# ME2614 – ASSIGNMENT REPORT

# NUMERICAL AND ANALYTICAL STUDY OF A T-SECTION ALUMINIUM BEAM SUBJECT TO BENDING, TORSION AND SHEAR

Student ID: *1911261*

Beam Length: *426 mm*

## 1. FEA

### 1.1. FEA model description

- **Introduction**

A T sectioned beam plays a vital role as a load carrying element in bridges, buildings and construction generally. The flange, at the top of the T beam section, vital role is to withstand the compression stress generated from the applied load. While the web of the section withstands and deals with shear stress. The study of stresses and strains generated on T section beam is important and aid in designing a safe beam and avoiding failures.
In this study, the stress, strains and deformation generated in T section cantilever beam will be studied when a load at top left corner is applied. This load will make the beam subjected to bending, shear and torsion.

- **Geometry**

A 3d model of T section beam was created to apply a FEA study on it. Firstly, the geometry was created by a sketch with the dimensions of the T-section then extruded to the specified length. The T section has flange width of 100 mm, web width of 10 mm and overall height of 90 mm. The length of the beam is 426 mm.
A 2d surface was created on the top left corner of the t section to apply load in it. This was made to avoid the stress singularities generated when a point load is applied on a point. In case of the point load on a point geometry the area is considered as nearly zero so high numerical stress singularity is generated. The surface has a length and width of 10 mm.
The final geometry of the beam is shown in the following figure.

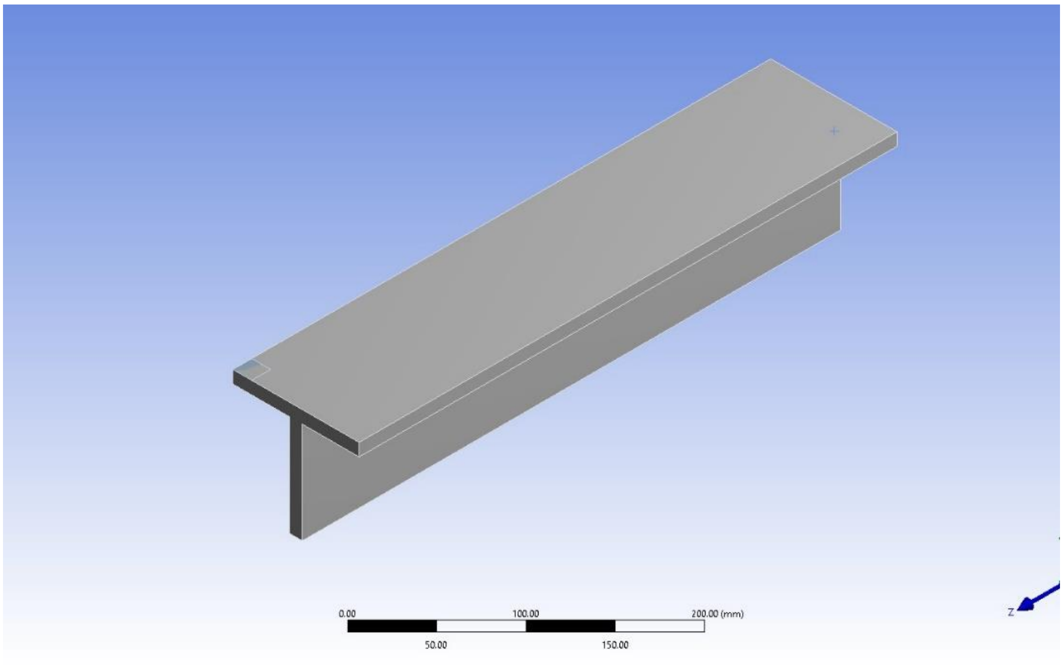


*Figure 1 T section beam geometry*

- **Material**

Aluminium alloy was used as the material of the beam. The model used in this simulation is linear isotropic elastic material model. That means that the relation between stress and strain is linear and the material properties is same in all directions. The material properties of Aluminium are summarized in the following table.

