

Elmer FEM – Parallelisation, nonlinear and time dependent problems

dr. Borut Černe

University of Ljubljana, Faculty of Mechanical Engineering

Univerza v Ljubljani



TECHNISCHE
UNIVERSITÄT
WIEN



CINECA

VSB TECHNICAL
UNIVERSITY
OF OSTRAVA

IT4INNOVATIONS
NATIONAL SUPERCOMPUTING
CENTER



Co-funded by the
Erasmus+ Programme
of the European Union

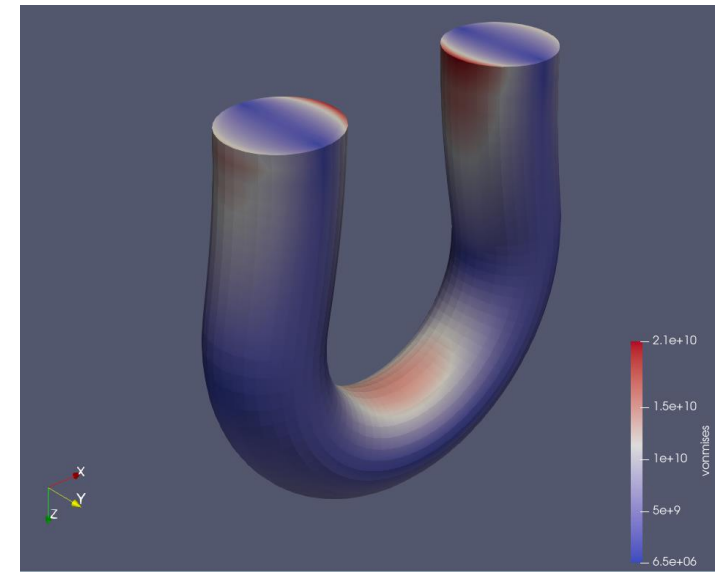
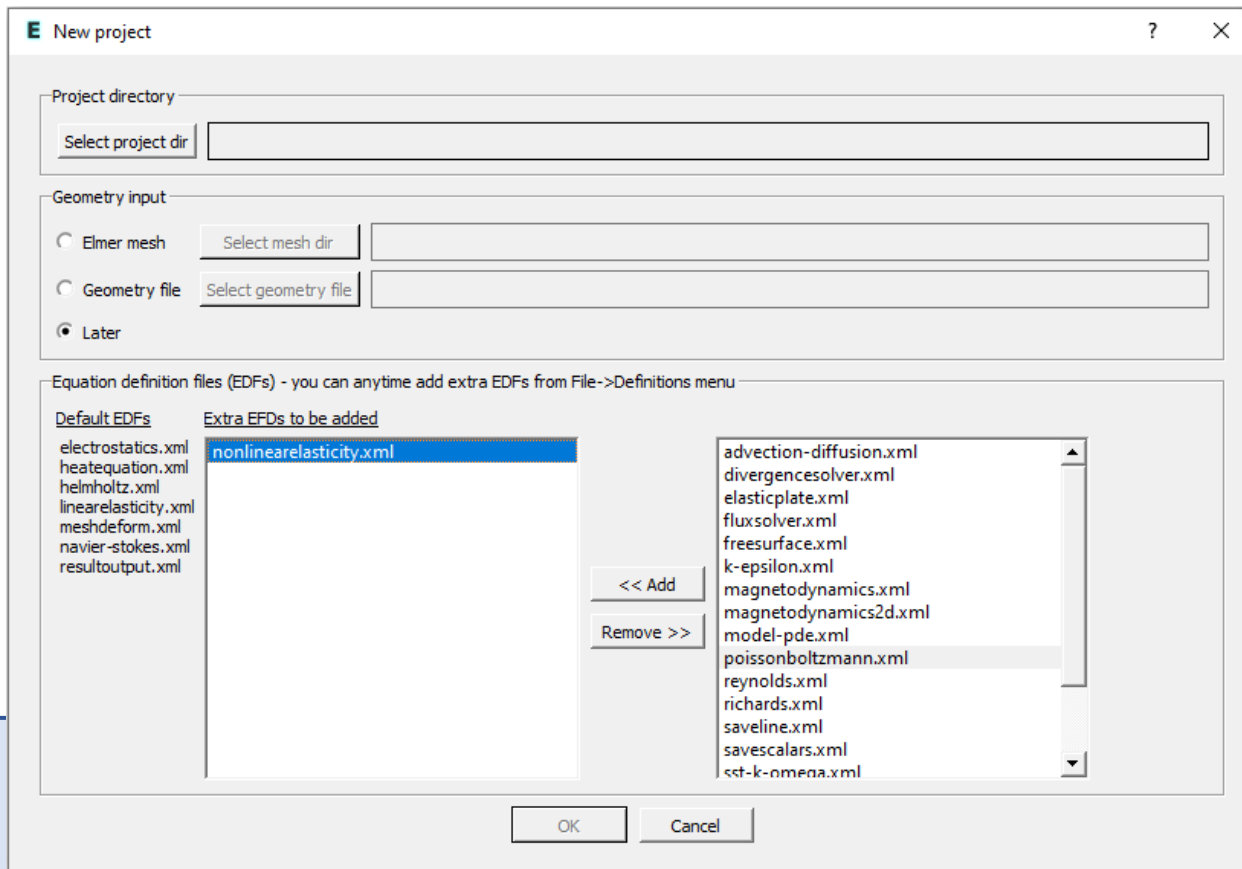
This project has been funded with support from the European Commission.

