

Applications of Finite Element Analysis

Portfolio Engineering Projects

EEC_6_701 2014/2015

Anthony Sophocli

3003254

School of Engineering

London South Bank University

Contents

1.0 - Problem Formulation

1.1 - Materials, Loads & restraints

2.0 - T-Beam (Original)

2.1 - I-Beam (Re-design)

3.0 - Mesh

3.1 - Coarse Mesh

3.2 - Fine Mesh

4.0 - Conclusion

1.0 - Problem formulation

The aim of this project involves the analysis and optimization of the design of structural elements of a crane which is cantilever loaded, this was done in ANSYS workbench using the static structural analysis system. The initial T-beam should be drawn with the initial dimensions given, and the beam should carry a static vertical load of 2kN without yielding. This beam will then be tested, and redesigned to optimize it. The mass, width, material, force and height will all be kept constant throughout this project whilst the thicknesses of the web and flanges will be variable. The stress analysis will be carried out in ANSYS and results for the vertical deflection of the beam and the direct stress due to bending on the top will be obtained for the T-beam.

The design of the beam will then be optimized in order to achieve better results. An I-beam profile will be used for the re-design of the beam, the thicknesses of the web and flanges will be altered in order keep the mass constant.

The aim of the re-design of the beam is to optimize it and minimize the total deformation and stress of the beam whilst either reducing or keeping the mass constant.

The mesh of the re-designed beam will be optimized in order to obtain more accurate results.

The re-designed beam will then be tested and the results will be analyzed in order to determine whether the beam has been optimized. The results of the I-beam should present a total deformation and equivalent stress lower than the original T-beam and with the mass kept the same, or lower.

1.1 – Materials, loads and restraints

The material that was chosen to be used was structural steel, which has tensile yield strength of 250MPa. This material will also be required to be used on the re-designed beam. Structural steel was chosen to be kept constant throughout this project in order to get a good comparison in the results.

The beam was loaded with 2kN of force on the top face (as seen in figure 1.0); the force was also kept constant throughout this project in order to produce accurate comparative results. A fixed support was added to one end face of the beam as shown in figure 1.1.

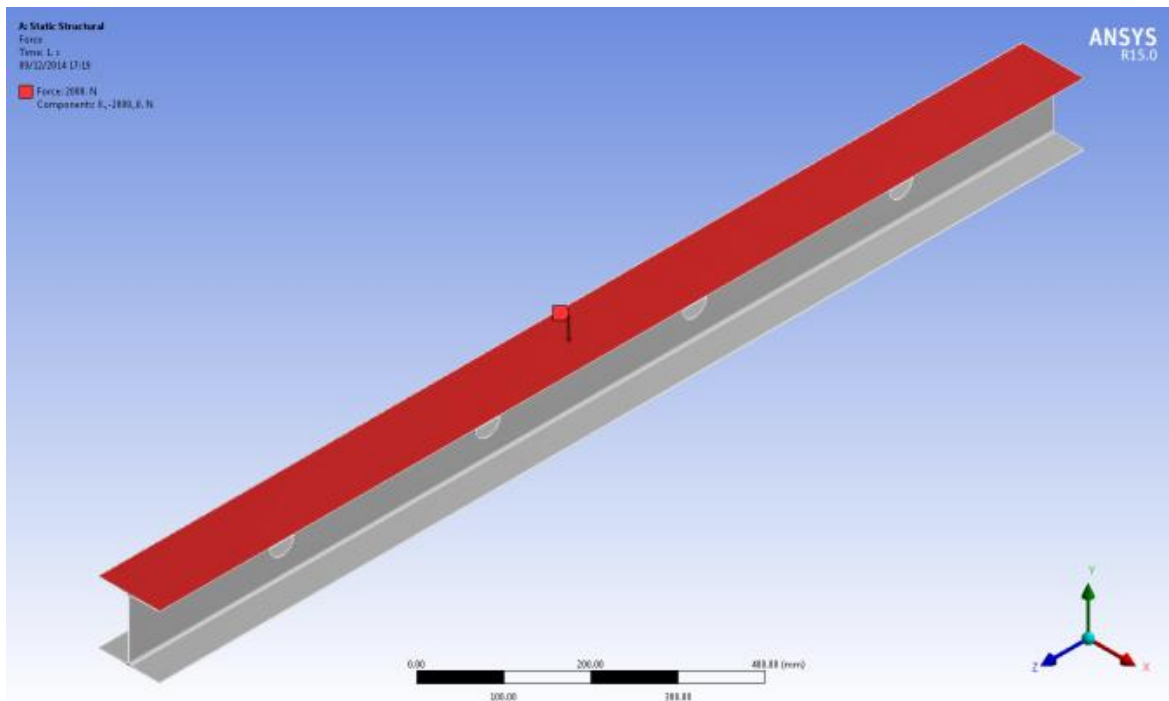


Figure 1.0 - Application of force on beam

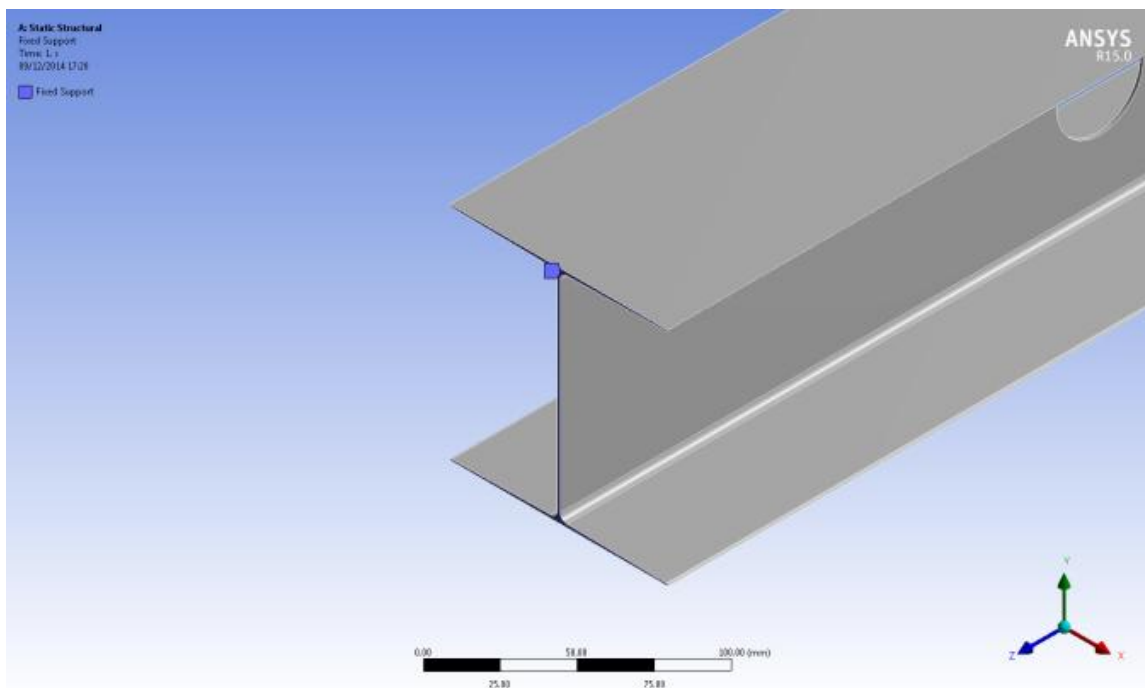


Figure 1.1 - Application of fixed support

Throughout this project there were many variables and constants, see the table below:

Constant	Variable
Mass (or less)	Profile
Width – 100mm	Web thickness
Height – 101.5mm	Holes-Diameters
Length – 1500mm	Distance between holes
Force – 2kN	Fillets/Chamfers
Material - Structural steel	Flange thickness

2.0 - T-Beam (Original)

The original beam was a T-Beam (See figure 1.2 & 1.3); the total deformation and equivalent stress of the T-Beam are shown below. The beams were modelled in the ANSYS modeller.



Figure 1.2 - T-Beam profile

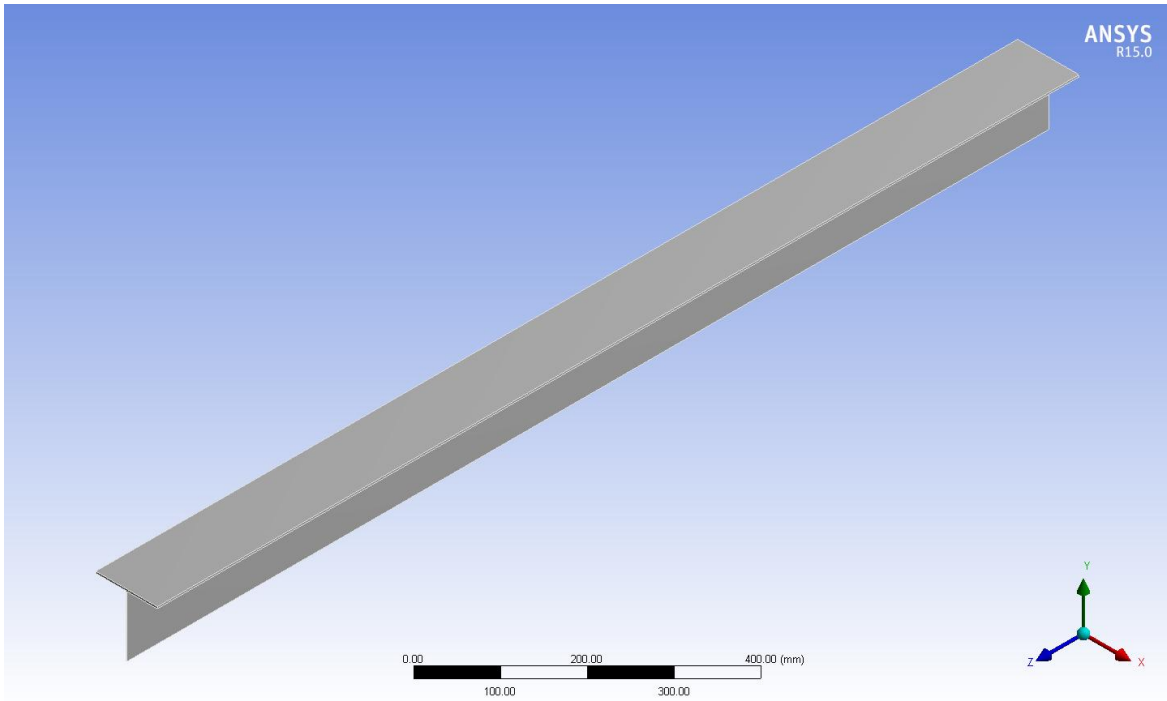


Figure 1.3 - T-Beam isometric view

The T-beam was then tested to produce results for the total deformation, the equivalent stress and the safety factor of the beam.