*Table 1 Material properties*

| Property | density | Youngs modulus | Poisson ratio |
|---|---|---|---|
| **Value** | 2770 Kg/m3 | 71 GPa | 0.33 |

- **Mesh**

A curvature and proximity-based mesh were applied to the entire domain with element size of 3 mm. A sweep method was applied to create a hexahedral mesh element with number of divisions of 30 between the start and end of the beam. The mesh has minimum orthogonal quality of 0.98 and average of 0.99. In addition to that, it has maximum skewness of 0.02 and average of 0.008. The mesh quality is high and acceptable for the simulation. The total number of nodes are 28033 while the total number of elements are 5425. The final fine refined mesh is shown in the following figure.

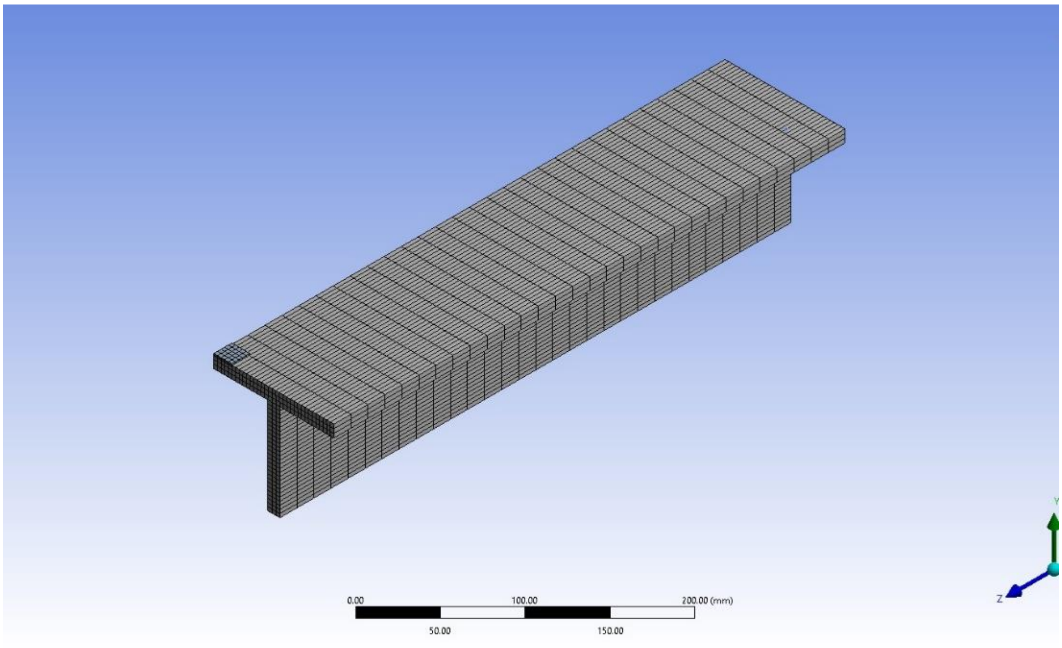


*Figure 2 fine mesh*

| Statistics | |
|---|---|
| Nodes | 28033 |
| Elements | 5425 |

*Figure 3 mesh statistics*

- **Setup and boundary conditions**

To model the behavior of the beam under the specified loads, boundary conditions must be added to the model. A fixed support was added to one end of the beam. This boundary condition constrains the motion and rotation of the face. This means that the face has zero degree of freedom. A force was applied to the surface which was created at top left corner of the beam. The force has value of 294 N. The following figure shows the boundary conditions schematic and location of their application. Finally, a bonded relation has been made between the surface and the T section beam.