This publication [communication] reflects the views only of the author, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

Case 1: finite deformations on a U-shaped rod

Step 1 – Open ElmerFEM - New project – Choose directory and analysis type

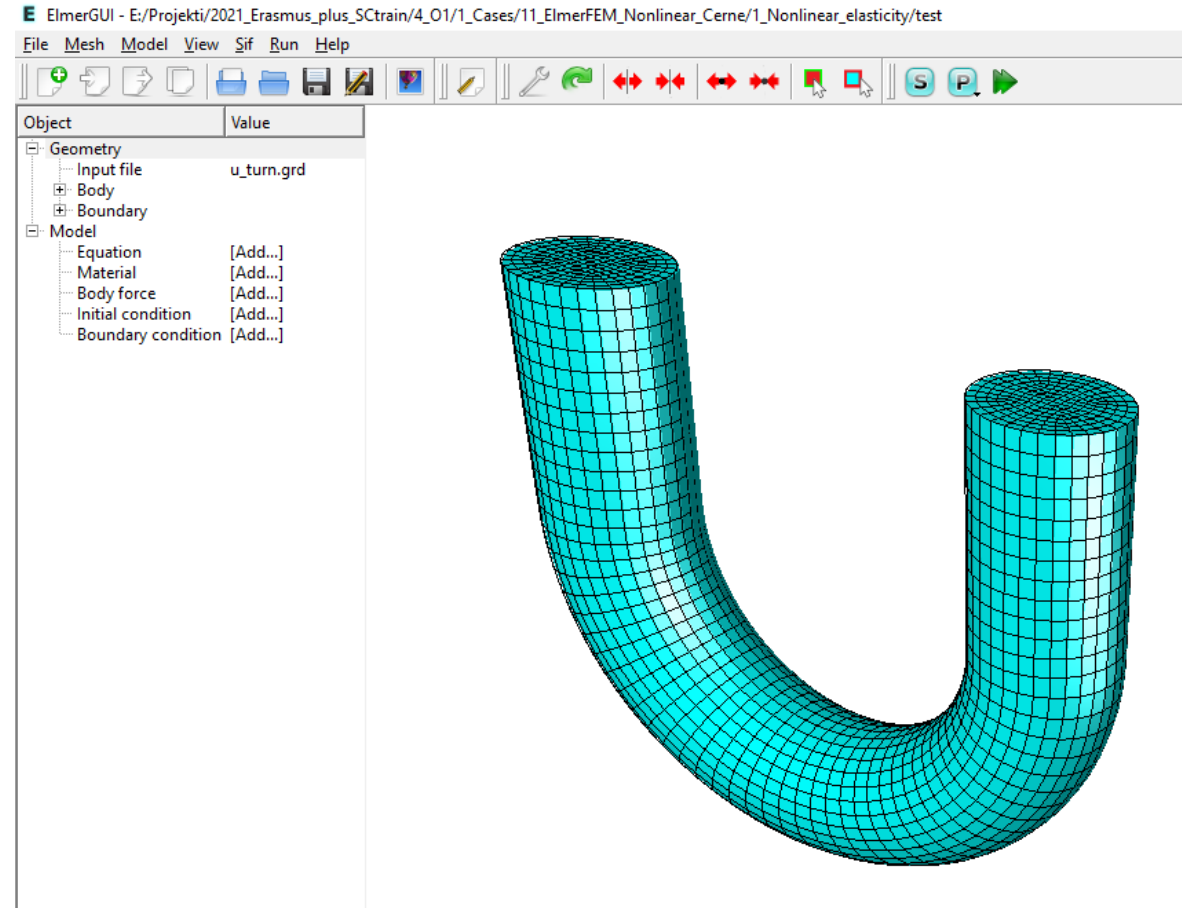
- module load elmer/foss-2018b
- ElmerGUI



