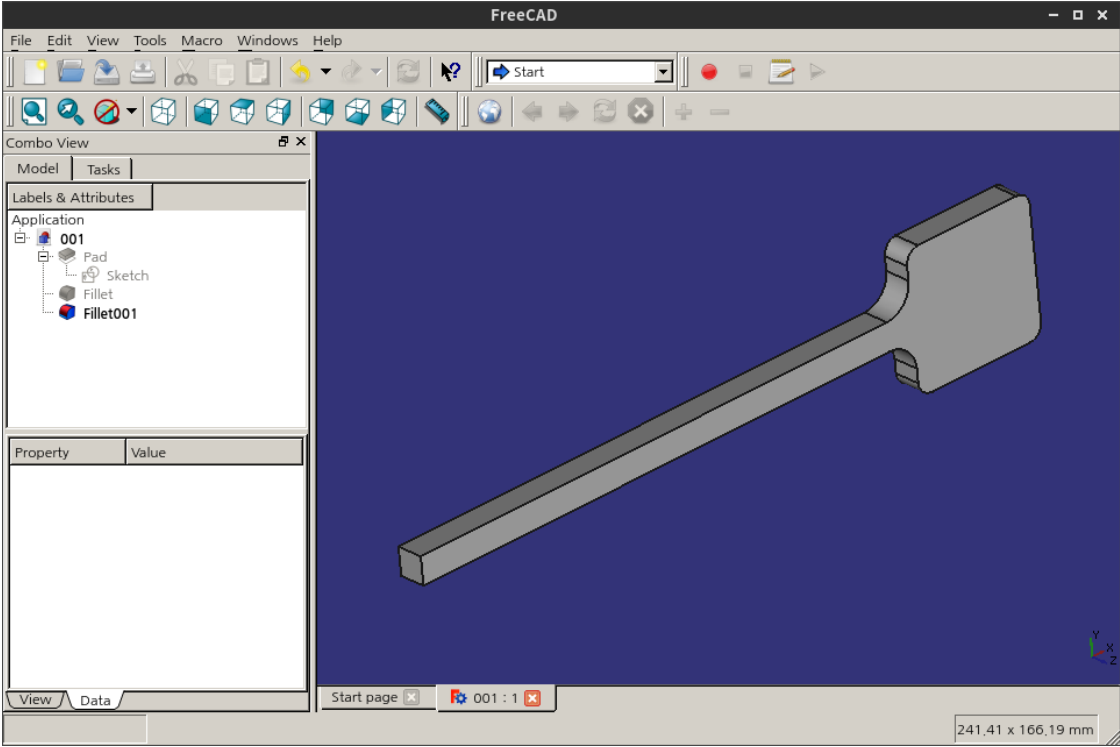
FreeCAD

- 3D Solid Modeling

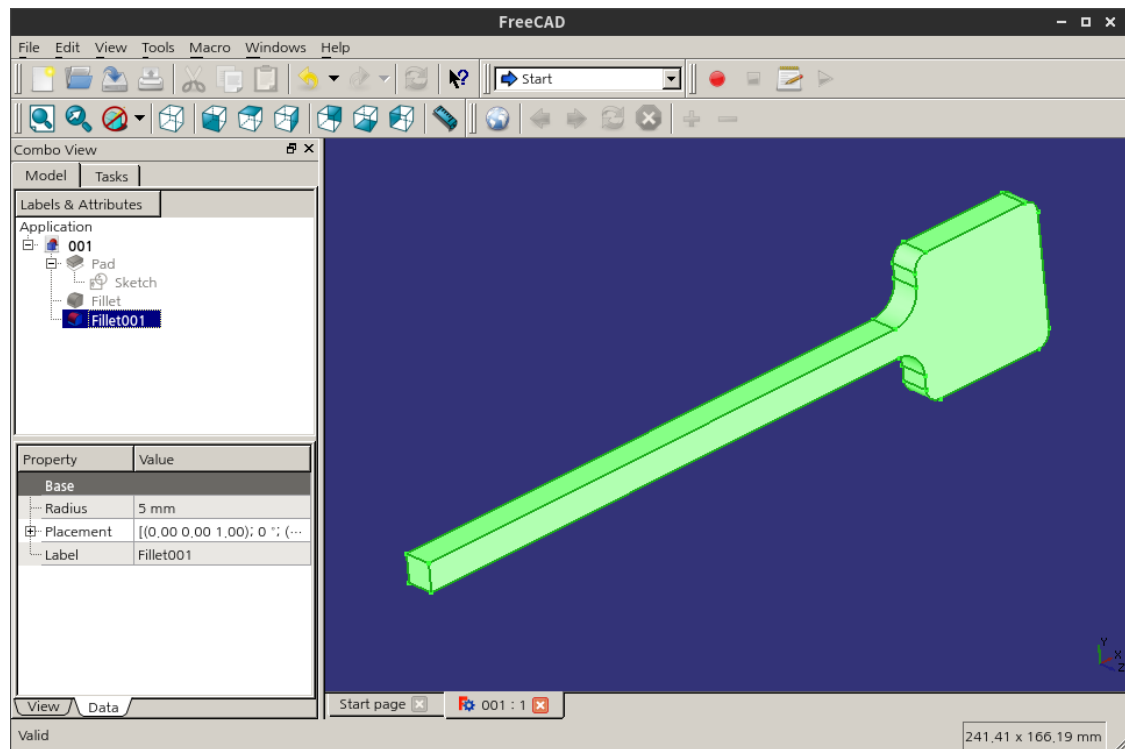
Sketch



Pad



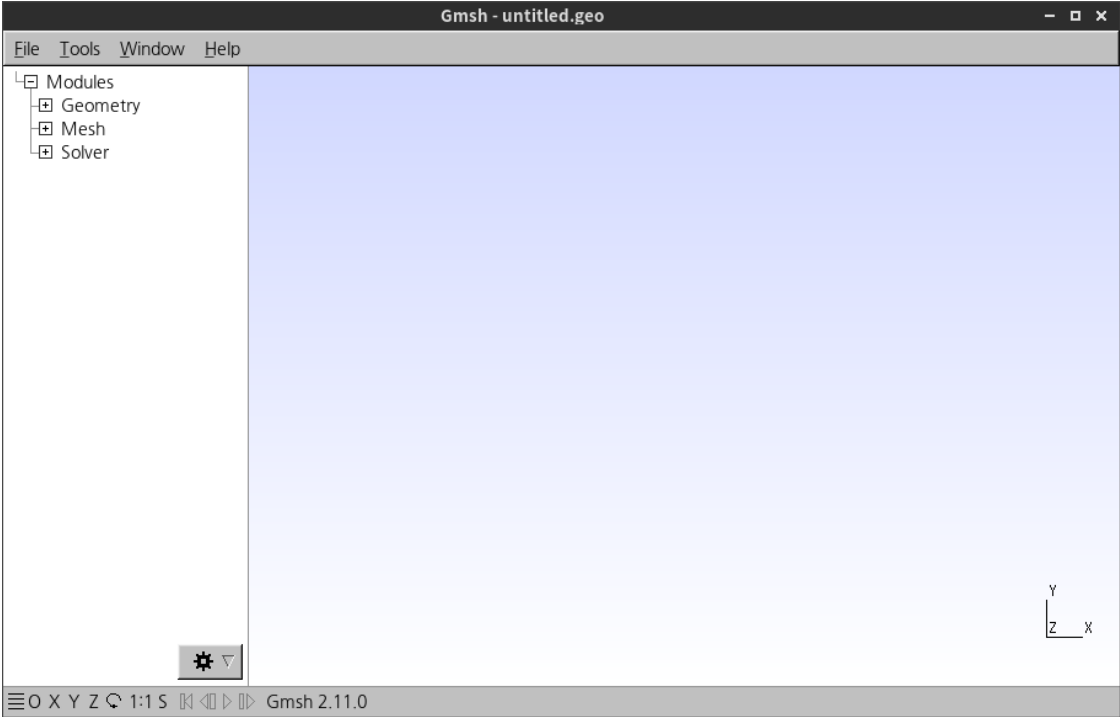
Export to STEP



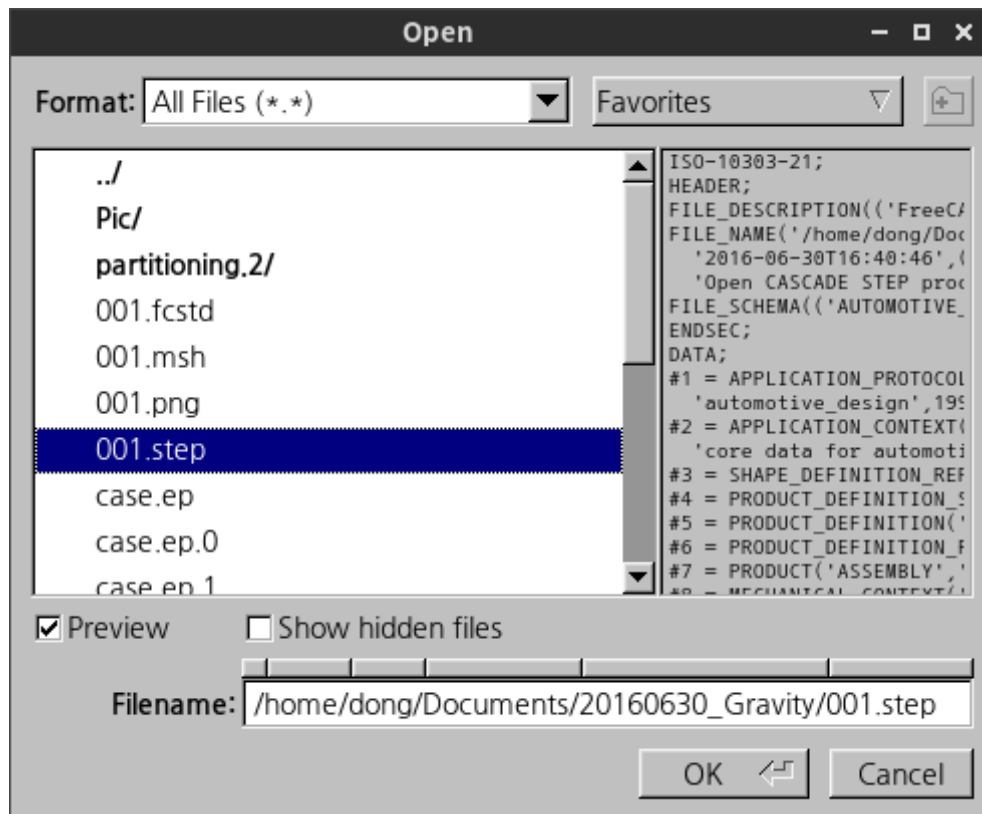
Gmsh

- Mesh Generator