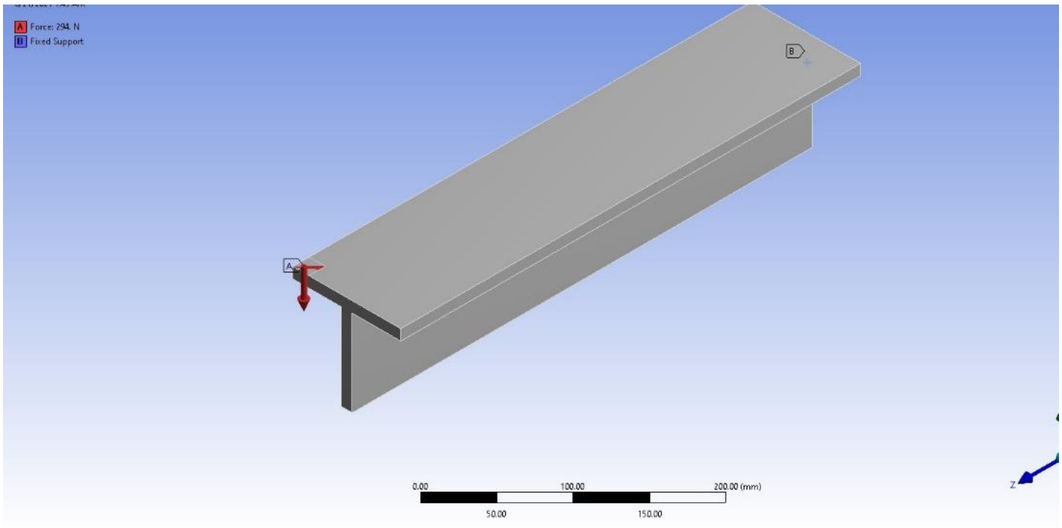


*Figure 4 boundary conditions*

## 1.2. FEA accuracy

To validate the results generated from the simulation, a mesh independence test must be done. In this test the mesh is refined gradually until a resulting testing parameter doesn't vary while further refining the mesh. In this simulation four different meshes were applied and monitoring the maximum von mises stress and deflection at 250 mm from end of the beam. The mesh convergence study was made by increasing the number of divisions of the sweep across the beam. The following table summarizes the mesh independence study.

*Table 2 mesh convergence results*

| Mesh | number of sweeps | number of elements | Maximum von mises stress | Deflection |
|---|---|---|---|---|
| **coarsest** | 5 | 925 | 11.896 | -0.0069584 |
| **coarse** | 10 | 1825 | 12.172 | -0.00693 |
| **medium** | 20 | 3625 | 12.85 | -0.0069641 |
| **fine** | 30 | 5425 | 13.501 | -0.0069643 |