Case 1: finite deformations on a U-shaped rod

Step 2 – Load Geometry

Open > select u_turn.grd file



Case 1: finite deformations on a U-shaped rod

Step 3 – Model setup

- Scale geometry to correct unit system
- Define time-dependent type analysis

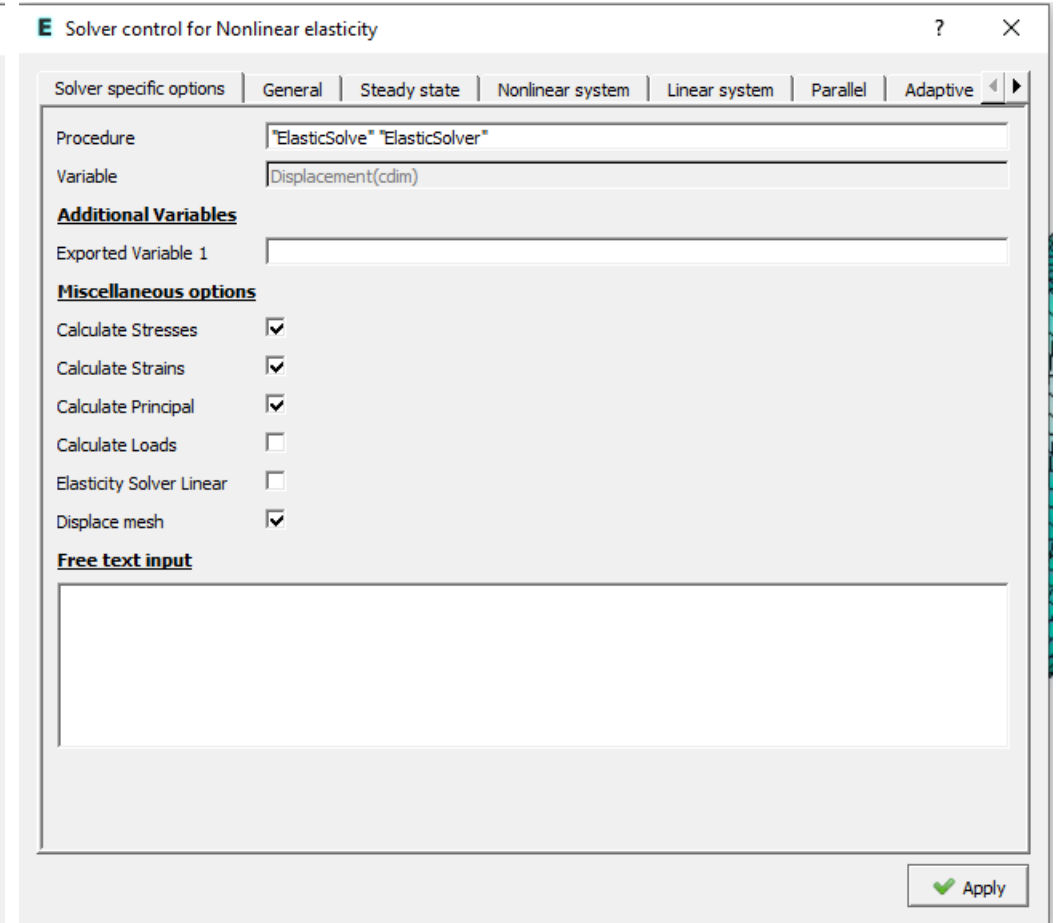
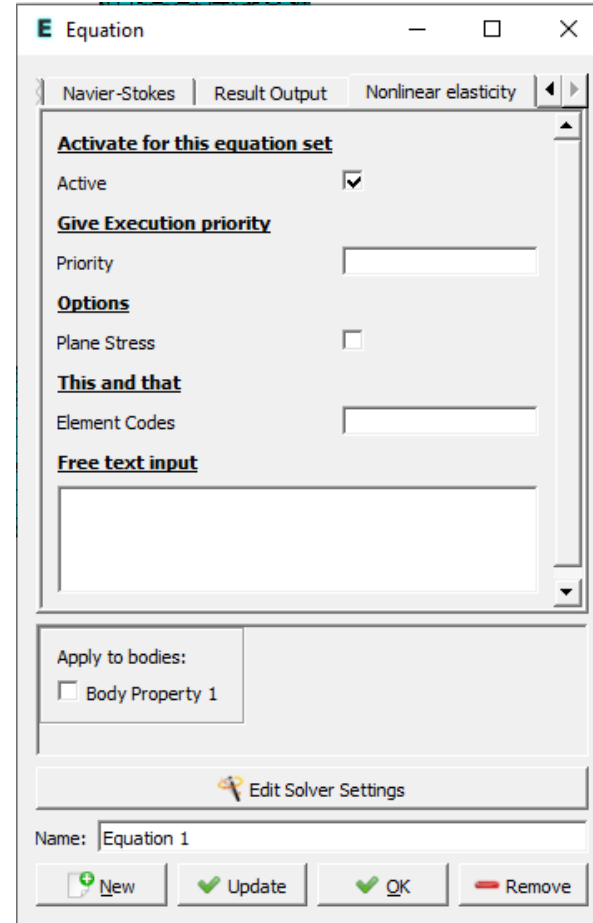
The screenshot shows the 'Setup' dialog box in SCtrain, divided into three main sections: Header, Simulation, and Constants. The 'Header' section includes a checked 'Check keywords warn' option and fields for MeshDB, Include path, Results directory, and Free text. The 'Simulation' section contains various parameters: 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 (Transient), Timestep intervals (20), Output intervals (1), Timestep sizes (0.05), Coordinate Scaling (0.01), Angular Frequency, Solver input file (case.sif), and Post file (case.vtu). The 'Constants' section includes Gravity (0 -1 0 9.82), Boltzmann (1.3807e-23), Stefan Boltzmann (5.67e-08), Unit charge (1.602e-19), and Vacuum permittivity (8.8542e-12). An 'Apply' button is located at the bottom right.

