PrePoMax v1.1.0 Examples

26th of June 2021

Written by Jakub Michalski and Matej Borovinšek

Contents

1 Pressure loaded ring segment

The first example is a ring segment (one quarter of a ring) subjected to pressure loading. The dimensions are:

- Outer diameter: 150 mm
- Inner diameter: 90 mm
- Thickness: 20 mm

1) Create a new file with [mm, ton, s, °C] units and import geometry in step format to PrePoMax. Then mesh the part. In this case maximum element size of 3 mm was selected while other settings were left with no changes (Figure 1).

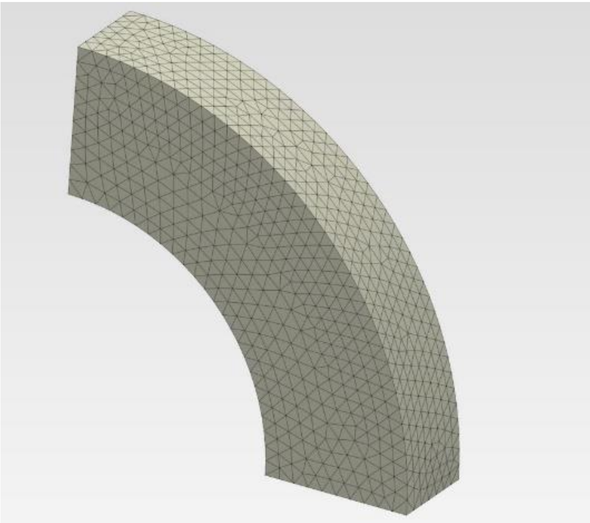

Figure 1: Ring segment – mesh

2) Create a node set fix (bottom side) and surface press (top side) for further use when applying boundary condition and load.

3) Define a new material named Steel, add Elastic behavior and specify Young's modulus of 210000 MPa and Poisson's ratio of 0.3. Create a new solid section named Steel_section, with Steel as a material and select the ring segment so that the section will be assigned to it.

4) Define static analysis step with default settings.

5) Add Fixed boundary condition to the node set named fix. Add Pressure load to the surface named press, specify the magnitude of 20 MPa (Figure 2).

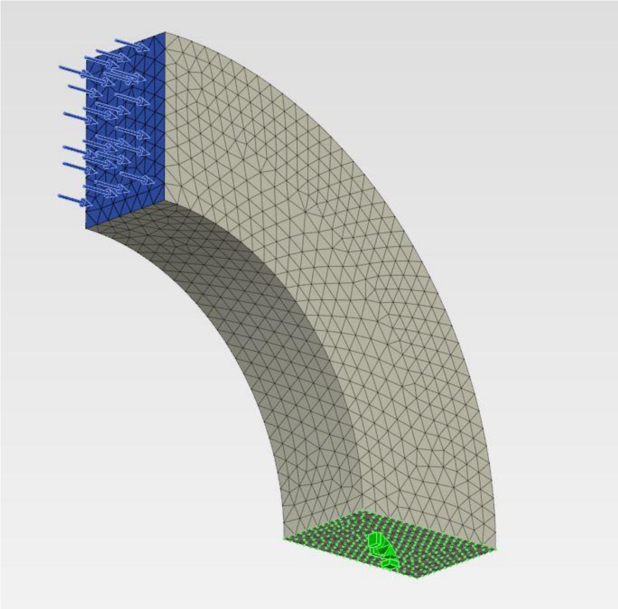

Figure 2: Ring segment – boundary conditions and loads

6) Since all the necessary definitions were made, the analysis can be submitted. Wait until it finishes and open the results.

7) Examine Mises stress and displacement contour plots. Change the color spectrum type to Rainbow. Hide max value label, change number format to general and set the deformation scale factor to 40 (Figure 3). Analytically calculated maximum stress value is 230.68 MPa and similar stresses can be found in simulation results when checking the outer shorter edge of the fixed face.

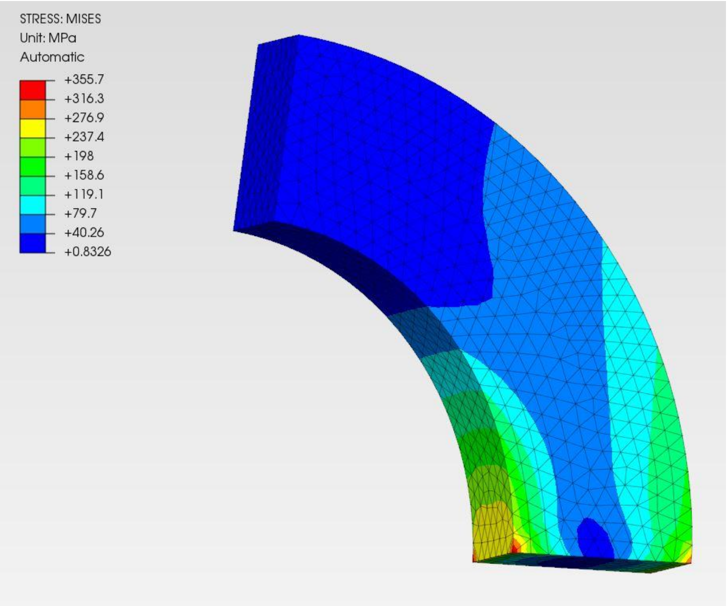

Figure 3: Ring segment – von Mises stress

2 Shaft with surface traction loading

In this example a short circular shaft is subjected to surface traction loading that causes bending. The dimensions are:

- Diameter: 50 mm
- Length: 200 mm

1) Create a new file with [mm, ton, s, °C] units and import geometry in step format to PrePoMax. Then mesh the part. In this case maximum element size of 4 mm was selected while other settings were left with no changes (Figure 4).

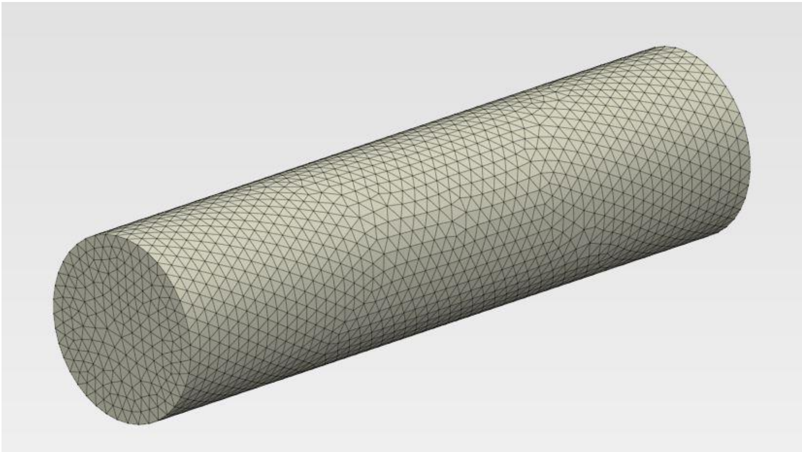

Figure 4: Shaft – mesh

2) Define a new material, add Elastic behavior and specify Young's modulus of 210000 MPa and Poisson's ratio of 0.3. Create a new solid section referencing previously created material and select the shaft so that the section will be assigned to this part.

3) Define a static analysis step with default settings. Assign Fixed boundary condition to one end of the shaft and Surface traction loading to the other one. Specify the value of -500 N in the direction perpendicular to the shaft's axis (Figure 5).

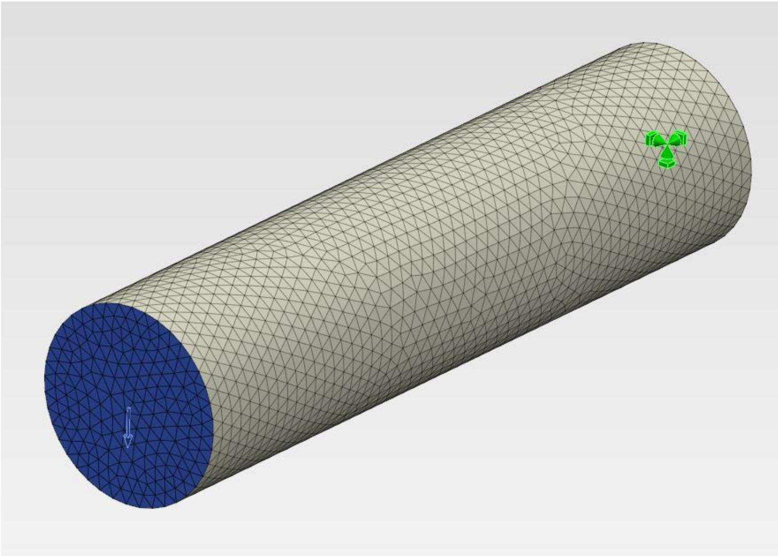

Figure 5: Shaft – boundary conditions and loads

4) Run the analysis and open the results when it completes. Change the deformation scale factor to 400 and turn on the option to draw undeformed model. Examine the displacements contour plot, hide undeformed model shape and check the Mises stress plot (Figure 6). Analytically calculated maximum stress is 8.17 MPa and similar values can be found when checking the results of the analysis.

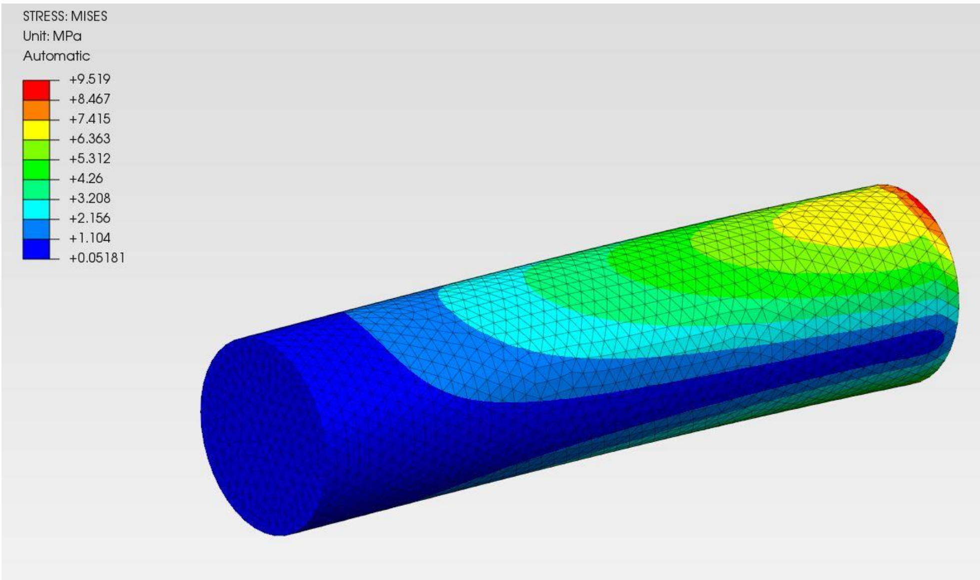

Figure 6: Shaft – von Mises stress

3 Elliptical bar torsion

In this case an elliptical section bar is subjected to torsion. The dimensions are:

- Major axis: 200 mm
- Minor axis: 100 mm
- Length: 1000 mm

1) Create a new file with [mm, ton, s, °C] units and import geometry in step format to PrePoMax. Then mesh the part. In this case maximum element size of 10 mm was selected while other settings were left with no changes (Figure 7).