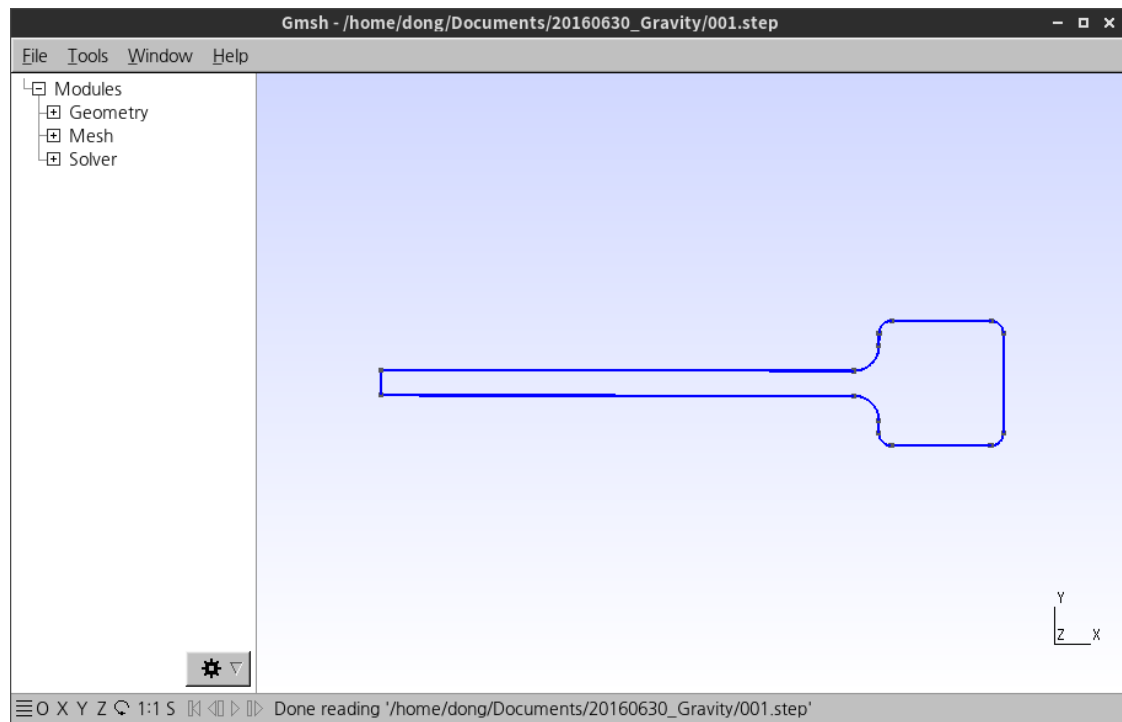
Start Gmsh



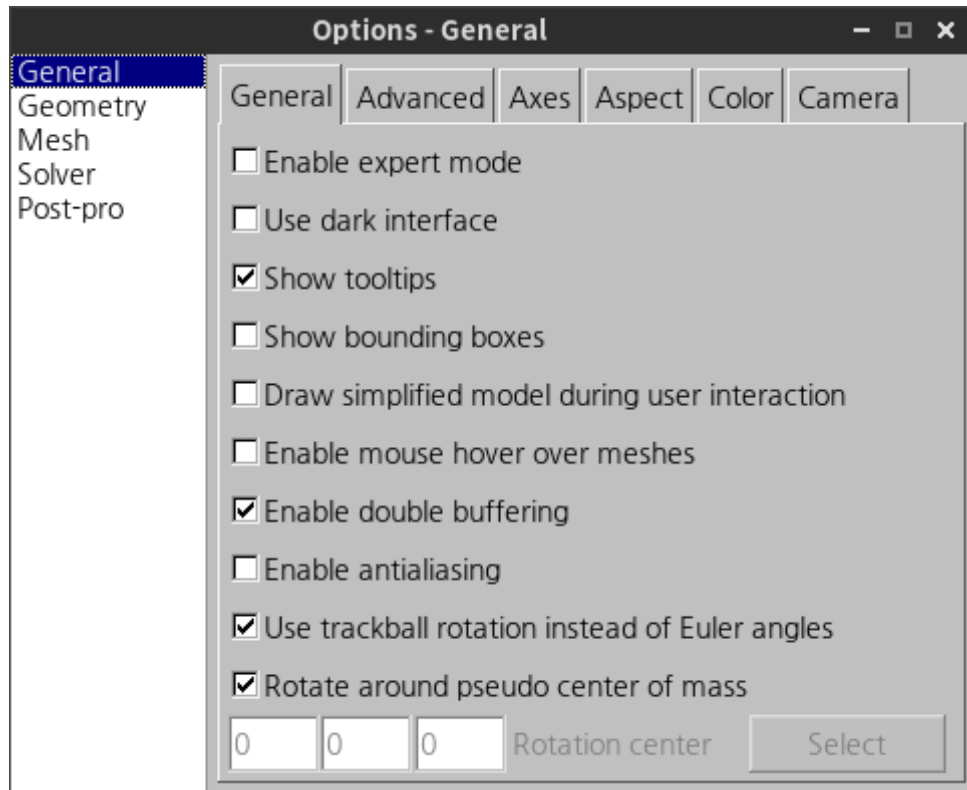
Open STEP file



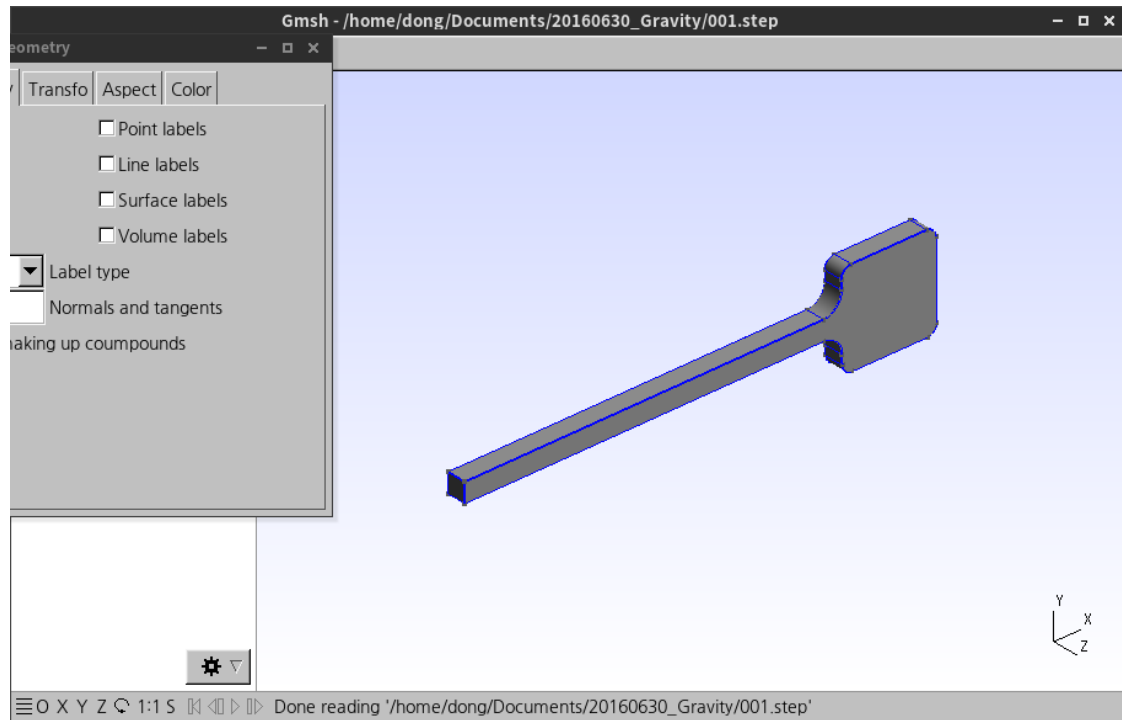
Check Geometry



Tools - Options

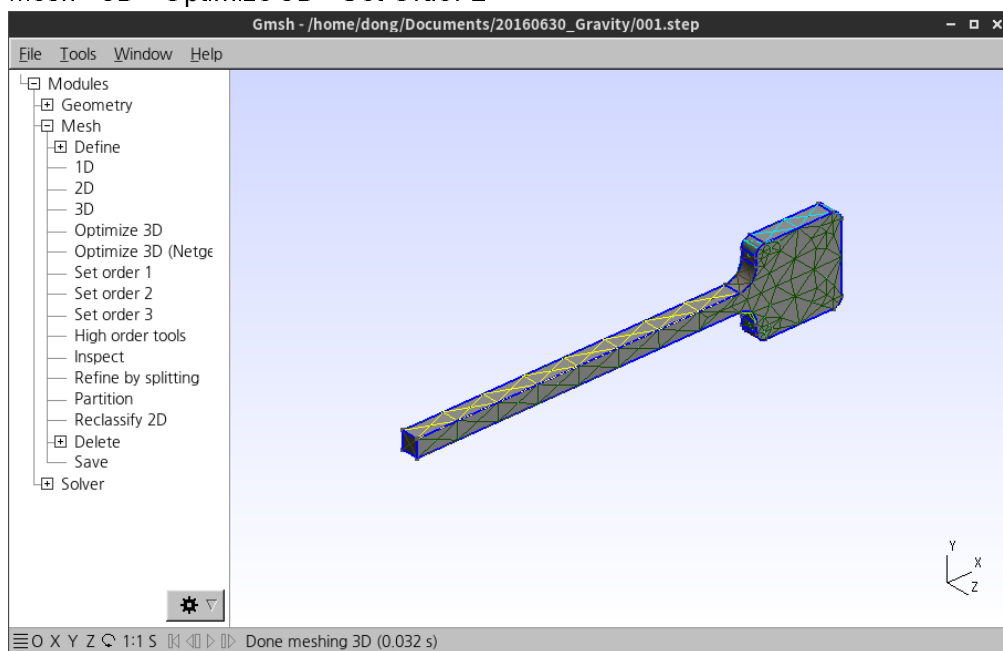


Check Geometry

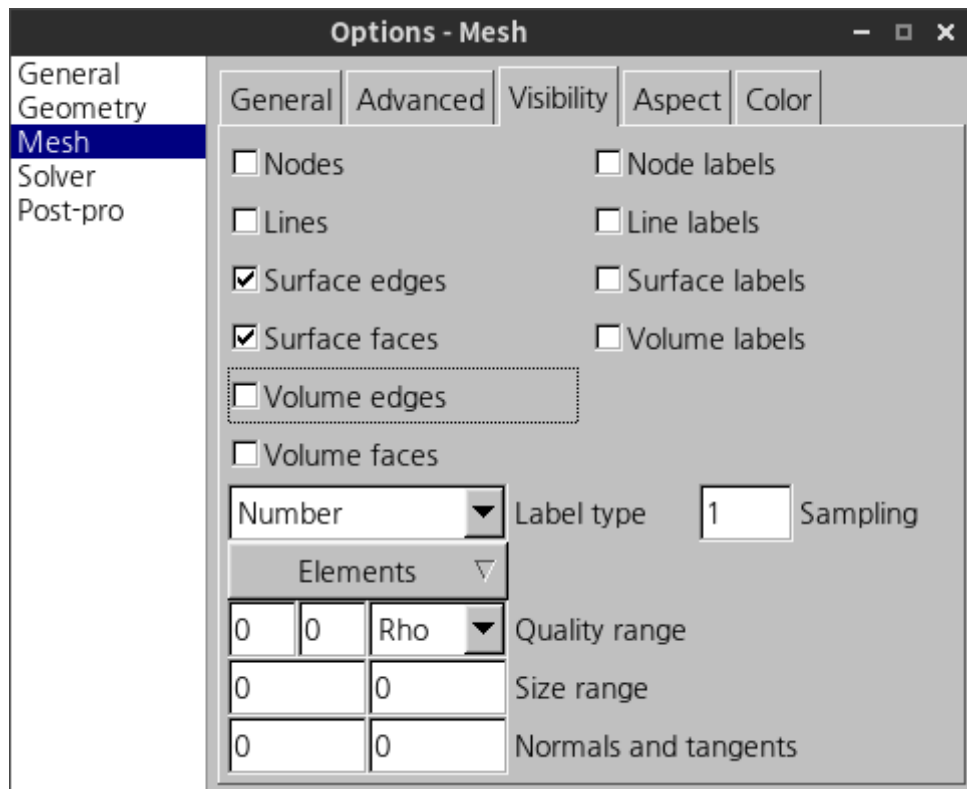