| Section | Parameter | Value |
|---------------------|------------------------|-------------------------------------|
| Header | Check keywords warn | <input checked="" type="checkbox"/> |
| | MeshDB | . |
| | Include path | |
| | Results directory | |
| 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 | Transient |
| | Timestep intervals | 20 |
| | Output intervals | 1 |
| | Timestep sizes | 0.05 |
| Constants | Coordinate Scaling | 0.01 |
| | Angular Frequency | |
| | Solver input file | case.sif |
| | Post file | case.vtu |
| | 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 | |

Case 1: finite deformations on a U-shaped rod

Step 4 – Select model

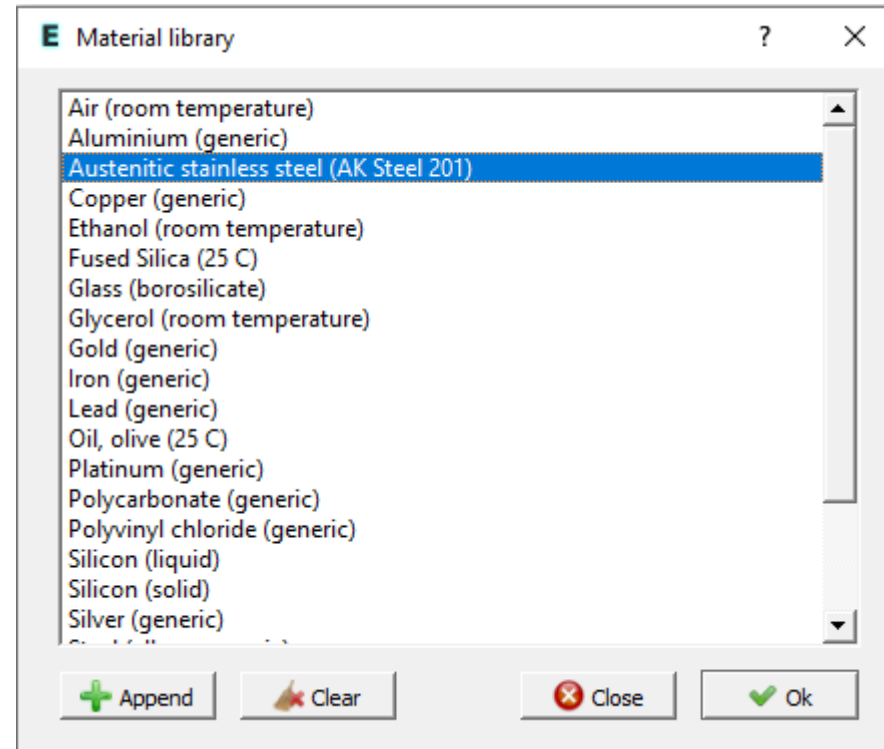
- *Equation* > Nonlinear elasticity
- Edit Solver Settings



Case 1: finite deformations on a U-shaped rod

Step 5 – material

- *Material* > Material library > Austenitic stainless steel



Case 1: finite deformations on a U-shaped rod

Step 6 – Boundary conditions

- Boundary condition > Left

The screenshot shows the 'BoundaryCondition' dialog box for the 'Left' boundary. The 'Normal-Tangential Coordinate System' section has 'Use normal-tangential coordinate system' unchecked and 'Change of variables' checked. Under 'Dirichlet Conditions', 'Displacement 1' is set to 'Variable "time"; Real MATC "0.006*tx"', while 'Displacement 2' and 'Displacement 3' are set to '0'. The 'Apply to boundaries' section has 'Boundary 1' and 'Boundary 2' unchecked, and 'Boundary 3' checked. The 'Name' field contains 'Left'. At the bottom, there are buttons for 'New', 'Update', 'OK', and 'Remove'.

