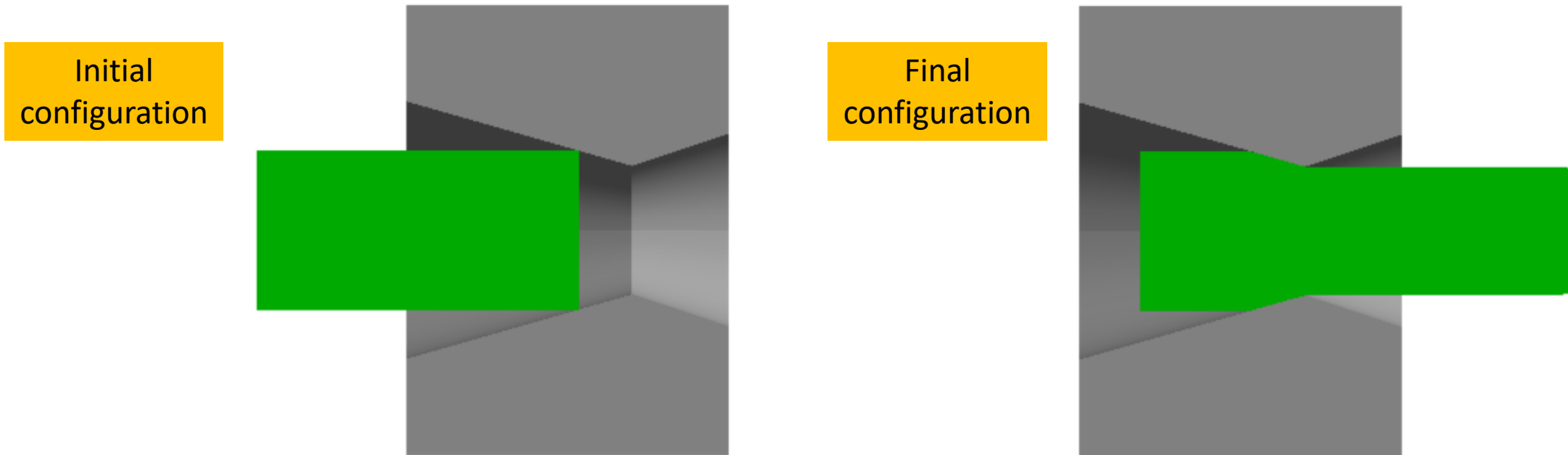


Wire Drawing Analysis using Ansys

ME756 - Numerical Modelling of
Manufacturing Processes

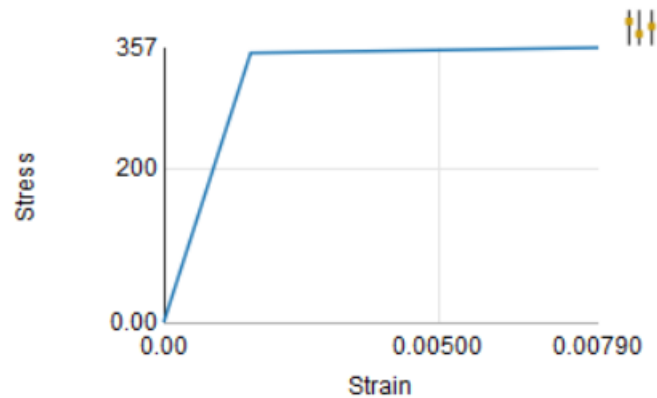
/ Problem Description

- 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



/ 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	-	-

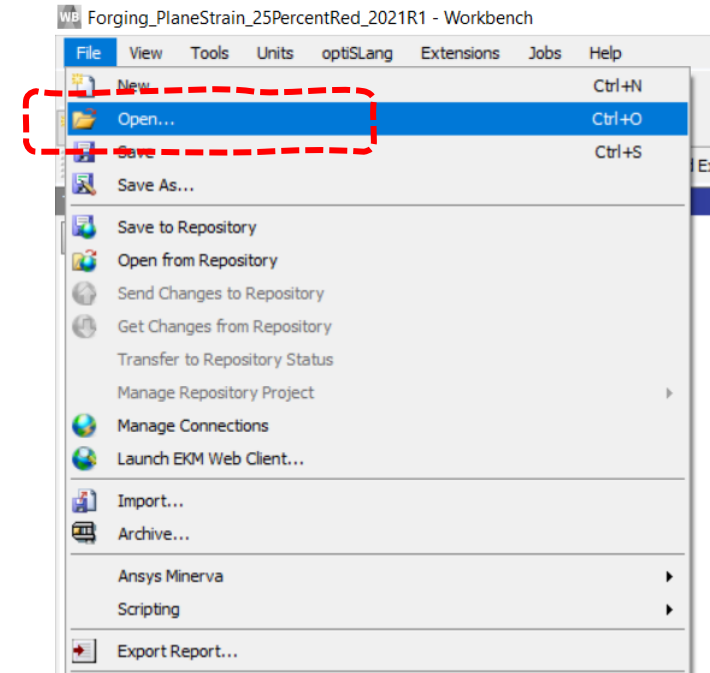
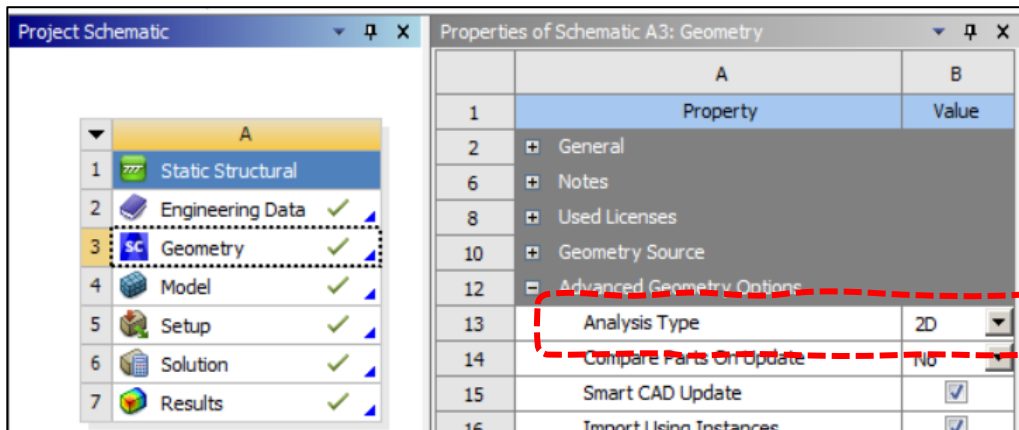


True-stress vs True-Strain Curve

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.

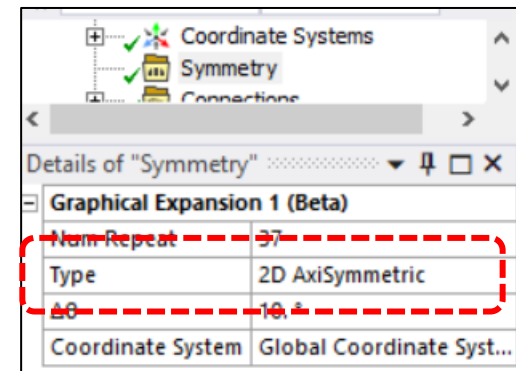
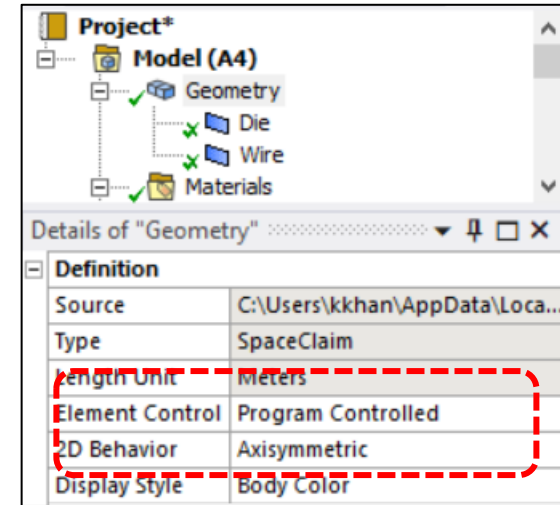
The screenshot displays the ANSYS Engineering Data interface. The top window, titled "Outline of Schematic A2: Engineering Data", shows a tree view with columns A (Contents of Engineering Data), B (Source), C (Description), D, and E. Row 3 is highlighted, showing "Carbon steel, 1040, annealed" with a red dashed box around it. The bottom window, titled "Properties of Outline Row 3: Carbon steel, 1040, annealed", shows a table of material properties. The table has columns A (Property), B (Value), C (Unit), D, and E. Row 9 is highlighted, showing "Isotropic Elasticity" with a red dashed box around it. Row 19 is also highlighted, showing "Bilinear Isotropic Hardening".