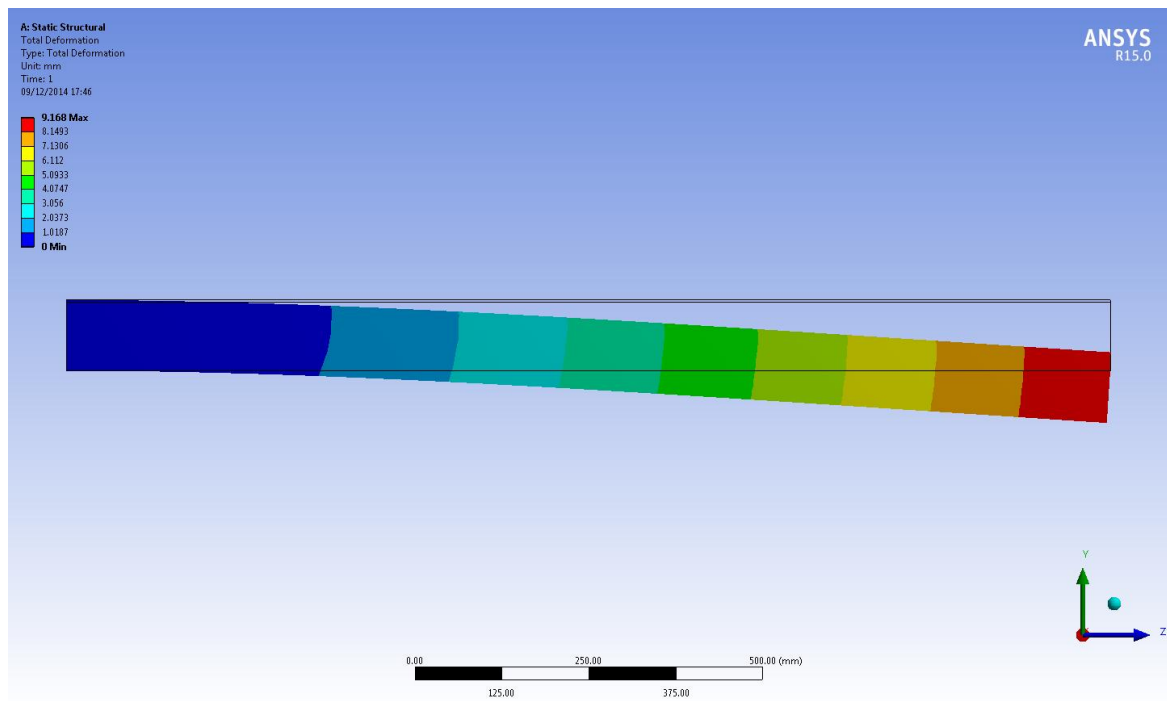


Figure 1.4 - T-Beam total Deformation

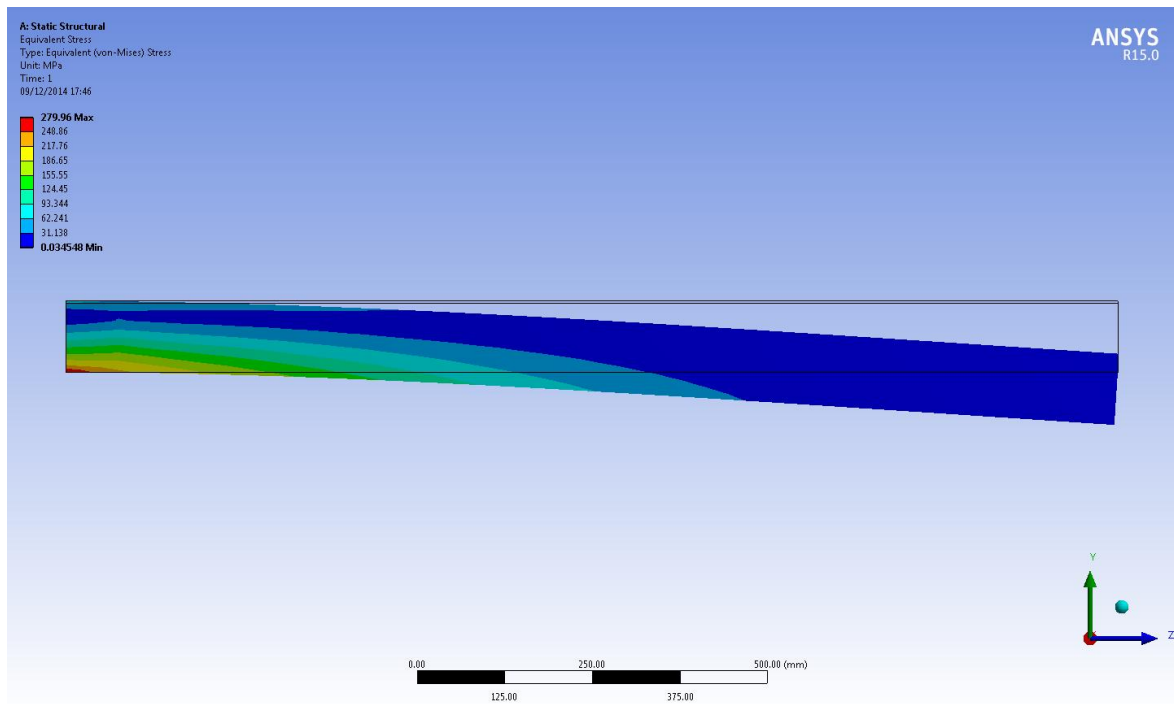


Figure 1.5 - T-Beam equivalent stress

Total deformation = 9.168mm

Equivalent stress (MAX) = 279.96MPa

Safety factor = 0.89298

Mass = 5.8522Kg

Figure 1.5 shows that there is high stress on the bottom edge of the fixed support; this will be considered in the redesign of the beam.

The data obtained from the results show that the maximum equivalent stress of the beam did not exceed the ultimate tensile strength of the material (460 MPa); however the beam was undergoing a large deformation. As a result of the equivalent stress being greater than the tensile yield strength of structural steel, the beam was permanently plastically deformed and would therefore fail in a real world environment.

Overall, the total deformation of the beam and the equivalent stress are very high for the T-Beam. The safety factor is also very low.

2.1 - I-Beam (Re-design)

An I-Beam profile was used for the re-design of the beam; this profile was chosen because it has a higher moment of inertia and is also good and dealing with shear due to its large web

area. (See figure 1.6 & 1.7) After viewing the pictures for the results of the T-Beam, it was noticed that there was high stress on the bottom of the fixed support, thus a bottom plate was added in order to dissipate the stress in that area.