Generate Mesh

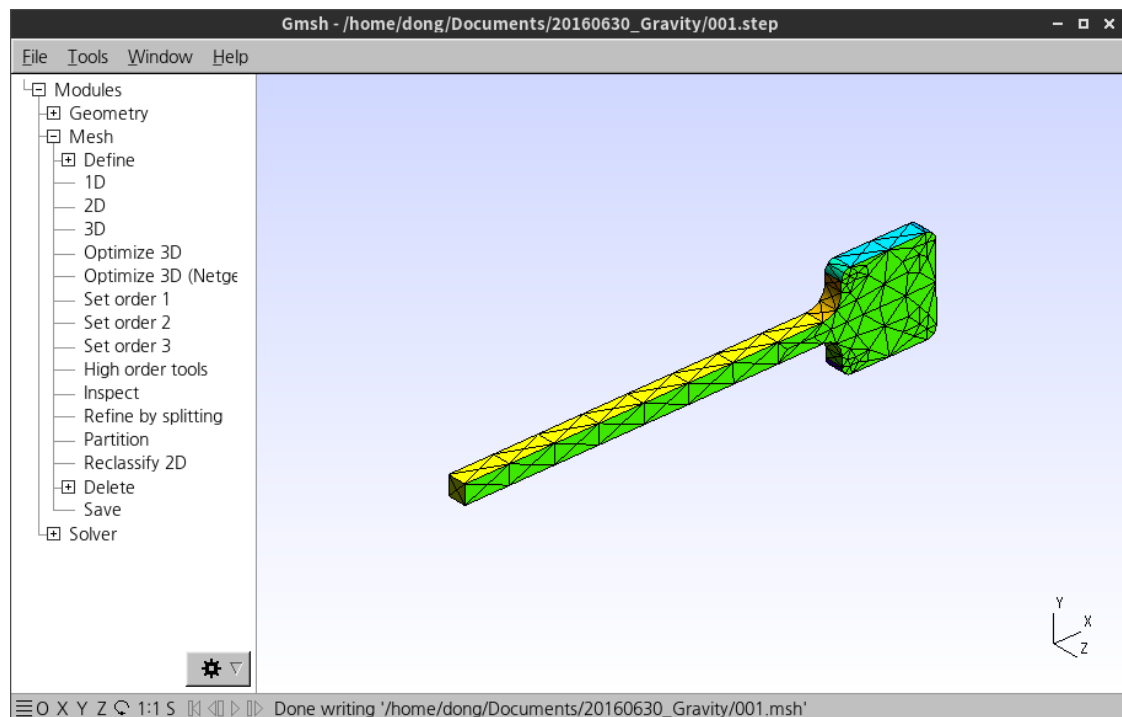
- Mesh - 3D - Optimize 3D - Set Order 2



Options



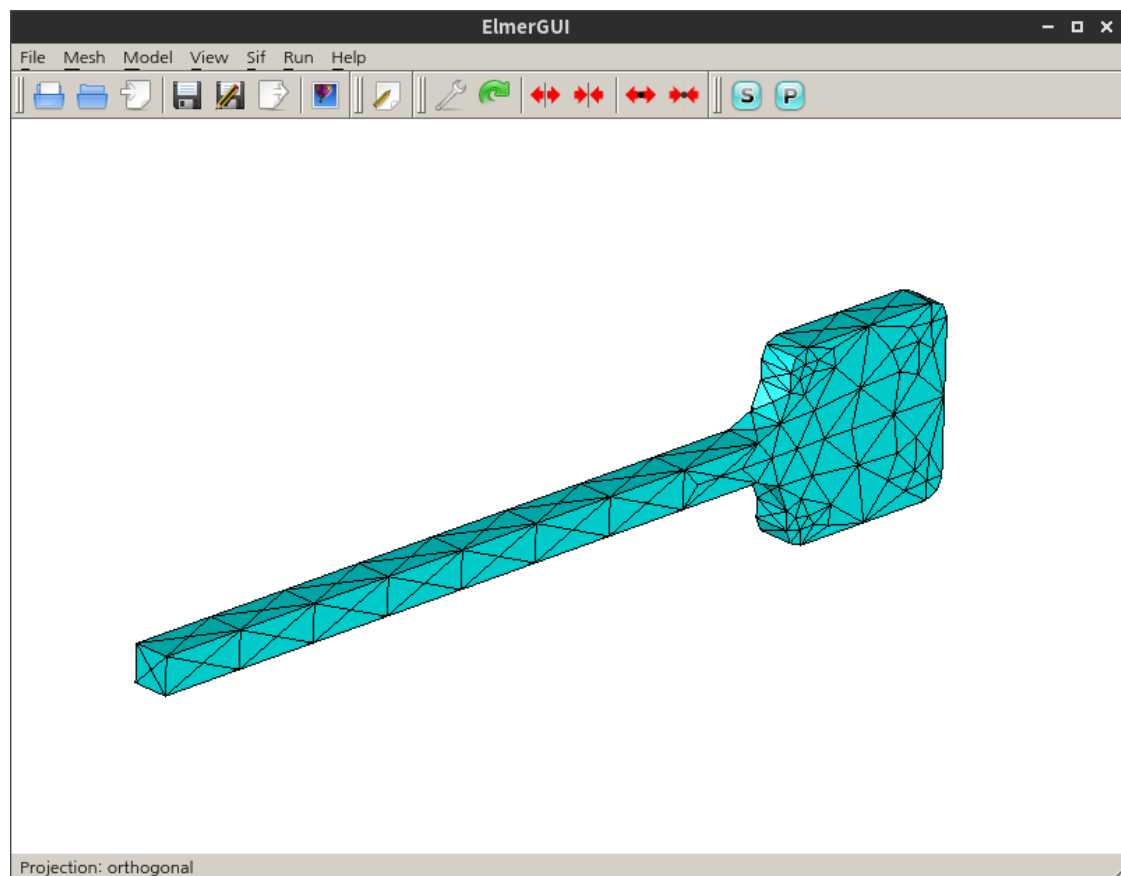
Check & Export Mesh



Elmer

- "StressSolver" has no gravity condition in default setting.

