COMPUTATIONAL ENGINEERING 2 (MT2527) ASSIGNMENT-3

Group Members:

- ⇨ Akhil Mora 95072-6954

- ⇨ Chetan Bhandare 941009-3273

- ⇨ Raghavendra Machipeddi 950606-6894

- ⇨ Pulicherla Yashpal Surendhar Goud 950419-4995

Abstract

Plasticity refers to the material response beyond the yield point. A ductile material when subjected to stresses beyond the elastic limit, it will yield acquiring larger permanent deformations. When Plasticity is involved, true stresses and true strains are considered rather engineering stresses and engineering strains.

A Conductor pipe is usually operated under internal pressures. They are pre-treated with internal pressures high enough to induce plastic deformation and when relieved from those pressures there are residual stresses induced in the material which may affect the durability of the pipe.

In the problem the conductor pipe's durability is tested before delivering it to the customer.
After the conductor pipe is undergone plastic deformation it is subjected to a working pressure to find the deformation and stress distribution and to determine the optimum pressure to pre-treat the conductor pipe.

Contents

1. Notations

P	Work Pressure
P_{in}	Internal Pressure
R_{in}	Inner radius
R_{out}	Outer Radius
L	Length of the test piece
b_0	Width of the test piece
h_0	Height of the test piece
L	Load acting on the sample
R	Engineering stress
E	Engineering strain
e_e	Elastic strain
e_p	Plastic strain
σ	True stress
ε	True strain
E	Young's modulus
E_{iso}	Isotropic hardening modulus
E_{Tiso}	Tangential Isotropic hardening modulus
σ_{y0}	Initial Yield stress of the material
σ_y	Yield stress of the material after plasticity is induced

2. Introduction

A ductile material when experiences stresses beyond the elastic limit it yields acquiring permanent deformations. This phenomenon is called Plasticity. It is considered important as an Energy-absorbing mechanism for structural service. Plastic response plays a prominent role while in metal forming operations.

Plastic deformation occurs when there is a slip between planes of atoms due to shear stresses (deviatory stresses). This results in unrecoverable strains or can say permanent deformation after the load is removed. Slipping doesn't generally result in any volumetric strains like in the condition of compressibility unlike elasticity.

While engineering stress-strain is used for small strain analyses, true stress-strain should be used for Plasticity as they are more representative measures of the state of the material. Also the introduction of 'plastic strain' is done while the material has undergone to plastic deformation.

When plastic deformation is induced in the material and then the load is removed the measured strain is divided into elastic strain and plastic strain. And when the material is again subjected to loading it will deform to a new yielding point which is observed to be increased than the material before undergoing plastic deformation. This betterment of yield strength of the material is due to strain hardening effects. This phenomenon is called 'Work Hardening' or 'Cold working'.

3. Problem:

3.1 Problem statement:

A conductor pipe with its dimensions is shown below in figure 1. The conductor pipe is made of mild carbon steel.

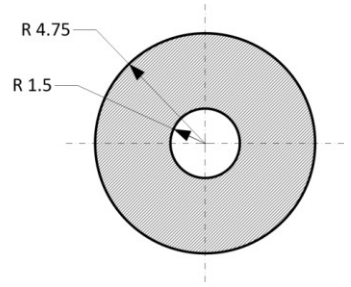


Figure 1: Cross-section of the pipe

A standard tensile test is performed to a test piece of the material to determine the characteristics of the material. The geometry of the test piece is shown in figure 2.

The data of time, respective forces and displacements are given.

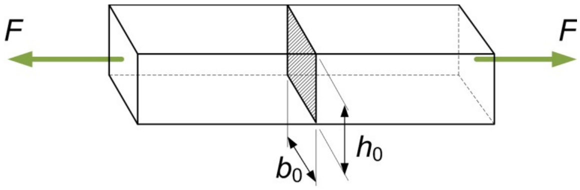


Figure 2; Dimensions of the test piece

The un-deformed width of the test piece, $b0$, was 19.01 mm, the height, $h0$, was 1.01 mm and the length was 80 mm.

The conductor pipe is pre-treated with an internal pressure which is capable of inducing Plastic deformation. As the pressure is released, residual stresses are left in the pipe, which affect the durability of the pipe.

The deformation and stress distribution in the pipe is to be determined during normal working conditions if the pipe is pre-treated with an internal pressure of 160 MPa and the work pressure is 125 MPa. Material properties such as Young's modulus, density and Poisson's ratio are required to calculate these values, which are obtained from the tests performed. The optimal pressure to pre-treat the pipe is also to be determined.

3.2 Method

The data file is provided with time, axial displacement due to corresponding loads of the sample test. These values are used to calculate engineering stresses and strains. After obtaining these engineering stress and strain, these values are used in the calculation of true stress and strain. From these true stresses and true strains, material properties like Young's modulus, Poisson's ratio are determined. All these are implemented in MATLAB.

The Engineering stress is calculated as

$R = \frac{F}{S_0}$

Where F is force and S_0 is cross-sectional area.

And the Engineering strain is calculated as,

$$e = \int_{l_0}^{l_1} \frac{dl}{l_0} = \frac{l_1 - l_0}{l_0} = \frac{\Delta l}{l_0}$$

Where *l* is length of the test piece and *dl* is the infinitesimal change in length.

