0301 : Gravity Condition
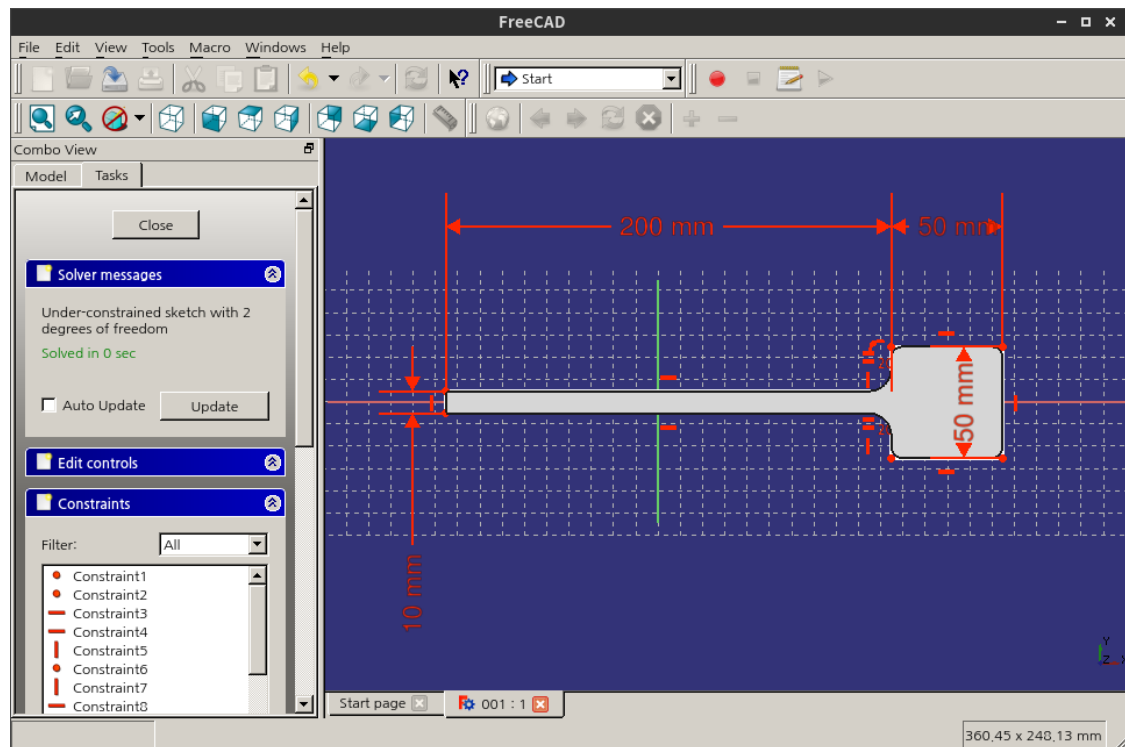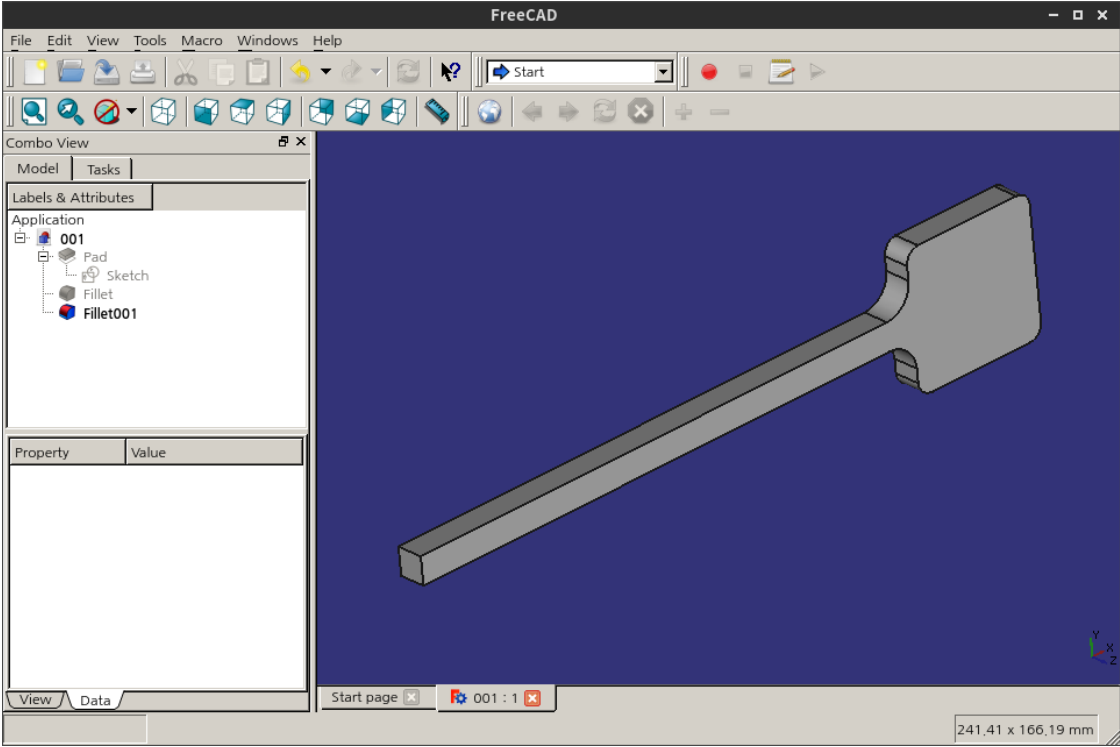
FreeCAD

- 3D Solid Modeling
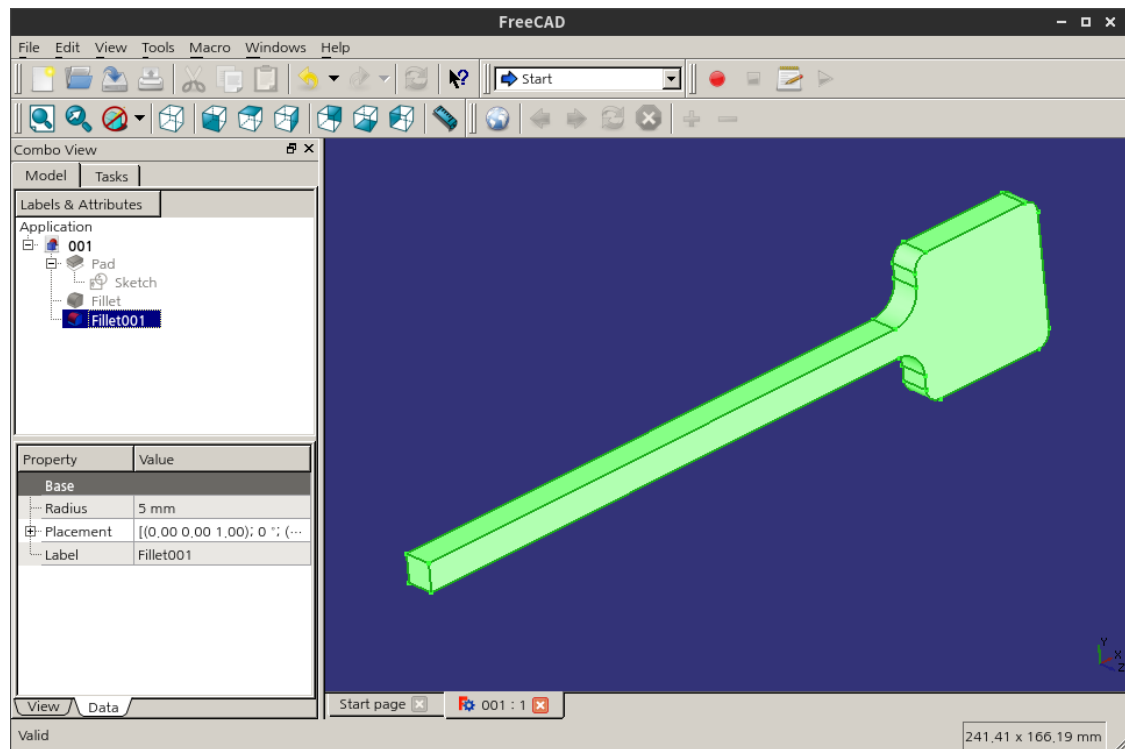
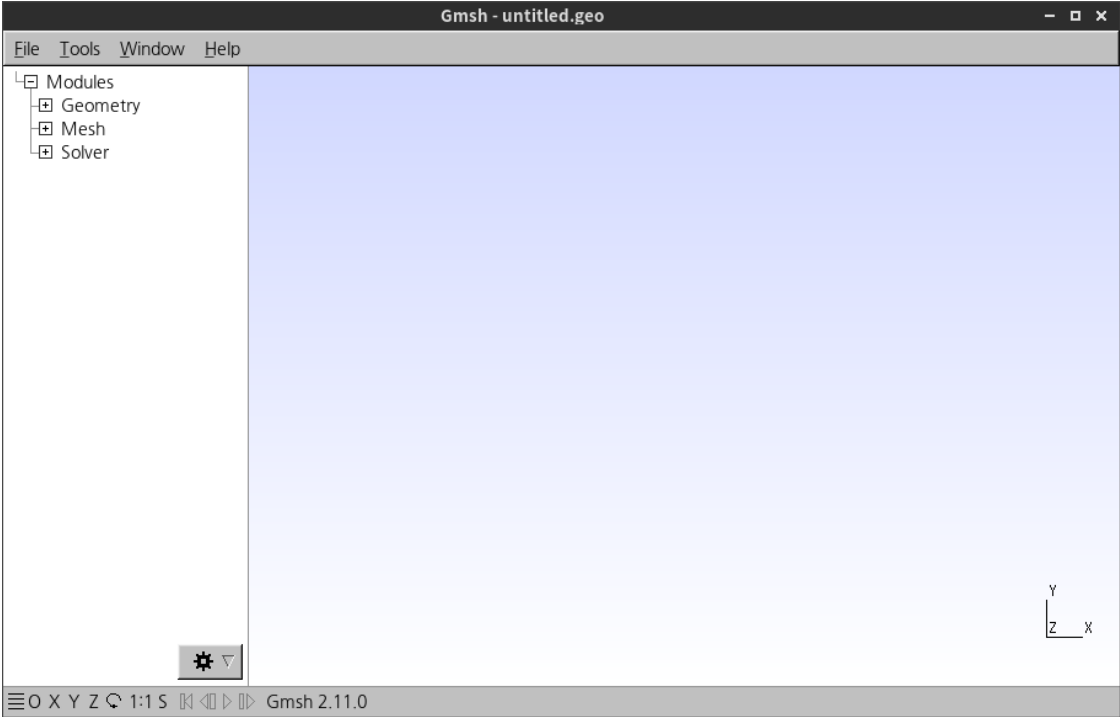## Sketch