Open Mesh



Setup

- Coordinate Scaling = Real 0.001

Setup

Header

☒ Check keywords warn

MeshDB

Include path

Results directory

Free text

Simulation

Max. output level

5

Steady state max. iter

1

Coordinate system

Cartesian

Timestepping method

BDF

Coordinate mapping

1 2 3

BDF order

1

Simulation type

Steady state

Timestep intervals

Output intervals

1

Timestep sizes

Solver input file

case.sif

Post file

case.ep

Free text

Coordinate Scaling = Real 0.001

Constants

Gravity

0 -1 0 9.82

Boltzmann

1.3807e-23

Stefan Boltzmann

5.67e-08

Unit charge

1.602e-19

Vacuum permittivity

8.8542e-12

Free text

✓ Apply

Equation - Liniear Elasticity

- Check Active, Options
- Check Body 1
- Press Edit Solver Setting

The screenshot shows a software window titled "Equation" with three tabs: "Helmholtz Equation", "Linear elasticity" (selected), and "Heat Equation". The "Linear elasticity" tab contains the following settings:

- Activate for this equation set**
 - Active: ☒
- Give Execution priority**
 - Priority:
- Options**
 - Calculate Stresses: ☒
 - Plane Stress: ☒
- This and that**
 - Element Codes:
- Free text input**
 -

Below the settings is a section "Apply to bodies:" with a checked checkbox and a text box containing "Body 1".

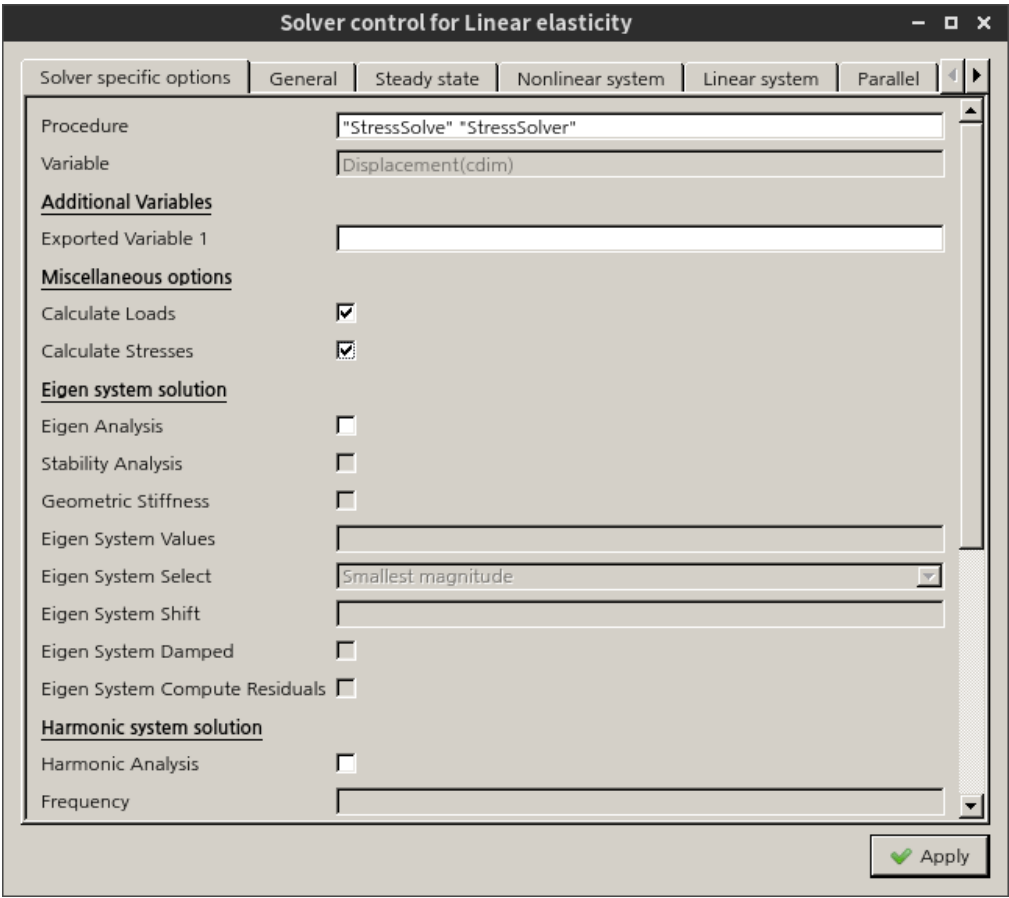
At the bottom of the dialog is a button labeled "Edit Solver Settings" with a wrench icon.

Below the dialog is a "Name:" field containing "Equation 1".

At the very bottom are four buttons: "New" (with a plus icon), "Add" (with a plus icon), "OK" (with a checkmark icon), and "Cancel" (with a red X icon).

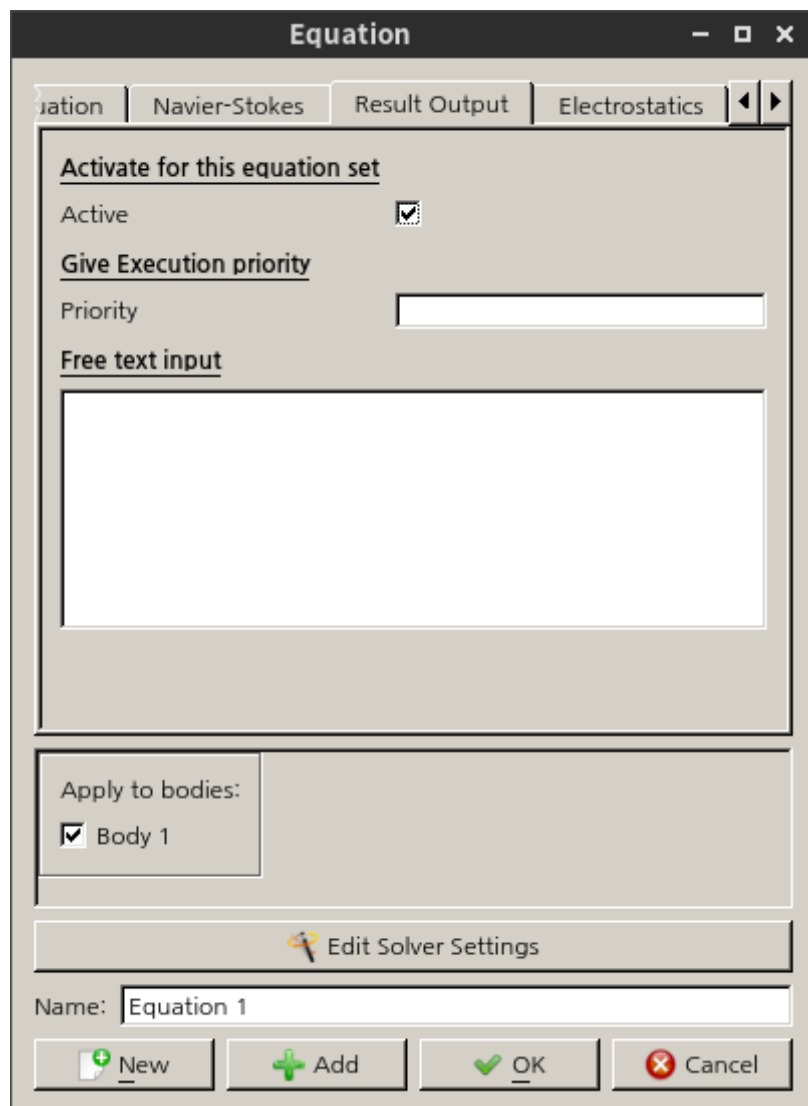
Solver

- Check Miscellaneous Options



Equation - Result Output

- Check Active
- Press Edit Solver Settings

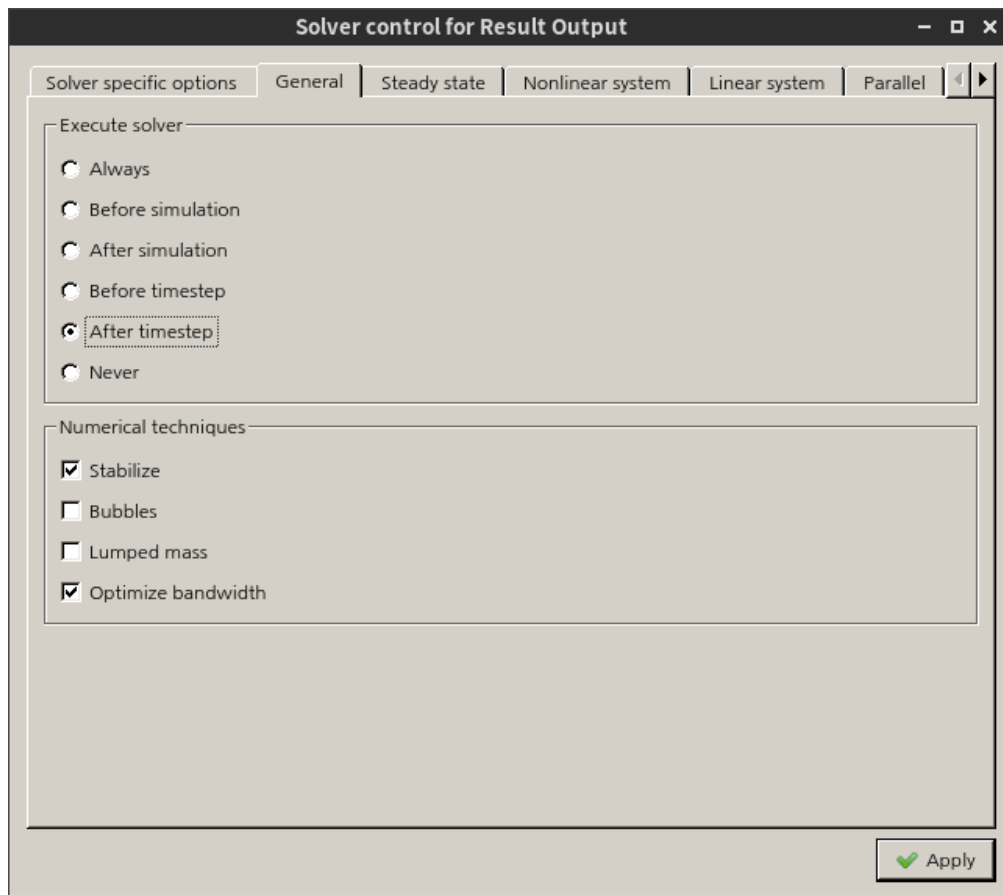


The image shows a software dialog box titled "Equation". It has four tabs: "Equation", "Navier-Stokes", "Result Output" (which is selected), and "Electrostatics". The "Result Output" tab contains the following elements:

- Activate for this equation set**: A checkbox labeled "Active" which is checked.
- Give Execution priority**: A text input field labeled "Priority" which is currently empty.
- Free text input**: A large, empty rectangular text area.
- Apply to bodies:**: A section containing a checked checkbox labeled "Body 1".
- Edit Solver Settings**: A button with a wrench icon.
- Name:**: A text input field containing "Equation 1".
- Buttons**: Four buttons at the bottom: "New" (with a plus icon), "Add" (with a plus icon), "OK" (with a green checkmark icon), and "Cancel" (with a red X icon).