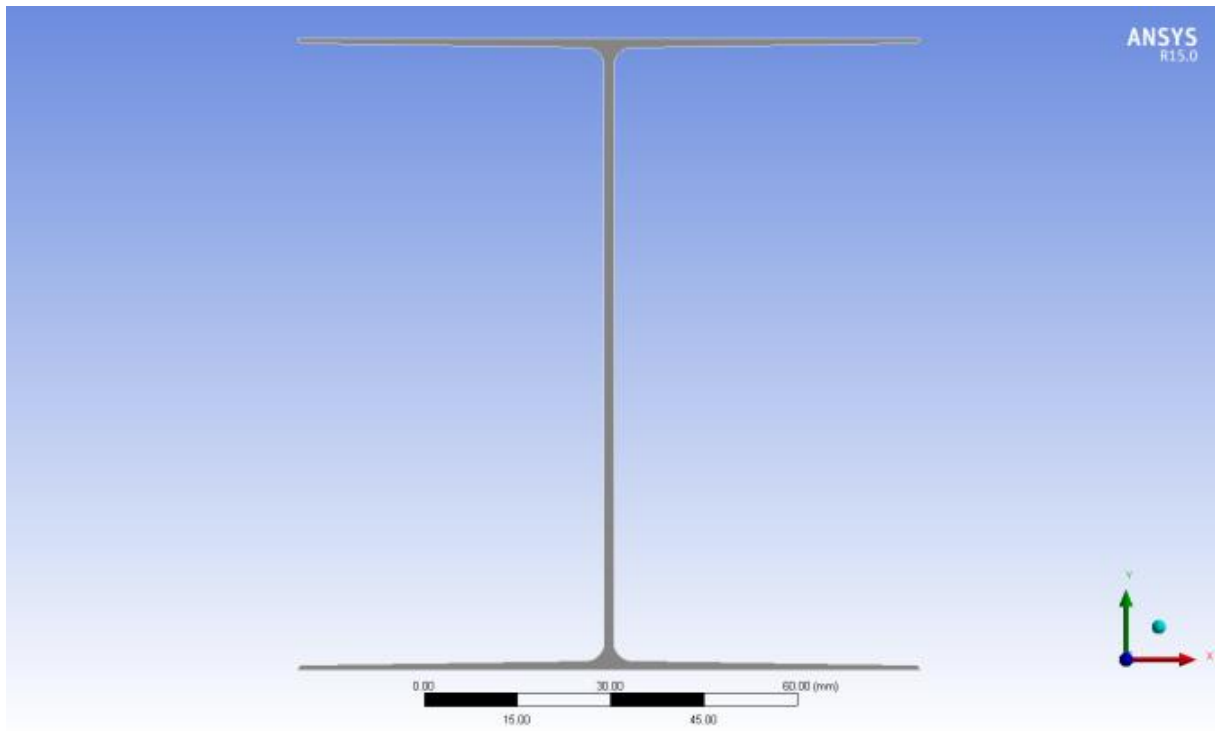


Figure 1.6 - I-Beam profile

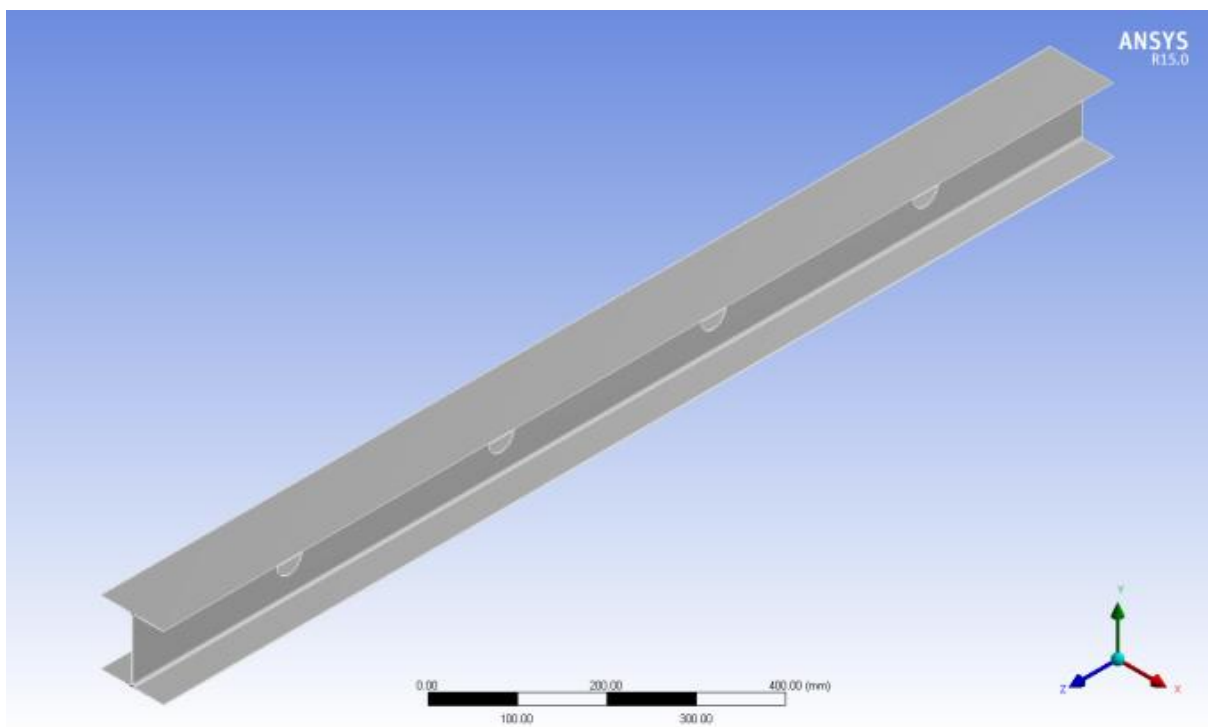


Figure 1.7 - I-Beam isometric view

The I-Beam profile had a thinner web than the T-Beam and also had chamfered flanges; these design implications were added in order to reduce the mass as much as possible. Fillets were added to the corners of the beam in order to relieve high stress points on corners. Holes were then added along the web of the beam in order to help bring the mass down.

A combination of using the I-Beam profile, the chamfered flanges and the incorporation of the holes helped to produce a stronger, lighter beam.

The results can be seen below:

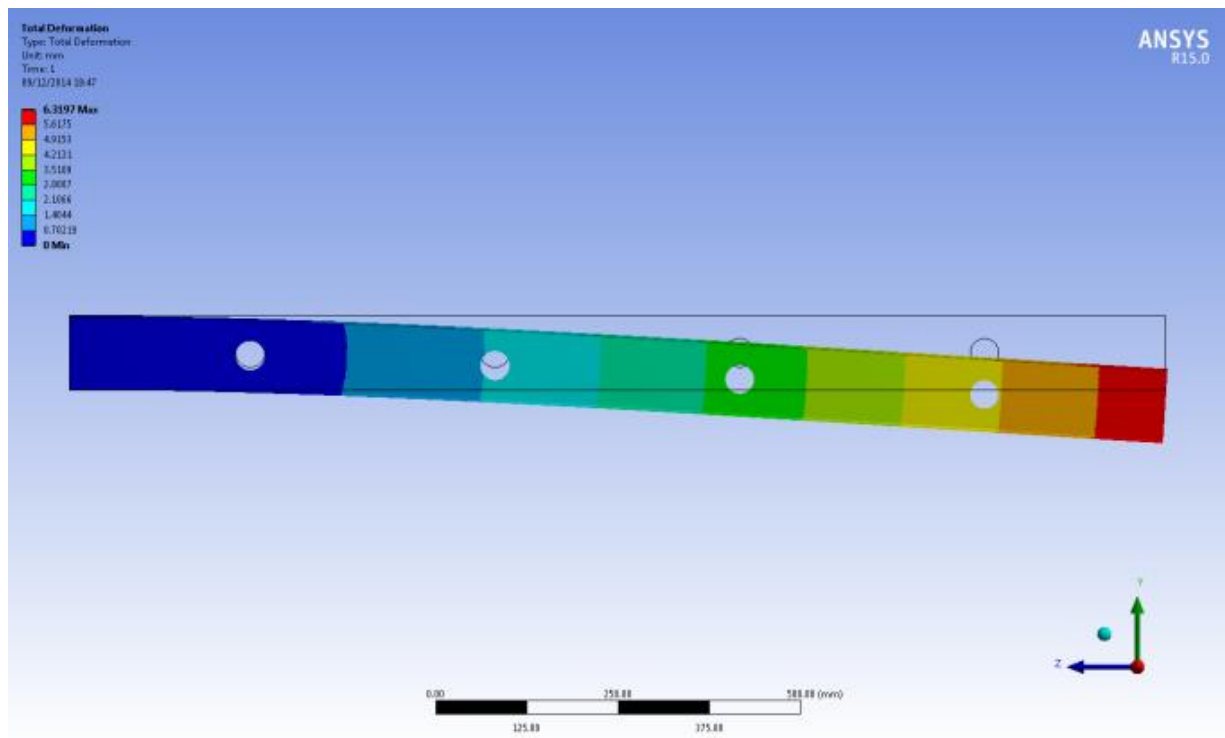


Figure 1.8 – I-Beam total Deformation

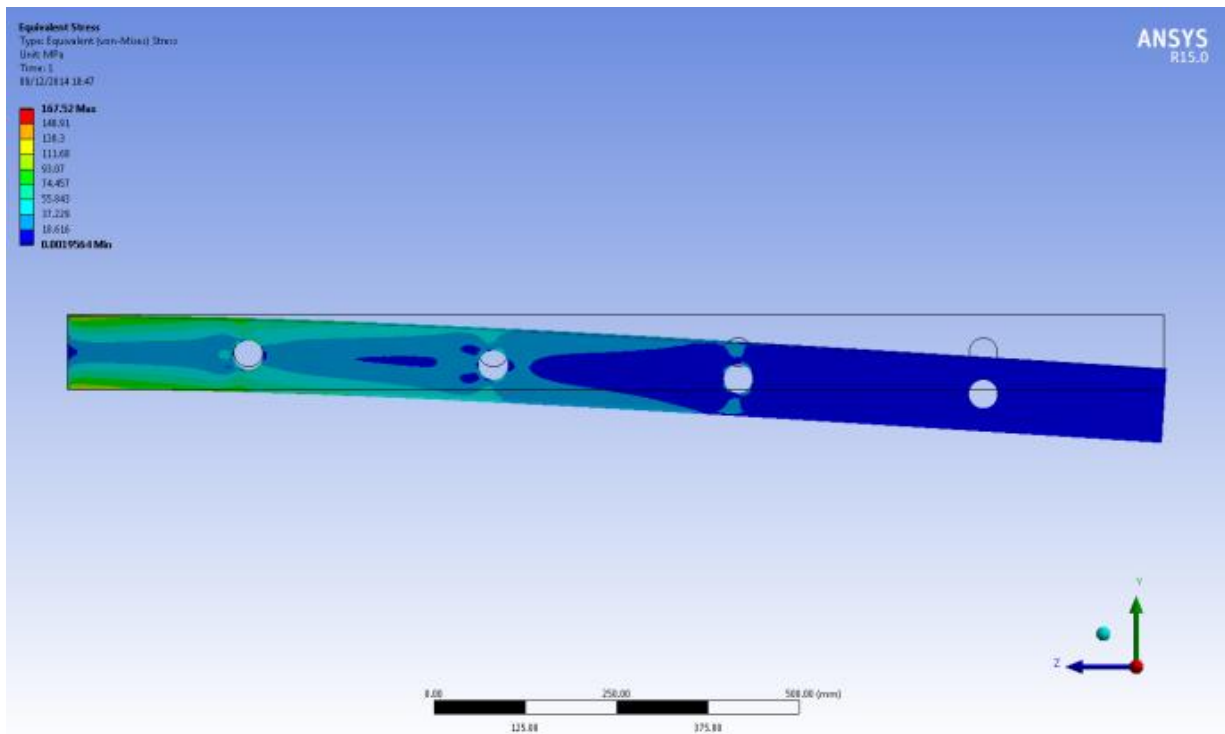


Figure 1.9 – I-Beam equivalent stress

Total deformation = 6.319mm

Equivalent stress (MAX) = 167.52MPa

Safety factor = 1.4923

Mass = 4.439Kg

Figure 1.8 shows a high deformation at the end of the beam, and figure 1.9 shows high stress on the top and bottom surfaces near the fixed support. Finer mesh will be generated in these areas in order to account for the high stress.

Goal driven optimization was run in order to optimize the design of the beam to achieve the best results, the parameters set are shown below, as well as the results from the optimization.

Parameters:

- Mass
- Equivalent stress-von mises (MAX)
- Hole diameters (x4)
- Spacing between holes
- Space between holes and edge of beam
- Total deformation

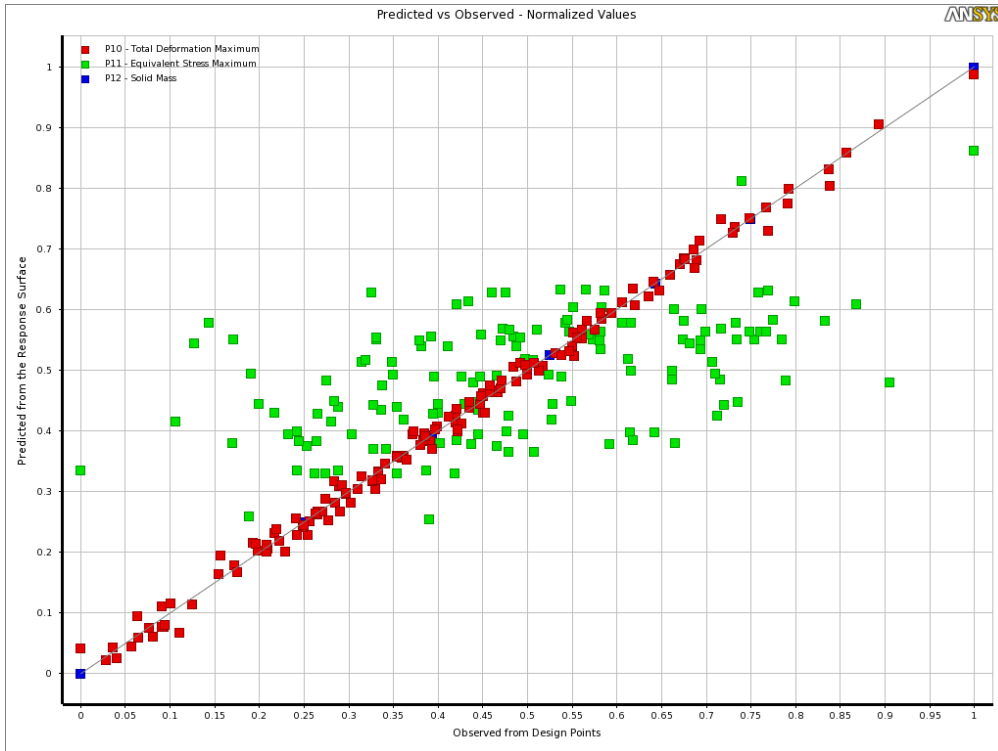


Figure 1.4-Goodness of fit table

The graph above shows ‘the goodness of fit’. It is a graph showing the results from my design points against predicted results from the response surface. Analysing this graph helped define the area and parameters in which I should be looking to implement in the I-Beam.