Pad

Export to STEP

Gmsh

- Mesh Generatoin

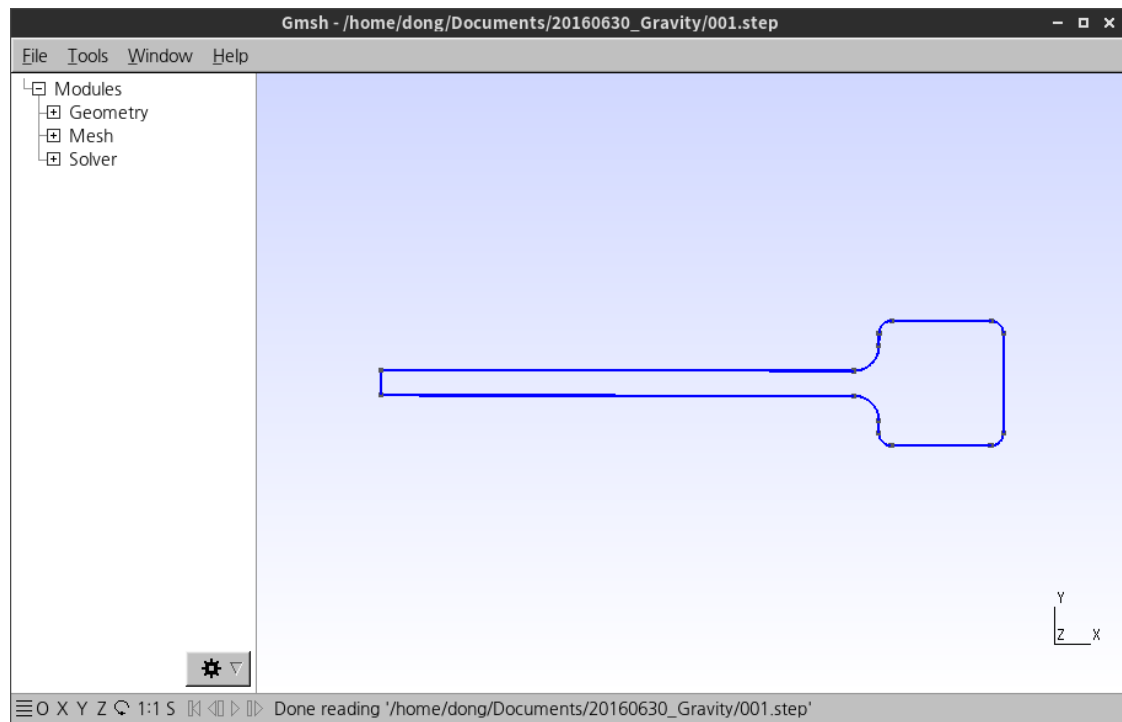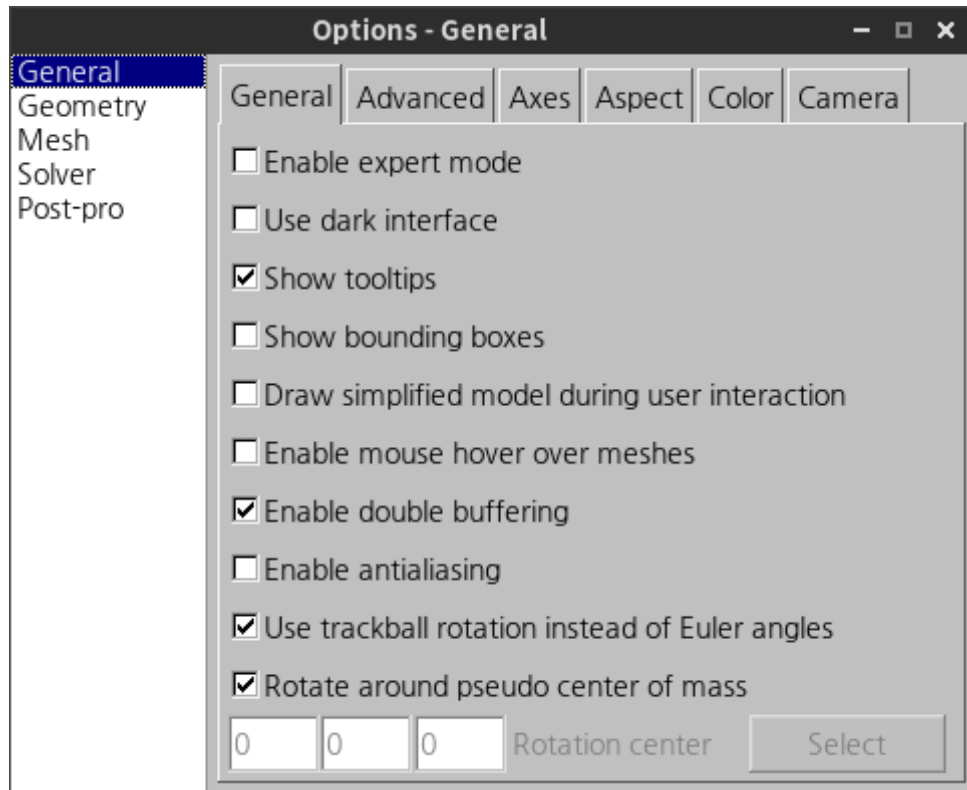2016. 7. 7.   0301_Gravity
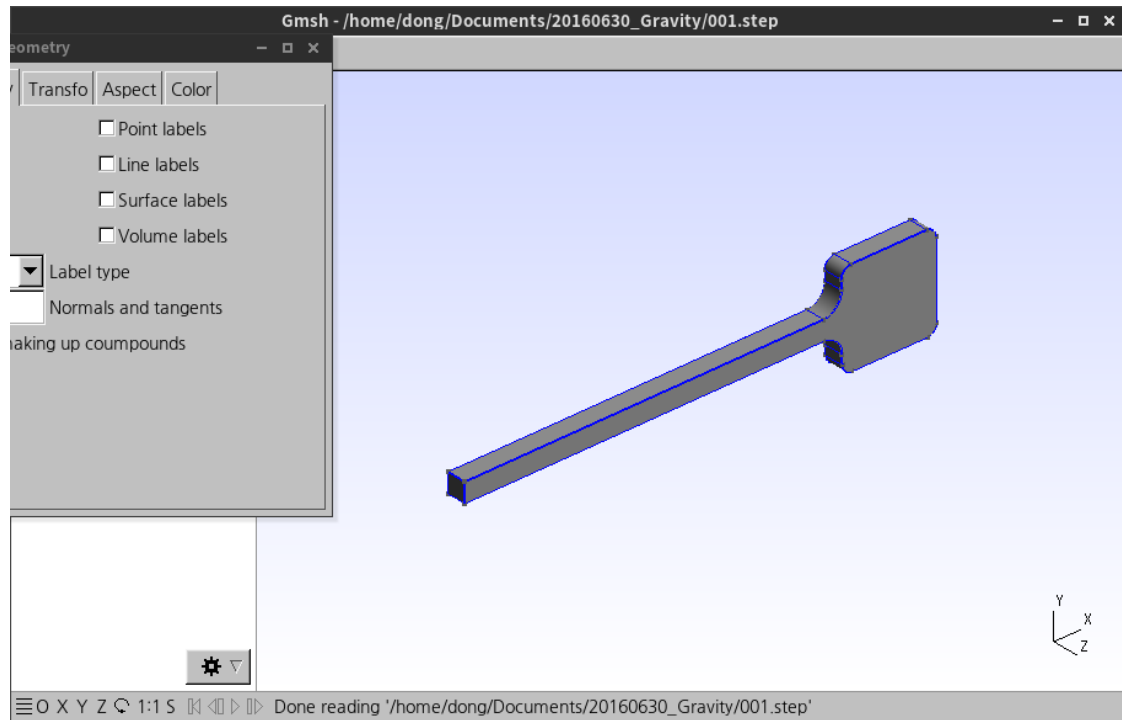
Start Gmsh