After obtaining the Engineering Stress and Engineering Strain, the true stress and true strains can be obtained by the following relation:

The true stress (σ) is

$$\sigma = R(e + 1)$$

The true strain (ε) is given by,

$$\varepsilon = \ln(1 + e)$$

For an elastic model, the Young's modulus is obtained by,

$$E = {}^{\sigma}/_{\varepsilon}$$

While for a plastic model the strain is given as,

$$\varepsilon = \varepsilon_e + \varepsilon_p$$

Where, ε_e is elastic strain and ε_p is plastic strain. The plastic part of the material starts at yield point.

The plastic strain is obtained by,

$$\varepsilon_p = \varepsilon - \frac{\sigma}{E}$$

The conductor pipe with obtained material properties from MATLAB is solved in COMSOL Multiphysics to determine the Stress and Deformation when the pipe is pre-treated with an internal pressure of 160MPa and then subjected to the working pressure of 125MPa. The optimal pressure to pre-treat the pipe is done using parametric sweep in COMSOL.

3.3. Solution

a) MATLAB

A data of the tensile test of the material for a test piece of length (l) 80mm, width (w) 19.01mm and height (h) of 1.01mm with different loads and corresponding axial displacements is in the form of text file. The text file is loaded in MATLAB which contains the data of time, load for corresponding axial displacements for all the load points. Engineering stresses and engineering strains for all the points are calculated using the below relations:

For engineering stress,

$$R = \frac{F}{S_0}$$

Where F is the force and S_0 is the cross-sectional area of the test piece.

For engineering strain,

$$e = \int_{l_0}^{l_1} \frac{dl}{l_0} = \frac{l_1 - l_0}{l_0} = \frac{\Delta l}{l_0}$$

Where *dl* is the change in length.

Engineering stresses given by the load acting on the initial cross section. These aren't good values to consider because when the load is applied, parameters like length, width, thickness might change. Thus the true stress is considered, as it the ratio of the load by the cross section at that instance. Similarly, engineering strain is defined as the deformation in the direction of force applied by initial length. Whereas deformation in the direction of force applied by length at that instant is considered as true strain.

The true stress and strain are more realistic values, hence they are considered. Thus the true stresses and true strains of the material for all the 859 points are calculated using the relations and plotted.

For true stress,

$$\sigma = R(e + 1)$$

For true strain,

$$\varepsilon = \ln(1 + e)$$

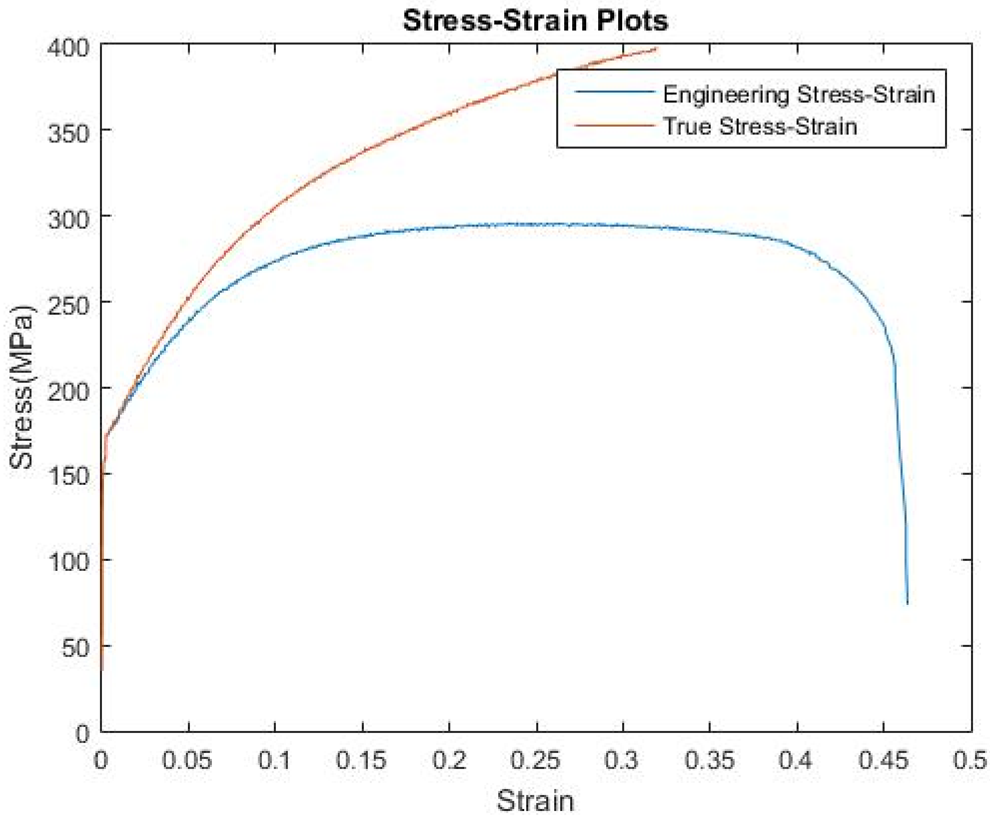


Figure 3: Stress-Strain plots

The modulus of elasticity is the initial linear part of the Stress-Strain Diagram. By observing the Stress-Strain Diagram we see that there are many data points in the elastic region. So the Young's modulus is calculated by averaging the ratio of stress and strain until the material behaves elastically.

$$E = {}^{\sigma}/_{\varepsilon}$$

The value of Young's modulus (E) obtained is 393.43 Gpa.