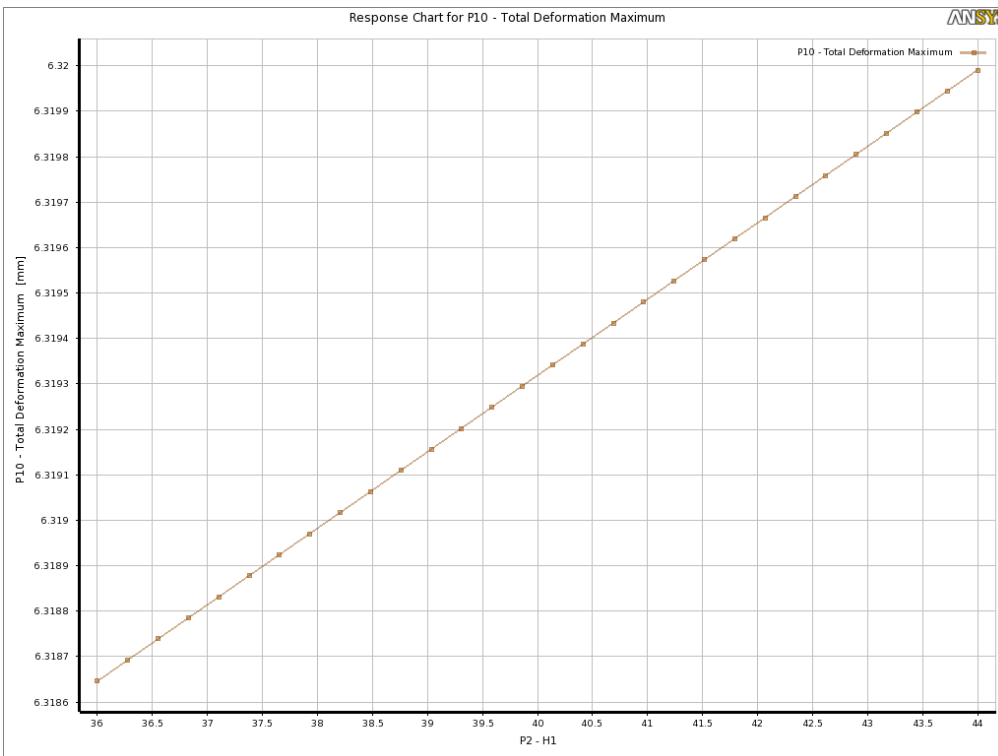


Figure 1.5-Response chart

Figure 1.5 shows a response chart, this produce results for total deformation, equivalent stress or the mass against a chosen parametre for each design point. The example shown, shows the total defromation against parametre 1 (Diammetre of hole 1), it shows 50 design points along the line and represents how the total deformation would change for each different diammetre of hole 1.

This chart was used to help view the results for particular dimensions for the different parametres on this project and to give a good idea of how this would effect particular calculations of this project. The use of the response chart helped give a good direction for the parametres of this project and for the overall design of the beam.

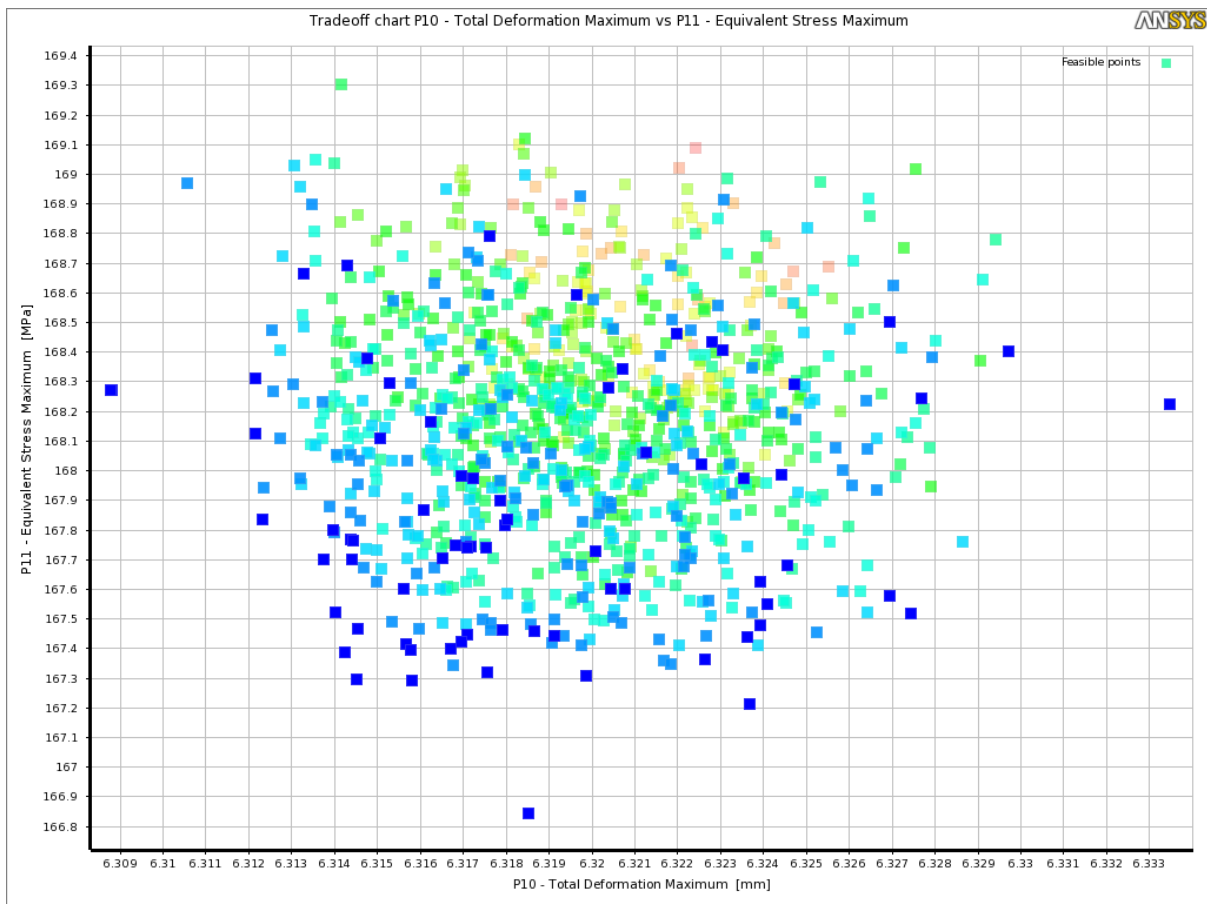


Figure 1.6-Trade-off chart

Figure 1.6 shows a trade-off chart, which again like the goodness of fit table helps to define the parameters in which the re-designed beam should incorporate. This chart shows the total deformation against the maximum equivalent stress. The turquoise squares show the feasible points and show results that would be optimum for the I-Beam.

It was the use of these two graphs that helped identify the area which should be looked at for the design of the beam. The areas in both graphs in the bottom left hand corners denote where the beam should be based, low equivalent stress, deformation and mass.

To begin, parameters were set, and design points were created from these in the design of experiments tool. After results were given through the design of experiments, the response surface was used to generate another design point; this gave the second design point to be considered. To finish, the goal driven optimization was used to define the last design point, this produce three candidates that the programme thought were a best match for the criteria, one (Candidate B) was chosen and copied as the last design point.

All three design points from the different tools were then compared against each other and the best one (the design point from goal driven optimization) was implemented into the final design, as seen in the table below.