	A	B	C	D	E
1	Contents of Engineering Data	Source	Description		
2	Material				
3	Carbon steel, 1040, annealed		Carbon steel, AISI 1040, annealed Data compiled by the Granta Design team at ANSYS, incorporating various sources including JAHM and MagWeb. ANSYS Inc. provides no warranty for this data.		
4	Structural Steel		Fatigue Data at zero mean stress comes from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1		
			Tungsten carbide		

	A	B	C	D	E
1	Property	Value	Unit		
2	Material Field Variables	Table			
3	Density	7850	kg m ⁻³		
4	Isotropic Secant Coefficient of Thermal Expansion				
9	Isotropic Elasticity	Tabular			
19	Bilinear Isotropic Hardening				
20	Yield Strength	3.505E+08	Pa		
21	Tangent Modulus	1.08E+09	Pa		

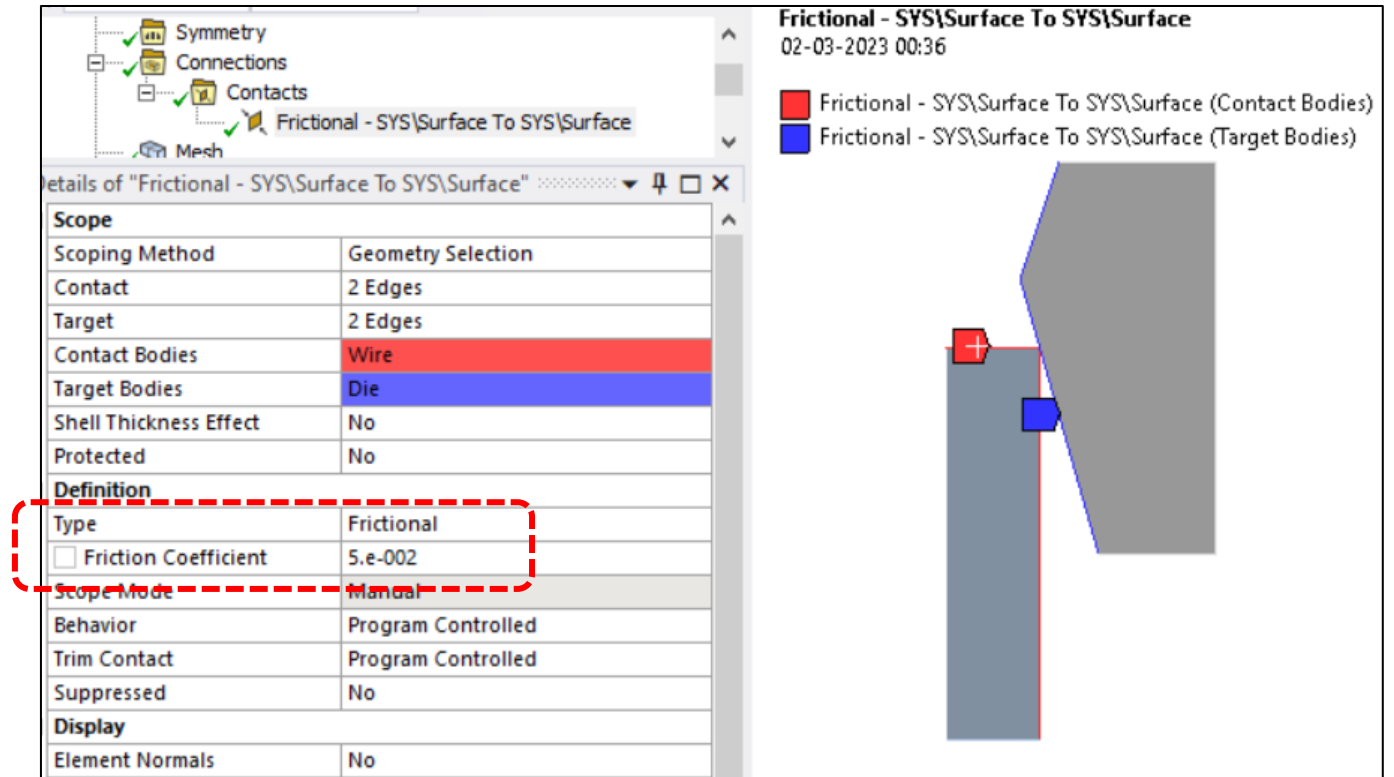
/ Mechanical Model

- 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

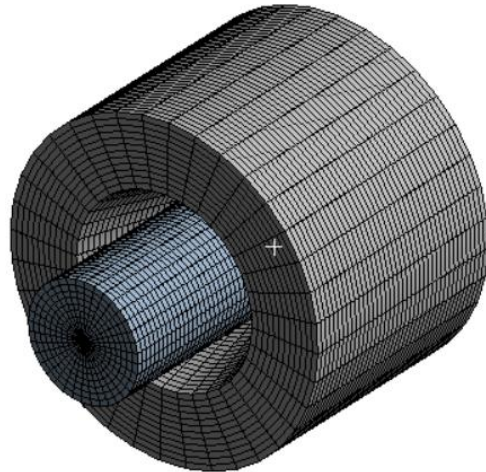
- The interaction between the wire and the die is defined using frictional contact with a friction coefficient of 0.05



