# Wire Drawing Analysis using Ansys

ME756 - Numerical Modelling of Manufacturing Processes

Problem Description

- 2D Axisymmetric analysis of drawing of a wire from an initial diameter of 5 mm to a final diameter of 4 mm i.e., a 20% reduction in the diameter.

- The material of the wire and die are Carbon Steel and Tungsten Carbide, respectively.

- Objective is to calculate the force required for the wire drawing process

Initial configuration

Final configuration
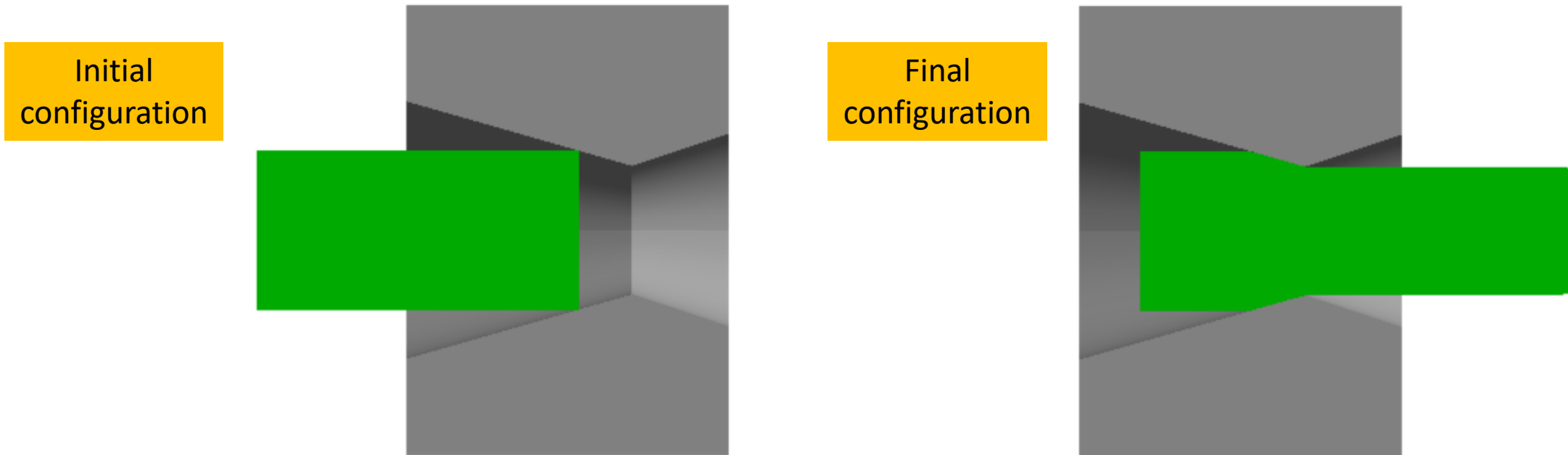
Material Properties

| Material	E (GPa)	ν	σ_y (MPa)	E_T (MPa)
Carbon steel, 1040, Annealed	210	0.29	350	1080
Tungsten Carbide	630	0.2	-	-
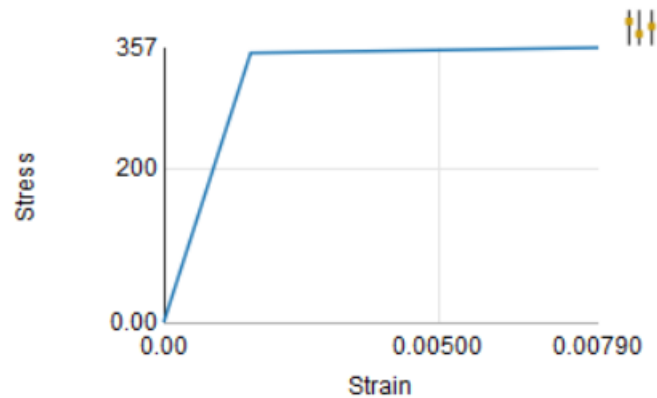
True-stress vs True-Strain Curve

E_T = Tangent Stiffness

Ansys Workbench

- Open Ansys Workbench 2021R1 or later version from Windows start button

- Open the project:
  Wire_Drawing_2D_Axisymm_20PercentRed_2021R1.wbpz

- A static structural system should appear.