Data	T-Beam	I-Beam (Design of experiments)	I-Beam (Response surface optimization)	* I-Beam (Goal driven optimization)
Mass (Kg)	5.8522	4.4251	4.4375	4.439
Volume, X(mm³)	7.455 10e+005	5.6371e+005	5.6529e+005	5.6547e+005
Total Deformation (mm)	9.168	6.329	6.316	6.319
Equivalent Stress (MPa)- MAX	279.96	168.07	168.77	167.52
Equivalent Stress (MPa)- MIN	3.4548 e-002	3.0198e-003	2.9246e-003	1.9564e-003
Safety Factor	0.89298	1.4989	1.4956	1.4923
Hole Diameters (mm)	-	44 44 44 44	40 40 40 40	39.255 38.739 40 40
Space Between Holes (mm)	-	230.36 301.5 345.84 368.5 222.75	247.5 335 335 335 335	247.5 335 335 335 247.5

*Chosen beam

After running the goal driven optimization, the results were compared, as shown above. It was apparent that all-round, the results from the I-Beam that were optimized using the goal driven optimization were the best and thus was taken forward to the current design in order to achieve the best results.

3.0 - Mesh

3.1 - Coarse mesh:

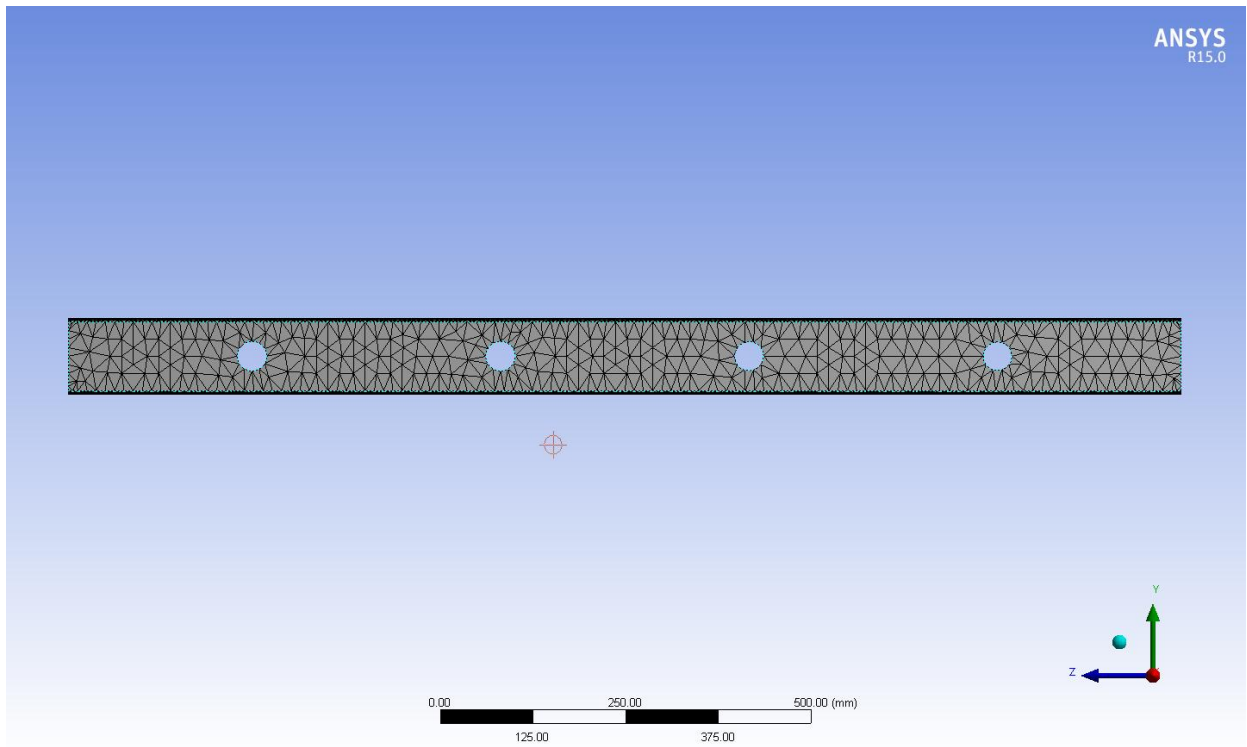


Figure 1.7-Meshing

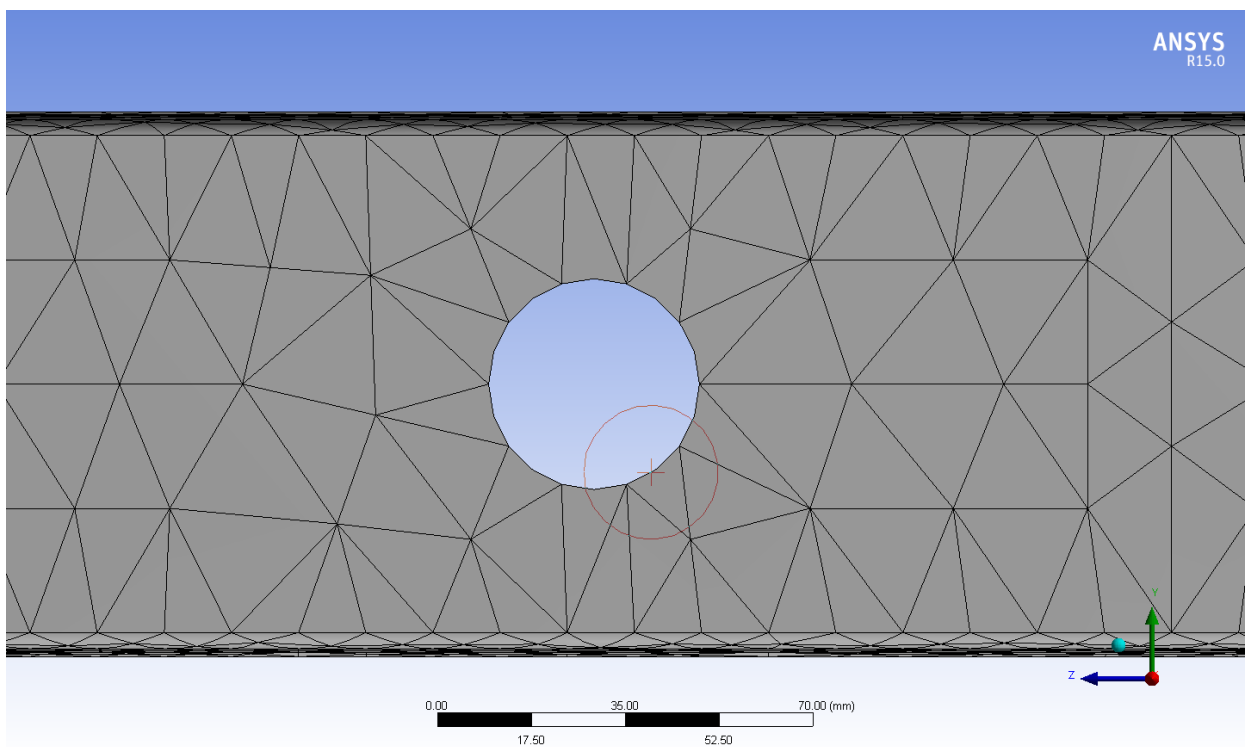


Figure 1.8-Meshing around hole

Originally the default mesh was coarse; the element size was at default (18mm), the relevance centre was set at coarse. The original mesh used for this beam was the default settings.