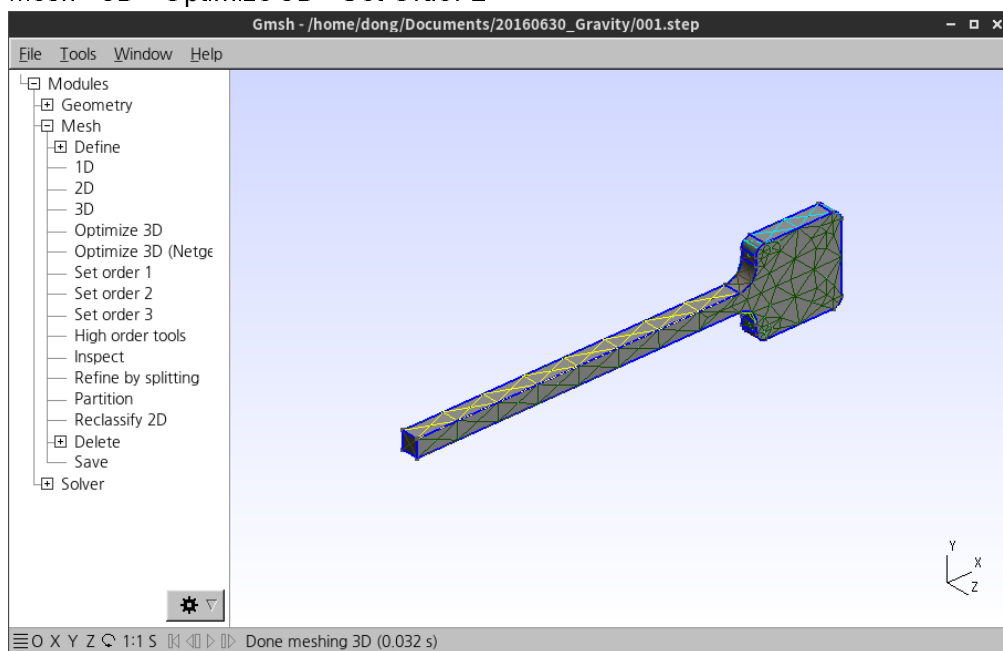## Open STEP file
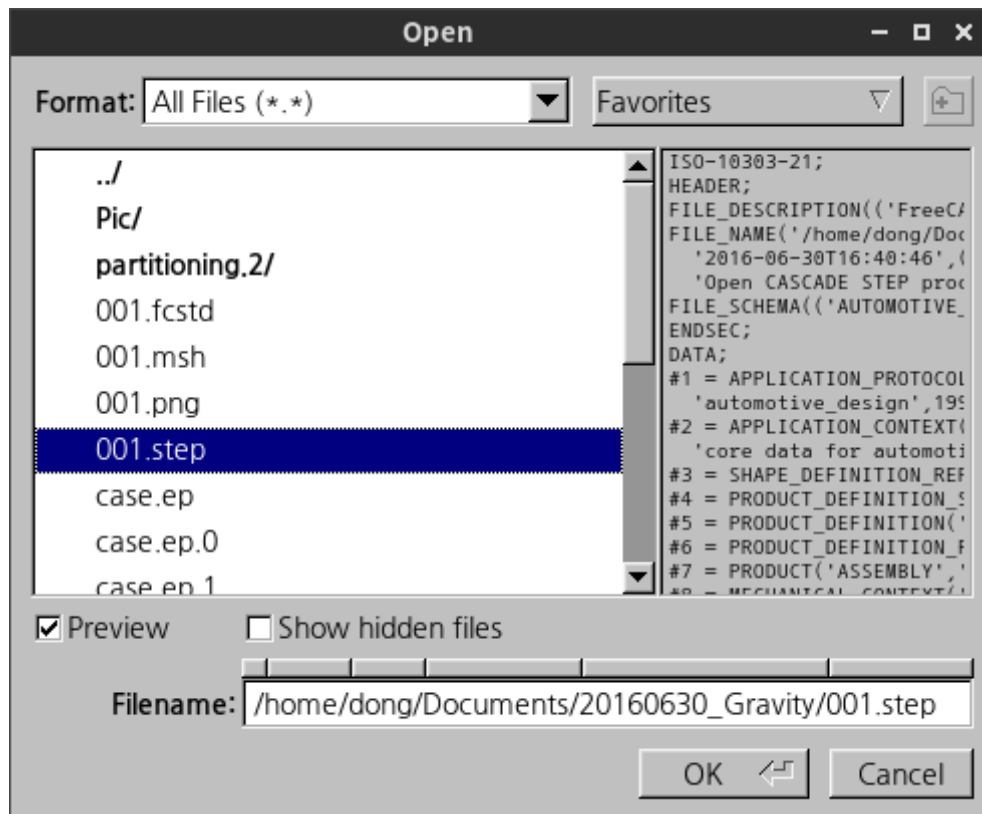
Check Geometry