After calculating the young's modulus, the plastic strain of the material is calculated using the below relation:

$$\varepsilon_p = \varepsilon - \frac{\sigma}{E}$$

Only elastic strains exist before yielding. Therefore, the yield strength of the material is the point where the plastic strain is equal to zero. By looking at the values of Plastic strain, we interpolate the data points to get the initial yield Stress.

The table showing the material model is shown below:

True stress (Mpa)	True plastic strain	Young's Modulus (Gpa)
125.2	0	393.43
134.49	$5.8066*10^{-5}$	
145.3	$2.2049*10^{-4}$	
149.08	$3.6079*10^{-4}$	
152.59	$4.8228*10^{-4}$	
154.58	$6.3657*10^{-4}$	
155.71	$6.4367*10^{-4}$	
157.19	$7.6978*10^{-4}$	

Table 1: True stress and true plastic strain

As we choose a plasticity model in Comsol, the isotropic tangent modulus is to be determined.

The isotropic hardening modulus is obtained in MATLAB using the below formula:

E_{Tiso}=ε (pl_start:uts)\(σ(pl_start:uts)- σ_{y0})

After solving, the value of E_{Tiso} obtained is 1125.56 Mpa.

The yield stress of the material after undergoing plastic deformation is found out by the relation

$$\sigma_y = \sigma_{y0} + \frac{E_{Tiso}}{1 - \frac{E_{Tiso}}{E}} * \varepsilon_p$$

where E_{Tiso}isotropic hardening modulus (elasto-plastic modulus) is, σ_{y0} is initial yield stress of the material i.e. before undergoing plastic deformation and σ_y is yield stress after undergoing plastic deformation (hardening).

Using the above properties, the model is solved in COMSOL Multiphysics.

b) COMSOL Multiphysics

The conductor pipe is pre-treated and solved for deformation and stress distribution in COMSOL Multiphysics.

The cross section of the pipe is designed with inner radius of 1.5mm and outer radius of 4.75mm. Only quarter part of the geometry is modelled because if the symmetry of the pipe and it takes less computational time.

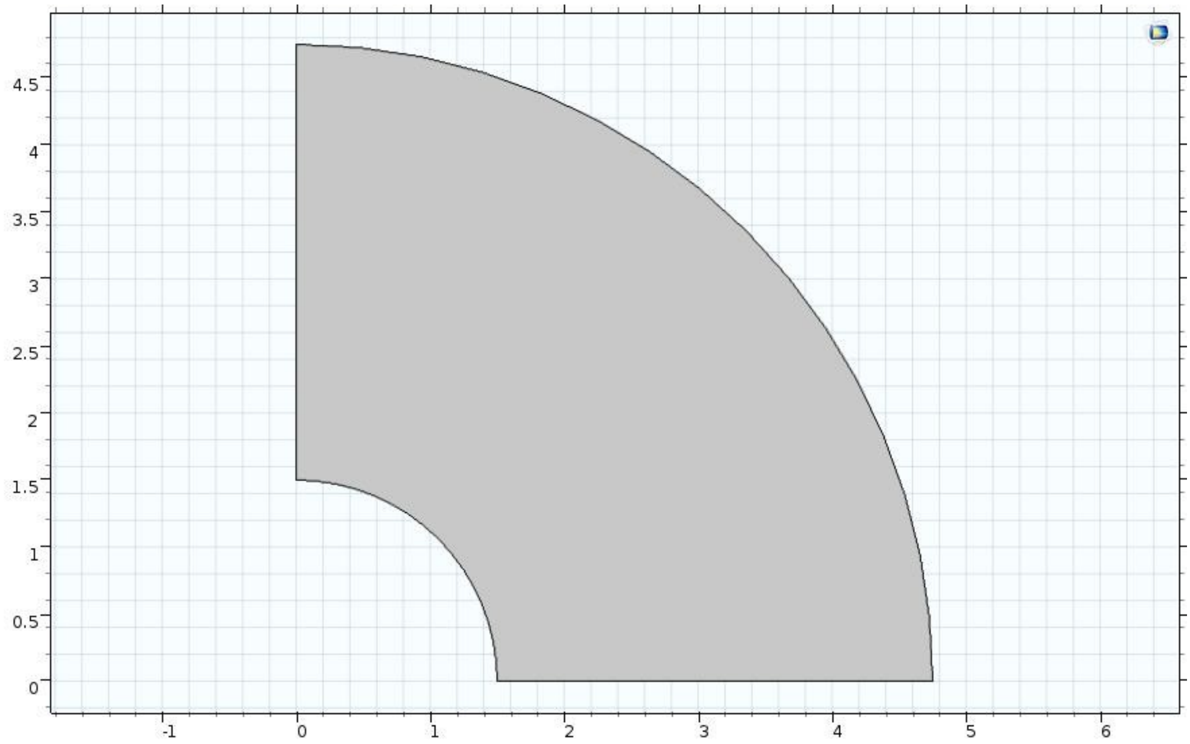


Figure 4: Geometry of the pipe (when symmetry is considered)

Under the tab Global definitions, the parameters and load groups are defined. The pre-treated pressure is given as a parameter as we later perform a parametric sweep to find the optimal pre-treatment pressure. Load groups are later assigned to Boundary loads.