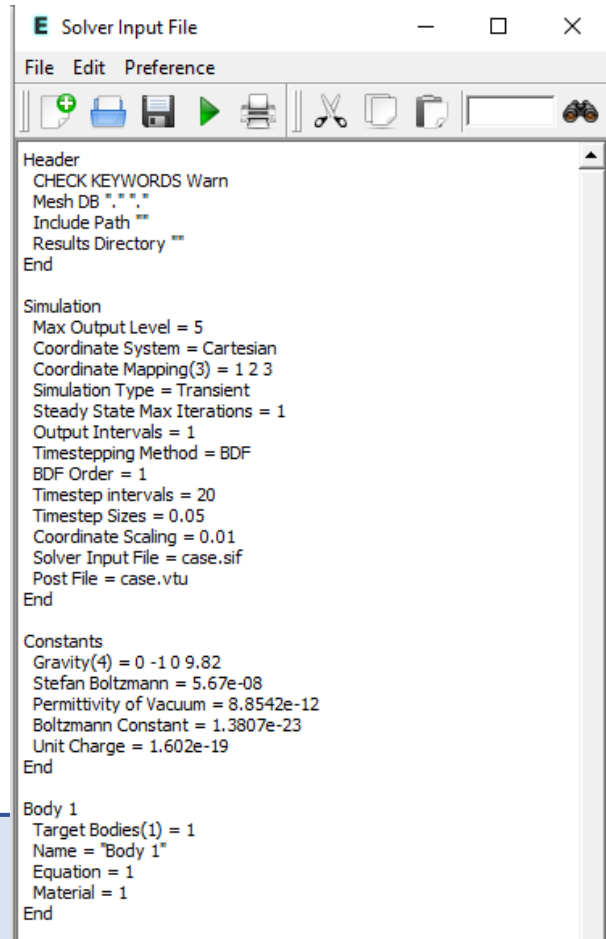
- Boundary condition > Right

The screenshot shows the 'BoundaryCondition' dialog box for the 'Right' boundary. The 'Normal-Tangential Coordinate System' section has 'Use normal-tangential coordinate system' unchecked and 'Change of variables' checked. Under 'Dirichlet Conditions', 'Displacement 1' is set to 'Variable "time"; Real MATC "-0.006*tx"', while 'Displacement 2' and 'Displacement 3' are set to '0'. The 'Apply to boundaries' section has 'Boundary 1' checked, 'Boundary 2' unchecked, and 'Boundary 3' unchecked. The 'Name' field contains 'Right'. At the bottom, there are buttons for 'New', 'Update', 'OK', and 'Remove'.

Case 1: finite deformations on a U-shaped rod

Step 7 – Run analysis

- Generate .sif file



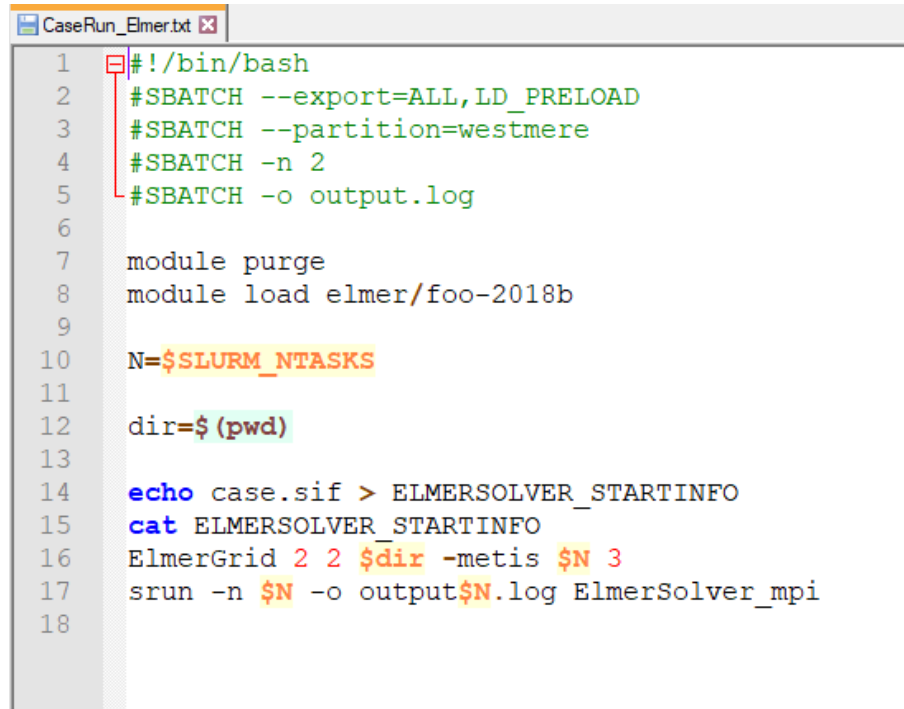
```
Header
CHECK KEYWORDS Warn
Mesh DB ""
Include Path ""
Results Directory ""
End

Simulation
Max Output Level = 5
Coordinate System = Cartesian
Coordinate Mapping(3) = 1 2 3
Simulation Type = Transient
Steady State Max Iterations = 1
Output Intervals = 1
Timestepping Method = BDF
BDF Order = 1
Timestep intervals = 20
Timestep Sizes = 0.05
Coordinate Scaling = 0.01
Solver Input File = case.sif
Post File = case.vtu
End

Constants
Gravity(4) = 0 -10 9.82
Stefan Boltzmann = 5.67e-08
Permittivity of Vacuum = 8.8542e-12
Boltzmann Constant = 1.3807e-23
Unit Charge = 1.602e-19
End

Body 1
Target Bodies(1) = 1
Name = "Body 1"
Equation = 1
Material = 1
End
```