- Note that the Analysis Type is set to 2D.
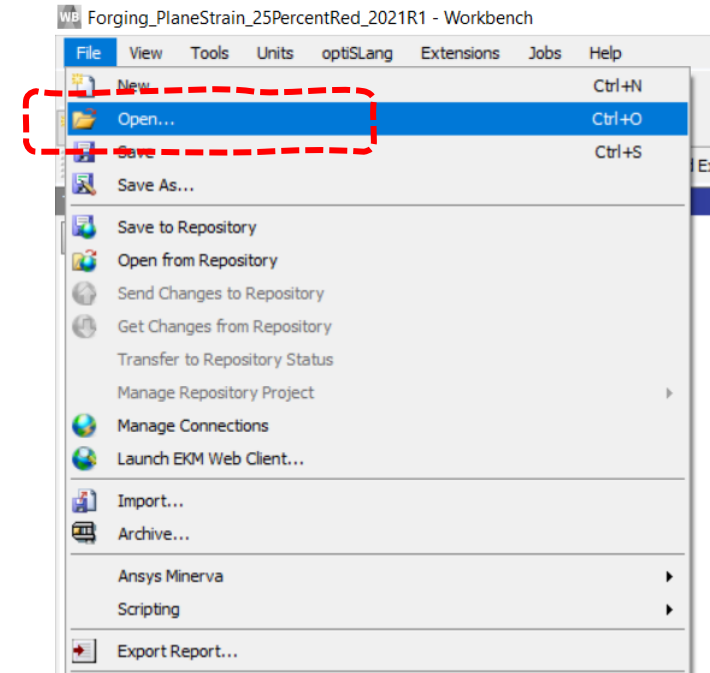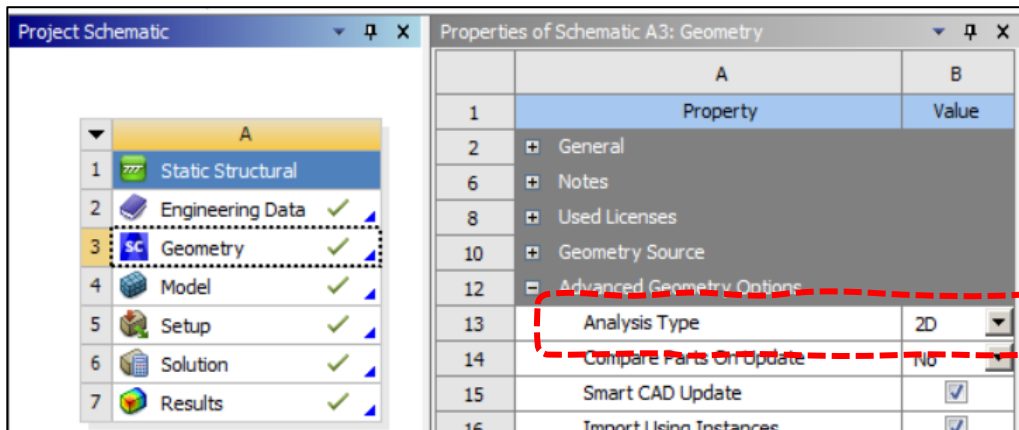
Engineering Data

- Double click on "Engineering Data"

- Note that Carbon steel, 1040 is defined with a Bilinear Isotropic Hardening material properties.

- Tungsten Carbide is considered as linear elastic.

Mechanical Model

- Double click on "Model"

- It will open the Mechanical GUI.

- The analysis is set to Axisymmetric.

- The wire is assigned with Carbon steel material and the Die is assigned with Tungsten Carbide.

