

## **Statement 1**

This work has not previously been presented in any form to Glyndwr University or at any other institutional body whether for assessment or for any other purposes. Save for any express acknowledgements, references, and/or bibliographies cited in the work, I confirm that the intellectual content of the work is the result of my own efforts and no other person.

## **Statement 2**

It is acknowledged that the author of this work shall own the copyright. However, by submitting this copyright work for assessment, the author grants to the University a perpetual royalty-free license to do all or any of those things referred to in section 16(l) of the copyright, designs, and patents act 1988 (copy work; to issue copies to the public; to perform or show or play the work in public, to broadcast the work or make an adaptation of the work).

Signed: \_\_\_\_\_

Date: \_\_\_\_\_

---

## Table of Content

Awareness of Sustainability .....	1
1. Question.....	1
2. Question A .....	4
3. Question B.....	6
4. Question C.....	10

---

## List of Figures

Figure 1: Schematic of Cantilever beam ..... 2

---

## **1. Introduction**

The Ansys Workbench is a project-management program which is based around a project schematic view which ties together the entire simulation process. It can be considered as the top-level interface linking all ANSYS software tools. Workbench handles the passing of data between ANSYS Geometry/Mesh/Solver/Post-processing tools. This greatly helps project management. You do not need worry about the individual files on disk (geometry, mesh etc). Graphically, you can see at-a-glance how a project has been built

## 2. Given Question

A cantilever beam of the length  $L$  is subjected to a load “ $F$ ” at its free end.

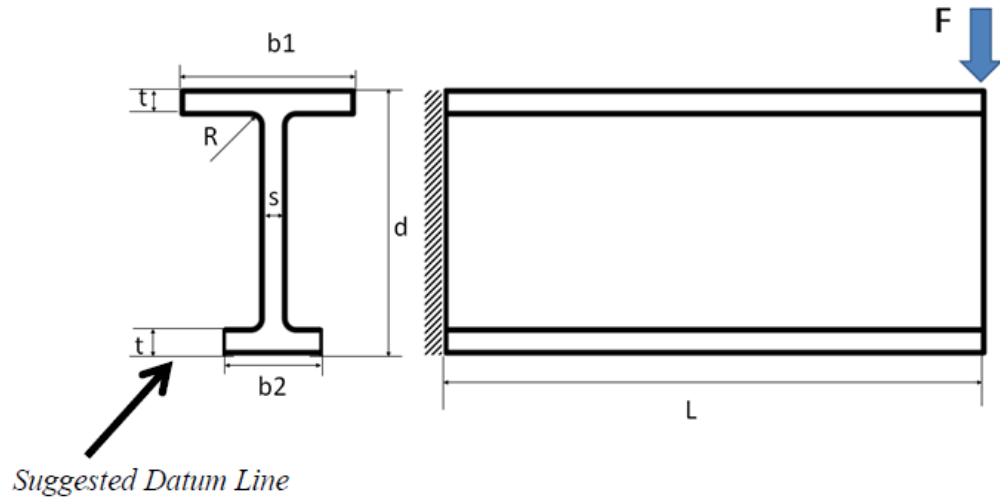


Figure 1: Schematic of Cantilever beam

The following dimensions are given:

Top width:	$b_1$	=	305 mm
Bottom width:	$b_2$	=	195 mm
Height:	$d$	=	320 mm
Beam length:	$L$	=	1800 mm
Web thickness:	$s$	=	32 mm
Flange thickness:	$t$	=	29 mm
Fillet Radius:	$R$	=	27 mm

The load  $F$  is perpendicular to the length  $L$  and is acting on the top edge of the beam:

Load:	$F$	=	1900N
-------	-----	---	-------

The beam is made of structural steel with the following properties:

Young's Modulus:	$E$	=	198 GPa
Poisson's Ratio:	$\nu$	=	0.3

The beam is operated in the following environmental conditions:

Standard Ambient Temperature:	$T_{amb}$	=	22°C
Gravity:	$g$	=	9.81 m/s <sup>2</sup>

---

Questions:

a) Identify the position of the Neutral Axis. Calculate the maximum bending stress by applying the Parallel Axis Theorem. (Fillet radii can be neglected).

**[10 Marks]**

b) Create a 3D CAD model (ANSYS Designmodeler) for the use with ANSYS Mechanical. Describe the different steps you have applied during the geometry preparation. (Fillet radii have to be included)1900.

**[15 Marks]**

c) Conduct a static structural FEA analysis of the beam using ANSYS Mechanical. Determine the maximum bending stress and its location as well as the maximum deflection for an ambient temperature of 22°C. Ensure that your computational mesh is suitable by applying a structural error estimate. These simulations shall be carried out including gravity (acting in the direction of force F). Describe the numerical set-up and discuss the results achieved.

**[15 Marks]**

d) Due to a change in the operating environment, the beam is now exposed to an ambient temperature of  $T_{amb}=85^{\circ}\text{C}$ . Discuss any differences in stress and/or deformation for this case in relation to the initial ambient conditions of  $T_{amb,ini}=22^{\circ}\text{C}$ .

**[10 Marks]**

---

### 3. Question A

Calculating the different Areas and the different moment of Intertia.