- Use batch file to run the cases

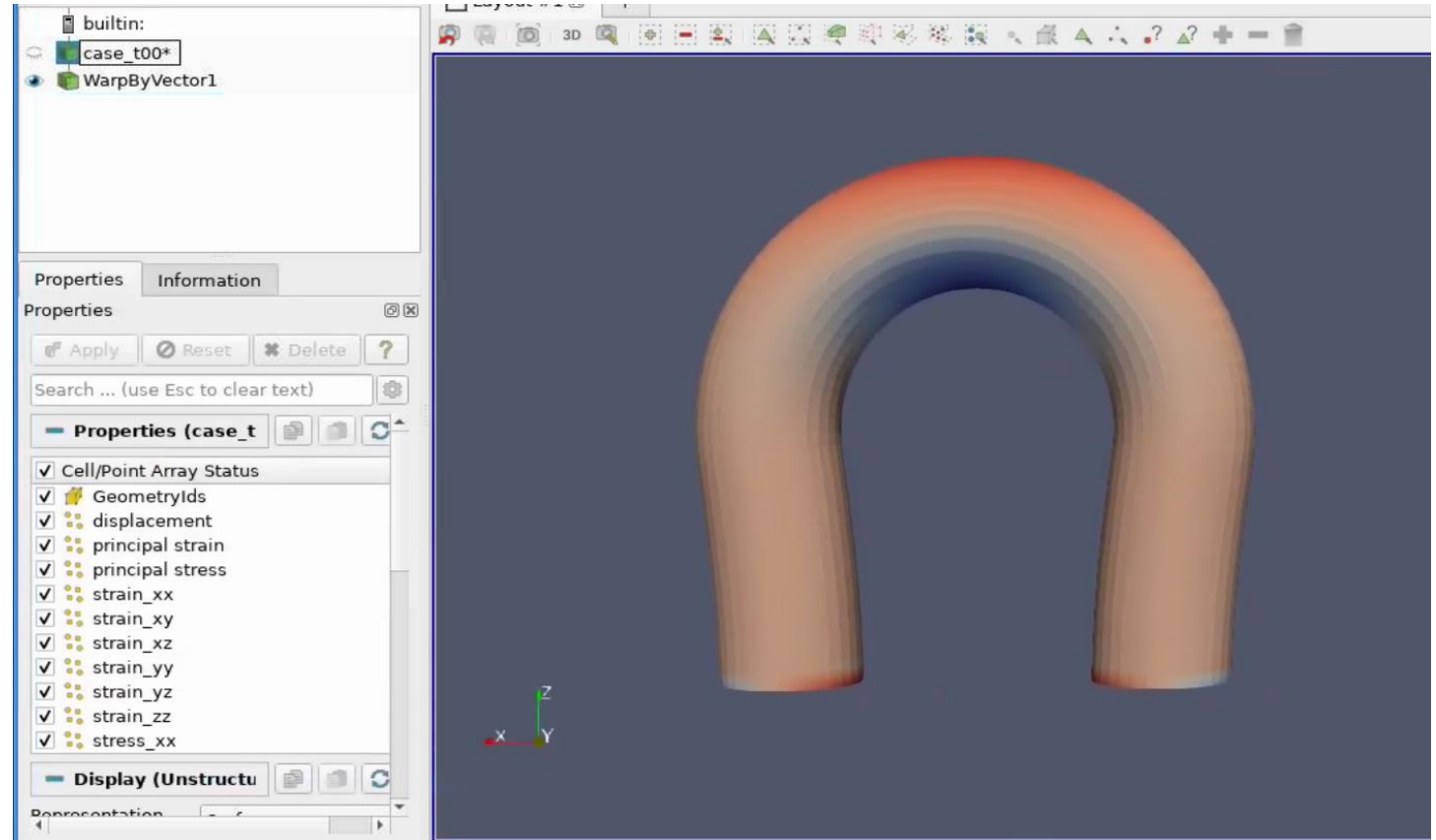


```
CaseRun_Elmer.txt
1  #!/bin/bash
2  #SBATCH --export=ALL,LD_PRELOAD
3  #SBATCH --partition=westmere
4  #SBATCH -n 2
5  #SBATCH -o output.log
6
7  module purge
8  module load elmer/foo-2018b
9
10 N=$SLURM_NTASKS
11
12 dir=$(pwd)
13
14 echo case.sif > ELMERSOLVER_STARTINFO
15 cat ELMERSOLVER_STARTINFO
16 ElmerGrid 2 2 $dir -metis $N 3
17 srun -n $N -o output$N.log ElmerSolver_mpi
18
```