- A symmetry object with "Type" as 2D Axisymmetric is added for visual expansion of the results to 3D. This is optional and requires "Beta" option to be on which is available in workbench under Tools → Options → Appearance .
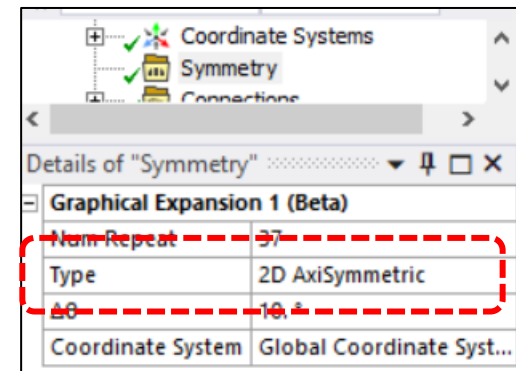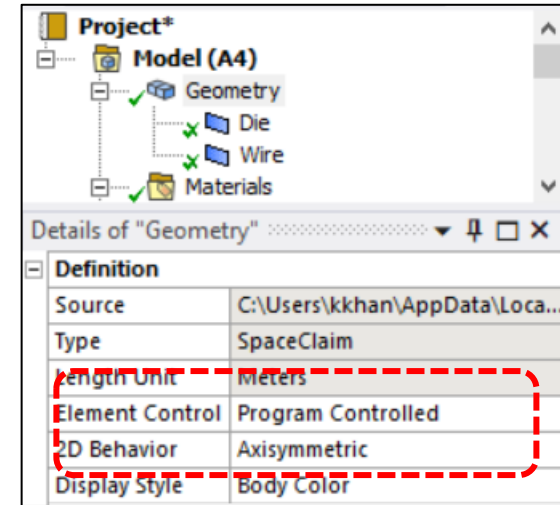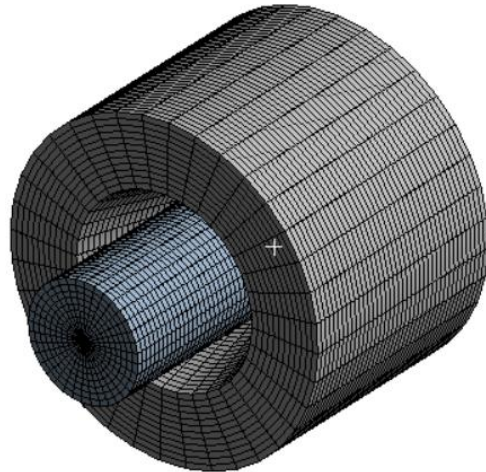
Connections

- The interaction between the wire and the die is defined using frictional contact with a friction coefficient of 0.05