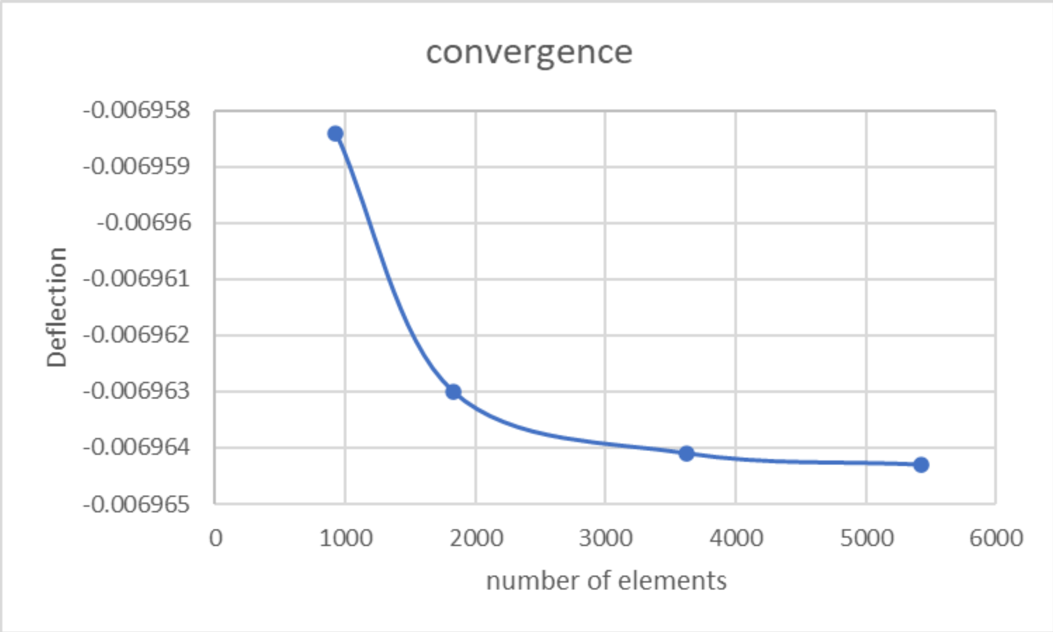


*Figure 5 mesh convergence graph*

As seen the results doesn't vary a lot and started to converge at medium mesh with number of elements of 3625. The remaining results were deduced from fine mesh for higher accuracy.

### 1.3. Description of FEA results, including those in terms of strains predicted in the direction of the gauges at the centre of the triaxial rosette (that would be used in an experiment, at the position given)

In this section, the FEA analysis results of the T section beam will be shown. Von mises stress distribution, total deformation, deflection and bending and shear stress will be plotted.

The following figure shows the total deformation plot on the entire beam domain. The deformation varies between the maximum value of 0.092 mm at top left corner and zero at fixed end.

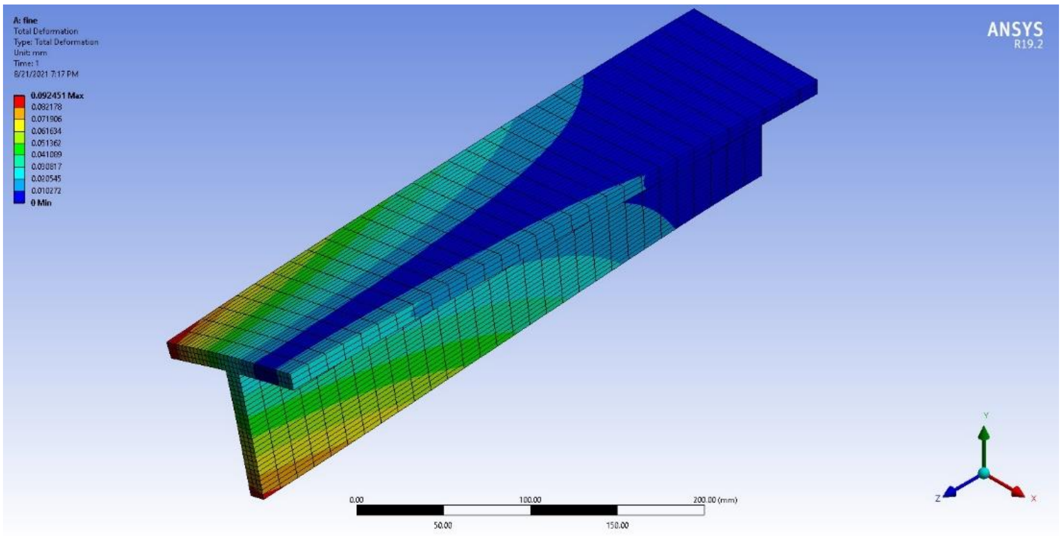


*Figure 6 total deformation*

In figure 7, the maximum deformation is 0.092 mm downward at the area of application of the load and 0.02 mm upward at the opposite side.