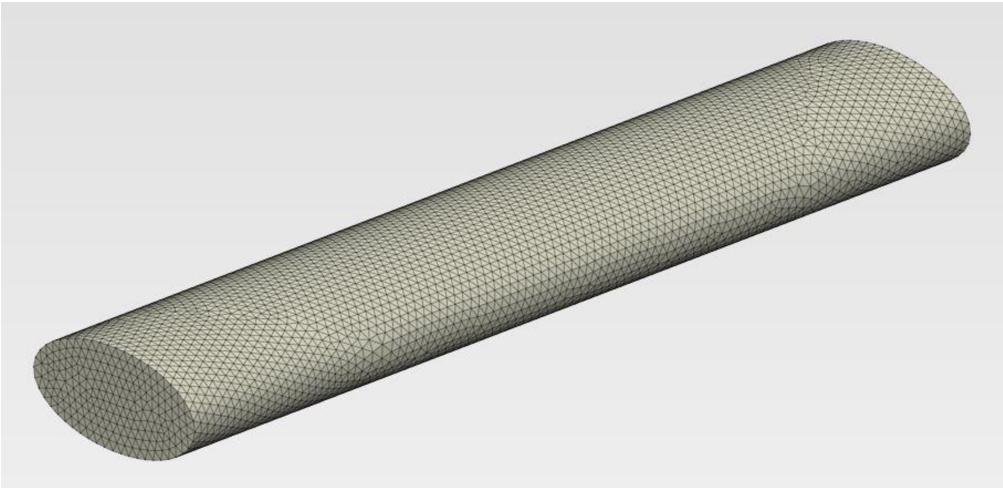

Figure 7: Elliptical bar – mesh

2) Define a new material, add Elastic behavior and specify Young's modulus of 210000 MPa and Poisson's ratio of 0.3. Create a new solid section referencing previously created material and select the shaft so that the section will be assigned to this part.

3) Create a reference point at (0, 0, 1000) coordinates. Create a rigid body constraint using this reference point and assign it to the surface of the shaft.

4) Define a static analysis step with default settings. Assign Fixed boundary condition to the surface of the shaft opposite to the one with reference point. Create Moment load and apply it to the reference point, specify the magnitude of 1000000 Nmm for the axis of the bar (Figure 8).

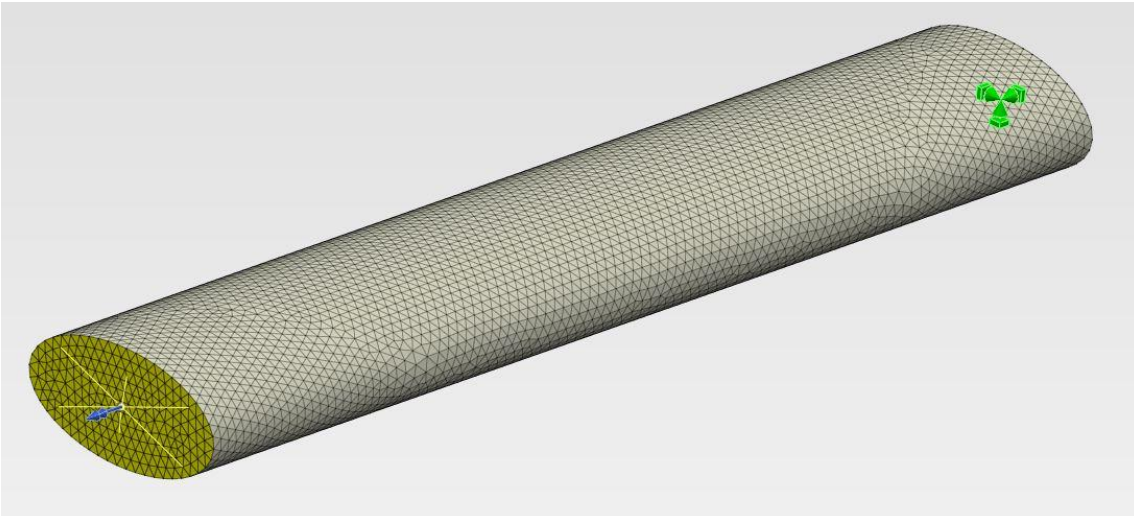

Figure 8: Elliptical bar – boundary conditions and loads

5) Run the analysis and open the results when it's completed. In this case analytically calculated maximum shear stress is 2.547 MPa. In order to compare the numerical result to it switch to proper stress component (depending on how your model is aligned with global coordinate system's axes). Use the Query → Point/Node tool to examine stress values in various locations on the bar. Also, show the label pointing to the node with maximum shear stress. In this case the result was 2.555 MPa, so very close to the analytical solution (Figure 9).

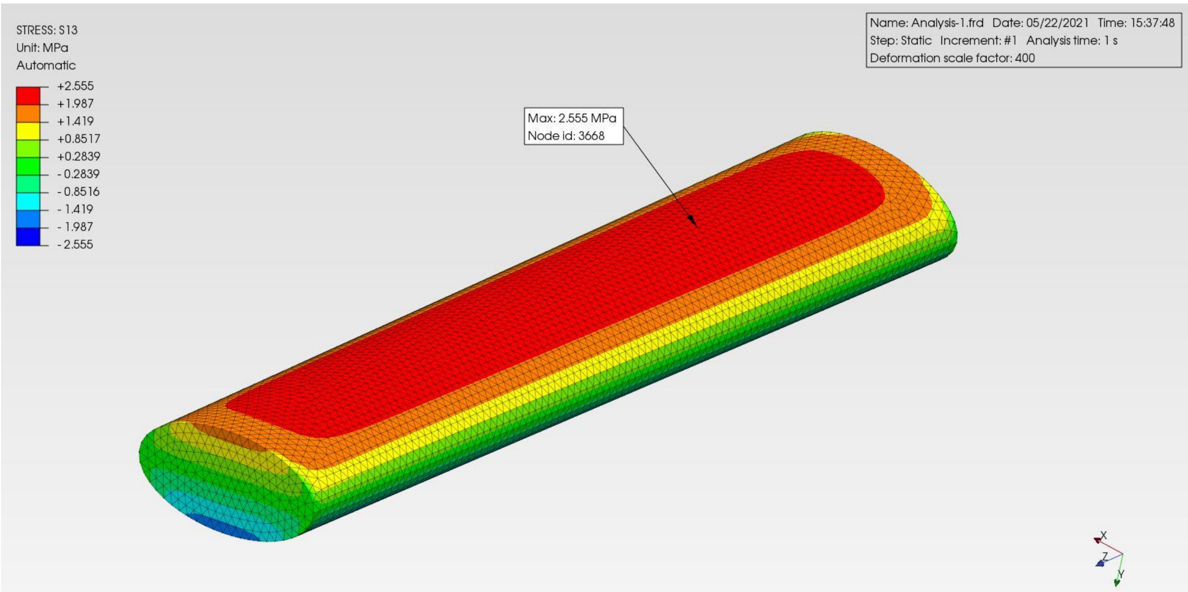

Figure 9: Elliptical bar – shear stress

4 Rectangular bar subjected to torsion and bending

In this case a rectangular section bar subjected to combined loading (torsion and bending) is analyzed. The dimensions are:

- Cross-section: rectangle 200x300 mm
- Length: 1400 mm

1) Create a new file with [mm, ton, s, °C] units and import geometry in step format to PrePoMax. Then mesh the part. In this case maximum element size of 30 mm was selected while other settings were left with no changes (Figure 10).

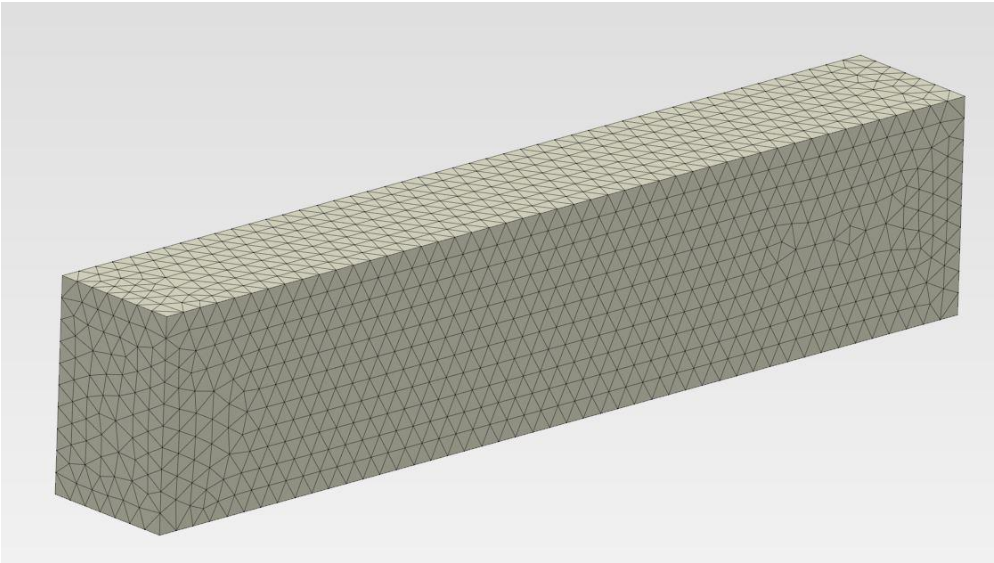

Figure 10: Rectangular bar – mesh

2) Define a new material, add Elastic behavior and specify Young's modulus of 180000 MPa and Poisson's ratio of 0.25. Create a new solid section referencing previously created material and select the bar so that the section will be assigned to this part.

3) Create a reference point at (500, 0, 1400) coordinates. Create a rigid body constraint using this reference point and assign it to the surface of the bar.