A=Area

I=Second Moment of Area

$$A1 = 305 * 29$$

$$A1 = 8845mm^2$$

$$I1 = \frac{305 * 29^3}{12}$$

$$I1 = 619887,08mm^4$$

$$A2 = 262 * 32$$

$$A2 = 8384mm$$

$$I2 = \frac{262^3 * 32^3}{12}$$

$$I2 = 47959274,67mm^4$$

$$A3 = 195 * 29$$

$$A3 = 5655mm^2$$

$$I2 = \frac{195^3 * 29^3}{12}$$

$$I2 = 396321,25mm^4$$

---

hn=Distance from neutral Axis

$$y = \frac{A1 * y1 + A2 * y2 + A3 * y3}{A1 + A2 + A3}$$
$$y = \frac{14,5 * 5655 + 160 * 8384 + 305,5 * 8845}{5655 + 8384 + 8845}$$
$$y = 180,2825mm$$

$$I_g = I1 + A1 * (hn - h1)^2 + I2 + A2 * (hn - h2)^2 + I3 + A3 * (hn - h3)^2$$

$$I_{g1} = 619887,08 + 8845 * (29 + 262 + 14,5 - 180,28)^2$$
$$I_{g1} = 139309915,1$$

$$I_{g2} = 47959274,67 + 8384 * (26 + 131 - 180,28)^2$$
$$I_{g2} = 51407432,78$$

$$I_{g3} = 396321,25 + 5655 * (14,5 - 180,28)^2$$
$$I_{g3} = 155812733,8$$

$$I_g = 346530081,1mm^4$$

$$I_g = 3,46 * 10^8$$

$$I_g = 3,46 * 10^{-4}m^4$$

Bending stress Top:

$$320 - 180,2825 = 139,72 \text{ mm}$$

$$\frac{1900N \times 1,8m \times 0,13972m}{3,46 \times 10^{-4}}$$
$$= 1381047,399 \text{ N/m}^2 \text{ or } 1381,047 \text{ MPa}$$

Bending Stress Bottom:

$$\frac{1900N \times 1,8m \times 0,1802825m}{3,46 \times 10^{-4}}$$
$$= 1781983,092 \text{ N/m}^2 \text{ or } 1781,983 \text{ MPa}$$

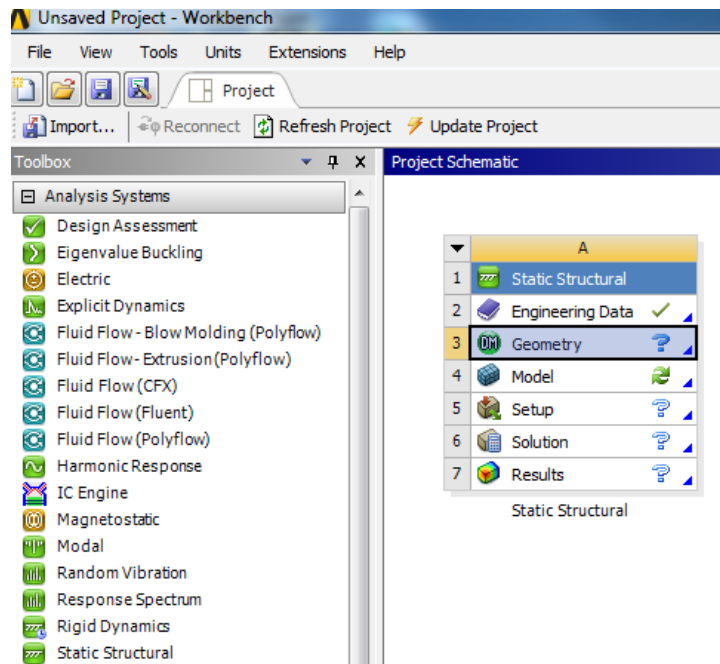


---

## 4. Question B









This assignment is done with the Ansys version 16.2 and is available on the computers in the L100 labs at the Glyndwr University.

Beginning with the Ansys Workbench. The given problem is a static structural problem, so dropping in the “Static Structural” and a green dashed box should appear where it is possible to find every part which is required for the assignment for example the “Engineering Data” for define or access material models for use in an analysis, the “Geometry” for constructing the beam and the “Model” for meshing the model.



After dropping in the mentioned box from the left register, clicking on the part “Engineering Data” for setting or changing the material properties which was given in the assignment.

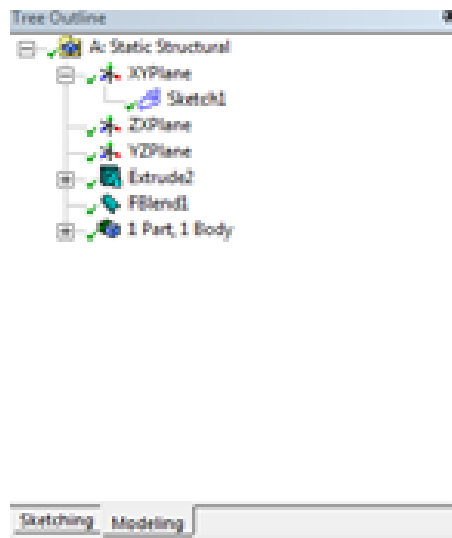
- Temperature: 22 °C
- Poisson’s Ratio: 0.3
- Young’s Modulus: 198 GPa
- Structural Steel