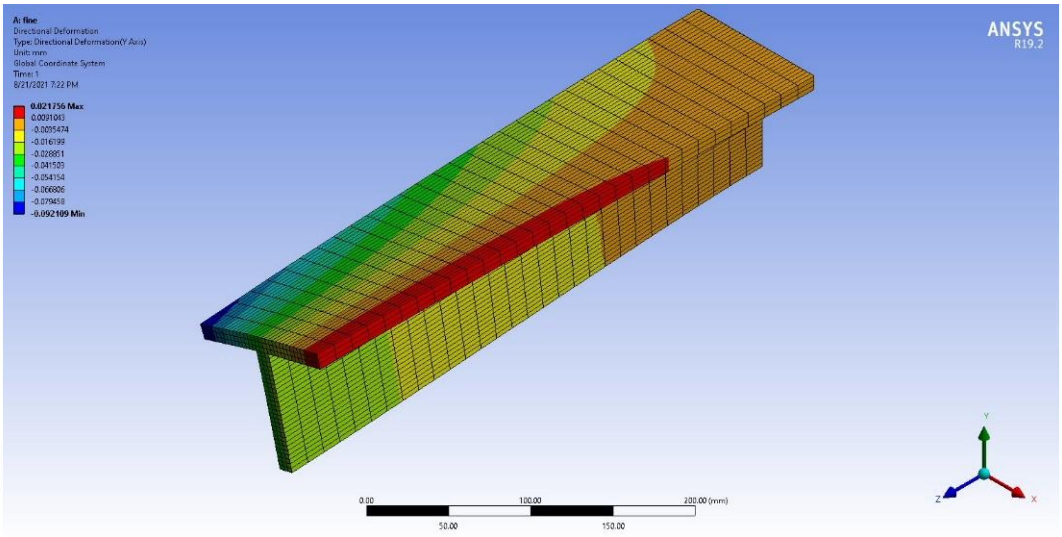


*Figure 7 directional deformation*

The normal bending stress, sigma z, were plotted on the beam. The maximum bending stress is 8.02 MPa in tension, while the maximum compression stress is 15.814 MPa. These values are shown in the following figure.

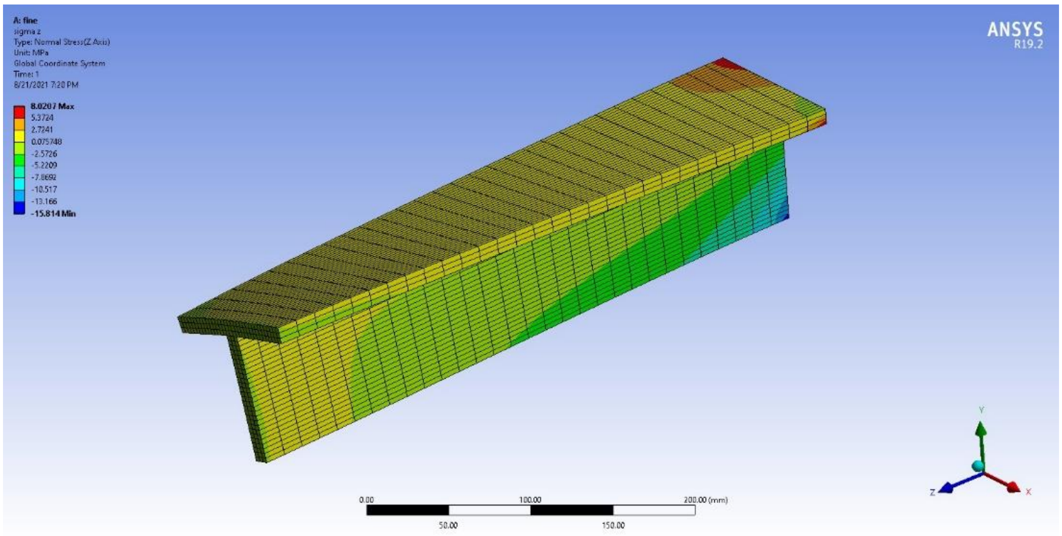


*Figure 8 sigma z stress*

The von mises stress were plotted on the beam, the maximum value occur near the fixed support and has maximum value of 13.501 MPa, while the minimum stress is 0.02 MPa. Von mises stress plot is shown below.