3) Define a static analysis step with default settings. Assign Fixed boundary condition to the end of the bar opposite to the one with reference point. Create Concentrated force loading to the other one. Specify the value of -8000 N in the direction perpendicular to the bar's axis (Figure 11).

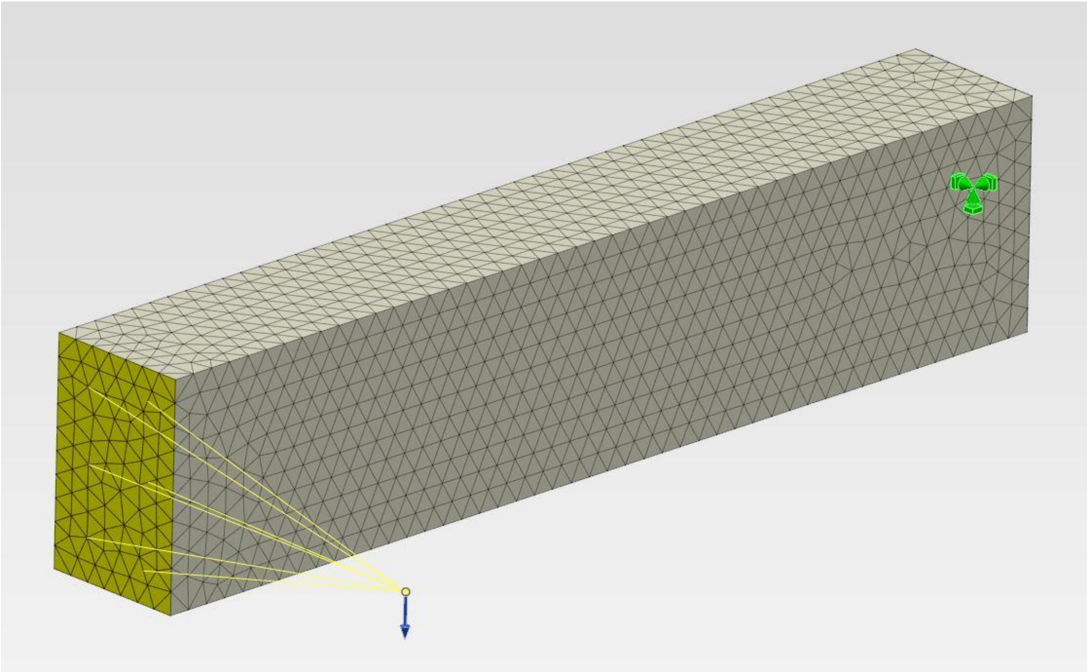

Figure 11: Rectangular bar – boundary conditions and loads

4) Run the analysis and examine the results. Set the deformation scale factor to 1000 and enable the option to draw undeformed model (Figure 12). You will notice both bending and twisting of the bar caused by the remote load. The analytical result is around 4.69 MPa. Results from simulation are affected by stress concentration and mesh sensitivity but the value read in a node right next to the corner with maximum stress is 4.77 MPa in this case.

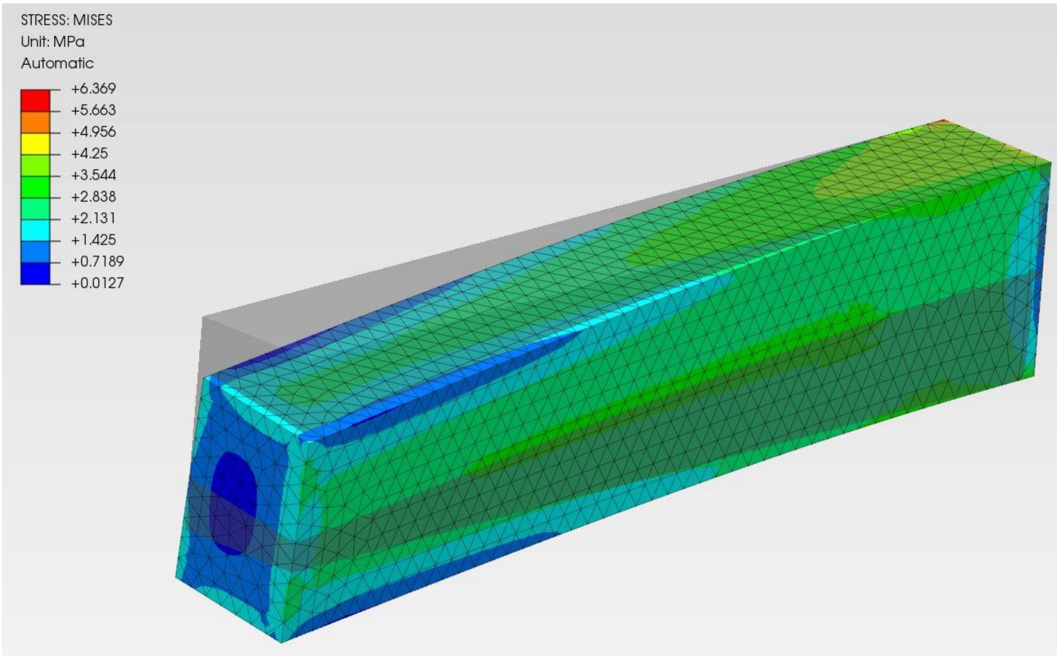

Figure 12: Rectangular bar – von Mises stress

5 Assembly of two cylinders

In this case a simple assembly consisting of two cylinders is analyzed. The dimensions are:

- Diameter of the first cylinder: 80 mm
- Length of the first cylinder: 100 mm
- Diameter of the second cylinder: 50 mm
- Length of the second cylinder: 120 mm

The cylinders are modelled in CAD software as separate parts and then their assembly has to be imported in step format.

1) Create a new file with [mm, ton, s, °C] units and import geometry in step format to PrePoMax. Then mesh both parts. In this case maximum element size of 5 mm was selected while other settings were left with no changes (Figure 13).

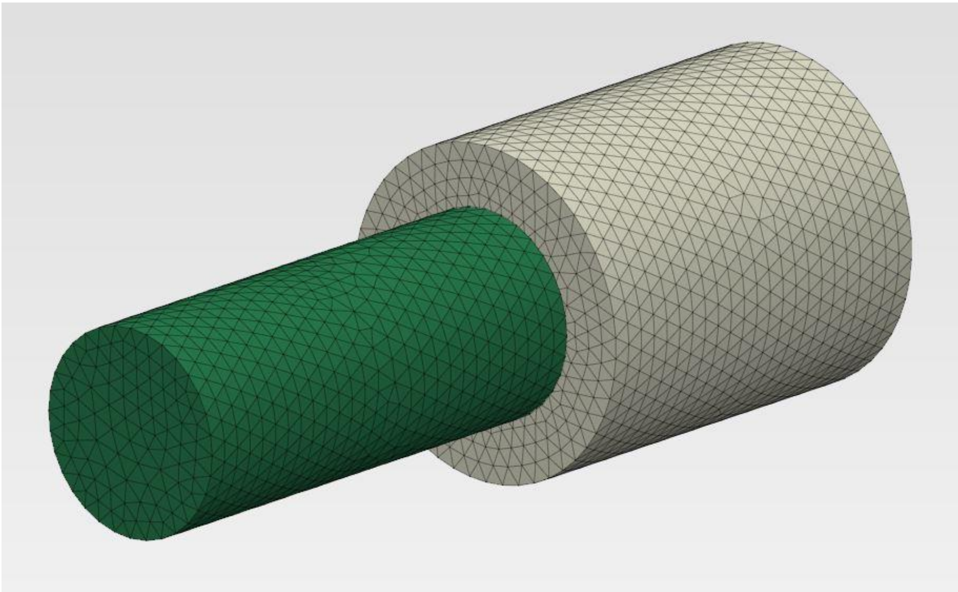

Figure 13: Two cylinders – mesh

2) Define a new material, add Elastic behavior and specify Young's modulus of 180000 MPa and Poisson's ratio of 0.25. Create a new solid section referencing previously created material and select both cylinders so that the section will be assigned to these parts.

3) Hide the wider cylinder and create surface named Tie1 for the back face of the narrower cylinder. Invert the part visibility (View → Invert Visible Parts) and create a surface named Tie2 for the front face of the wider cylinder. Show both parts again. Define a tie constraint, use previously created surfaces as master and slave regions.

4) Create a static step with default settings. Apply Fixed boundary condition to the back face of the wider cylinder and Pressure load (magnitude 30 MPa) to the front face of the narrower cylinder (Figure 14).

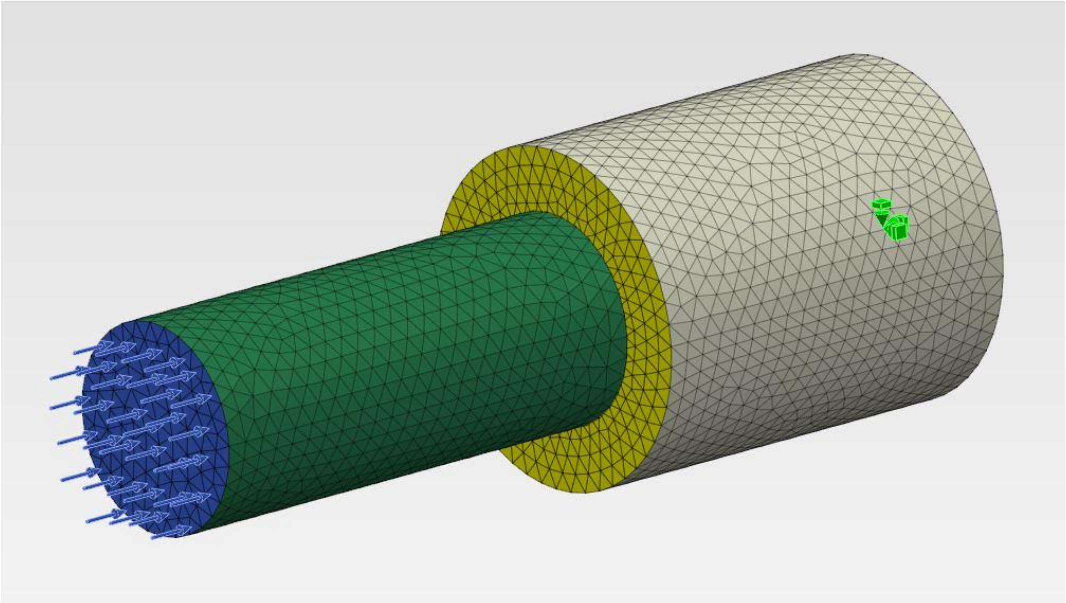

Figure 14: Two cylinders – boundary conditions and loads

5) Run the analysis and display the results. Examine the stresses and displacements with the deformation scale factor of 100 (Figure 15).