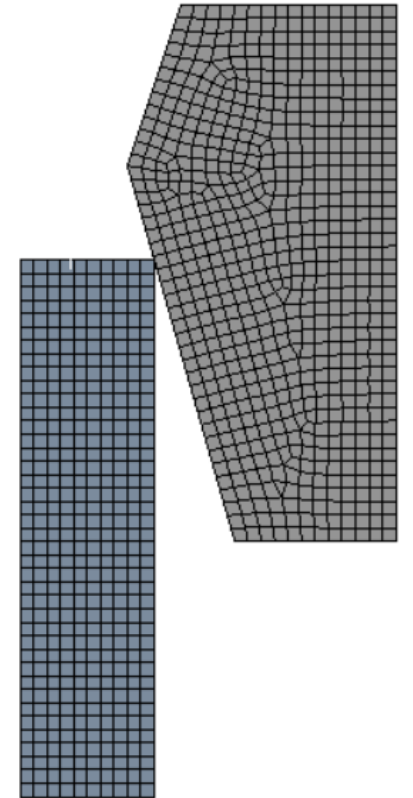
/ Meshing

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

With graphical expansion on

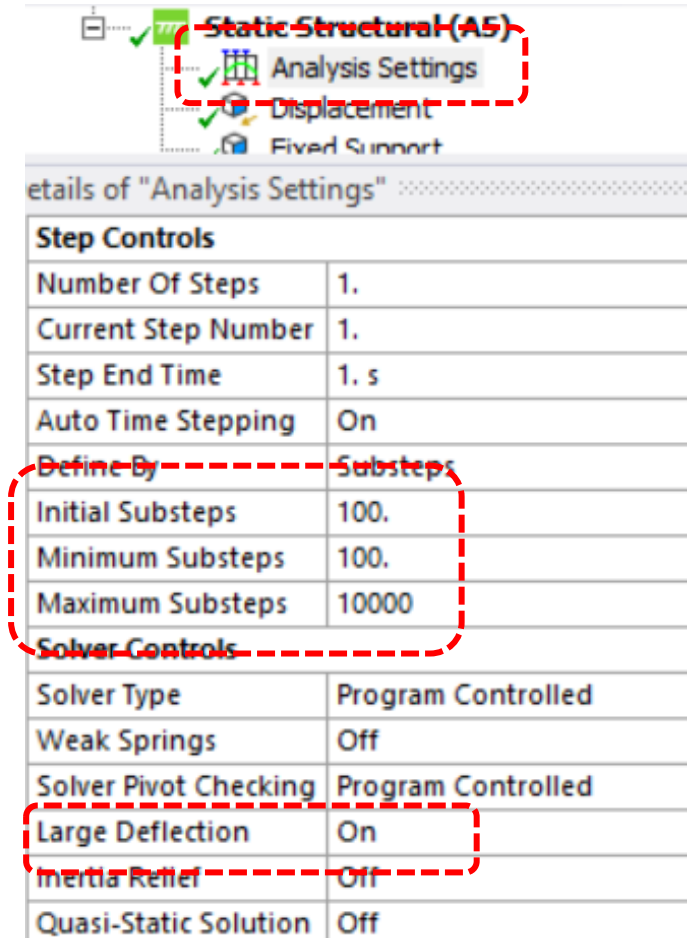


Mesh	
Static Structural (A5)	
Details of "Mesh"	
Display	
Defaults	
Physics Preference	Mechanical
Element Order	Program Controlled
<input type="checkbox"/> Element Size	0.25 mm
Sizing	
Quality	
Inflation	
Batch Connections	
Advanced	
Statistics	