Tools - Options
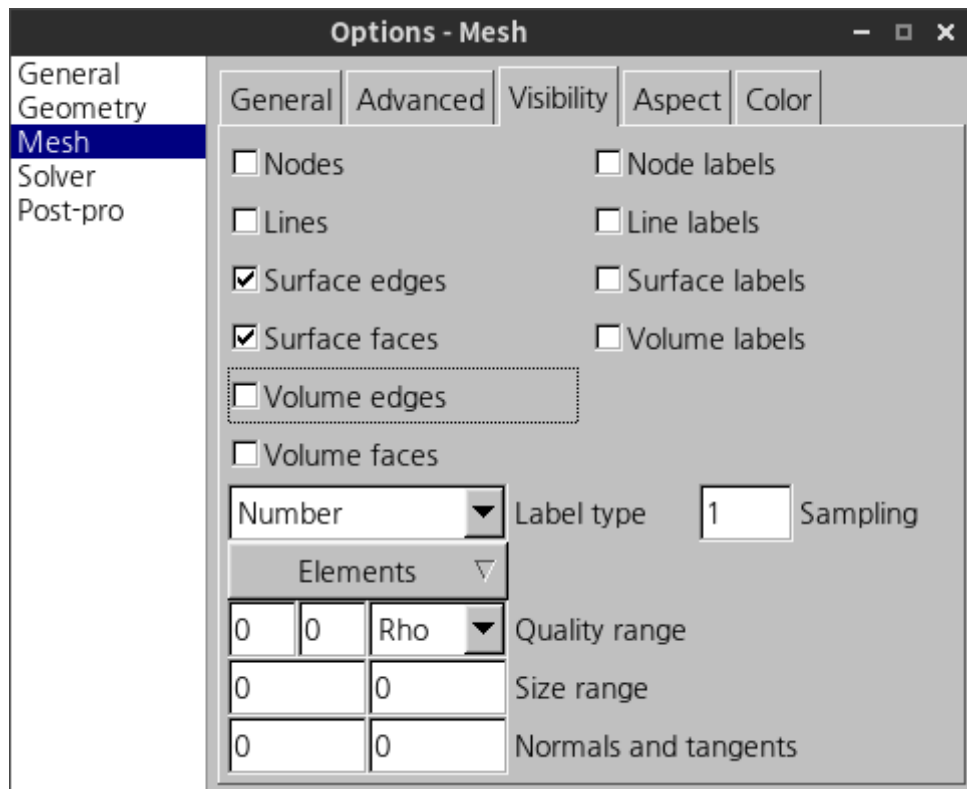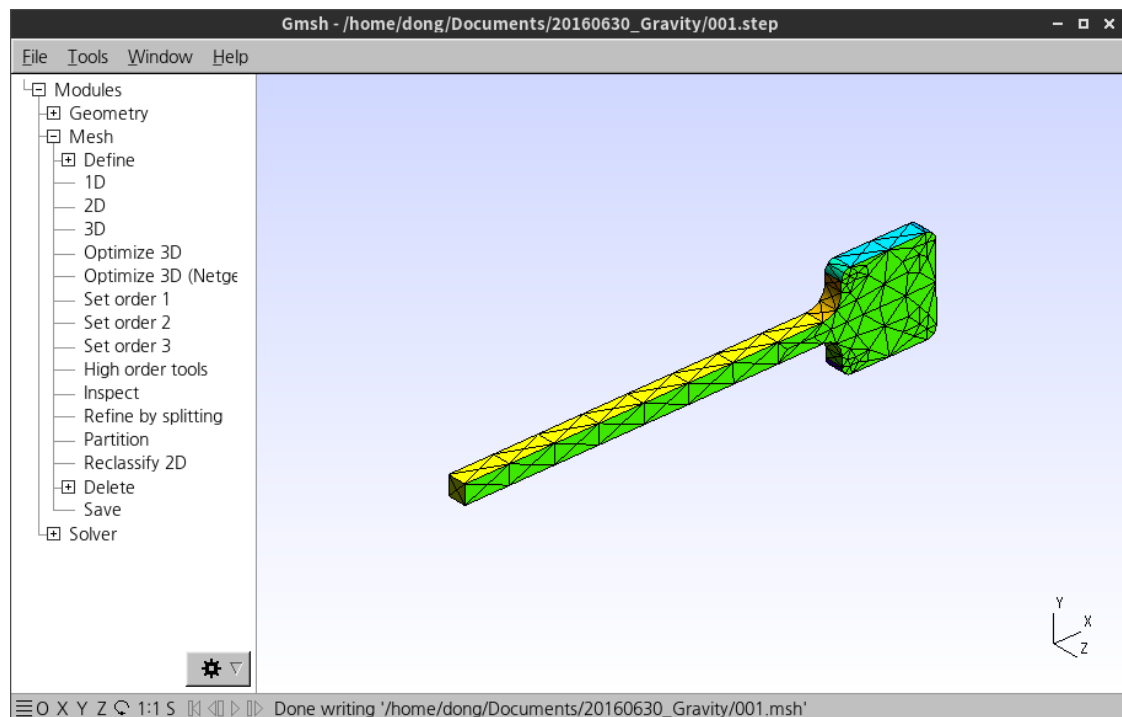
Check Geometry

Generate Mesh

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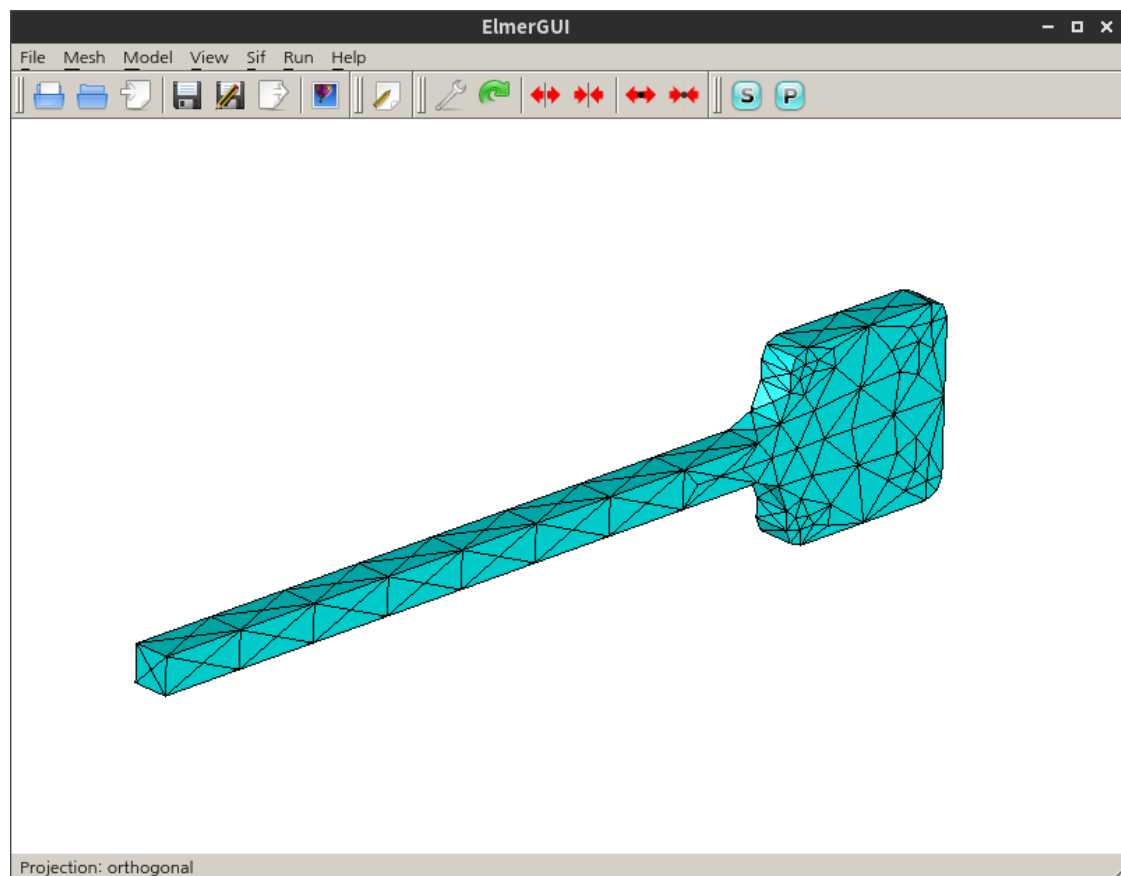
Options