Parameters

Parameters

Name	Expression	Value	Description
pressure	160[MPa]	1.6E8 Pa	

Figure 5: Parameters

Material properties are obtained by studying the data from the tensile test performed on the material test piece. These are calculated in MATLAB and are given in COMSOL by creating a blank material and adding the required properties to the material.

Young's modulus	E	393.43e9	Pa	Basic
Poisson's ratio	nu	0.3	1	Basic
Density	rho	7850	kg/m³	Basic
Initial yield stress	sigmags	125e6	Pa	Elastoplastic material model
Isotropic tangent modulus	Et	1126.56e6	Pa	Elastoplastic material model

Figure 6: Material properties

Under the physics solid mechanics, linear elastic material with a plasticity model and is solved assuming isotropic hardening as tangent data is selected. The boundary loads (three for each load group) are applied for the inner circle edge and the respective load groups are assigned to each boundary load.

$$\sigma_y = \sigma_{y0} + \frac{E_{Tiso}}{1 - \frac{E_{Tiso}}{E}} * \varepsilon_p$$

Constraints are given to the edges, i.e. symmetry for vertical and horizontal edges.

The geometry is then meshed.

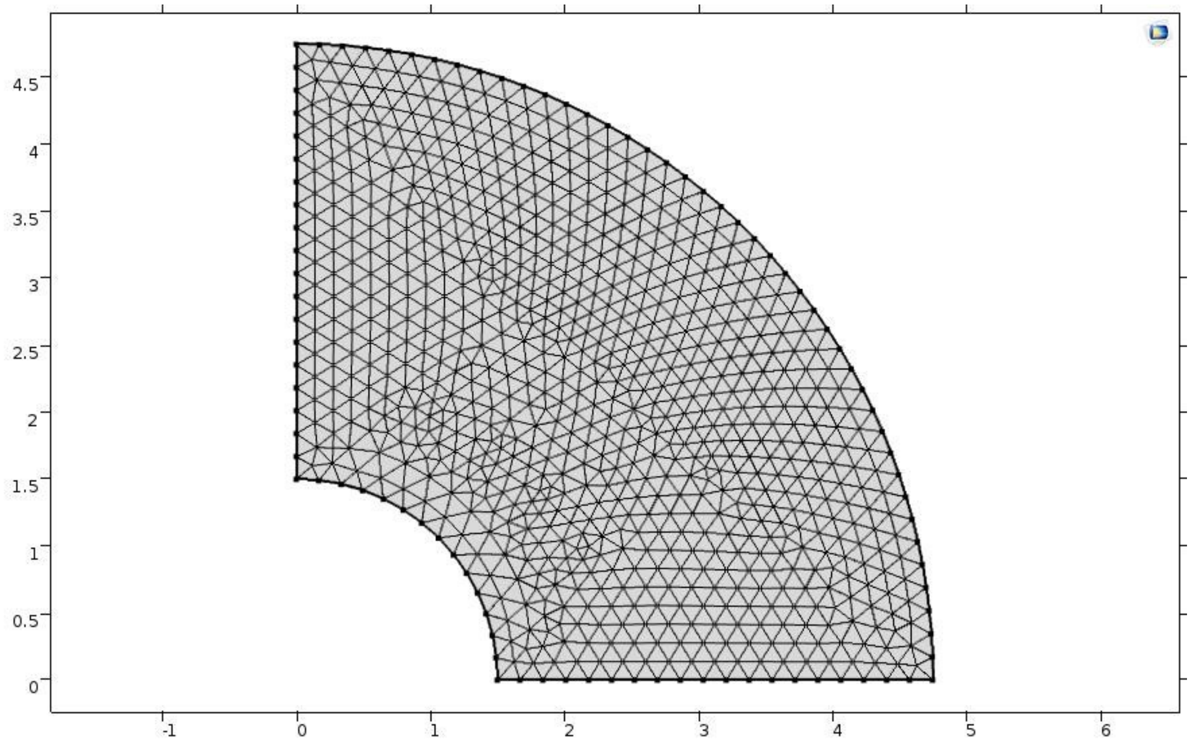


Figure 7: Model after Meshing

Before solving, under the stationary study step, in the tab study extensions three load cases are defined and are linked to the three load groups created before. The solution from each step is reused for the next step. So, by this our pipe is first pre-treated with an internal pressure of 160Mpa and is at rest and then a working pressure of 125MPa is applied. Under the study, a parametric sweep option is chosen to find out the optimal pressure to pre-treat the pipe.

The parameter ‘pressure’ created in the first step is added in the parametric sweep with a user-defined range and step size. The results from the sweep can be later used to find the optimal pressure.