Case 1: finite deformations on a U-shaped rod

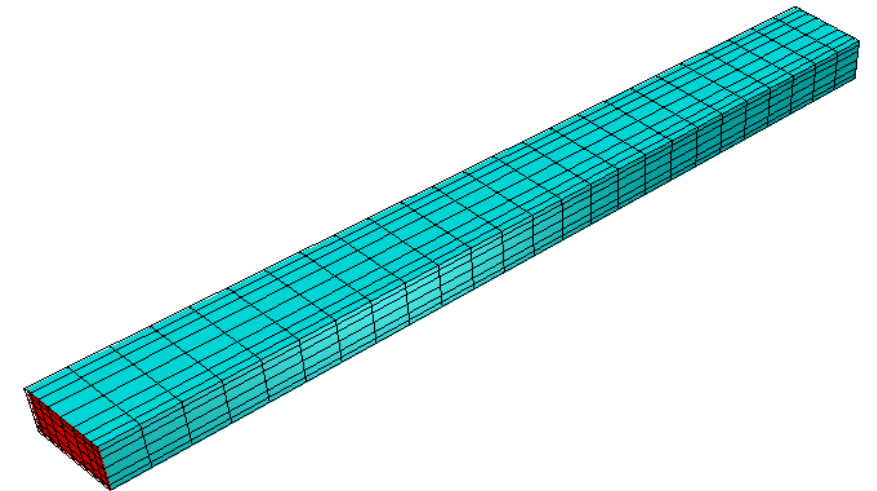
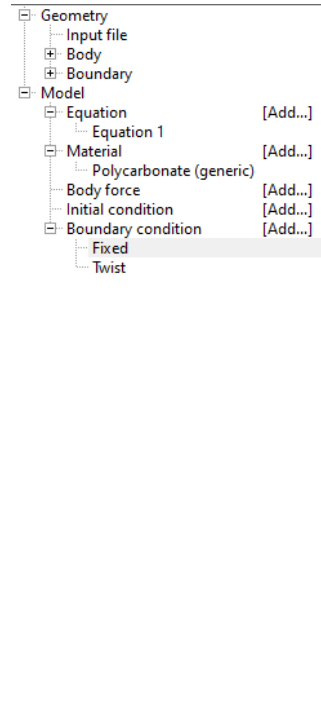
Step 8 – Postprocessing - Paraview

- module load
ParaView/5.8.1-foss-
2020b-mpi
- paraview



Case 2: Beam twisting

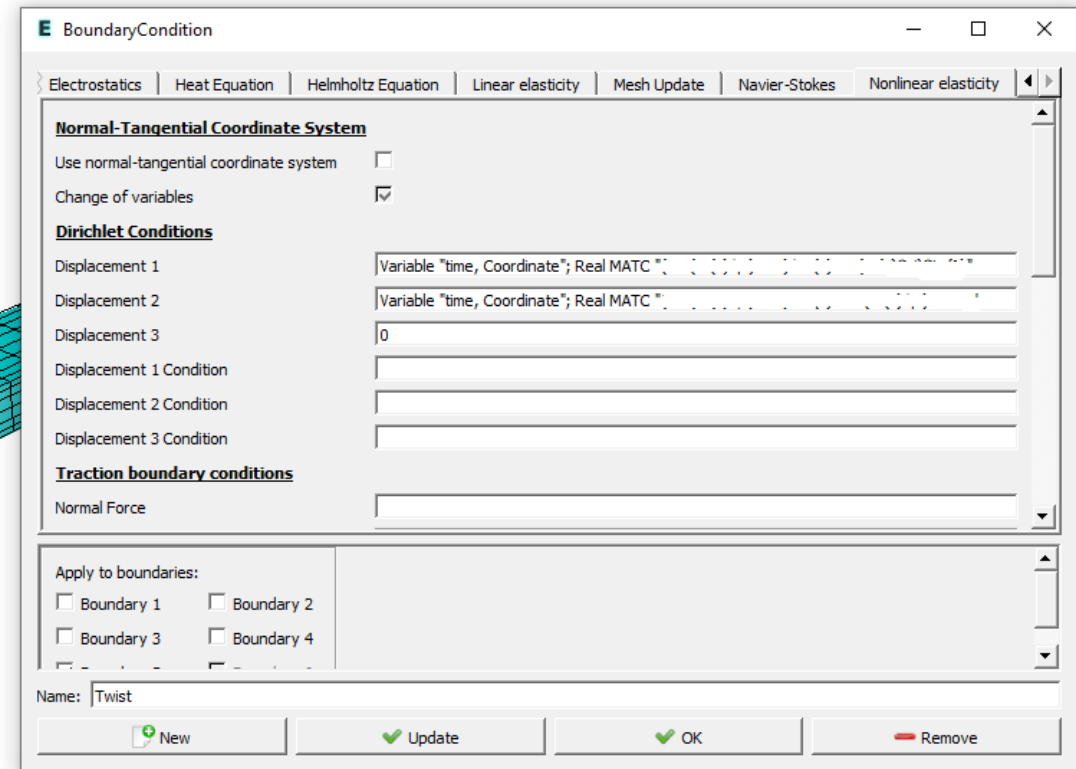
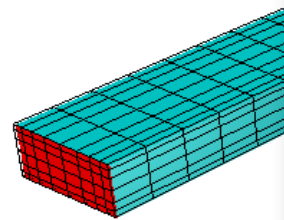
- Use similar procedure as in the previous example
- Analysis is again time-dependent: use 10 x 0.1 s time steps
- Mesh file: beam3d.grd
- Use nonlinear elasticity model
- The material in this case is Polycarbonate
- One side is fixed