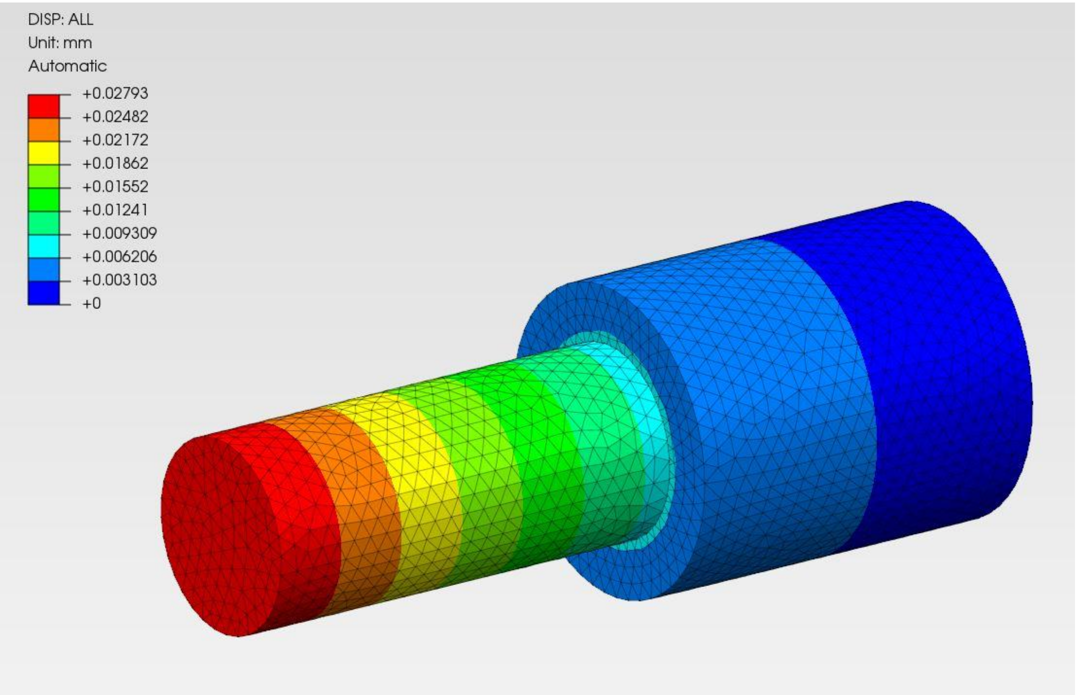

Figure 15: Rectangular bar – von Mises stress

6 Beam modal analysis

In this example a simply-supported rectangular beam is analyzed in terms of natural frequencies. The dimensions are:

- Cross-section: square 40 mm
- Length: 1000 mm

1) Create a new file with [mm, ton, s, °C] units and import geometry in step format to PrePoMax. Then mesh the part. In this case maximum element size of 10 mm was selected while other settings were left with no changes (Figure 16).

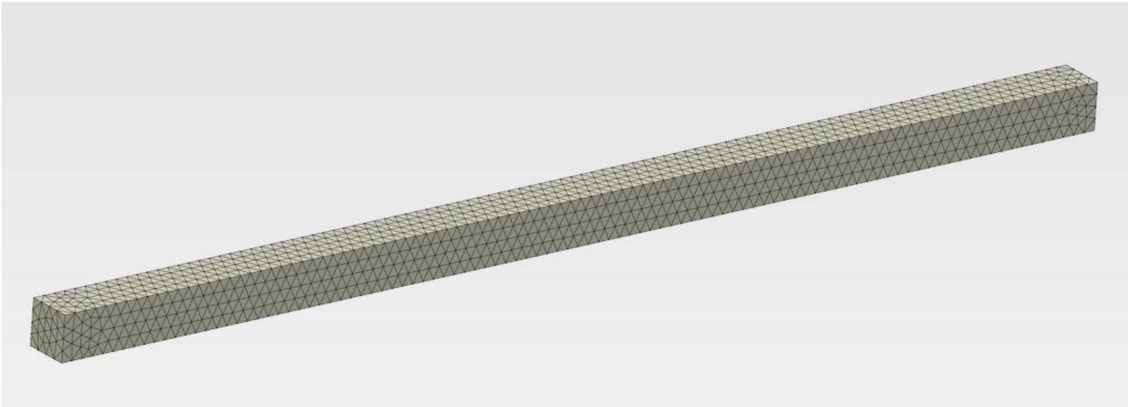

Figure 16: Beam – mesh

2) Define a new material, add Elastic behavior and specify Young's modulus of 210000 MPa and Poisson's ratio of 0.3. Also, add Density and specify the value of 7.85e-9 tonne/mm^3. Create a new solid section referencing previously created material and select the beam so that the section will be assigned to this part.

3) Create a frequency step with the default settings (extraction of the 10 natural frequencies). Apply Displacement/Rotation boundary condition to the bottom edge of cross-section at one end. Fix the degrees of freedom U1, U2 and U3. Create another Displacement/Rotation boundary condition applied to the bottom edge of the cross-section on the opposite side. Fix the degrees of freedom like before but leave one translation free (the one in the beam's axis) to simulate roller support (Figure 17).

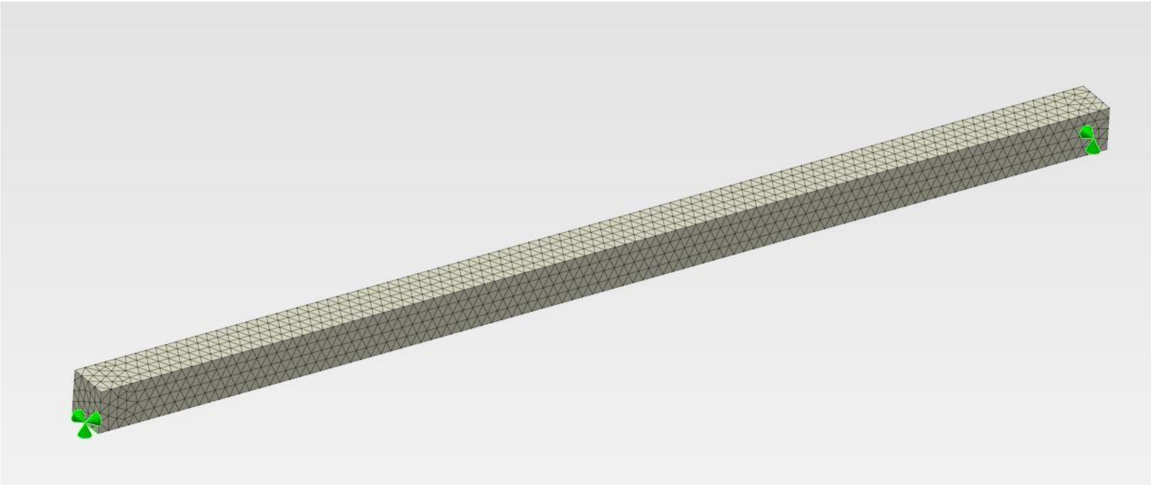

Figure 17: Beam – boundary conditions

4) Run the analysis and open the results when it completes. Examine the natural frequencies and mode shapes. To switch between mode shapes use Previous/Next Increment buttons or drop-down list in the shortcuts toolbar. Analytical values of natural frequencies are listed below (Table 1).

Table 1: Beam – theoretical natural frequencies

No.	Natural frequency [Hz]
1	93.82
2	375.46
3	844.07
4	1501.83
5	2347.80

while values obtained from the analysis are given in the next table (Table 2):

Table 2: Beam – natural frequencies from the analysis

No.	Natural frequency [Hz]
1	93.39
2	121.28
3	339.68
4	413.17
5	785.76
6	850.47
7	1066.7
8	1223.69
9	1478.86
10	1609.22

Analytical calculations are simplified (they account only for the deformation in one plane) and can't predict all the natural frequencies that were obtained using modal analysis. That's one of the main reasons of differences in results in this case.

One of the higher mode shapes (#9) can be seen in the image below (Figure 18).

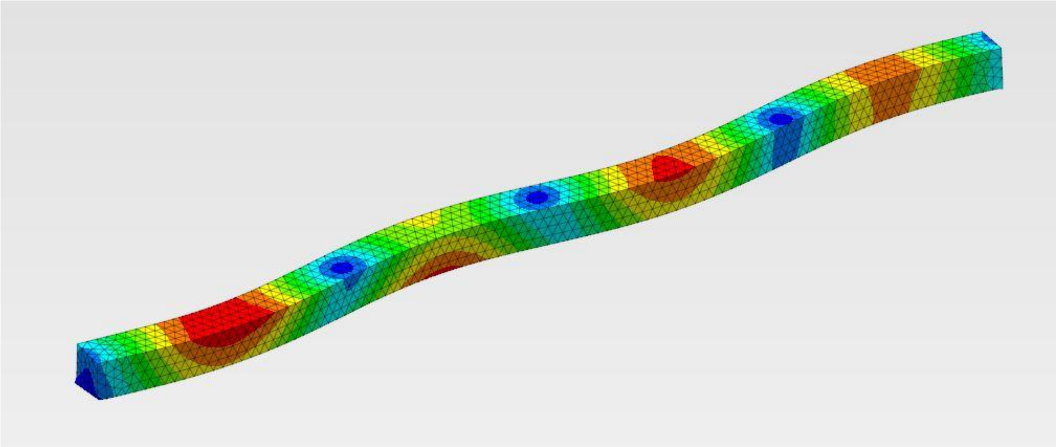

Figure 18: Beam – mode shape #9

7 Elasto-plastic plate in tension

In this example a rectangular plate with a hole in the middle is subjected to tension. The plate is made of elasto-plastic material. Due to symmetry, only one quarter of the plate is analyzed. The dimensions are:

- Plate size: 300x150 mm
- Hole diameter: 60 mm
- Thickness: 10 mm

1) Create a new file with [mm, ton, s, °C] units and import geometry in step format to PrePoMax. Then mesh the part. In this case maximum element size of 2 mm was selected while other settings were left with no changes (Figure 19).