Check & Export Mesh

Elmer

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

Open Mesh

Setup

- `Coordinate Scaling = Real 0.001`

Equation - Liniear Elasticity

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

Solver

- Check `Miscellaneous Options`
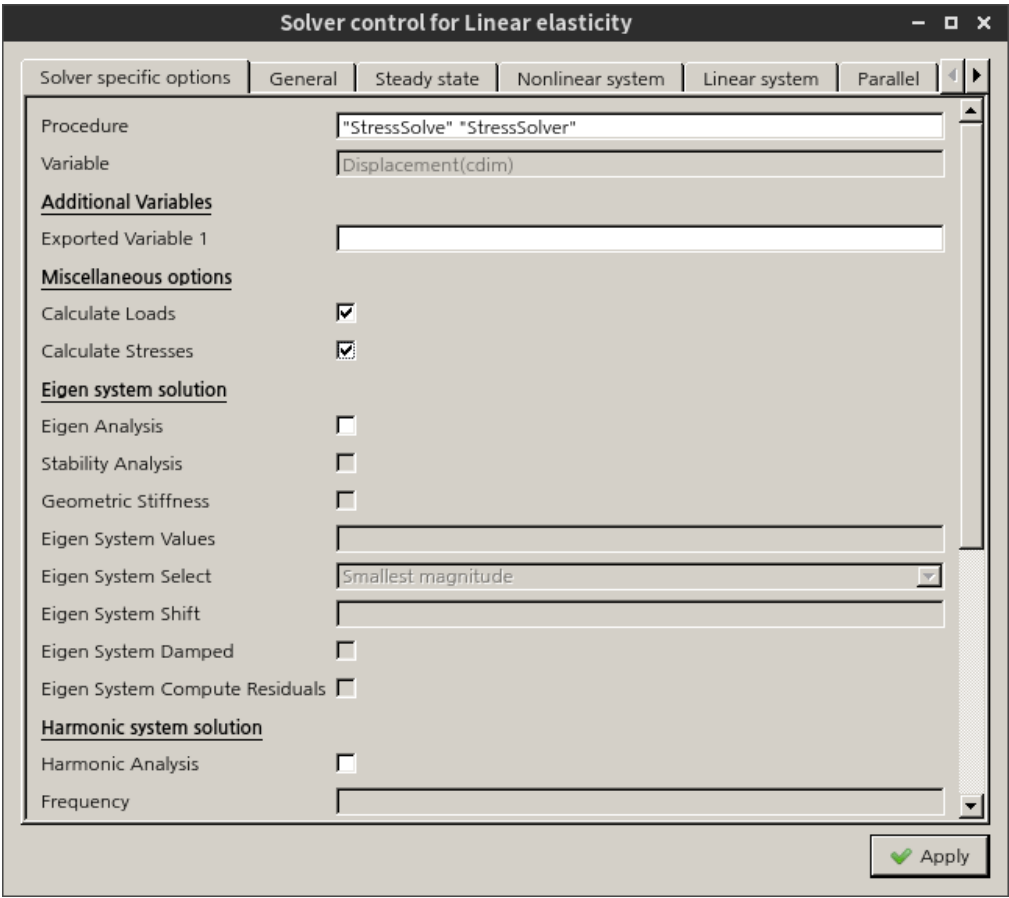
Equation - Result Output

- Check `Active`
- Press `Edit Solver Settings`

Solver

- Tab `General` - Check `After Timestep`
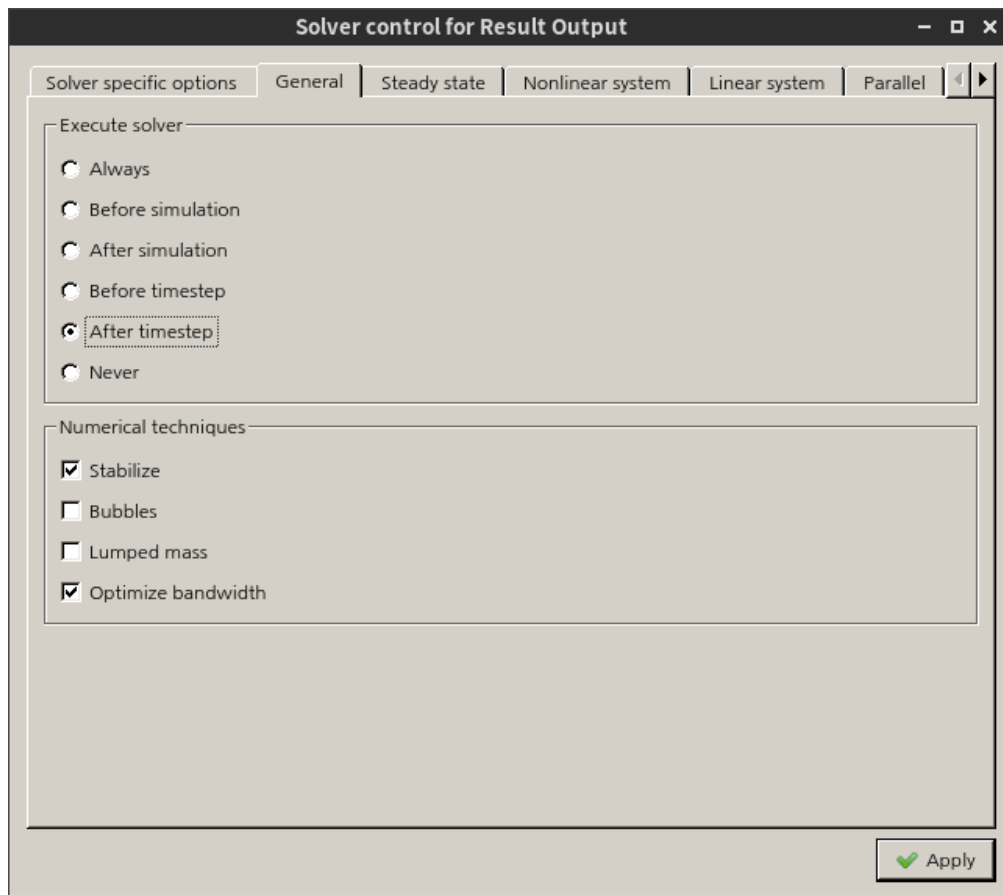
Material

- Check Body 1
- Press Material Library

Material Library

- Choice

Apply Material

- Check

Body Force

- Check Body 1

Body Force - Linear Elasticity

- Tab `Linear Elasticity`
- Apply Gravitation like that :

Boundary Condition

- Apply Boundary Condition like that :

Boundary Condition

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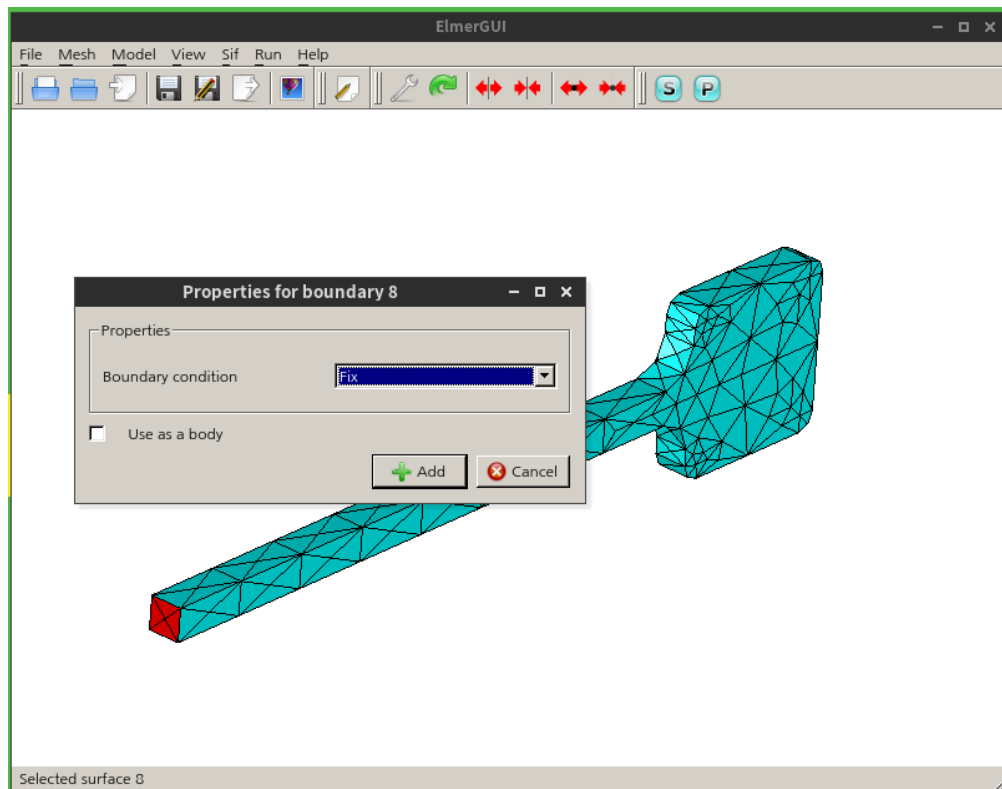
Check Models