Properties of Outline Row 3: Structural Steel				
	A	B	C	D E
1	Property	Value	Unit	 
2	 Density	7850	kg m <sup>-3</sup>	<input type="checkbox"/> <input type="checkbox"/>
3	 Isotropic Secant Coefficient of Thermal Expansion			<input type="checkbox"/>
4	 Coefficient of Thermal Expansion	1.2E-05	C <sup>-1</sup>	<input type="checkbox"/>
5	 Reference Temperature	22	C	<input type="checkbox"/>
6	 Isotropic Elasticity			<input type="checkbox"/>
7	Derive from	Young...		
8	Young’s Modulus	1.98E+05	MPa	<input type="checkbox"/>
9	Poisson’s Ratio	0.3		<input type="checkbox"/>
10	Bulk Modulus	1.65E+11	Pa	<input type="checkbox"/>
11	Shear Modulus	7.6154E+10	Pa	<input type="checkbox"/>
12	 Field Variables			
13	Temperature	Yes		
14	Shear Angle	No		

---

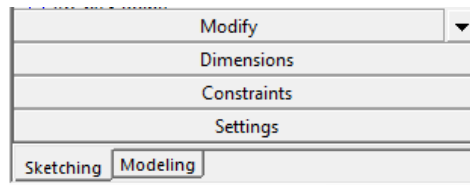
When the material settings are correct, close the window and open the part “Geometry” via double click to launch the design modeller in Ansys. Before going ahead with this part, the most important point is to set the units into mm, because units can’t be set during session. There are two ways to design the desired beam, drawing manual it in the sketching mode or open the point “Concept”, choosing “Cross section” and clicking on “I-Shape”. In this case it was done manually with the tool “Line” because that was the way we learned it.

Before starting the constructing, it is necessary to choose a plane on the left tree, which is shown in the next Figure. In this case I chose the “XY-Plane”.

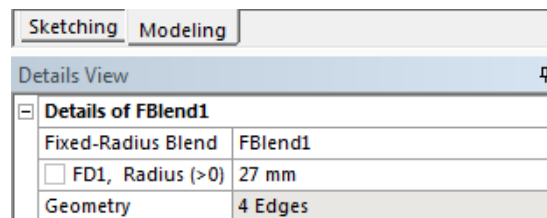


After selecting the desired plane, another point drops out which is called “Sketch1”. For a good overview press the icon “Look at face” and start drawing in the Sketching Mode Tab

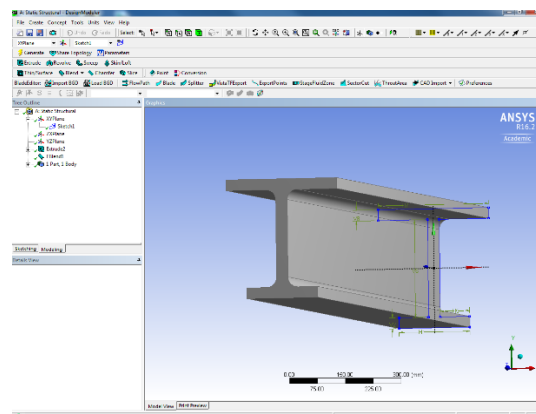
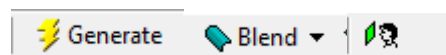
During the drawing part it is helpful to use the “Constraints” to get less dimensioning and using the feature “symmetry” (press CTRL, click on the desired axis where you would like to have the symmetry and select the other two lines). Has every length the given dimension, which were insert with the tool “General” from the “Dimension Toolbox” go to extrude and apply the figure and set the given length.



If everything is done correctly, press the icon “Generate” and the beam should be illustrated. After extruding the beam, there appear another two points on the left tree which are called in this case “Extrude2” and “Blend1” which can be used to access, modify or delete modelling operations or sketches created during the session. Therefore press the icon “Blend” in the toolbox and add the radius at the desired positions. For this action press and hold CTRL and select each edge.



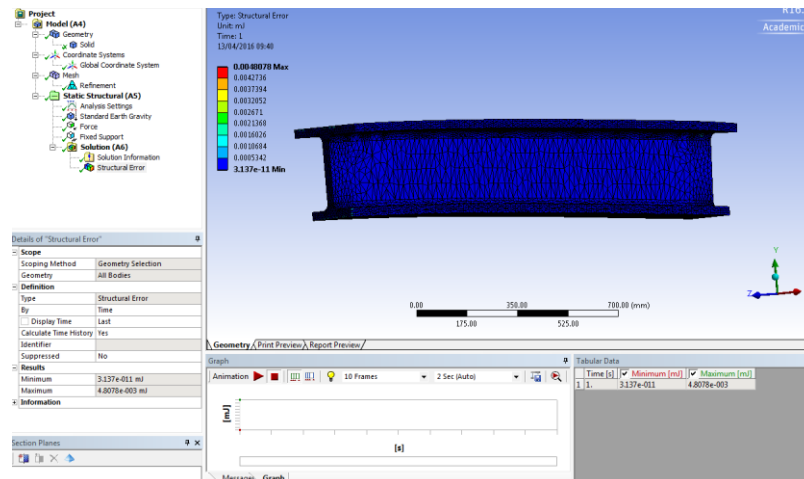
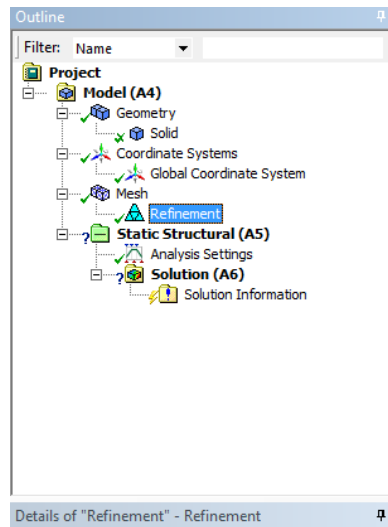
Important buttons in this part of the assignment:



## 5. Question C

Go to the workbench and check that there is a green tick sign beside the Geometry and then double click on the “Mesh” button.

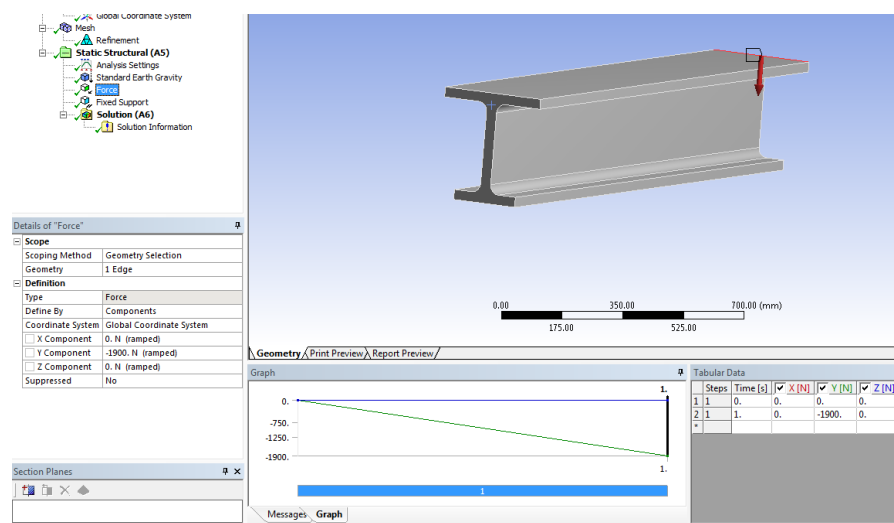
Right click on Mesh and chose Insert and then chose the method, in this case “Refinement.”



Details of "Refinement" - Refinement

<b>Scope</b>	
Scoping Method	Geometry Selection
Geometry	18 Faces
<b>Definition</b>	
Suppressed	No
<input type="checkbox"/> Refinement	3

After meshing go to the point “Static Structural” in the left tree and right click there and insert the “Standard earth gravity”, the “Force” with the desired direction and the given value and the “Fixed support”. At this point, there should be a details of "Force" window in the lower left corner of the setup window. Expand definition and then change Define by to Components. We would also like to look at the stresses and deflection in an explicit direction. Therefore change the the orientation to Z axis.



Are all the given things in the static structural part, go ahead to the point “Solution” in the tree and insert via clicking right mouse click the “Error Estimate”, the “Normal stress” and the “Maximum deflection”.

The software Ansys is actually looking how the load is distributed throughout the part. This is based on the geometry obviously and also the material (atomic structure and all its physical properties). The accuracy of the distribution also is dependent on the chosen mesh size. A mesh was applied to the part and the resolution of this mesh will determine how accurate the distribution is.

---

## 5.1 Error Estimation

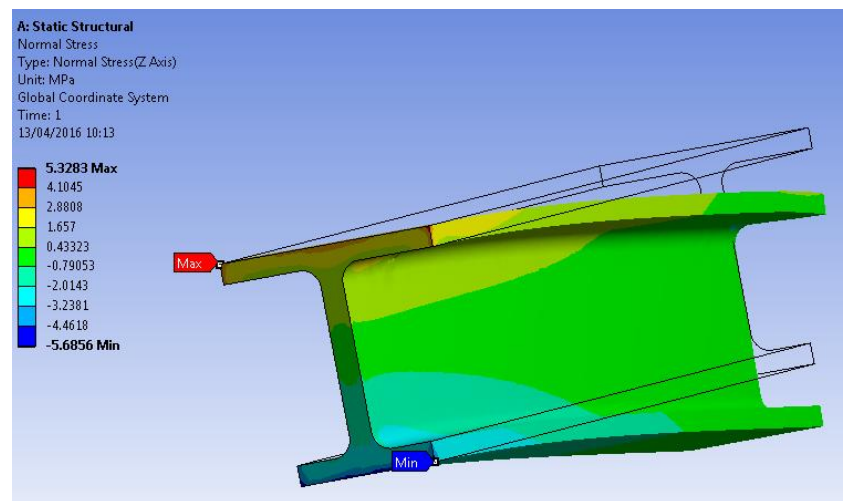
The finite element solution calculates stresses on a basis, i.e., stresses are individually calculated in each element. If you plot the element solution, you will see unaveraged data, which shows the discontinuity between elements. The difference between averaged and unaveraged stresses gives an indication of how “fine” or how “bad” the mesh is. This is the basis for error estimation.