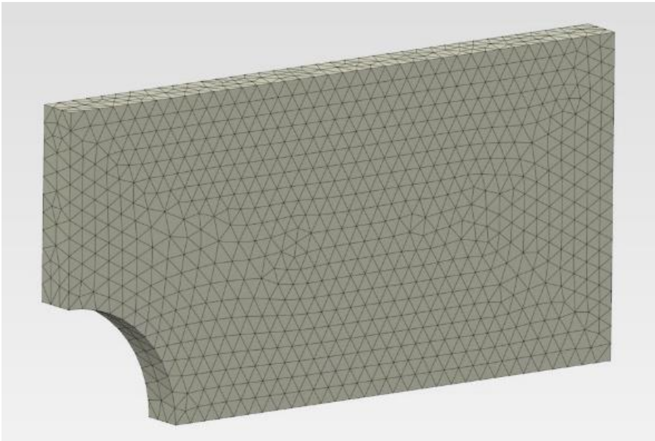

Figure 19: Plate – mesh

2) Define a new material, add Elastic behavior and specify Young's modulus of 210000 MPa and Poisson's ratio of 0.3. Also, add Plasticity and specify data points given below (Table 3).

Table 3: Plate – data points for plasticity definition

Yield stress [MPa]	Plastic strain [-]
235	0
335	0.12

Create a new solid section referencing previously created material and select the beam so that the section will be assigned to this part.

3) Define a static step with default settings. Apply Displacement/Rotation boundary conditions to faces that were cut due to symmetry. For each of these BCs fix the degree of freedom corresponding to the direction normal to specific face. Create another Displacement/Rotation BC, apply it to the two largest faces and constrain displacements in normal direction to prevent out-of-plane deformations (the problem is in fact two-dimensional). Apply a Pressure load of -200 MPa to the side were the plate is pulled (Figure 20).

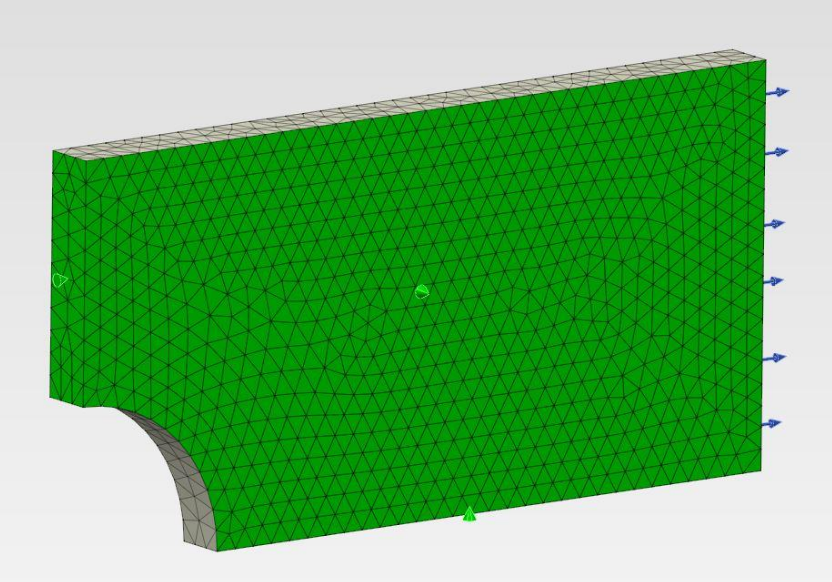

Figure 20: Plate – boundary conditions

4) Add new field output request, select element output and PEEQ variable.

5) Run the analysis and examine the results when it completes. Take a closer look at Mises stress and plastic strain (PE) distribution (Figure 21).

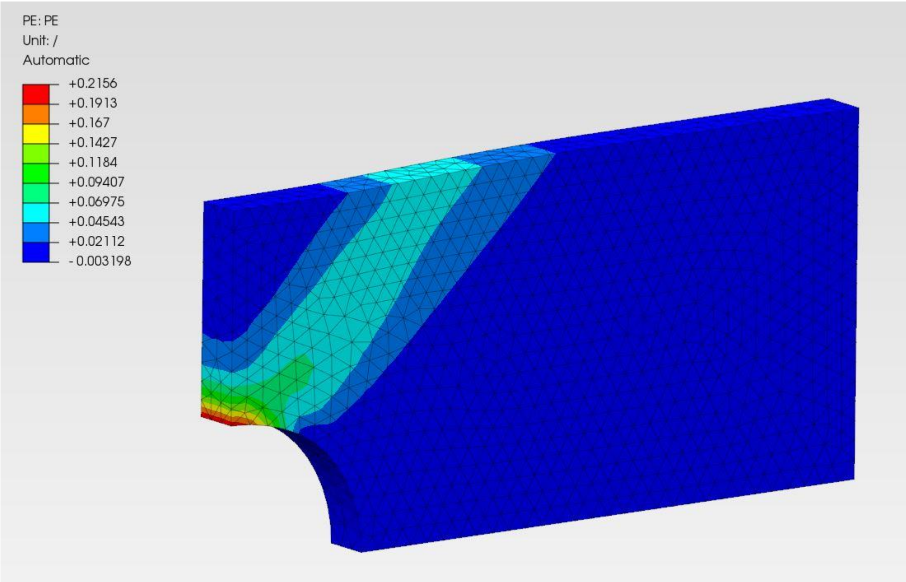

Figure 21: Plate – plastic strain

6) Use the Transformation → Symmetry option to mirror the results and display them for the whole plate (Figure 22).

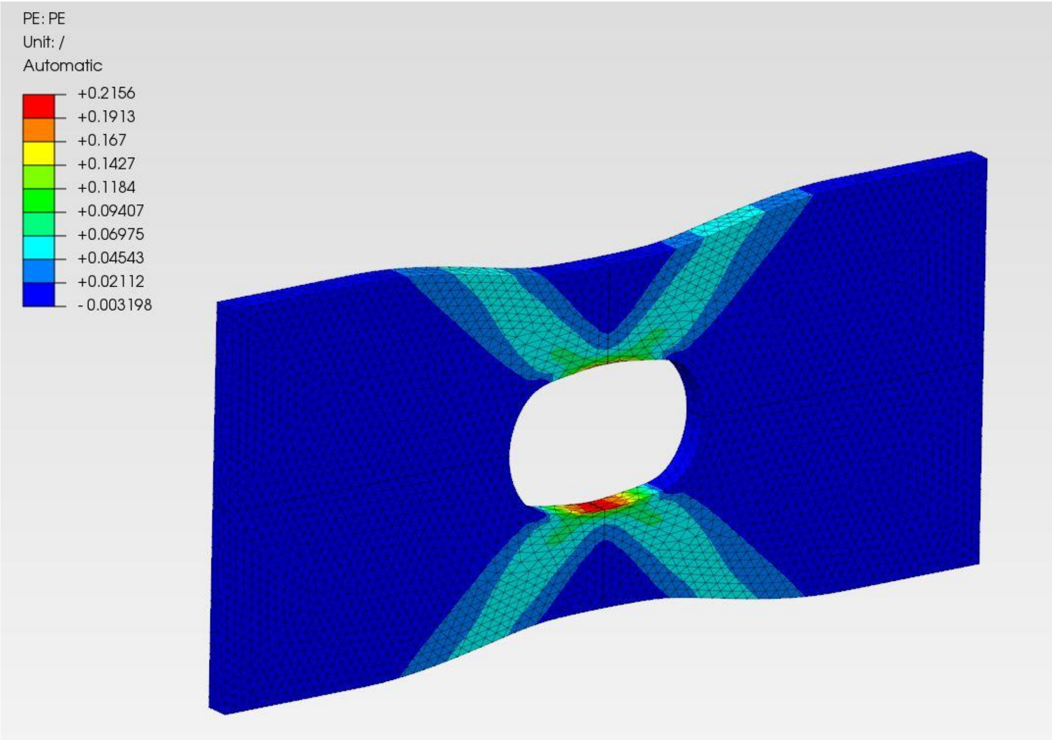

Figure 22: Plate – plastic strain (mirrored)

8 Hertz contact of two spheres

The example discussed here is based on Code_Aster benchmark problem SSNV104. In this case two spheres in contact are pressed against each other. Symmetry is utilized so that only 1/8 of each sphere has to be analyzed.

The geometry is modelled in CAD software as separate parts and then their assembly has to be imported in step format.

1) Create a new file with [mm, ton, s, °C] units and import geometry in step format to PrePoMax. Then mesh both parts. In this case maximum element size of 5 mm was selected while other settings were left with no changes. In addition, local mesh refinement of 0.5 mm was applied to the contact region (for better refinement of the mesh it would be necessary to partition the spheres, creating edges to which the refinement can be assigned. Otherwise, the user has to use vertices at the point of contact (Figure 23).

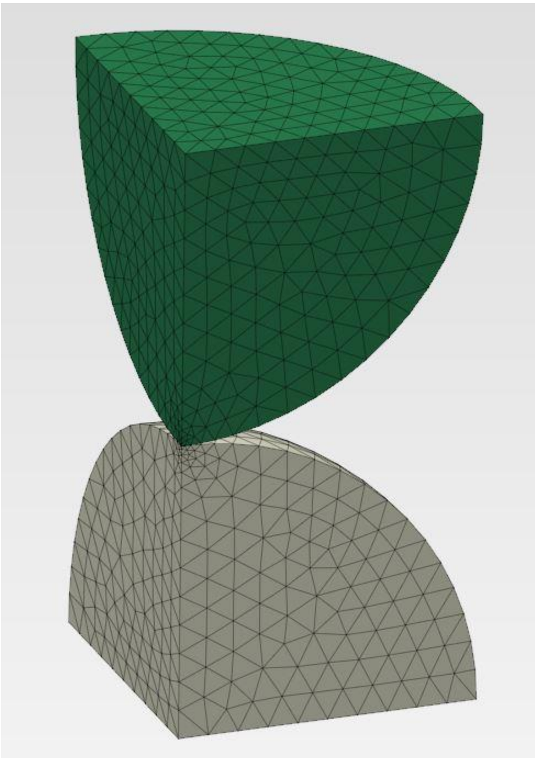

Figure 23: Two spheres – mesh

2) Define a new material, add Elastic behavior and specify Young's modulus of 20000 MPa and Poisson's ratio of 0.3. Create a new solid section referencing previously created material and select both sphere segments so that the section will be assigned to these parts.

3) Create a surface interaction with default surface behavior (hard). Create a contact pair referencing this interaction and select the two surfaces that will come into contact during the analysis.