3.4 Results

The resulted stress distribution and the deformation of the conductor pipe with a pre-treated pressure of and a working pressure of 125MPa is shown in the following images:

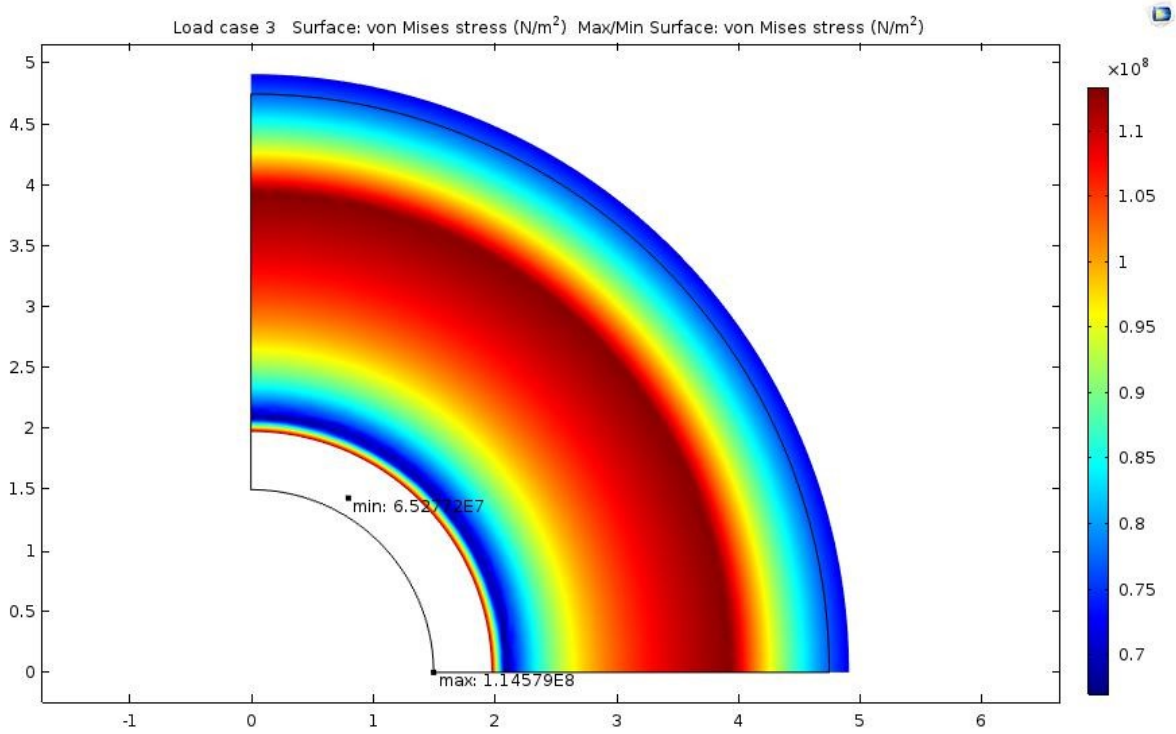


Figure 8: Von Mises stress

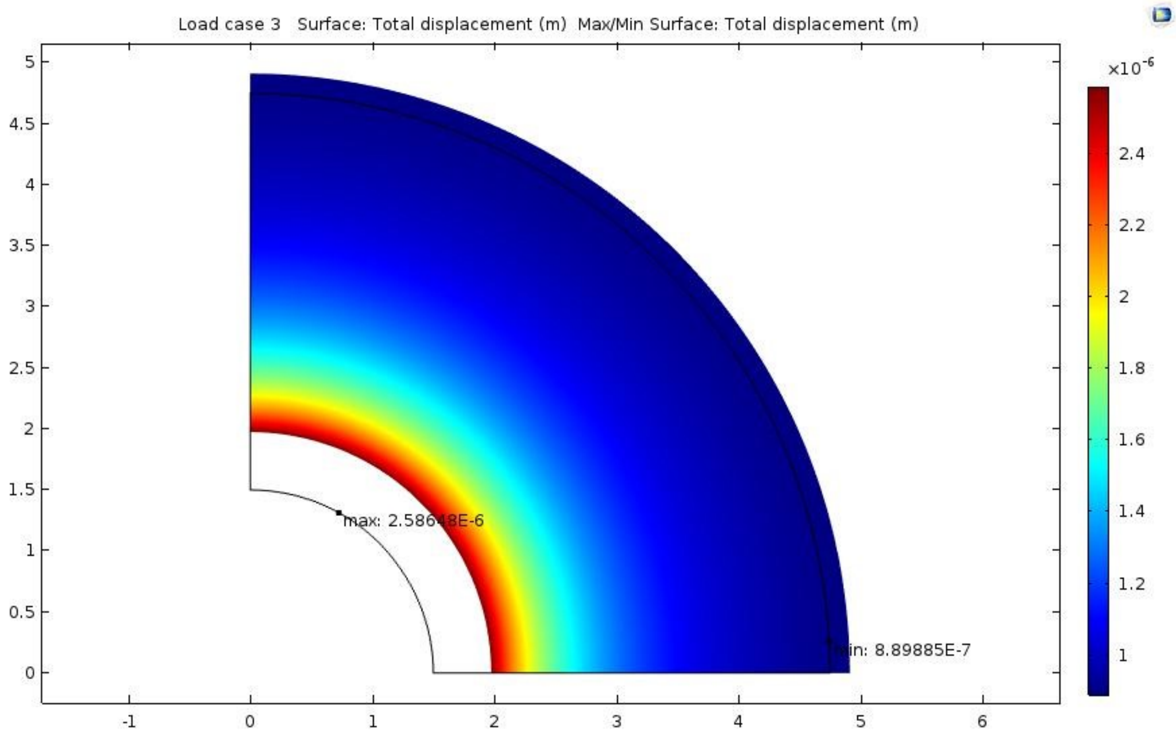


Figure 9: Deformation

The plot of parametric sweep between pre-treated pressures and von-mises stresses.