©2023 ANSYS, Inc.
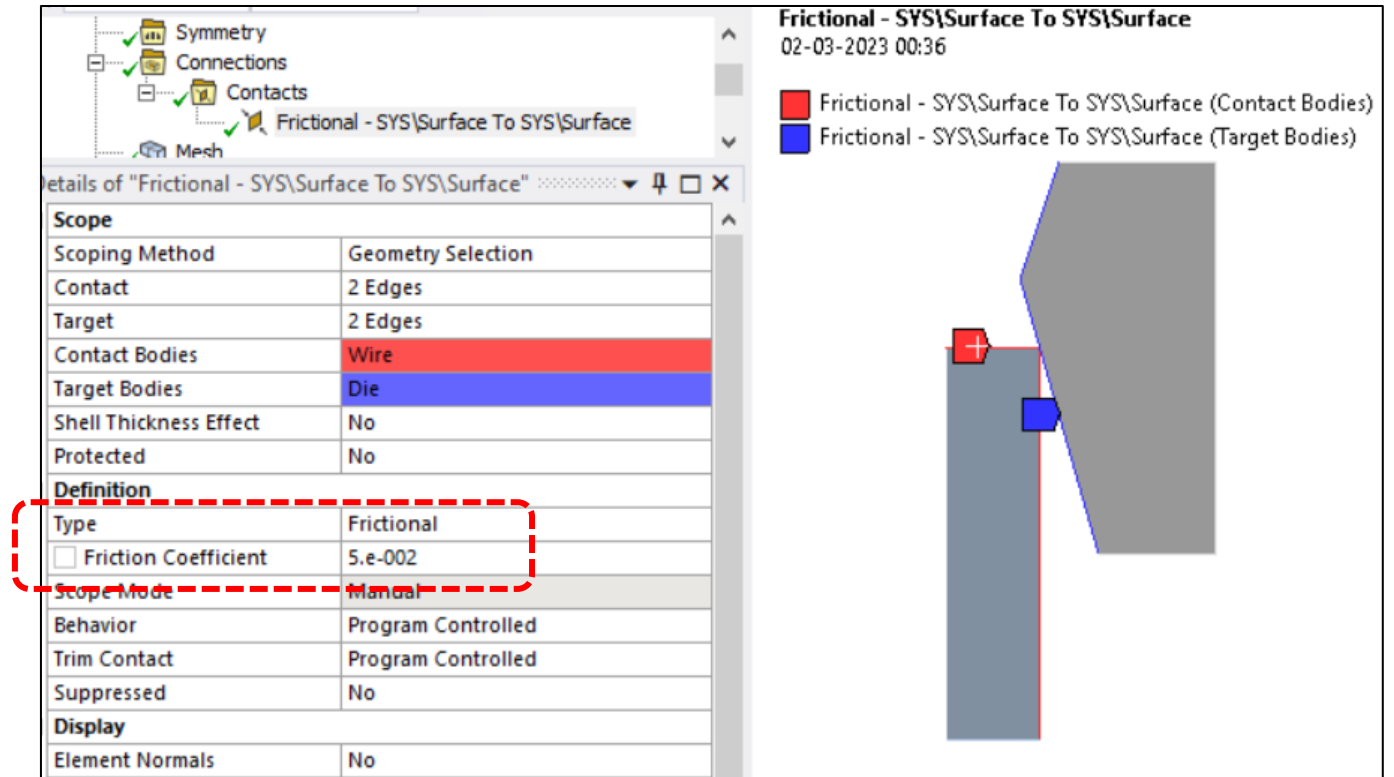
Meshing

- Both the surfaces are meshed with 10 noded quadrilateral elements with an element size of 0.25 mm.
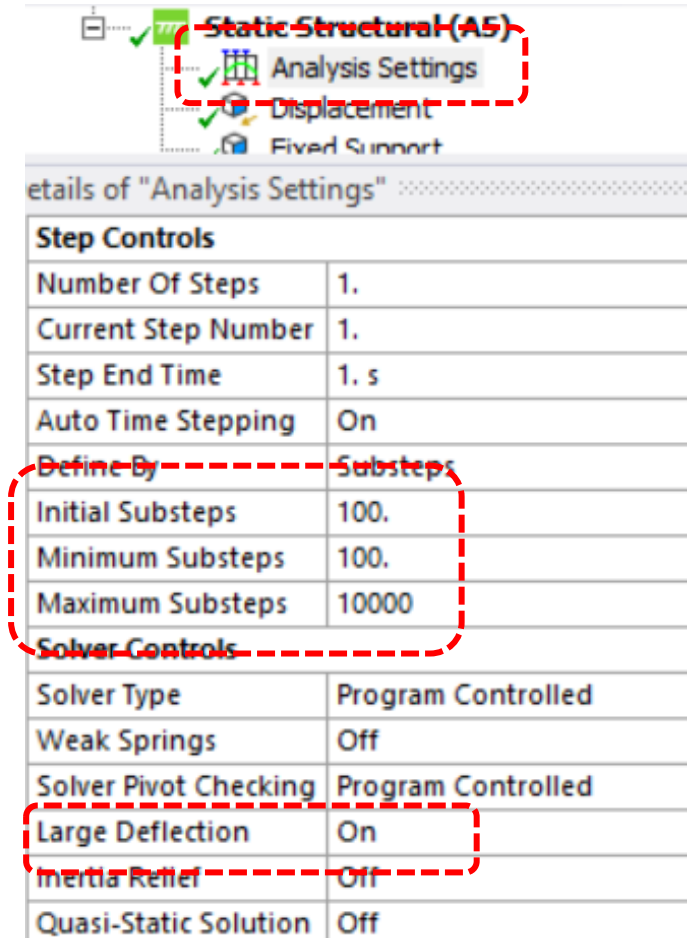
With graphical expansion on