---

## 5.2 Normal Stress

The colour bar relates to the drawing. There it is possible to see min. and max. listed, those correlate to the min. (-5.6856) and max. (5.3283) values (both in MPa). The colours on the bar give the values of each colour region on the part. The distribution is based on the mesh created during analysis. The finer the mesh, the more accurate the distribution will be.

The normal stress is shown in the next Figure and with clicking the icon “min. & max” it is easier to see at which point the most stress occurs.



The Figure shows detailed that the most stress occurs at the top of the beam in the corners.



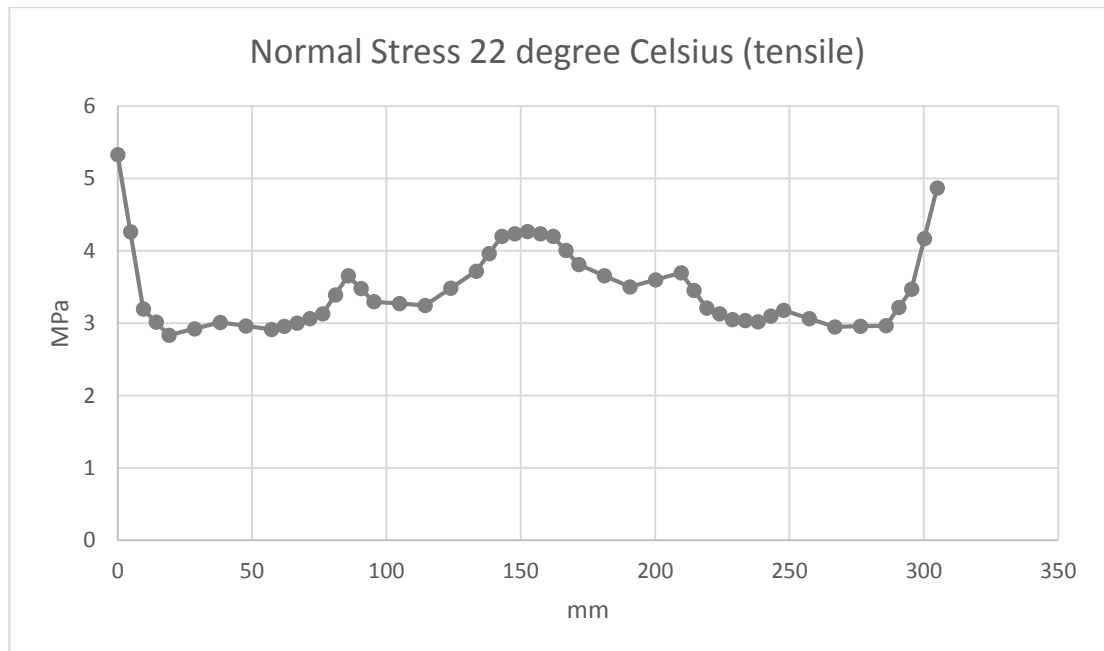
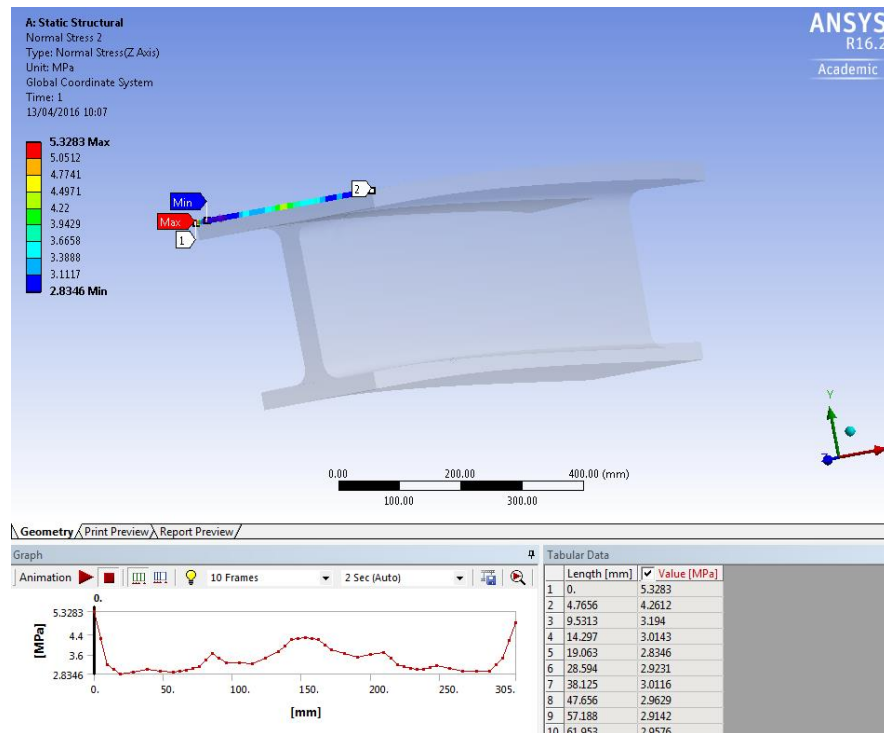