- Check directions with `Compas`
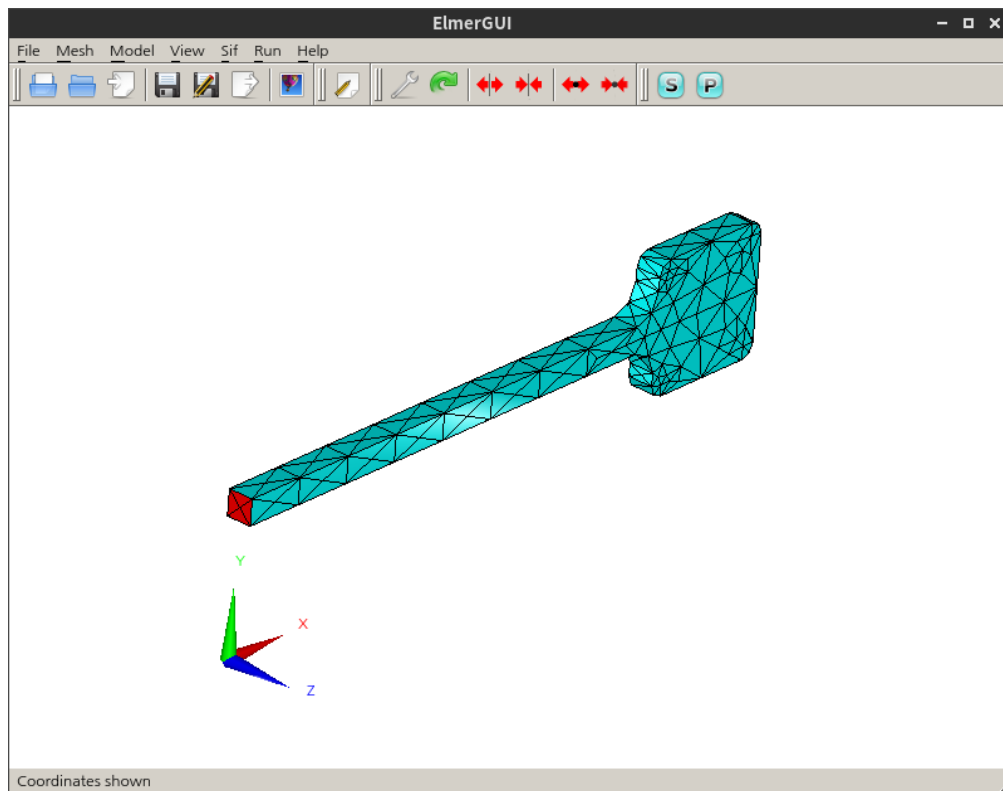
Run Solver