4) Create a new static step, turn Nlgeom to On (to account for geometric nonlinearity). Define two Displacement/Rotation boundary conditions, apply them to 2 sets consisting of 2 side faces where the symmetry is (constrain only displacements in the direction normal to each set of the faces). Create another two Displacement/Rotation BCs and assign them to top and bottom face, prescribing the displacement of 2 mm and -2 mm, so that the arrows point to each other and the spheres will be pressed against each other (Figure 24).

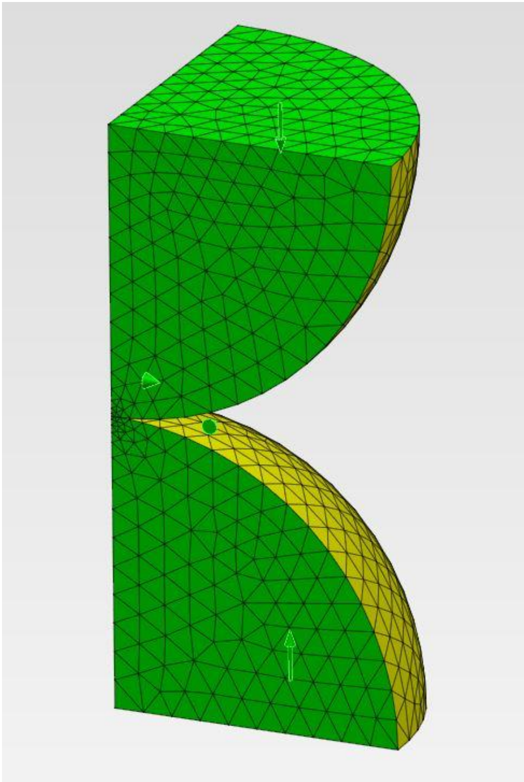

Figure 24: Two spheres – boundary conditions

5) Run the analysis and check the results when it completes. Notice specific stress distribution around the point of contact (Figure 25). Analytically calculated value of the normal stress in the direction in which spheres are compressed (at the point of contact) is 2798.3 MPa while the result of simulation in this case is 2750 MPa. Analyses involving contact are very sensitive to mesh density so better result could be obtained with finer mesh.

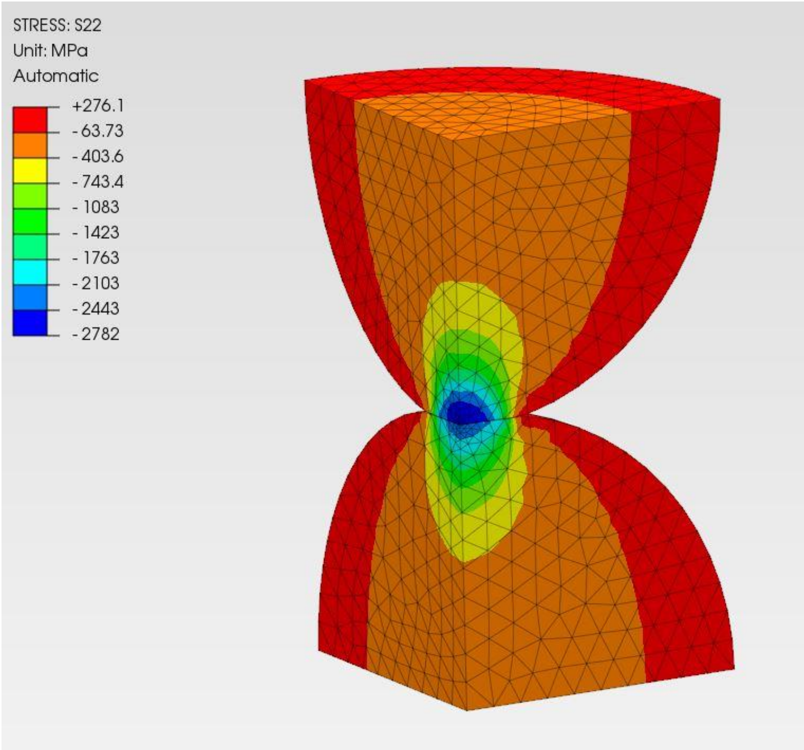

Figure 25: Two spheres – normal stress

9 Cylindrical shell buckling

In this example linear buckling analysis of an axially compressed cylindrical shell is performed. The dimensions are:

- Diameter: 300 mm
- Length: 600 mm

The geometry is modelled in CAD software as a surface.

1) Create a new file with [mm, ton, s, °C] units and import geometry in step format to PrePoMax. Then mesh the part. In this case maximum element size of 10 mm was selected and quad-dominated mesh was enabled while other settings were left with no changes (Figure 26).

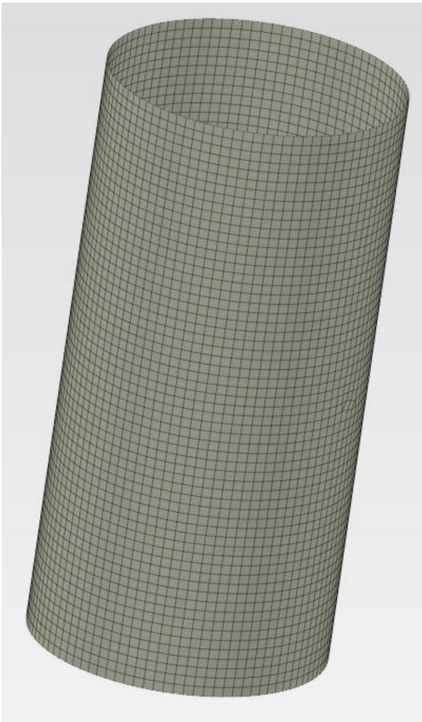

Figure 26: Cylindrical shell – mesh

2) Define a new material, add Elastic behavior and specify Young's modulus of 210000 MPa and Poisson's ratio of 0.3. Create a new shell section referencing previously created material and select the cylindrical shell so that the section will be assigned to this part. Specify the thickness of 5 mm.

3) Create a reference point at coordinates (0, 0, 600). Define a rigid body constraint using this reference point and shell edge.

4) Add a new buckle step and set the number of buckling factors to 5. Create a Fixed boundary condition assigned to the bottom edge of the shell. Define a concentrated force load and apply it to the reference point connected with top edge of the shell. Specify the value of -1 N for this load (Figure 27).

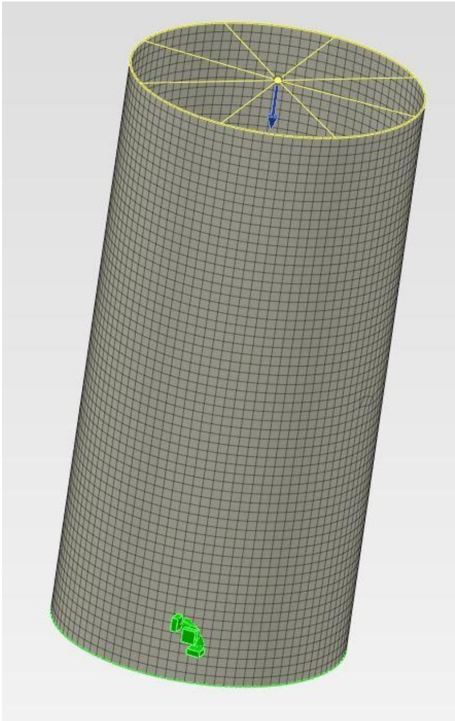

Figure 27: Cylindrical shell – boundary conditions and loads

5) Run the analysis and open results when it finishes. Set the deformation scale factor to 10000 and examine buckling mode shapes. One of them (#4) is shown in the image below (Figure 28). Analytically calculated critical load is $1.9964 \cdot 10^7$ while from the simulation we get $1.9024 \cdot 10^7$.

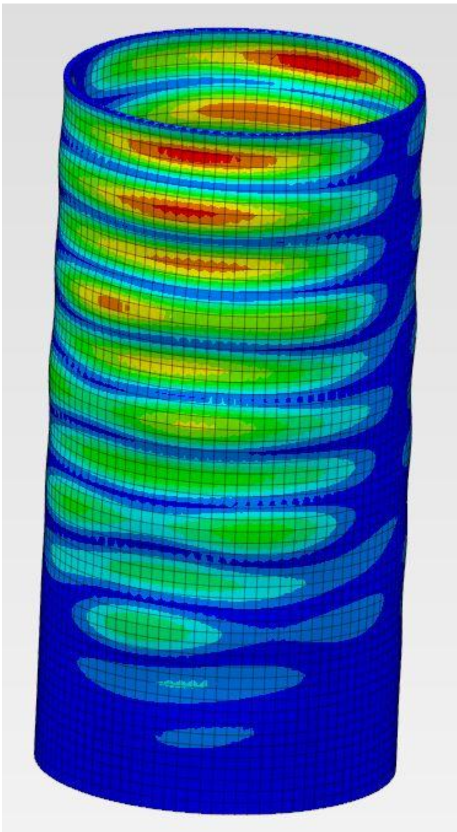

Figure 28: Cylindrical shell – buckling mode shape #4

10 Buckling of a simply-supported plate in compression

In this example a simply-supported rectangular plate subjected to compression on all edges is analyzed in terms of buckling. The dimensions are:

- Plate size: 200x150 mm

The geometry is modelled in CAD software as a surface.

1) Create a new file with [mm, ton, s, °C] units and import geometry in step format to PrePoMax. Then mesh the part. In this case maximum element size of 5 mm was selected and quad-dominated mesh was enabled while other settings were left with no changes (Figure 29).

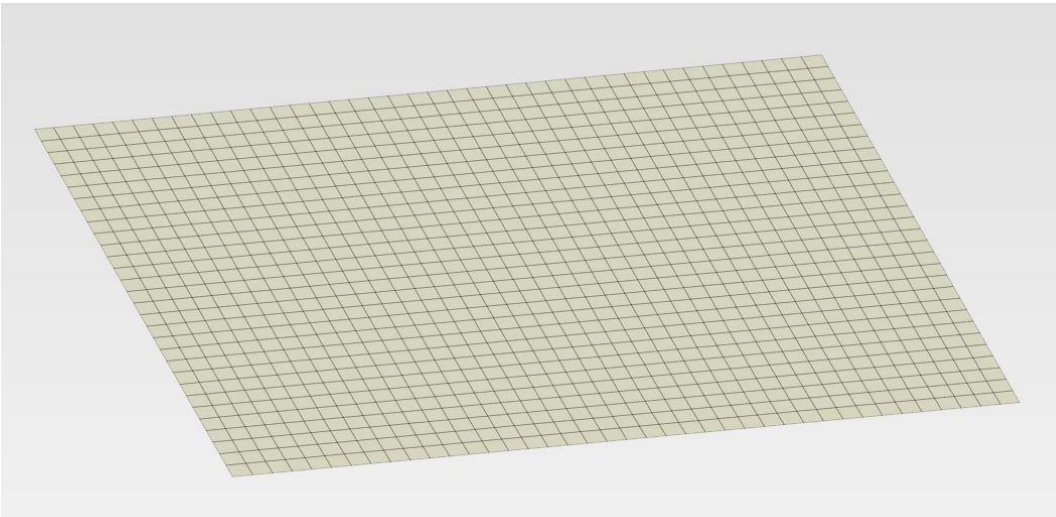

Figure 29: Plate – mesh

2) Define a new material, add Elastic behavior and specify Young's modulus of 200000 MPa and Poisson's ratio of 0.3. Create a new shell section referencing previously created material and select the cylindrical shell so that the section will be assigned to this part. Specify the thickness of 4 mm.