To get a better view of the two “min.” and “max” points go into the tree, choose the point “Construction Geometry” and insert “path”. In this case I did this two times to get two Figures which allows to see what happens at the both edges where the “min.” and “max” is. When inserting the “path” you have to set the following data:

Details of "Normal Stress 2"	
<b>Scope</b>	
Scoping Method	Path
Path	Path
Geometry	All Bodies
<b>Definition</b>	
Type	Normal Stress
Orientation	Z Axis
By	Time
<input type="checkbox"/> Display Time	Last
Coordinate System	Global Coordinate System
Calculate Time History	Yes
Suppressed	No
<b>Integration Point Results</b>	
Display Option	Averaged
Average Across Bodies	No
<b>Results</b>	
<input type="checkbox"/> Minimum	2.8346 MPa
<input type="checkbox"/> Maximum	5.3283 MPa
<b>Graph Controls</b>	

To calculate the average value and to get the diagram, go to the tree on the left side, go to the “Solution” right click on the desired item and export the data in a “Excel” file. For getting the average value, use the formula of “Excel” and calculate it.

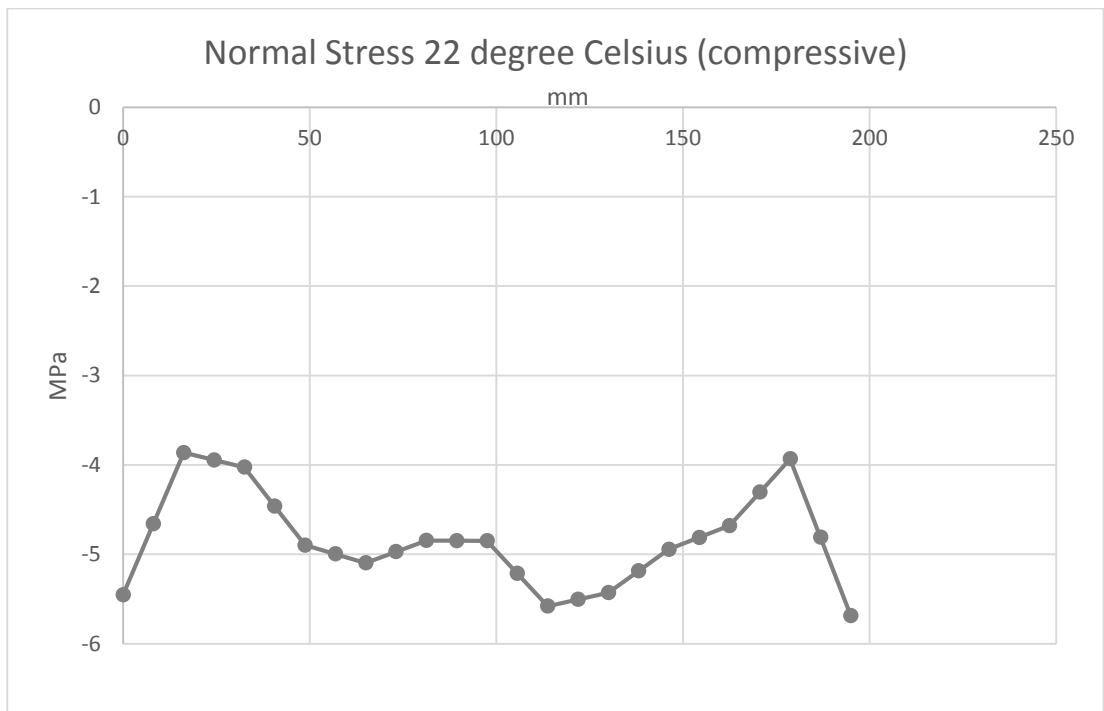
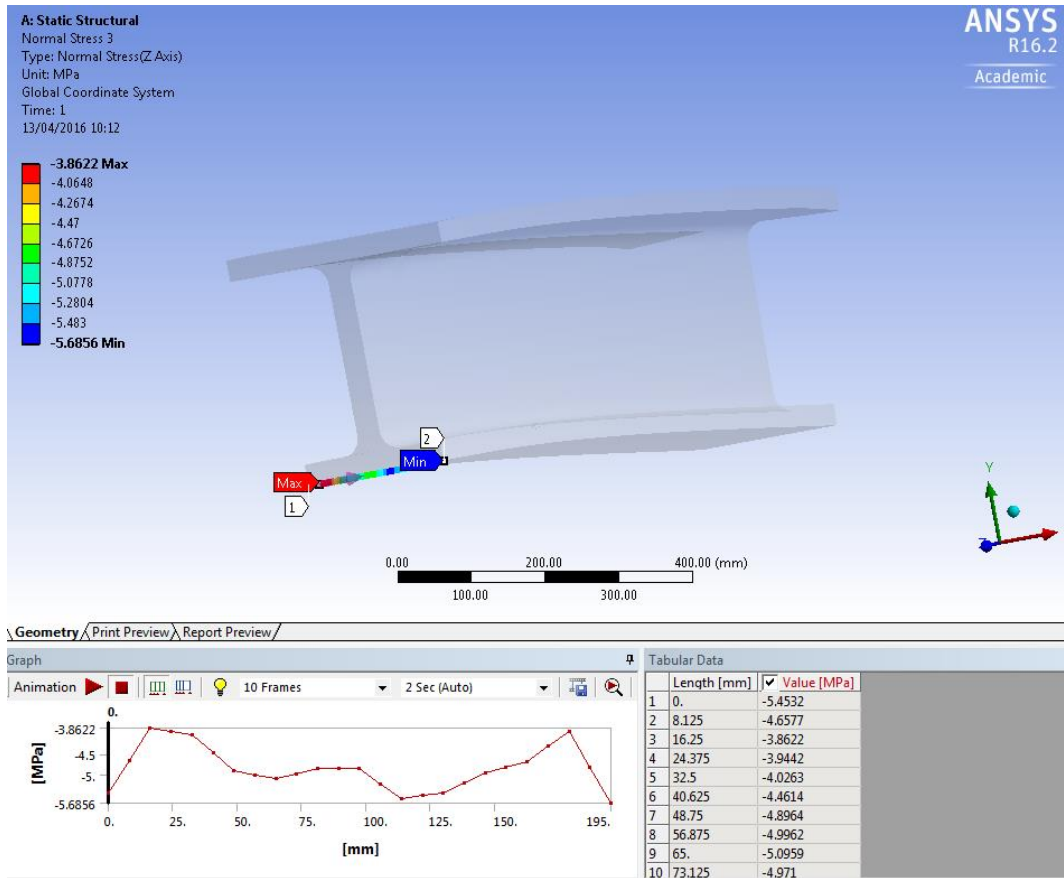
The Figure below shows the stress in the edge of the beam where the material is expanded.