Mesh
Static Structural (A5)

Details of "Mesh"

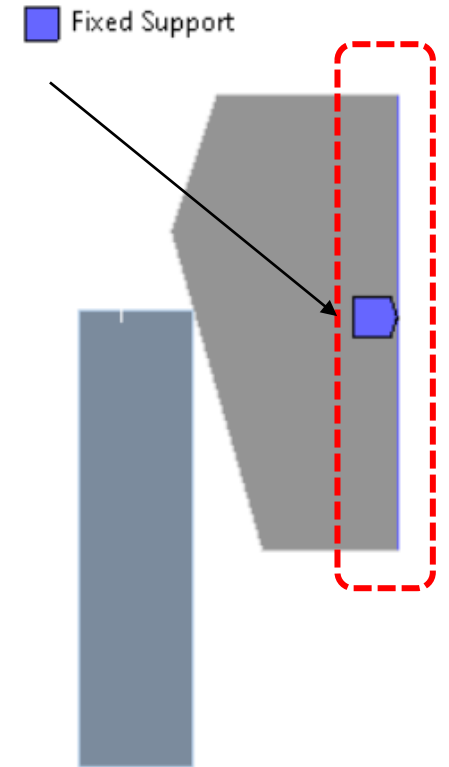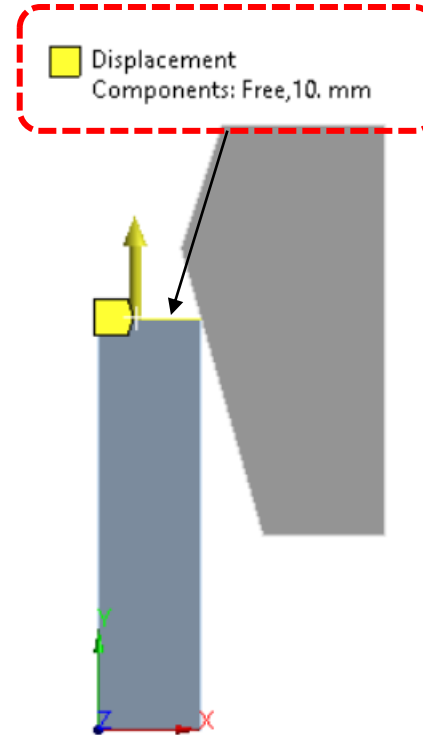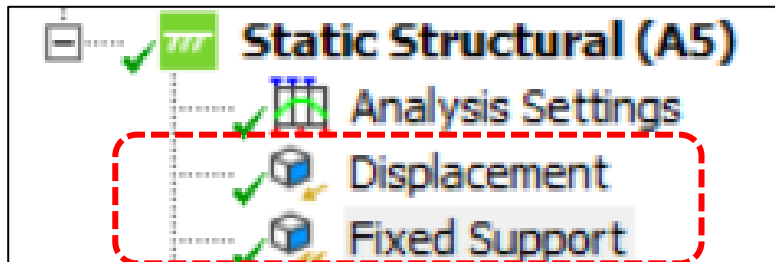| Display	
Defaults	
Physics Preference	Mechanical
Element Order	Program Controlled
☐ Element Size	0.25 mm
Sizing	
Quality	
Inflation	
Batch Connections	
Advanced	
Statistics	
☐ Nodes	3414
☐ Elements	1068
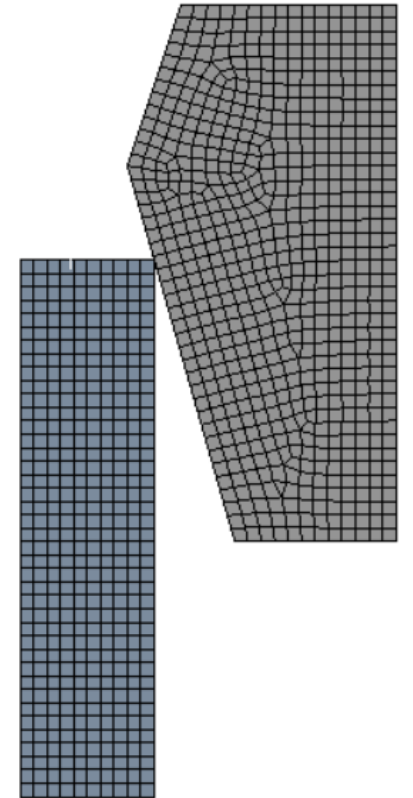
Analysis Setting