3) Add a new buckle step and set the number of buckling factors to 3. Create a Displacement/Rotation boundary condition assigned to all edges of the plate, constrain only the out-of-plane translation. Add two more Displacement/Rotation BCs and apply them to 2 perpendicular edges of the plate, constrain the direction normal to each edge. Define a normal shell edge load with magnitude 1 N/mm and assign it to 2 edges opposite to those with BCs in normal directions (Figure 30).

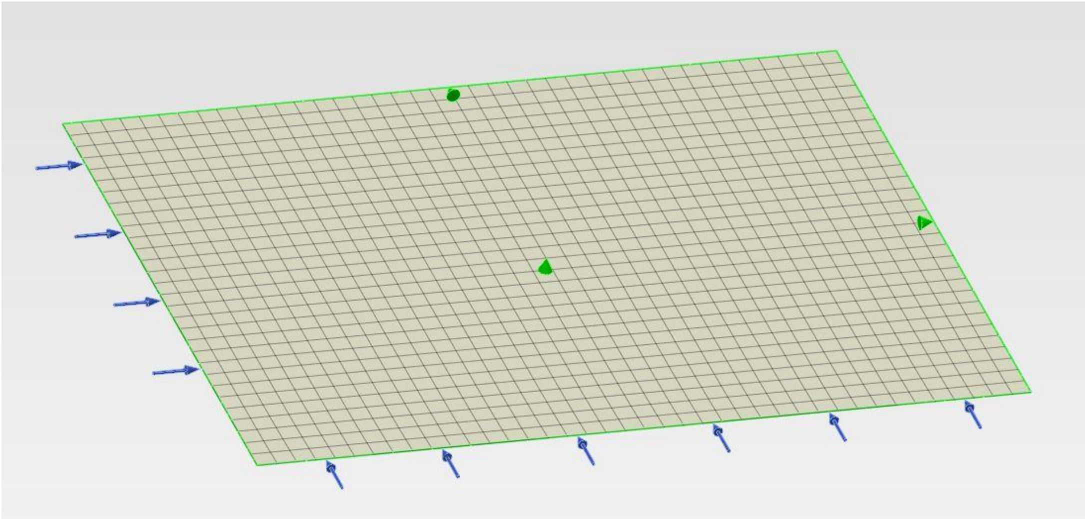

Figure 30: Plate – boundary conditions and loads

4) Run the analysis and examine results when it finishes. Check each buckling mode shape. The second one is shown in the image below (Figure 31). Analytically calculated critical compressive stress is 200.85 MPa while from the simulation we get 197 MPa.

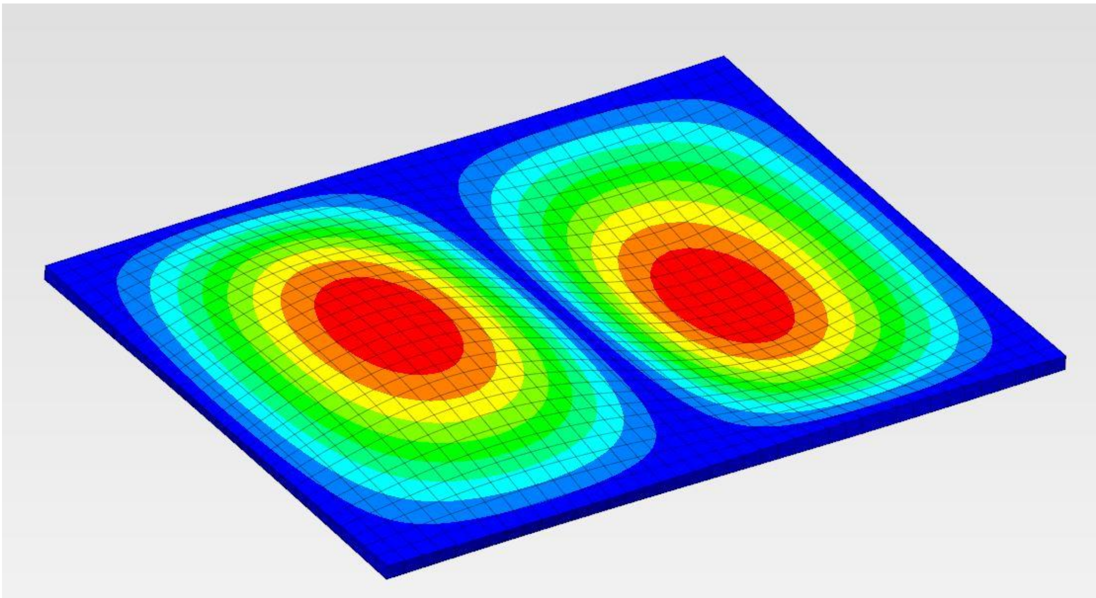

Figure 31: Plate – buckling mode shape #2

11 Heat transfer in insulated pipeline

In this example an insulated pipeline is subjected to thermal loading. The dimensions are:

- Outer diameter: 57 mm
- Inner diameter: 50 mm
- Insulation thickness: 28 mm
- Length: 200 mm

1) Create a new file with [mm, ton, s, °C] units and import geometry in step format to PrePoMax. Then mesh both parts. In this case the following maximum element sizes were selected: 2 mm for insulation and 3 mm for pipe. Other settings were left with no changes (Figure 32).

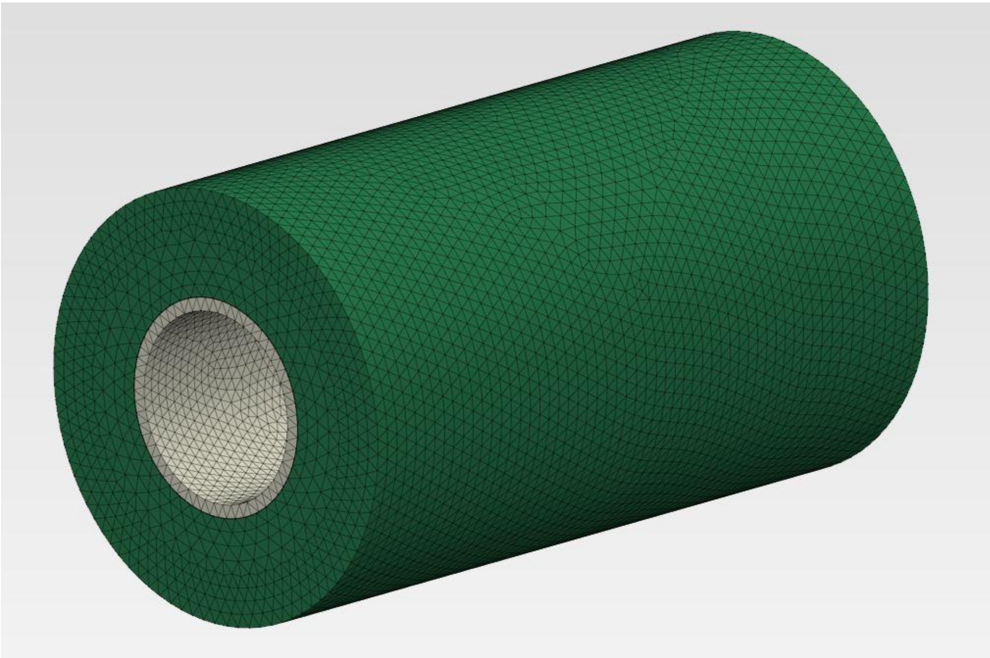

Figure 32: Pipeline – mesh

2) Define a new material named Pipe_material, add Thermal conductivity and specify the value of 45 mW/(mm·°C). Define another material, name it Insulation_material, add Thermal conductivity and specify the value of 0.05 mW/(mm·°C). Create a new solid section named Pipe_section, with Pipe_material as a material and select the inner part (pipe) so that the section will be assigned to it. Create another section, name it Insulation_section, pick Insulation_material as a material and select the outer part (insulation).

3) Hide the pipe and create surface named Tie1 for the inner face of the insulation. Invert the part visibility (View → Invert Visible Parts) and create a surface named Tie2 for the outer face of the pipe. Show both parts again. Define a tie constraint, use previously created surfaces as master and slave regions.

4) Define heat transfer analysis step with default settings (steady state).

5) Create Convective film load and assign it to the outer surface of the insulation. Specify sink temperature of 20 °C and film coefficient of 0.015 mW/(mm^2·°C). Create another Convective film load and assign it to the inner surface of the pipe. Specify sink temperature of -30 °C and film coefficient of 0.8 mW/(mm^2·°C) (Figure 33).

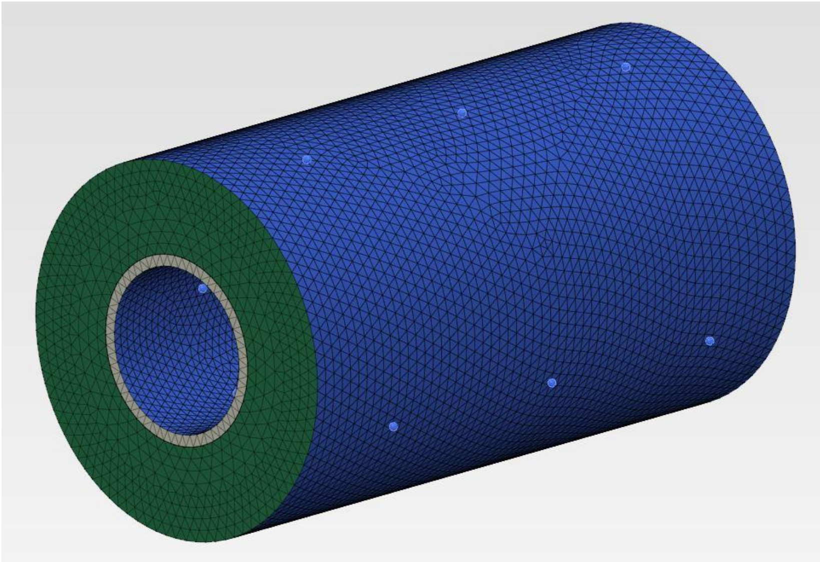

Figure 33: Pipeline – thermal loads

6) Since all the necessary definitions were made, the analysis can be submitted. Wait until it finishes and open the results.