Average value: 3.476197959 MPa

Beschreibung u. excl diagram von den zahlen

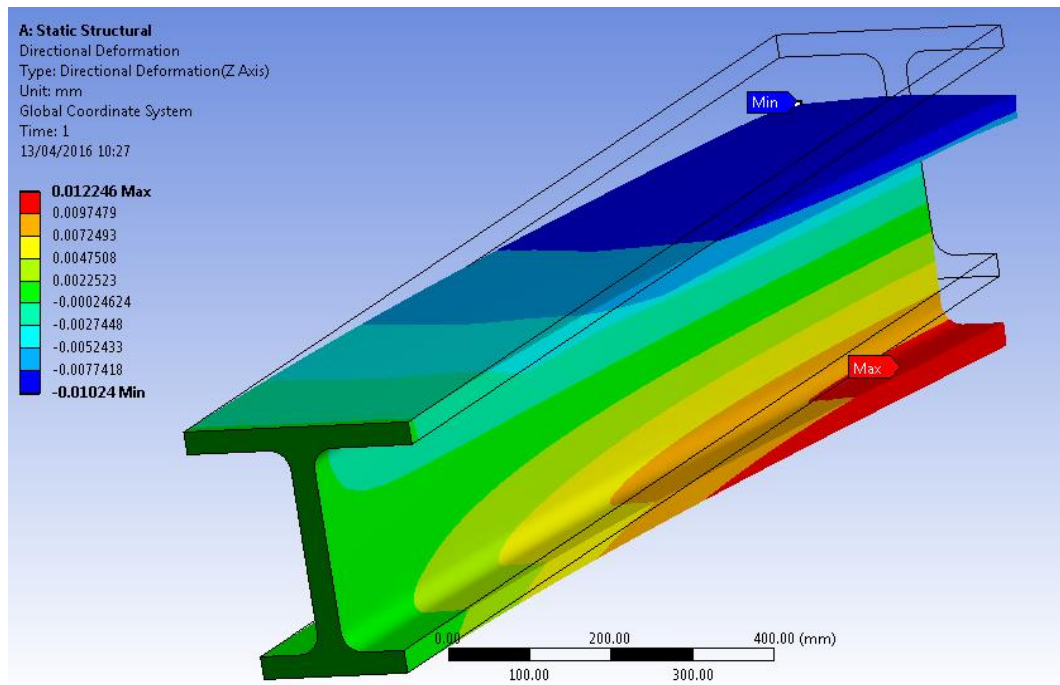
The Figure below shows what happens to the beam in the lower edge where the material is compressed.



Average Value: -4.83918 MPa

### 5.3 Maximum Deflection

Deflection is the degree to which a mechanical component is displaced under a force. The next graphic shows the total directional deformation about the Z-axis of the beam and the min. and the max. values in mm. The maximum deflection of the beam occurs in the lower part adverse to the fixed support and the minimum deflection on the top of the beam adverse to the fixed support.