<input type="checkbox"/> Nodes	3414
<input type="checkbox"/> Elements	1068



/ Analysis Setting

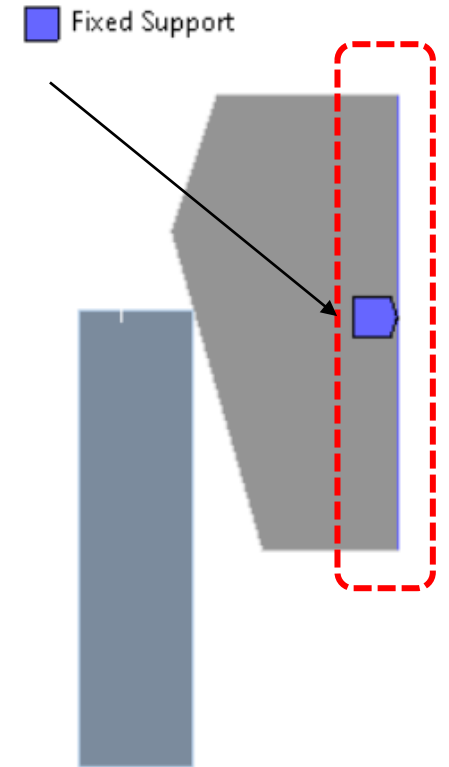
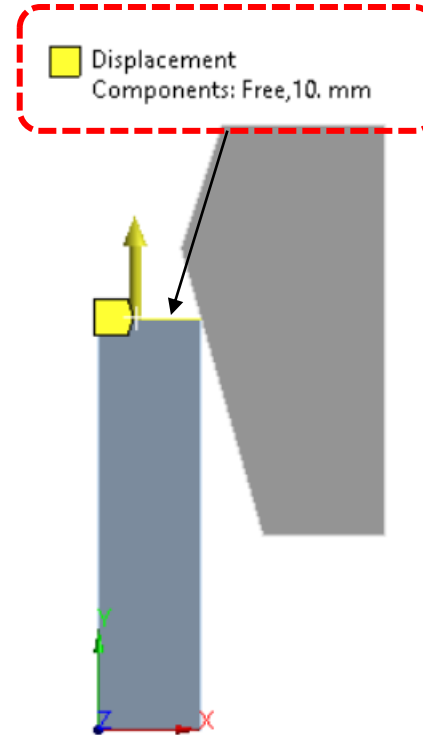
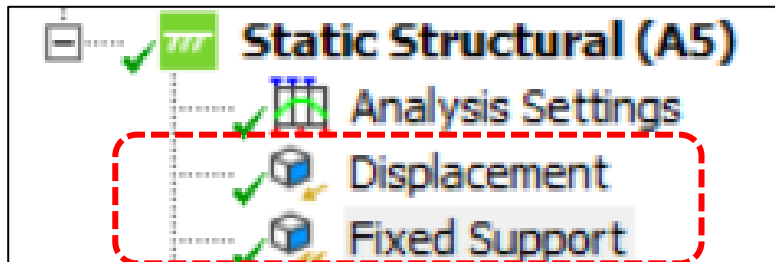
- Note that the problem is solved using multiple substeps as it is a nonlinear problem
- Also the “Large Deflection” option is set to “on” as it is a large strain problem going to the plastic region.