Finally, the results from parametric sweep are converted into a table and a graph is plotted between the pre-treating pressure and the von Mises stress obtained. From this graph the optimal pressure is chosen.

Pressure($x10^8$) Pa	Stress ($x10^8$) N/m^2
1.4	1.1881
1.4242	1.1896
1.4525	1.1862
1.4768	1.1658
1.5010	1.1544
1.5253	1.1514
1.5495	1.1497
1.577	1.1472
1.6020	1.1456
1.625	1.1444
1.6343	1.1410
1.6505	1.1497
1.7030	1.1568
1.7515	1.1612
1.8	1.1632

Table 2: Parametric sweep results

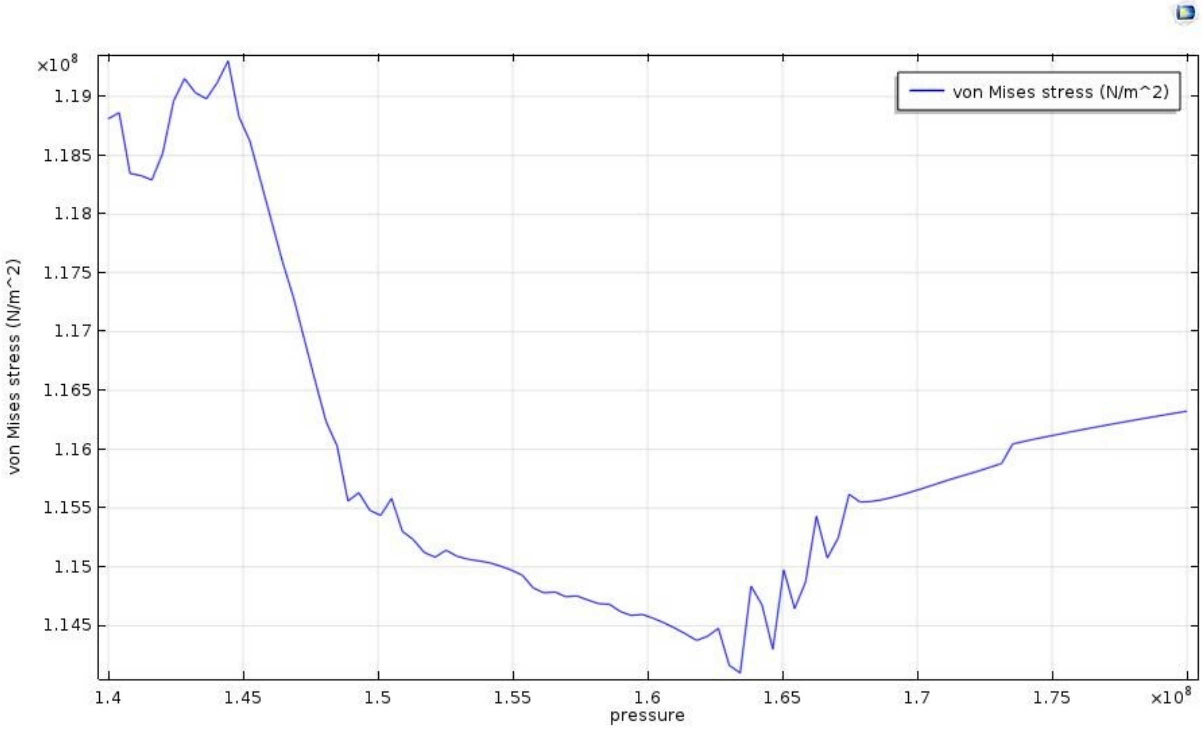


Figure 10: Results from Parametric sweep

The optimal pressure to pre-treat the conductor pipe is obtained 1.6343e8 Pa i.e. 163.43MPa.

The stress distribution of the conductor pipe at optimal pre-treated pressure of 163.43MPa after applying working pressure of 125MPa is shown below.