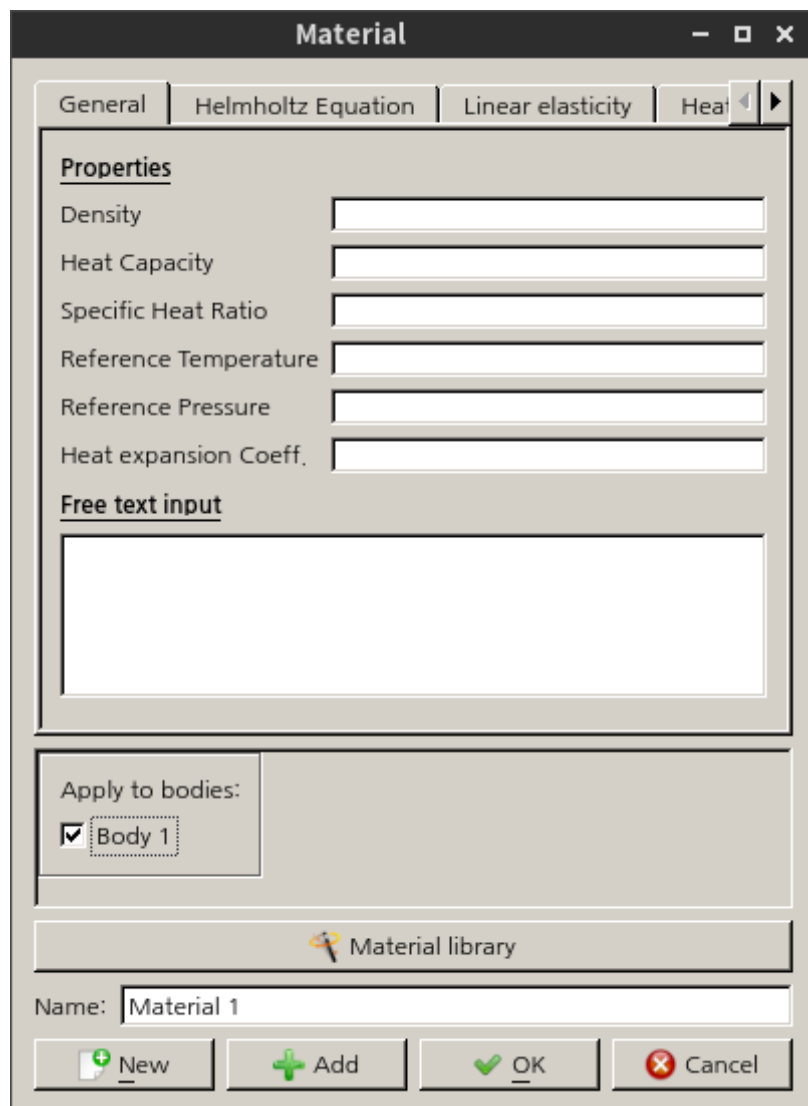
Solver

- Tab General - Check After Timestep



Material

- Check Body 1
- Press Material Library



The image shows a software dialog box titled "Material". It has four tabs: "General", "Helmholtz Equation", "Linear elasticity", and "Heat". The "General" tab is selected. Inside the dialog, there is a section titled "Properties" with six input fields: "Density", "Heat Capacity", "Specific Heat Ratio", "Reference Temperature", "Reference Pressure", and "Heat expansion Coeff.". Below this is a "Free text input" section with a large text area. At the bottom, there is a section "Apply to bodies:" with a checked checkbox and the text "Body 1". Below that is a "Material library" section with a magnifying glass icon. At the very bottom, there is a "Name:" label followed by a text field containing "Material 1". The bottom of the dialog features four buttons: "New" (with a plus icon), "Add" (with a plus icon), "OK" (with a green checkmark icon), and "Cancel" (with a red X icon).

Material

General | Helmholtz Equation | Linear elasticity | Heat

Properties

Density

Heat Capacity

Specific Heat Ratio

Reference Temperature

Reference Pressure

Heat expansion Coeff.

Free text input

Apply to bodies:

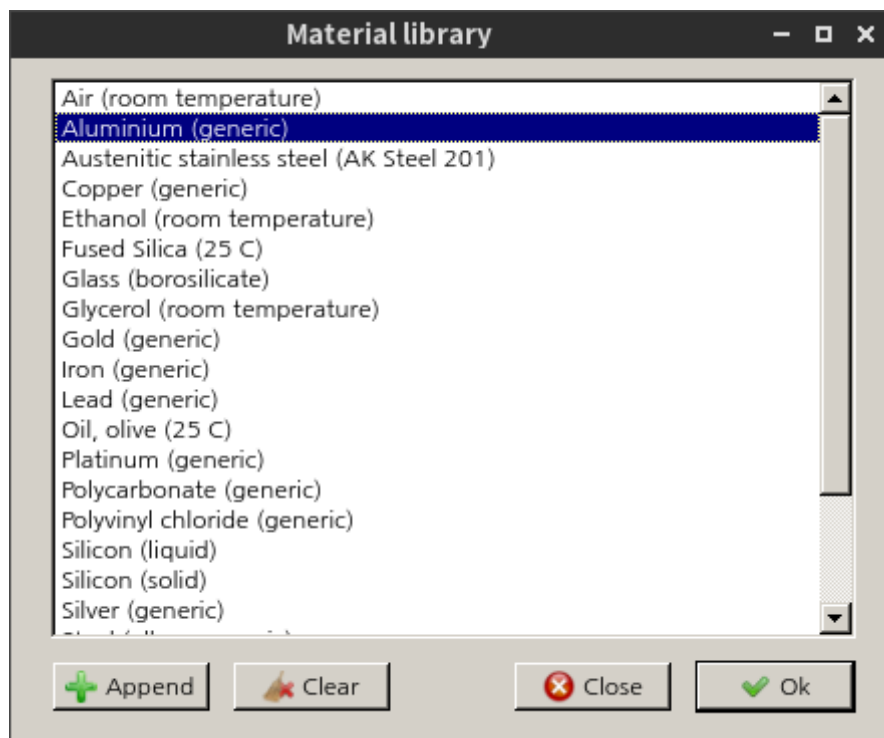
☒ Body 1

Material library

Name:

Material Library

- Choice



Apply Material

- Check

Material

GeneralHelmholtz EquationLinear elasticityHeat

Properties

Density2700.0

Heat Capacity897.0

Specific Heat Ratio

Reference Temperature

Reference Pressure

Heat expansion Coeff.23.1e-6

Free text input

Apply to bodies:

☒ Body 1

Material library

Name: Aluminium (generic)

NewAddOKCancel

Body Force

- Check Body 1



The image shows a software dialog box titled "BodyForce". It has three tabs: "Helmholtz Equation", "Linear elasticity", and "Heat Equation". The "Linear elasticity" tab is currently selected. Inside the dialog, there are two input fields for "Volume source 1 (real part)" and "Volume source 2 (imag part)". Below these is a large "Free text input" area. At the bottom, there is a section "Apply to bodies:" with a checked checkbox next to "Body 1". Below this is a "Name:" field containing "BodyForce 1". At the very bottom are four buttons: "New" (with a plus icon), "Add" (with a plus icon), "OK" (with a checkmark icon), and "Cancel" (with a red X icon).

BodyForce

Helmholtz Equation | Linear elasticity | Heat Equation

Volume sources

Volume source 1 (real part)

Volume source 2 (imag part)

Free text input

Apply to bodies:

☒ Body 1

Name:

Body Force - Linear Elasticity

- Tab Linear Elasticity
- Apply Gravitation like that :

BodyForce

tion Linear elasticity Heat Equation Navier-Stokes

Volume forces

Force 1

Force 2

Force 3

Pressure

Stress 6-vector

Strain 6-vector

Bodywise Dirichlet Conditions

Displacement 1

Displacement 2

Displacement 3

Displacement 1 Condition

Apply to bodies:

☒ Body 1

Name:

New Add OK Cancel

Boundary Condition

- Apply Boundary Condition like that :

BoundaryCondition

GeneralHelmholtz EquationLinear elasticityHeat

Normal-Tangential Coordinate System

Use normal-tangential coordinate system

Change of variables

Dirichlet Conditions

Displacement 1

Displacement 2

Displacement 3

Displacement 1 Condition

Displacement 2 Condition

Displacement 3 Condition

Traction boundary conditions

Normal Force

☐

☒

Apply to boundaries:

☐ Boundary 1

☐ Boundary 2

☐ Boundary 3

☐ Boundary 4

Name:

New

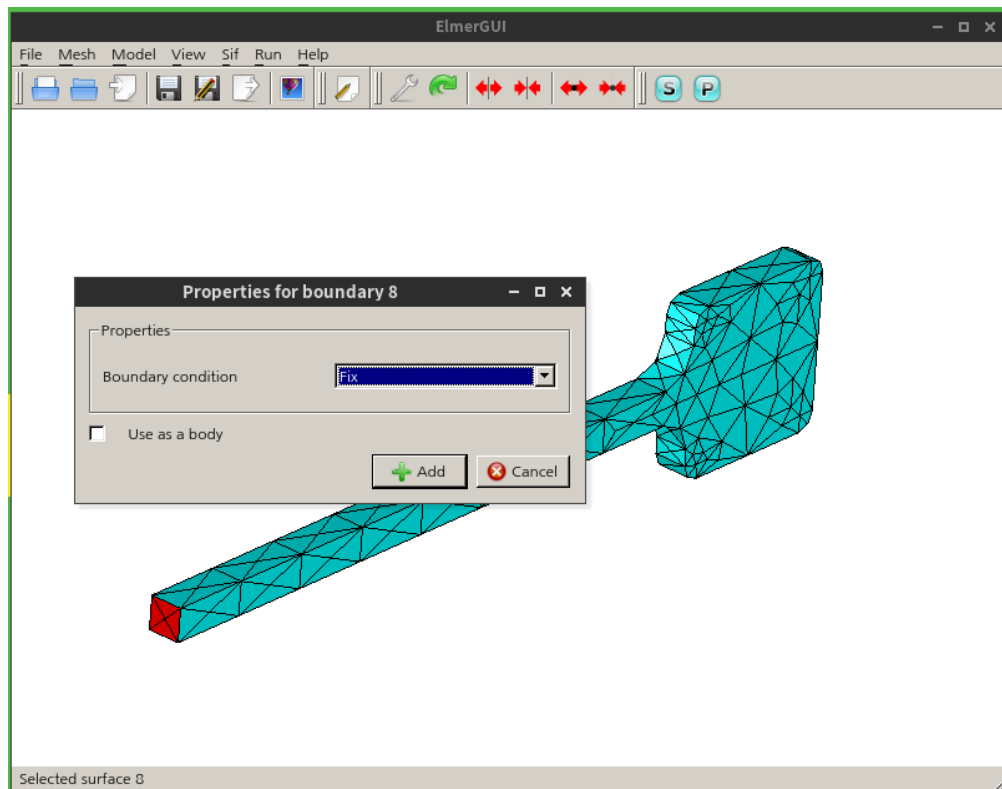
Add

OK

Cancel

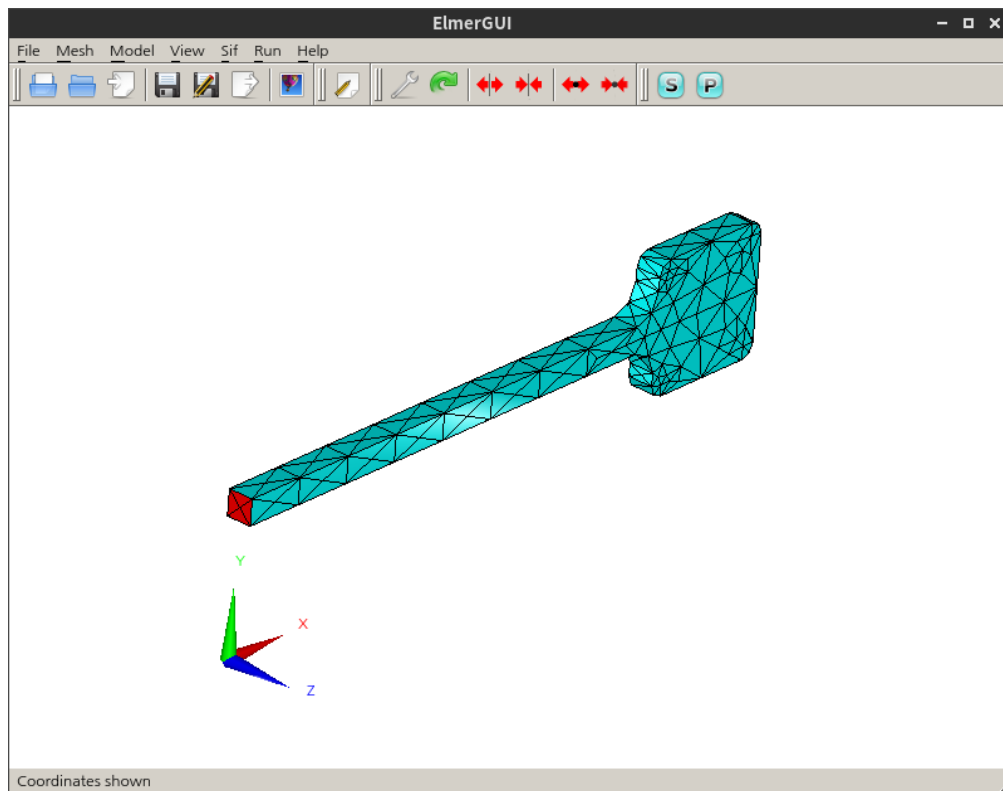
Boundary Condition

- Check Model - Set Boundary Properties
- Double Click on desired Surface & Choice :