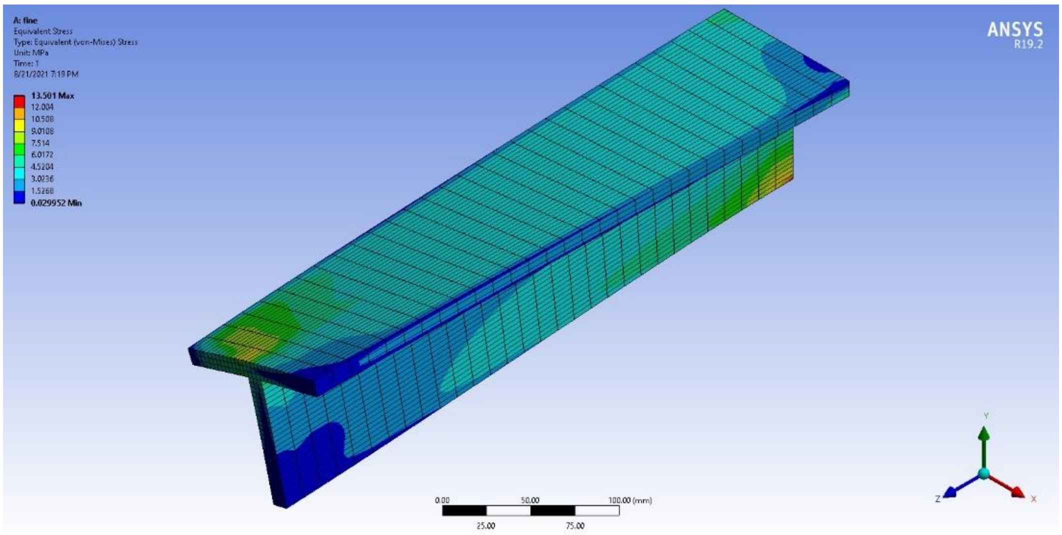


*Figure 9 von mises stress plot*

To measure the strain at the location of the strain gauges, 200 mm from end of the beam and 30 mm from the center, a coordinate system was created in the specified location to create a probe on it. Strain in x and z direction is measured to show the strain of the rosette strain gauge. The strain in x direction is -1.53E-06 mm/mm, while in z direction is 5.42E-06 mm/mm.

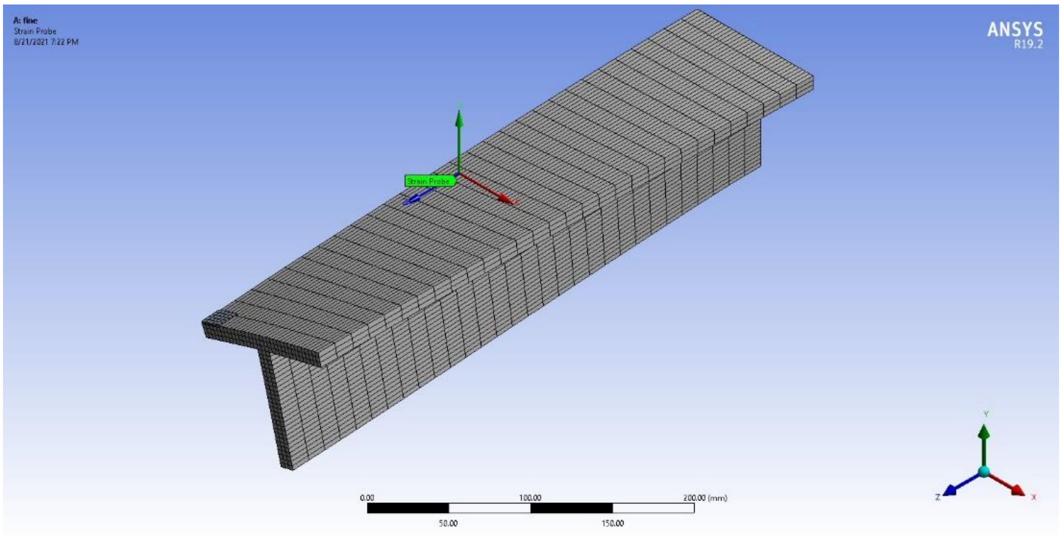


*Figure 10 probe location*

### 1.4. Qualitative analysis of FEA results

As the beam is clamped from one end, the deformation of the beam was expected to have a zero displacement at the fixed face. So, no deformation occurs at this part of the beam. This is due to the fixed support applied which constrain the six degrees of freedom of each element. In addition to that since the force is applied with eccentricity. It was expected that one side will deflect downward while the other part deflects upward. Both phenomena occur in the simulation results.