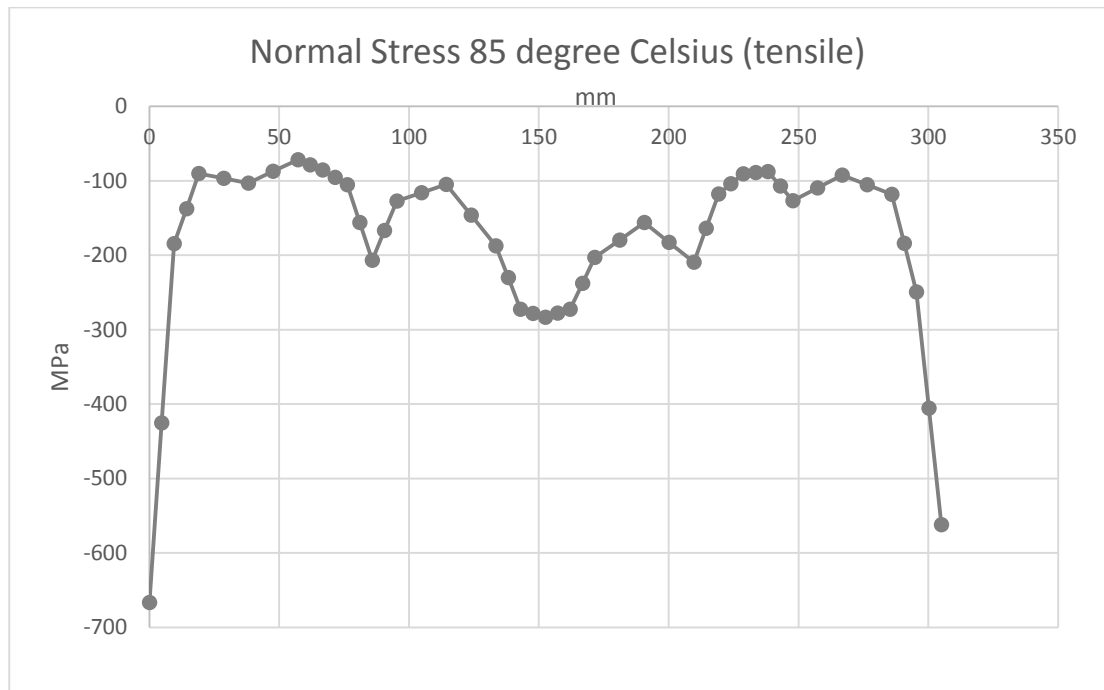
---

## 6. Question D

Due to a change in the operating environment, the beam is now exposed to an ambient temperature of 85 degree Celsius.

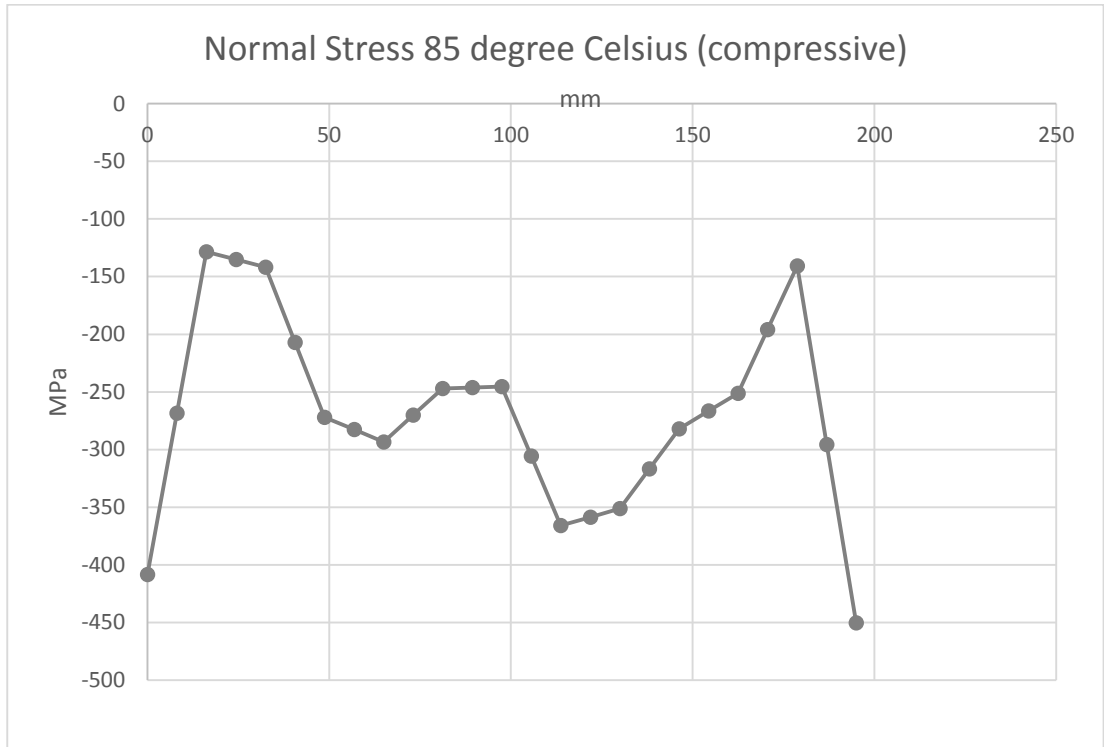
To set the boundary condition of 85 degree Celsius go to the left tree, right click on the “Static Structural” and insert “Thermal Condition” and change the temperature from 22 to 85.

The next Figure illustrated the same Normal Stress (tensile) as shown in Chapter 5.2 with the new temperature of 85 degree.



Average Value: -182.362 MPa

The next Figure illustrated the same Normal Stress (compressive) as shown in Chapter 5.2 with the new temperature of 85 degree.



Average Value: -269.0388 MPa