Case 2: Beam twisting

- On the other side apply rotational (twist) boundary condition
- Use set of equations below
- Use variables tx(0) for time and tx(1), tx(2) for coordinate displacements

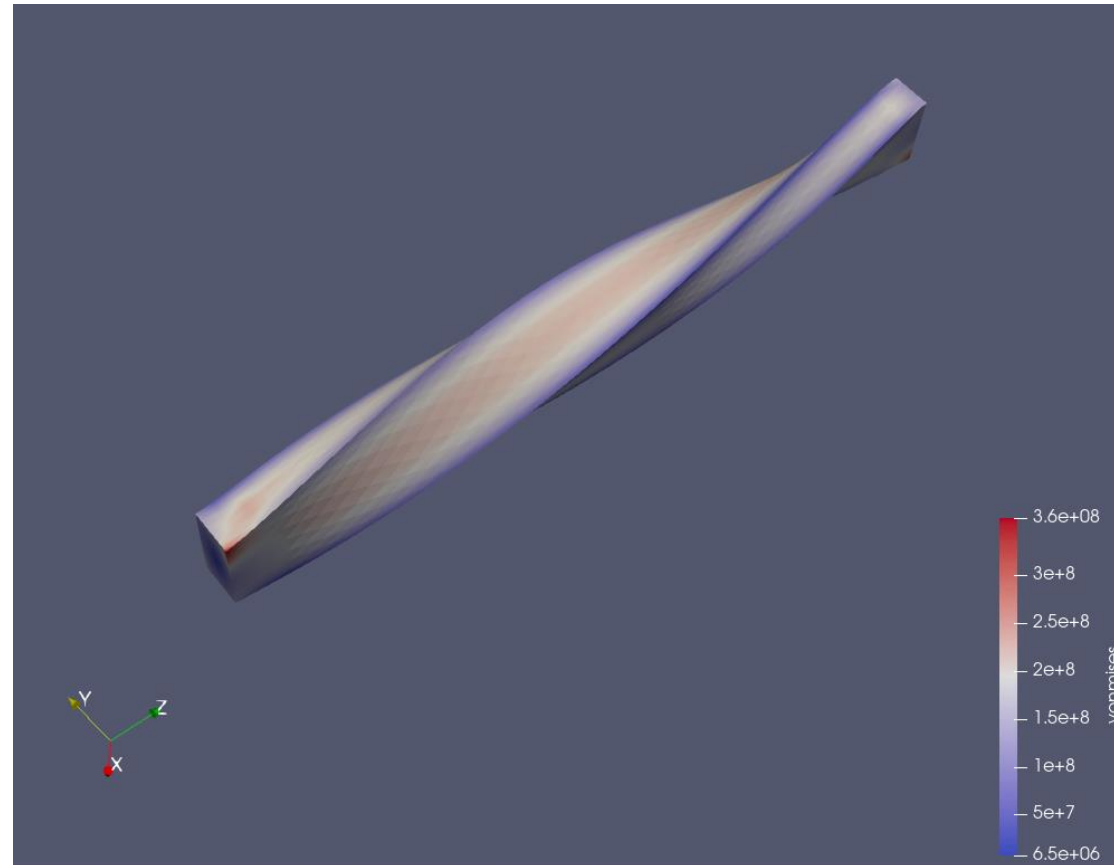
```
etry  
put file  
ody  
oundary  
:!  
quation [Add...]  
--- Equation 1  
aterial [Add...]  
--- Polycarbonate (generic)  
ody force [Add...]  
itial condition [Add...]  
oundary condition [Add...]  
--- Fixed  
--- Twist
```



$$u_x = (\cos(\phi) - 1)x - \sin(\phi)y$$
$$u_y = (\cos(\phi) - 1)y + \sin(\phi)x$$

Case 2: Beam twisting

- Perform analysis using .sif input file and CRun_Elmer.txt
- Visualize time-dependent results in ParaView



Thank you for your attention!

<http://sctrain.eu/>

Univerza v Ljubljani



TECHNISCHE
UNIVERSITÄT
WIEN



VSB TECHNICAL
UNIVERSITY
OF OSTRAVA

IT4INNOVATIONS
NATIONAL SUPERCOMPUTING
CENTER



Co-funded by the
Erasmus+ Programme
of the European Union

This project has been funded with support from the European Commission.

This publication [communication] reflects the views only of the author, and the Commission cannot be held responsible for any use which may be made of the information contained therein.