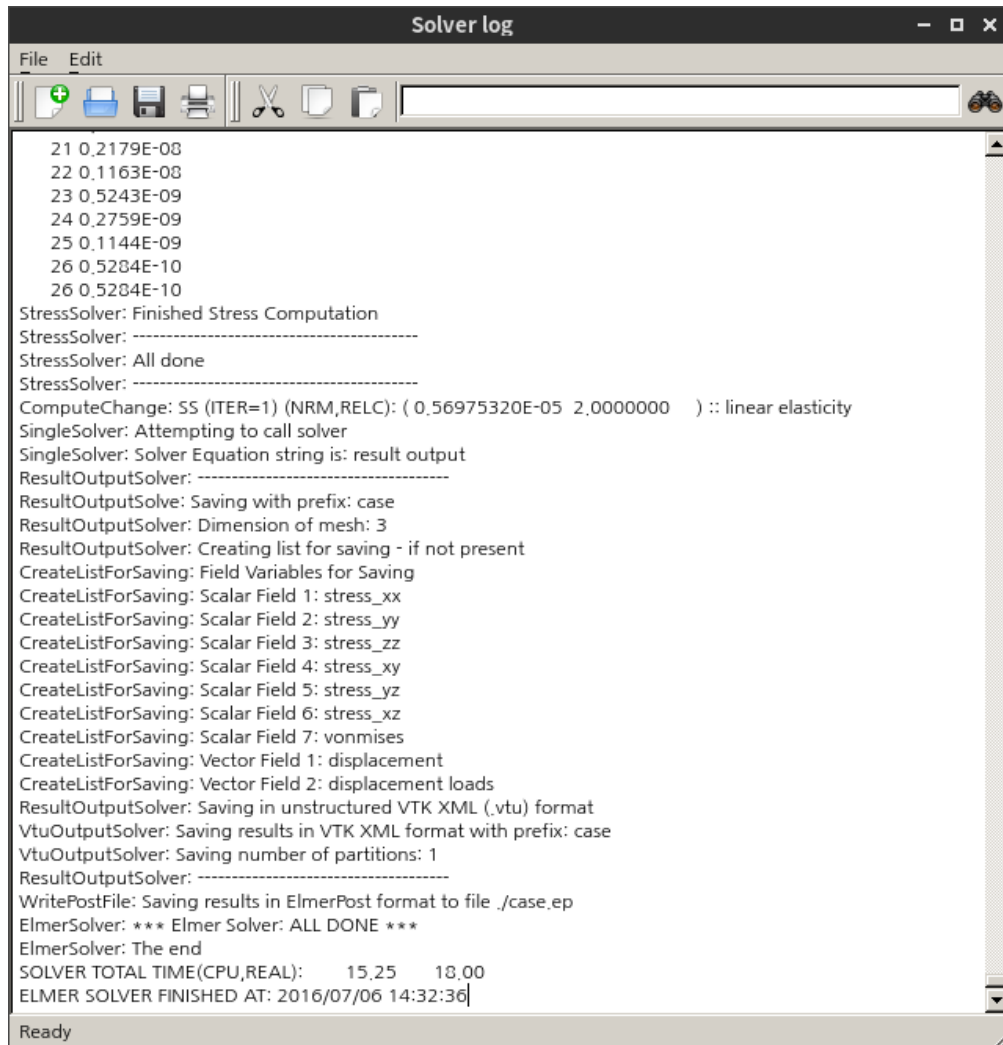
Check Models

- Check directions with Compas



Run Solver

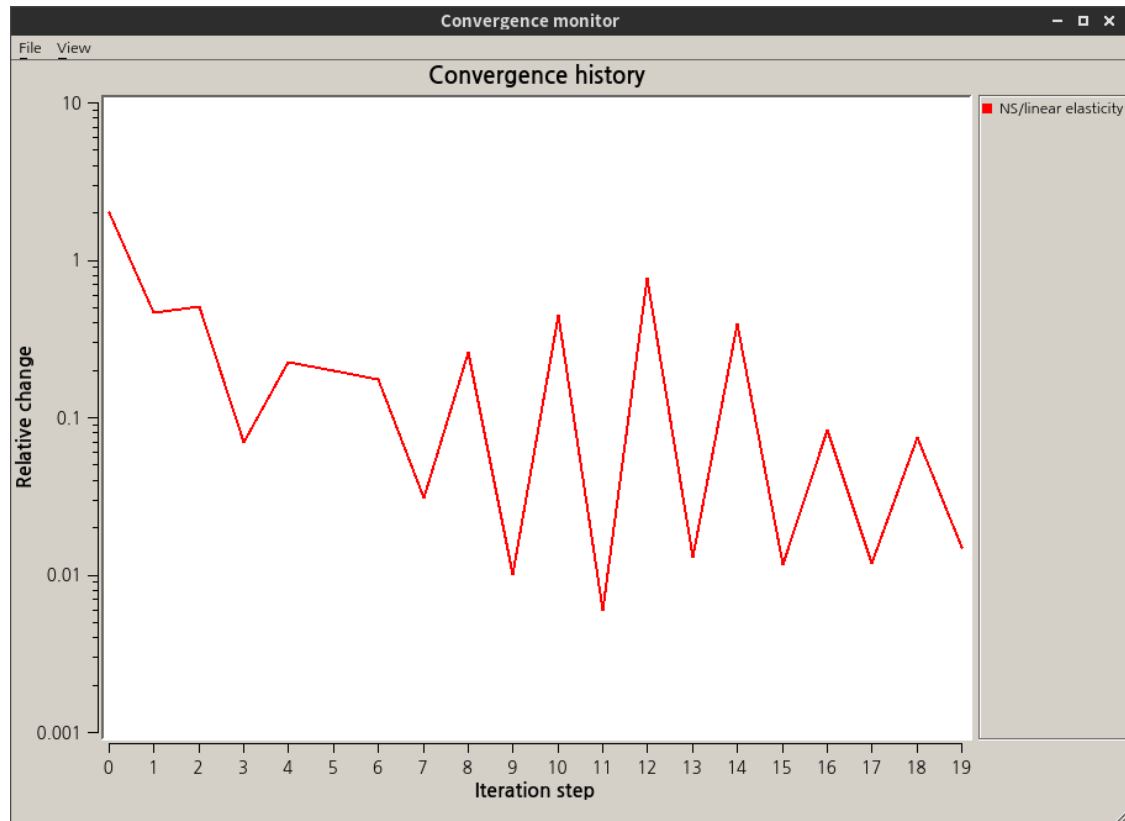
- Run - Start Solver



```

21 0.2179E-08
22 0.1163E-08
23 0.5243E-09
24 0.2759E-09
25 0.1144E-09
26 0.5284E-10
26 0.5284E-10
StressSolver: Finished Stress Computation
StressSolver: -----
StressSolver: All done
StressSolver: -----
ComputeChange: SS (ITER=1) (NRM,RELC): ( 0.56975320E-05  2.0000000 ) :: linear elasticity
SingleSolver: Attempting to call solver
SingleSolver: Solver Equation string is: result output
ResultOutputSolver: -----
ResultOutputSolver: Saving with prefix: case
ResultOutputSolver: Dimension of mesh: 3
ResultOutputSolver: Creating list for saving - if not present
CreateListForSaving: Field Variables for Saving
CreateListForSaving: Scalar Field 1: stress_xx
CreateListForSaving: Scalar Field 2: stress_yy
CreateListForSaving: Scalar Field 3: stress_zz
CreateListForSaving: Scalar Field 4: stress_xy
CreateListForSaving: Scalar Field 5: stress_yz
CreateListForSaving: Scalar Field 6: stress_xz
CreateListForSaving: Scalar Field 7: vonmises
CreateListForSaving: Vector Field 1: displacement
CreateListForSaving: Vector Field 2: displacement loads
ResultOutputSolver: Saving in unstructured VTK XML (.vtu) format
VtuOutputSolver: Saving results in VTK XML format with prefix: case
VtuOutputSolver: Saving number of partitions: 1
ResultOutputSolver: -----
WritePostFile: Saving results in ElmerPost format to file ./case.ep
ElmerSolver: *** Elmer Solver: ALL DONE ***
ElmerSolver: The end
SOLVER TOTAL TIME(CPU,REAL):   15.25   18.00
ELMER SOLVER FINISHED AT: 2016/07/06 14:32:36
Ready
  