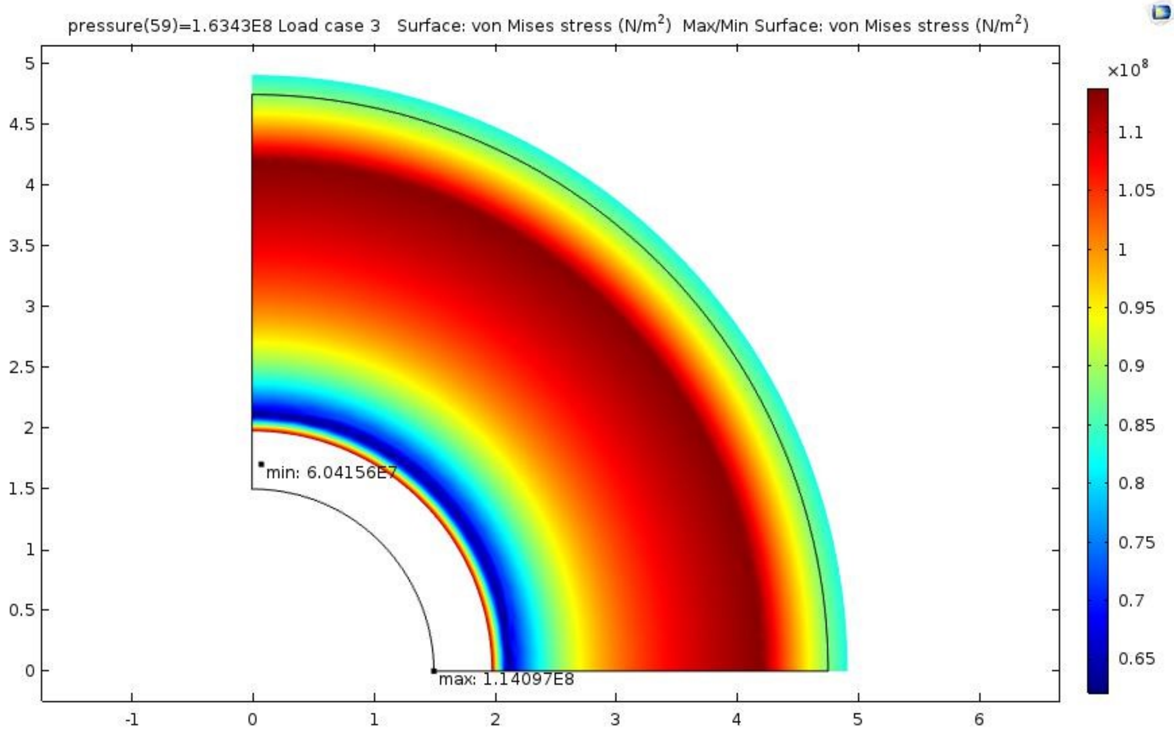


Figure 11: Stress distribution with optimal pre-treated pressure

The deformation of the conductor pipe at optimal pre-treated pressure of 163.43MPa after applying working pressure of 125MPa is shown below.