Details of "Analysis Settings"	
Step Controls	
Number Of Steps	1.
Current Step Number	1.
Step End Time	1. s
Auto Time Stepping	On
Define By	Substeps
Initial Substeps	100.
Minimum Substeps	100.
Maximum Substeps	10000
Solver Controls	
Solver Type	Program Controlled
Weak Springs	Off
Solver Pivot Checking	Program Controlled
Large Deflection	On
Inertia Relief	Off
Quasi-Static Solution	Off

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.

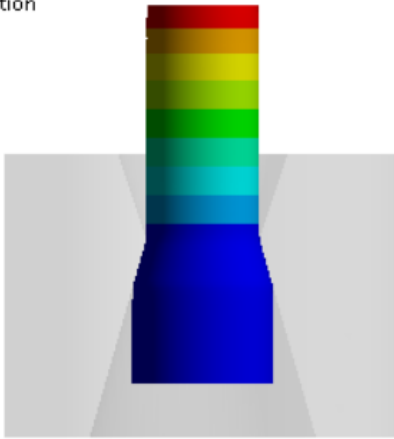


Results

Total Deformation

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

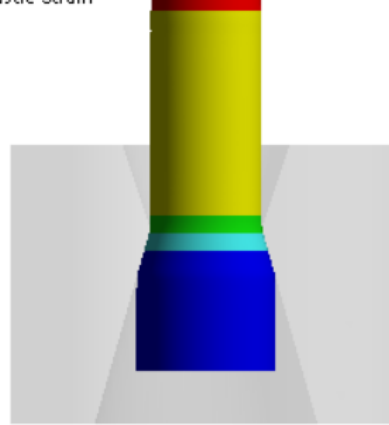
10.015 Max
9.6397
9.2648
8.8899
8.5151
8.1402
7.7653
7.3904
7.0155
6.6406 Min



Eq. Plastic Strain

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

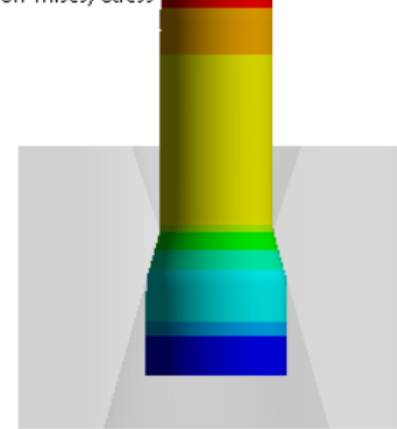
0.80857 Max
0.6
0.45
0.3
0.15
0 Min



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