- Note that the problem is solved using multiple substeps as it is a nonlinear problem

- Also the "Large Deflection" option is set to "on" as it is a large strain problem going to the plastic region.

Boundary Conditions

- The die is fixed in all degrees of freedom (DOF) from outside.

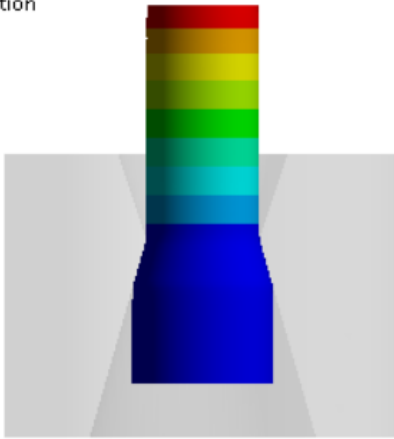- A displacement of 10 mm is applied on the one end of the wire in the Y-direction.

Displacement
Components: Free,10. mm

Fixed Support

Static Structural (A5)
Analysis Settings
Displacement
Fixed Support

Results

Total Deformation

Type: Total Deformation
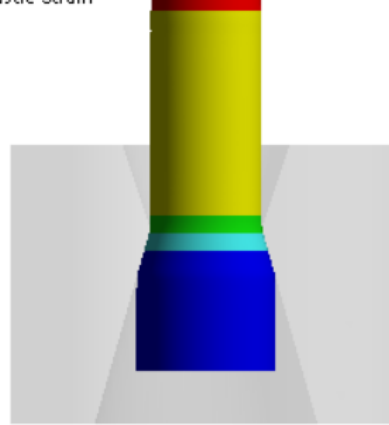Unit: mm
Time: 1
02-03-2023 00:42

10.015 Max
9.6397
9.2648
8.8899
8.5151
8.1402
7.7653
7.3904
7.0155
6.6406 Min

[s]

Eq. Plastic Strain

Type: Equivalent Plastic Strain
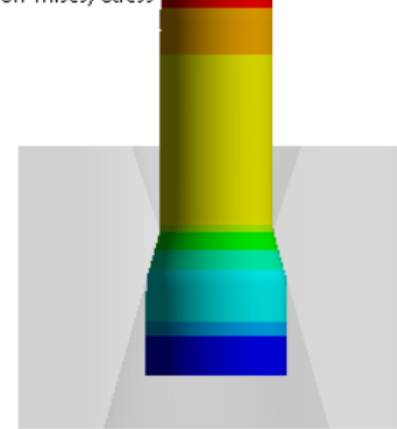Unit: mm/mm
Time: 1
02-03-2023 00:44

0.80857 Max
0.6
0.45
0.3
0.15
0 Min

Von-Mises Stress

Type: Equivalent (von-Mises) Stress
Unit: MPa
Time: 1
02-03-2023 00:45

1265.7 Max
1125.2
984.79
844.33
703.87
563.41
422.95
282.49
142.03
1.573 Min

Drawing Force

Max = 13 kN