```

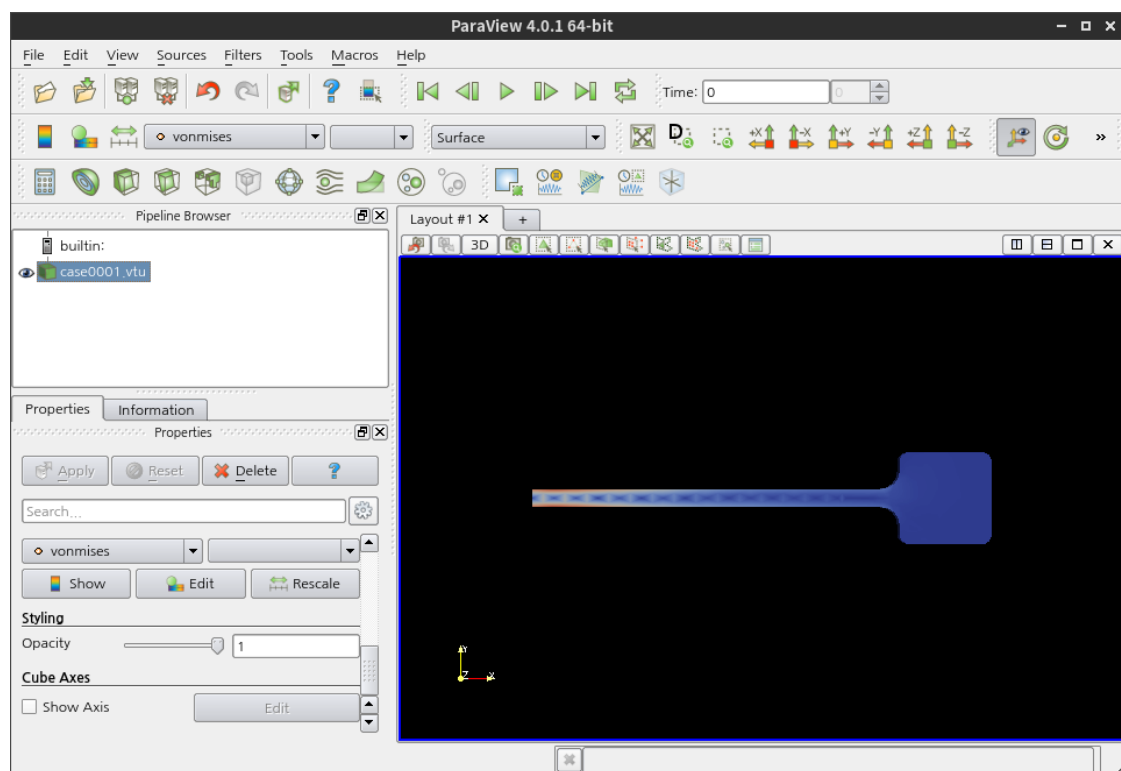

Run Solvere - Convergence



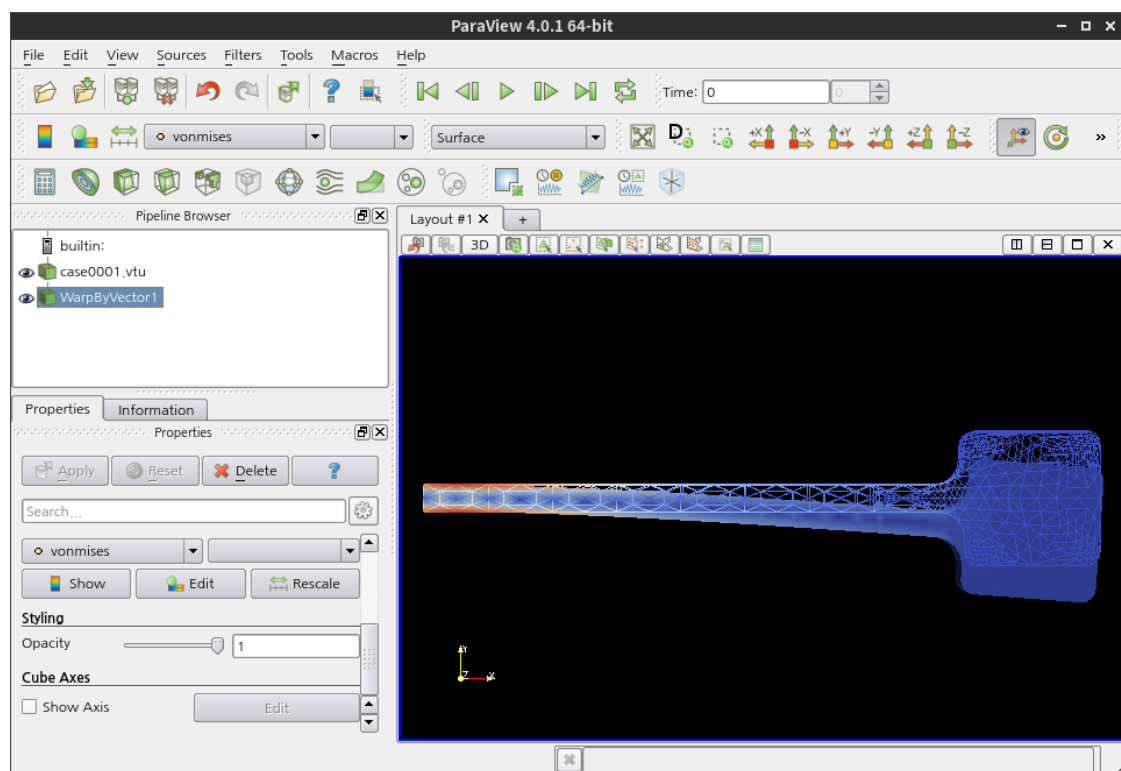
ParaView

- 해석 결과 데이터 보기

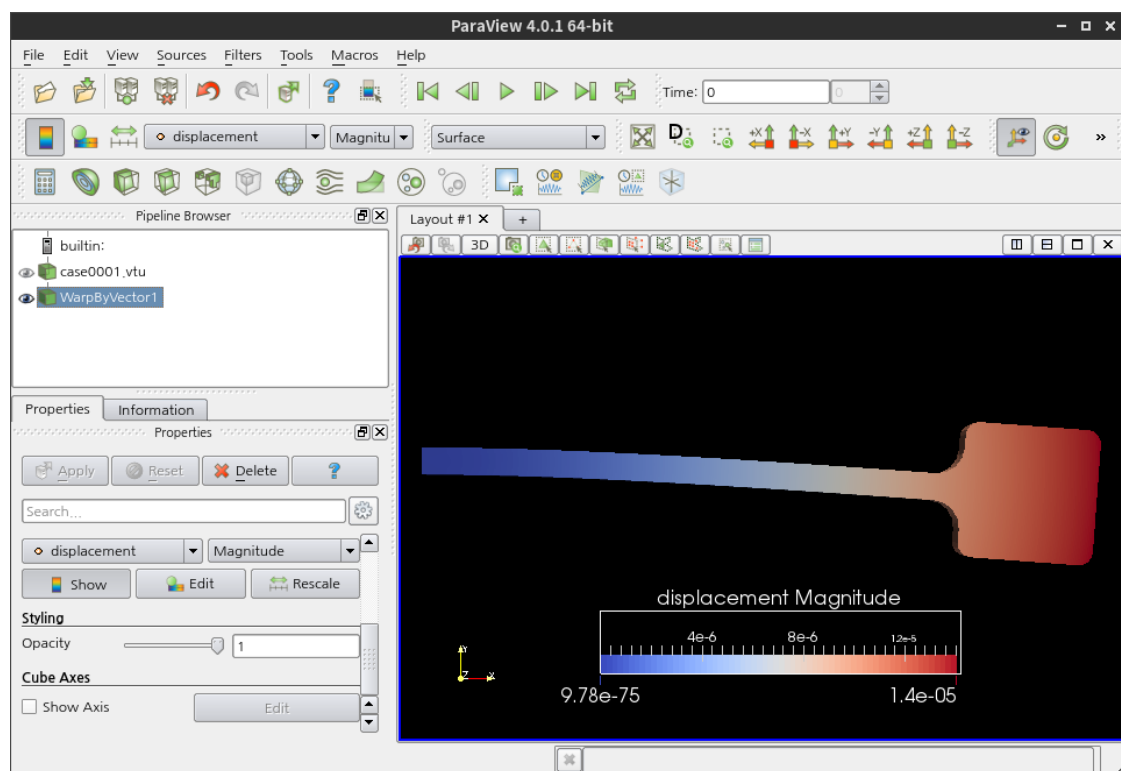
VonMises Stress



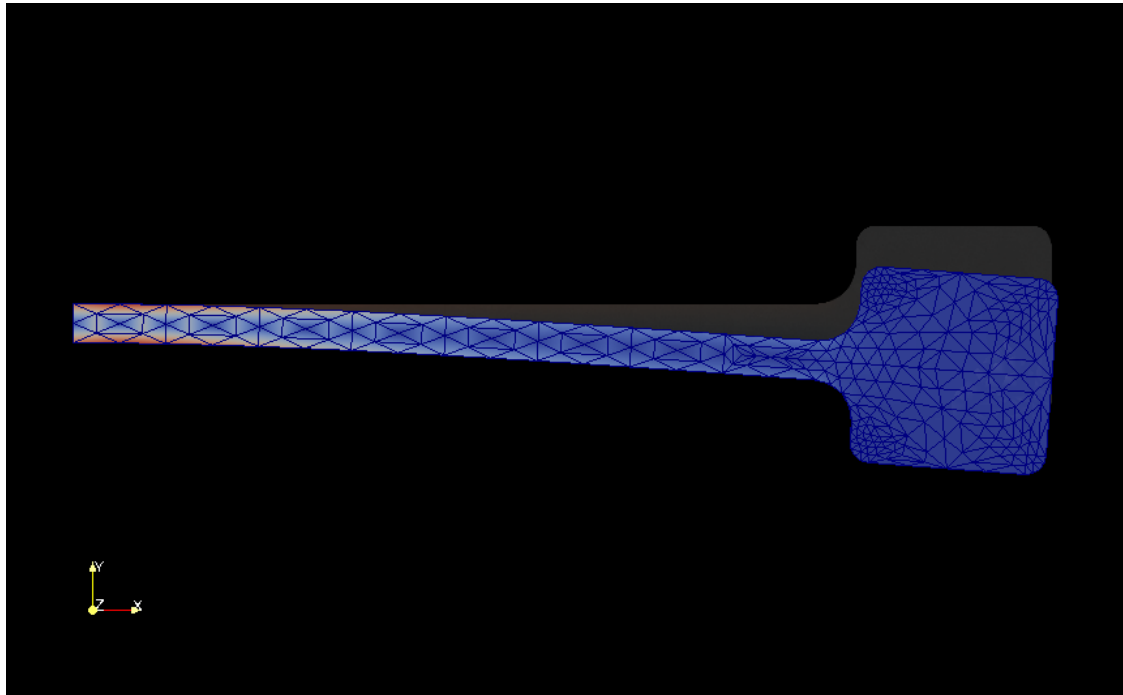
Displacement with Stress



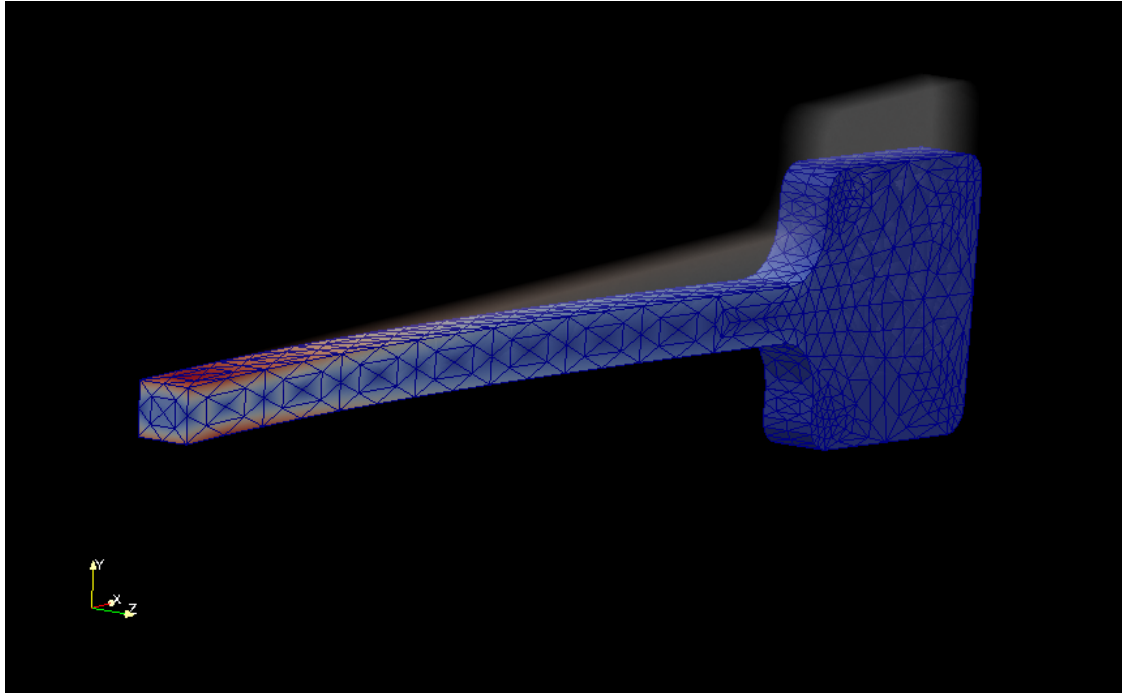
Displacement



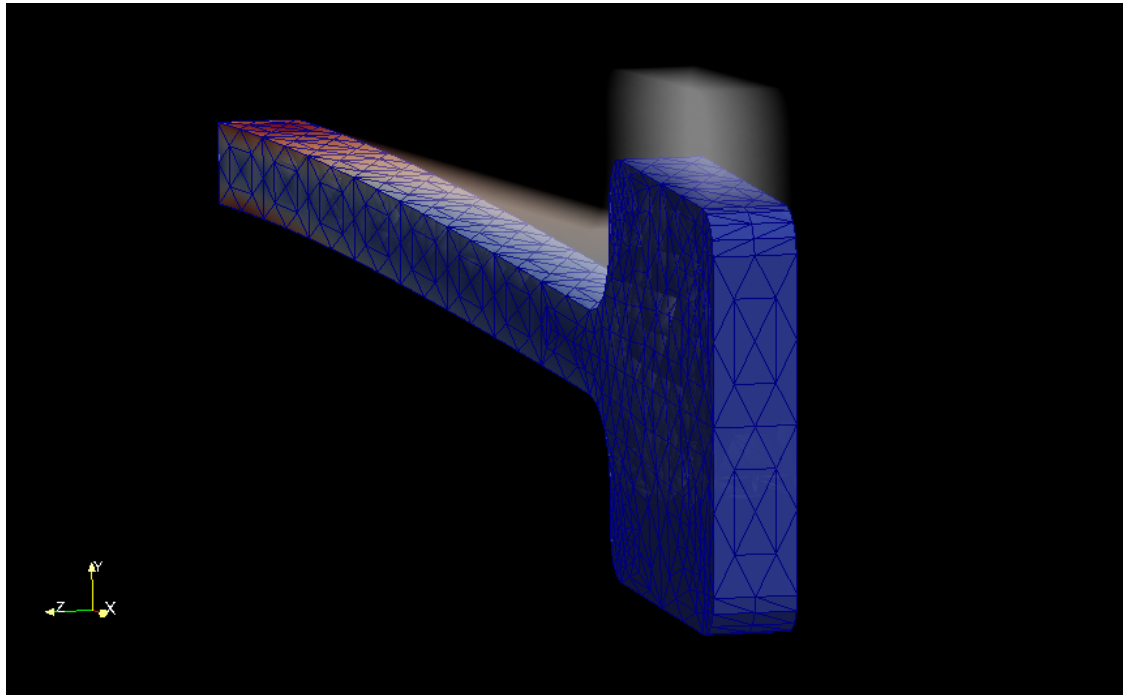
Details



Details



Details



Thank you!