- Run - Start Solver

```
                                   Solver log                        —  □  ✕
 File   Edit

  ┌─────────────────────────────────────────────────────────────────────┐
  │  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: -------------------------------------            │
  │ ResultOutputSolve: 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
```
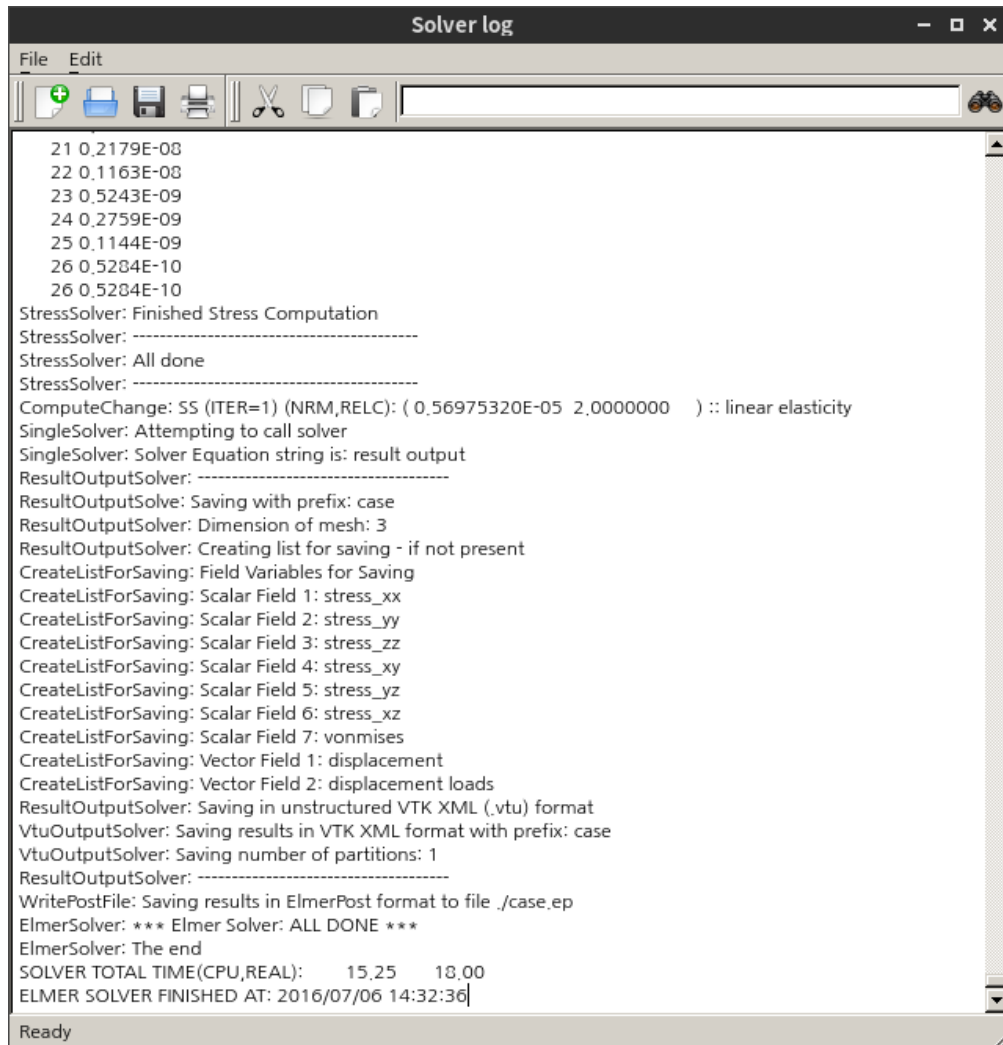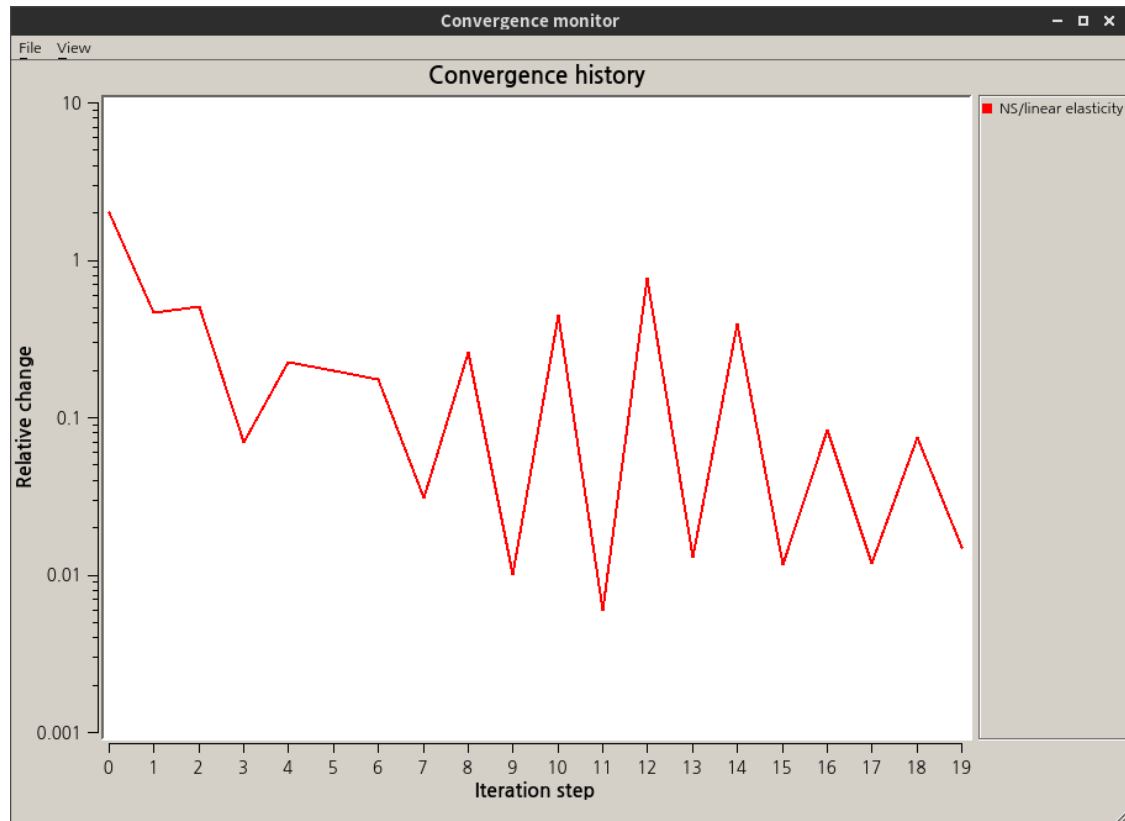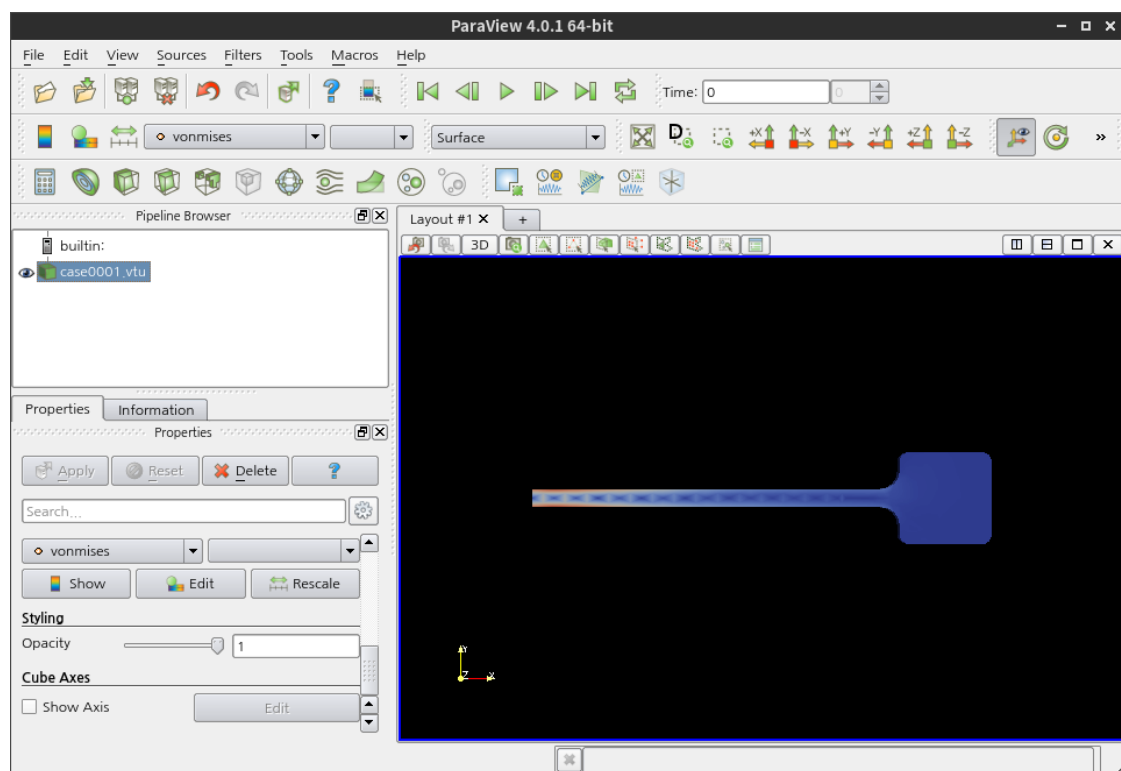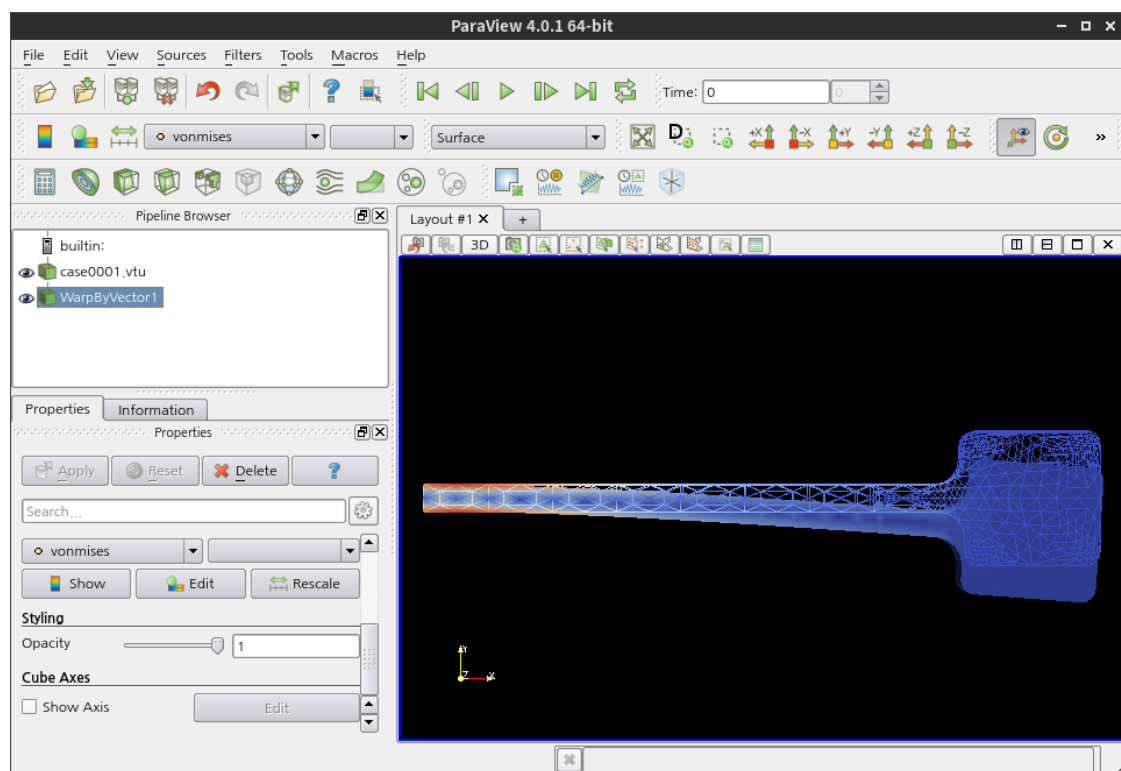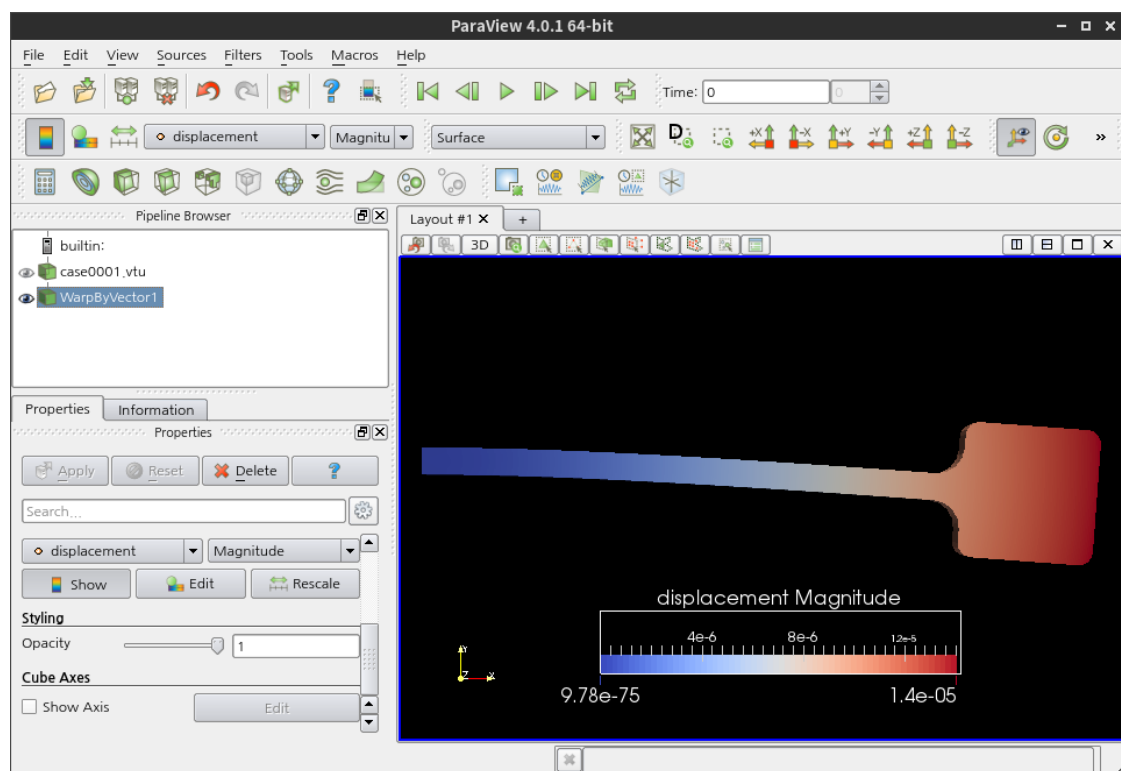
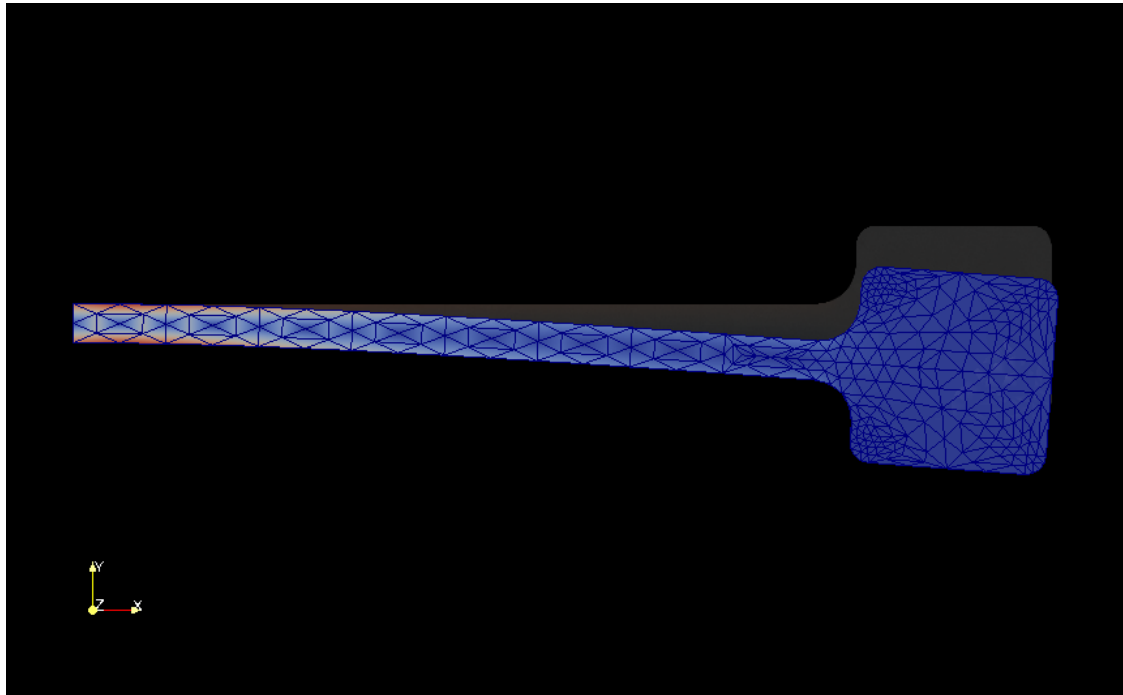## Run Solvere - Convergence