7) Examine temperature contour plot (Figure 34). Analytically calculated temperature at the inner face of the pipe is -29.83 °C while at the outer face of the pipe it's 16.05 °C. Similar values can be found in the analysis (use Query tool to check them).

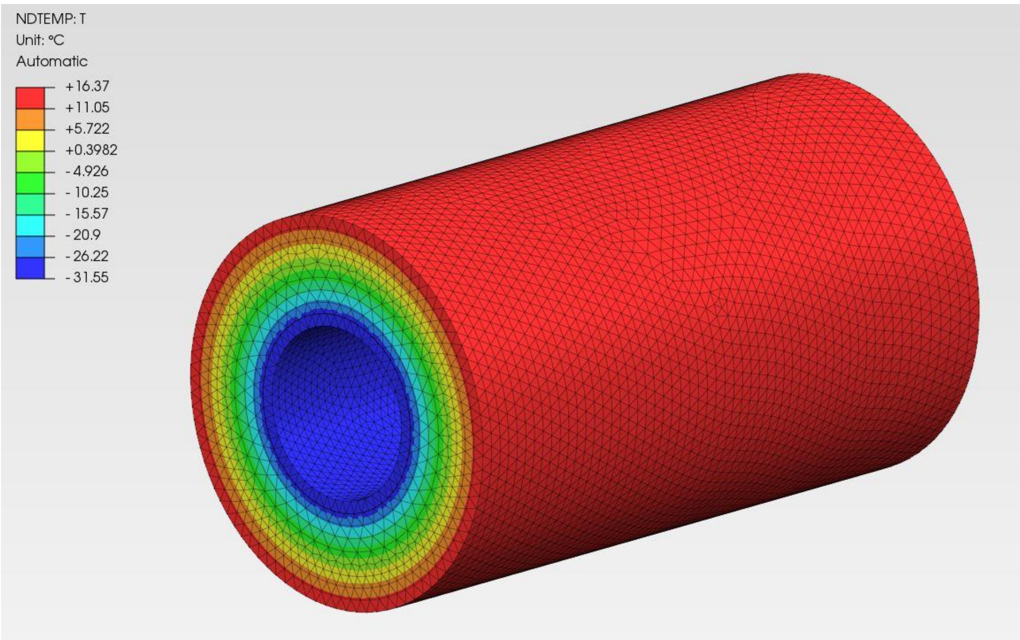

Figure 34: Pipeline – temperature

12 Termomechanical analysis of a bimetallic strip

In this example a bimetallic strip bends due to different thermal expansion of each layer. The model consists of two beams (120x20x4 mm) placed on top of each other.

1) Create a new file with [mm, ton, s, °C] units and import geometry in step format to PrePoMax. Then mesh both parts. In this case maximum element size of 1 mm was selected for both parts while other settings were left with no changes (Figure 35).

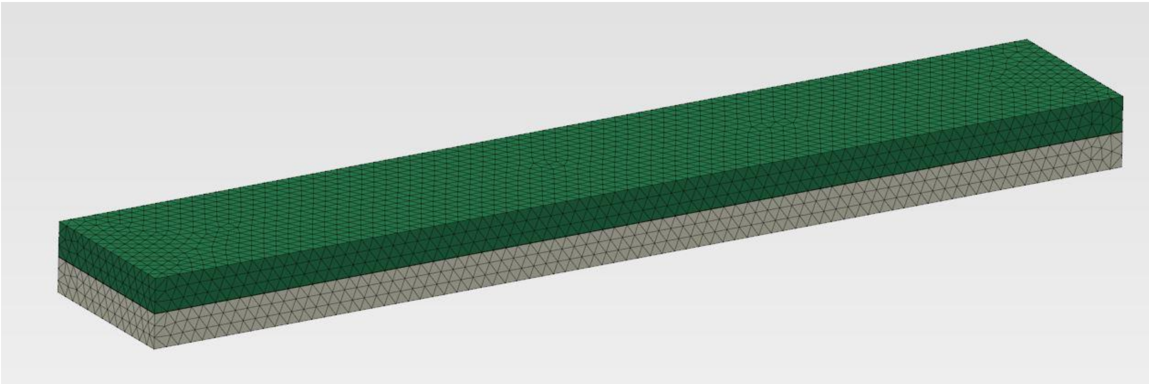

Figure 35: Bimetallic strip – mesh

2) Define a new material named Copper, add Elasticity (Young's modulus 130000 MPa, Poisson's ratio 0.34), Thermal conductivity (385 mW/(mm·°C)) and Thermal expansion (17e-6 (1/°C), zero temperature 20 °C). Define another material, name it Steel, add Elasticity (Young's modulus 210000 MPa, Poisson's ratio 0.3), Thermal conductivity (45 mW/(mm·°C)) and Thermal expansion (12.3e-6 (1/°C), zero temperature 20 °C). Create a new solid section named Top_layer, with Copper as a material and select the top beam so that the section will be assigned to it. Create another section, name it Bottom_layer, pick Steel as a material and select the bottom beam.

3) Hide the top beam and create surface named Tie1 for the upper face of the bottom beam. Invert the part visibility (View → Invert Visible Parts) and create a surface named Tie2 for the lower face of the top beam. Show both parts again. Define a tie constraint, use previously created surfaces as master and slave regions.

4) Define coupled temperature-displacement analysis step with default settings (steady state).

5) Assign Fixed boundary condition to the back faces of both beams. Add Temperature boundary condition with magnitude of 80 °C to all external faces apart from the fixed one (Figure 36).

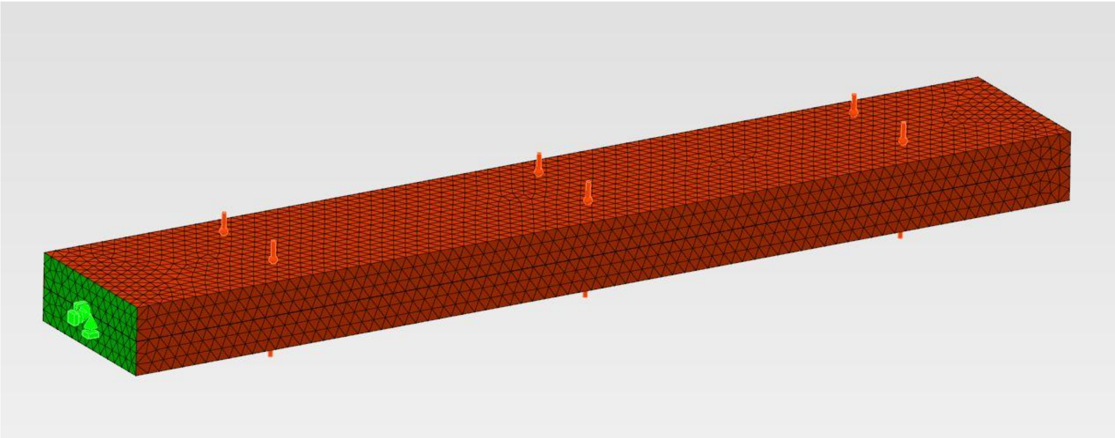

Figure 36: Bimetallic strip – boundary conditions

6) Since all the necessary definitions were made, the analysis can be submitted. Wait until it finishes and open the results.

7) Examine the vertical displacement contour plot (Figure 37). Analytically calculated deflection of the bimetallic strip is 0.3807 mm. The analysis may give a slightly higher prediction.

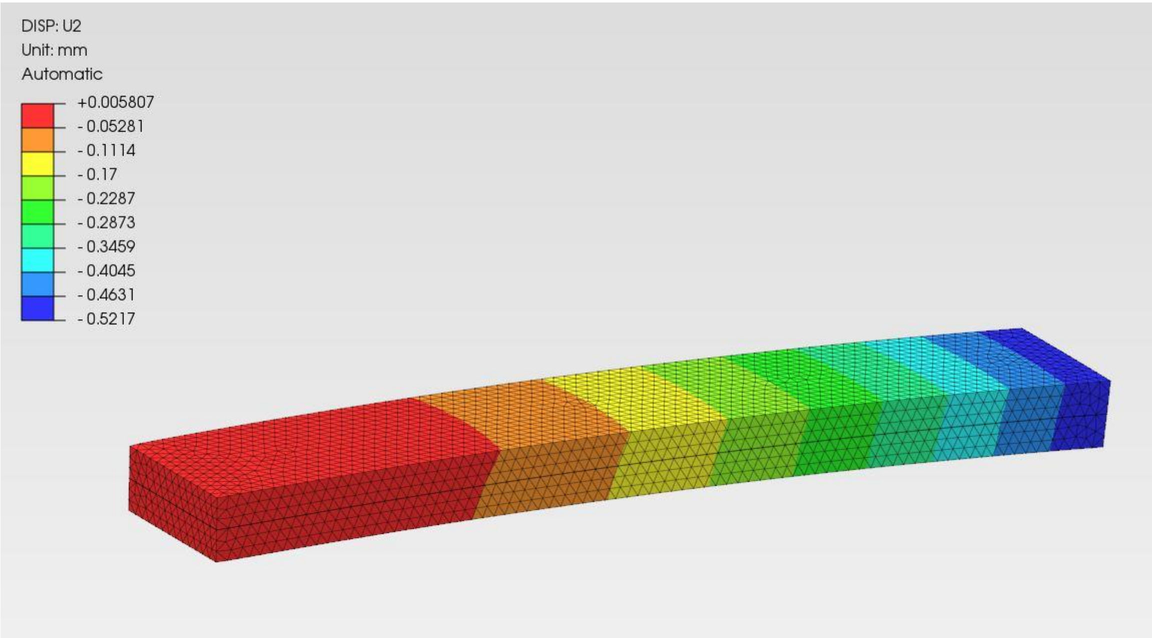

Figure 37: Bimetallic strip – deflection