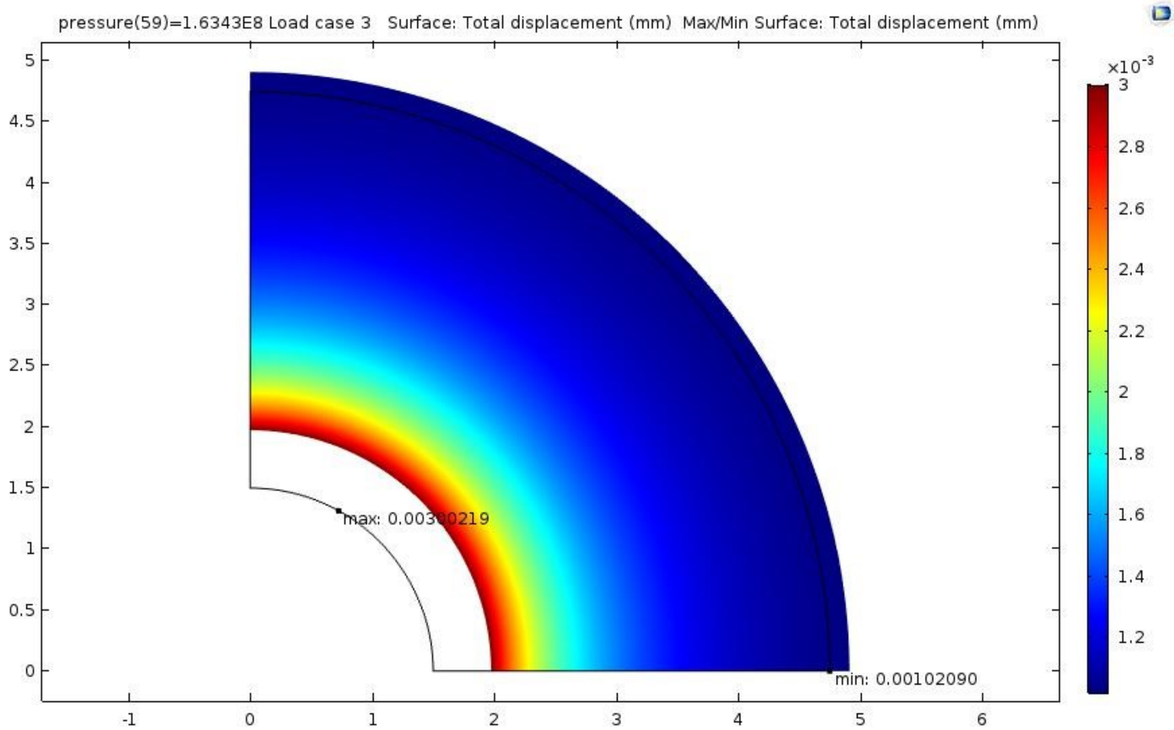


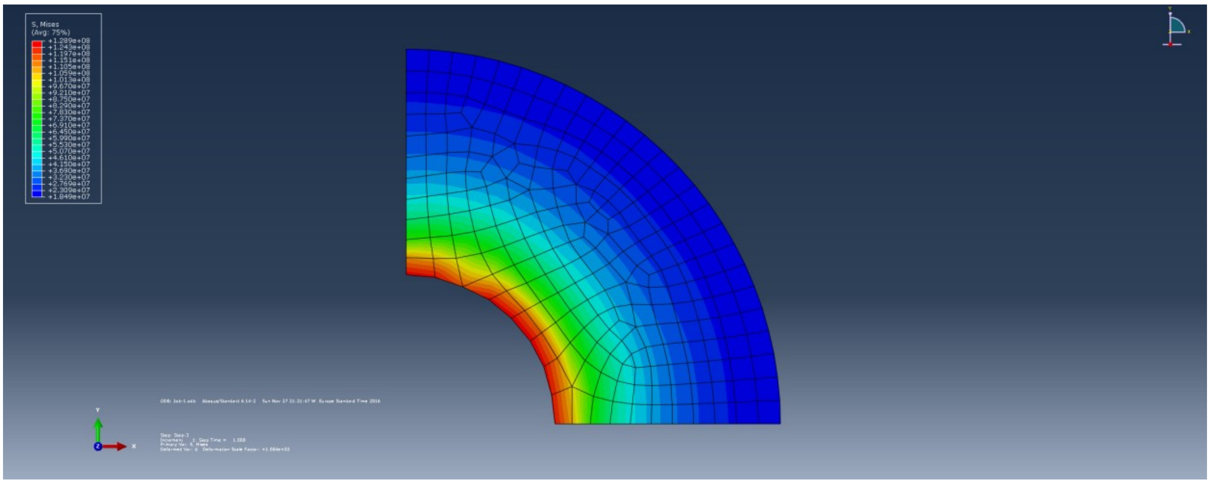
Figure 12: Deformation with optimal Pre-treated Pressure

3.5 Discussions and conclusions

We assumed that the maximum stress will be obtained at the inner surface of the pipe. But the results were not obtained as expected. We found that there are higher stresses also in between the pipe i.e. between the inner and outer radius of the pipe. This may be due to the residual stresses left when the pipe is pre-treated.

The results obtained from COMSOL are validated using ABAQUS. The results obtained from ABAQUS can be seen in the below images. The maximum pressure is only seen at the inner surface of the pipe in ABAQUS which is little different from the one obtained in COMSOL. But the magnitude of stresses obtained in both the cases is similar. The change in the distribution may be because of the post processing method in ABAQUS. We assume that the results are only shown for the final working pressure and the previous stresses induced because of pre-treating are not shown. But looking at the magnitude of the stress we guess there is no mistake done during the pre-processing stage.

From this assignment we learnt about the theory of plasticity. We learnt that pre-treating an elastic material until plastic deformation increases the yield strength of the material. The yield strength is a very important value because the stresses acting on the material depend on this value.



4. References

[1] Elastic-plastic material modelling for FE analysis, Course material.

[2] Computational Engineering, Goran Broman.

[3] COMSOL Multiphysics, User guide.

[4] ANSYS Mechanical Structural Nonlinearities.
(http://inside.mines.edu/~apetrell/ENME442/Labs/1301_ENME442_lab7.pdf)

5. Self-evaluation:

Name/Evaluation	Much More	More	Equal	Less	Much Less	Absent
Akhil			X			
Chetan			X			
Raghavendra			X			
Surendhar			X			

Table 3: Self-evaluation Table

Appendix

```
clc
close all
clear all
%%
load ass3data.txt
forcekN=ass3data(:,2);
dispmm=ass3data(:,3);
length=80; %mm
width=19.01; %mm
height=1.01; %mm
csa=width*height; %mm2
force1=forcekN*1e3; %N

for i=1:859
   stress(i)=force1(i)/csa; %N/mm2
   strain(i)=(dispmm(i))/length;
   truestress(i)=stress(i)*(strain(i)+1);
   truestrain(i)=log(1+strain(i));
   E= 393433.91; %n/mm2
   trueplasticstrain(i)=truestrain(i)-(truestress(i)/E);
 end
sstress=stress';
sstrain=strain';
mat=[sstrain;sstress];
%% youngs modulus
for i=1:12
   e(i)=truestress(i)/truestrain(i);
end
Eyoung=sum(e)/12
%% plots

plot(strain,stress)
hold on
plot(truestrain(1:747),truestress(1:747))
title('Stress-Strain Plots')
xlabel('Strain')
ylabel('Stress(MPa)')

legend('Engineering Stress-Strain','True Stress-Strain')
hold on
[num] = max(truestress(:));
[x y] = ind2sub(size(truestress),find(truestress==num));

%% yield
```

```
yieldinitial=truestress(5)+([(134.4968-121.5131)/((5.8066+2.8892)*10^-5)]*2.8892e-5)

%% elasto plastic modulus
truestrain1=truestrain';
truestress1=truestress';
Etiso=truestrain1(6:747)\(truestress1(6:747)-yieldinitial)

yield=yieldinitial+((Etiso/(1-(Etiso/E)))*trueplasticstrain);

%% extrapolation
x=linspace(0.3191,0.5,1000);
y=397.6+(0.7*((x-0.3166)/(2.5e-3)));

extruestress=[truestress(1:747) y(1) y];
extruestrain=[truestrain(1:747) x(1) x];
plot(extruestrain,extruestress)
xlabel('True Strain')
ylabel('True Stress(MPa)')
title('Extrapolated True Stress-True Strain Plot')
legend('Linear Extrapolation')
```