As the beam behave as a cantilever beam, it is expected that the maximum stress occurs at fixed end, this was found in the simulation.

Since the beam is subjected to torsion, the maximum stress due to compression will be higher than the stress resulted from tension and as the following figure show that maximum stress due to tension is 8.02 MPa, while the maximum stress due to compression is 15.814 MPa.

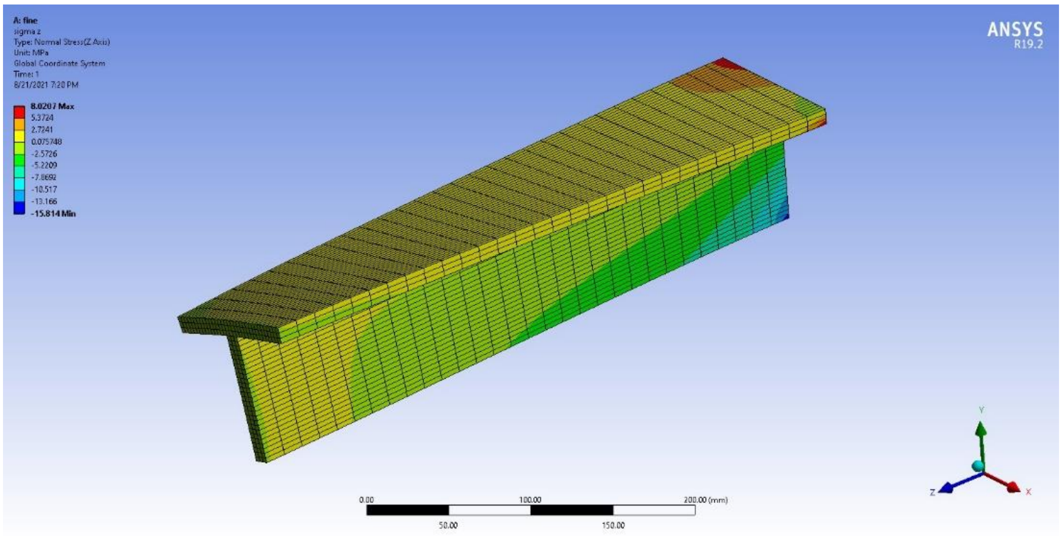


*Figure 11 sigma z stress*

As shown by results validation and different results can be plotted and determined we can run an FEA simulation with results comparable to experimental results.

## 2. Analytical calculation of strains in the triaxial rosette and comparison with FEA predictions

To calculate the strain on the cantilever T section beam, the stress needs to be calculated first, as the relation between stress and strain in linear elastic model material is given below.

$$\sigma = E\varepsilon$$

The centre of gravity of the T section beam is given by:

$$y = \frac{\sum Ay_i}{\sum A} = \frac{1000 * 85 + 800 * 40}{1800} = 65\ mm$$

The moment of inertia of the beam is as shown below

$$I = \sum (I_i + A_i d_i^2)$$

$$= \left(\frac{1}{12} * 100 * 10^3 + 1000 * 20^3\right) + \left(\frac{1}{12} * 10 * 80^3 + 800 * 10^3\right) = \ 1335000\ \mathrm{mm}^4$$

The moment is given by

$$M = P * L = 294 * 426 = 125244 \frac{N}{mm}$$

$$\sigma = \frac{MY}{I} = \frac{125244 * 25}{1335000} = 2.345\ MPa$$

Finally,

$$\varepsilon = \frac{\sigma}{E} = \frac{2.345}{71 * 10^3} = 3.3 * 10^{-6}$$

The above results are comparable to simulation results

## 3. References


- Das, Sandeep & Sarangi, Saroj Kumar. (2016). Static Analysis of Functionally Graded Composite Beams. IOP Conference Series: Materials Science and Engineering.
- DR.CH. S. NAGA PRASAD. Design and Analysis of Cantilever Beam
- Li, Yan Wei, and Lin Hai Zhang. “Finite Element Model Updating of T Beam Bridge Based on ANSYS.” Applied Mechanics and Materials
- Hibbler R.C. Engineering mechanics statics