ParaView
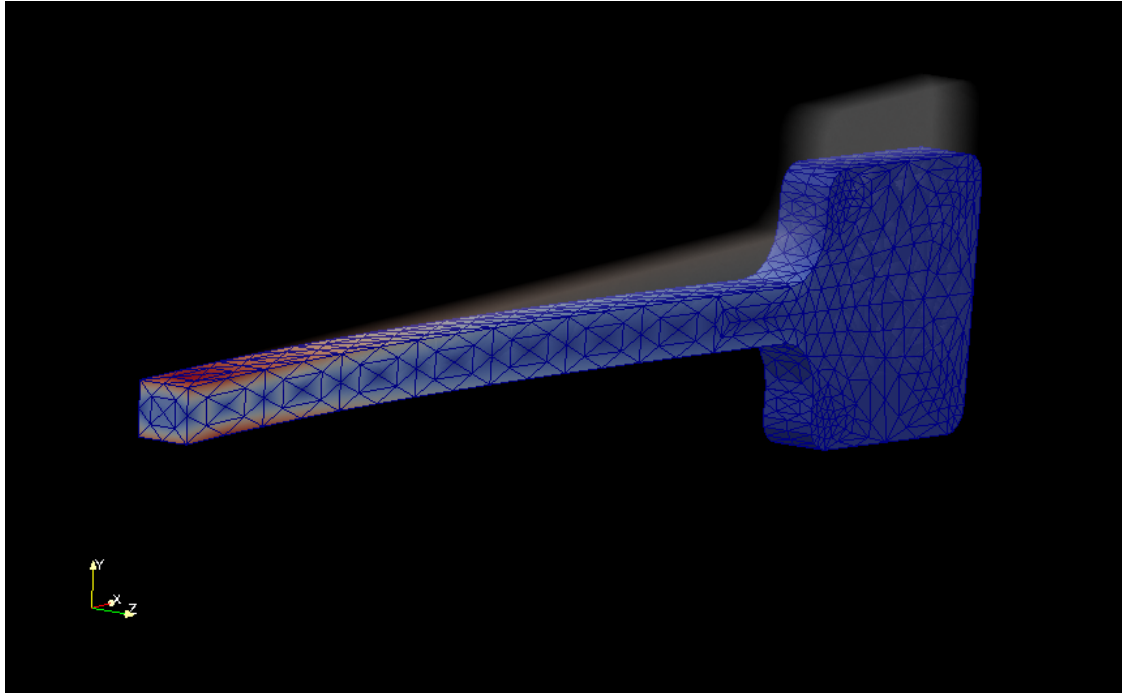
- 해석 결과 데이타 보기
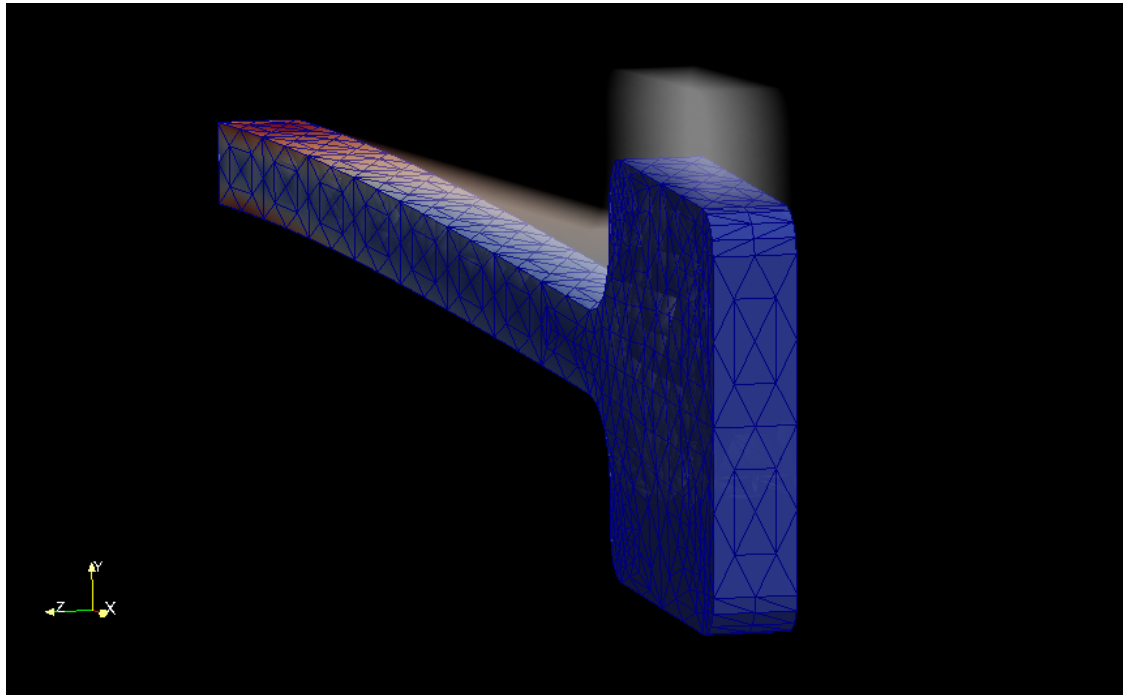
VonMises Stress

Displacement with Stress

Displacement

Details

Details

Details

Thank you!