Total deformation: 6.2793mm

Equivalent stress: 120.50MPa

The more coarse mesh produced better results in terms of deformation and stress, but the results are less accurate. Therefore a higher quality mesh is needed in order to obtain more accurate results. As can be seen from the element quality plot graph for coarse mesh on the I-Beam, the mesh was of a poor quality resulting in inaccurate results.

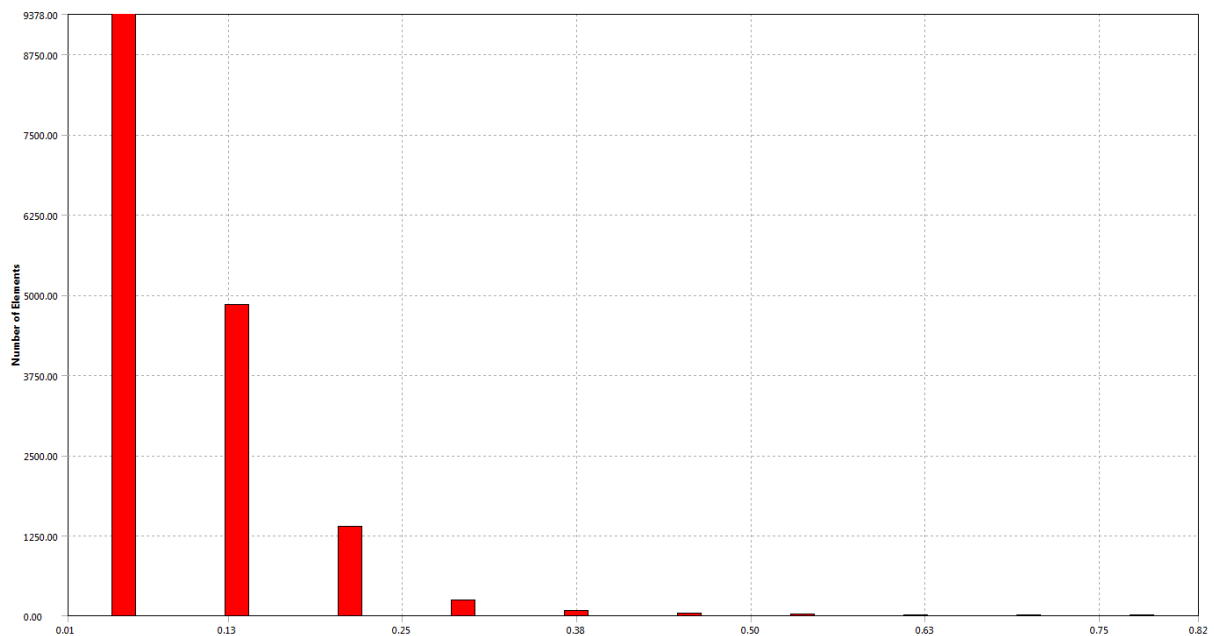


Figure 1.9- Element Quality Plot Graph for Coarse Mesh on I-Beam Re-design

3.2 - Fine mesh:

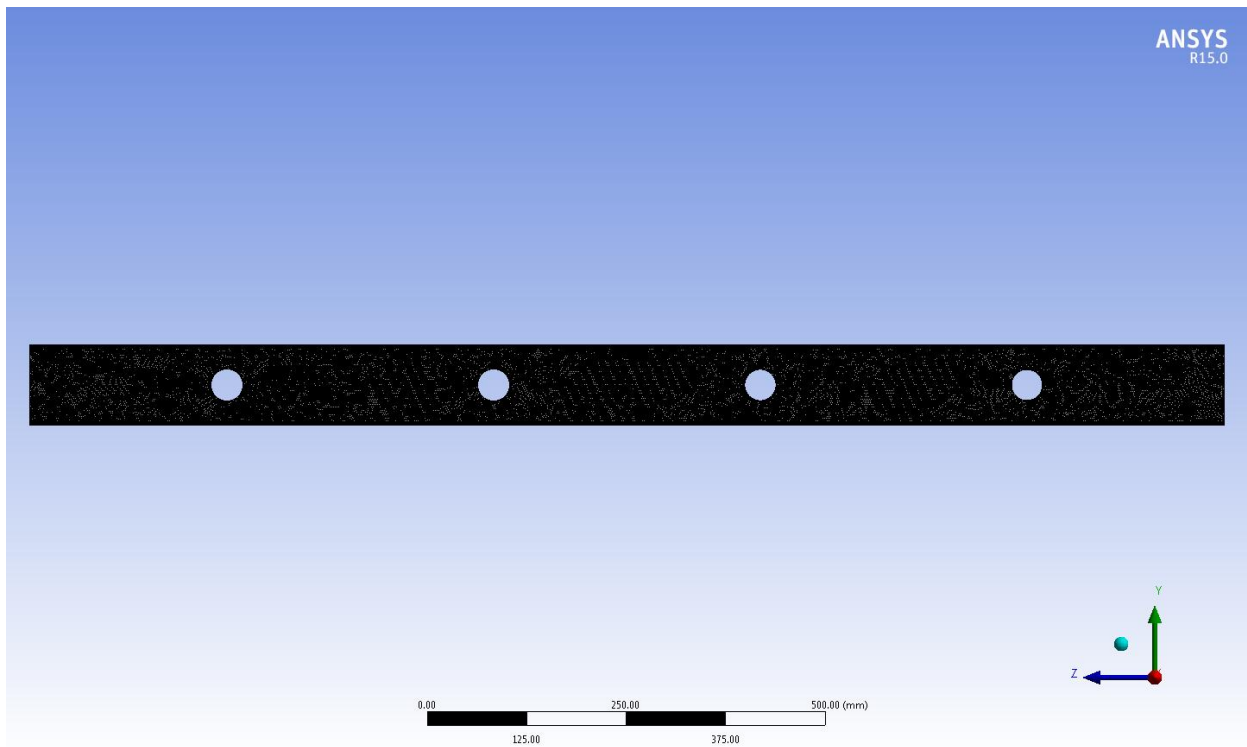


Figure 2.0-Meshing

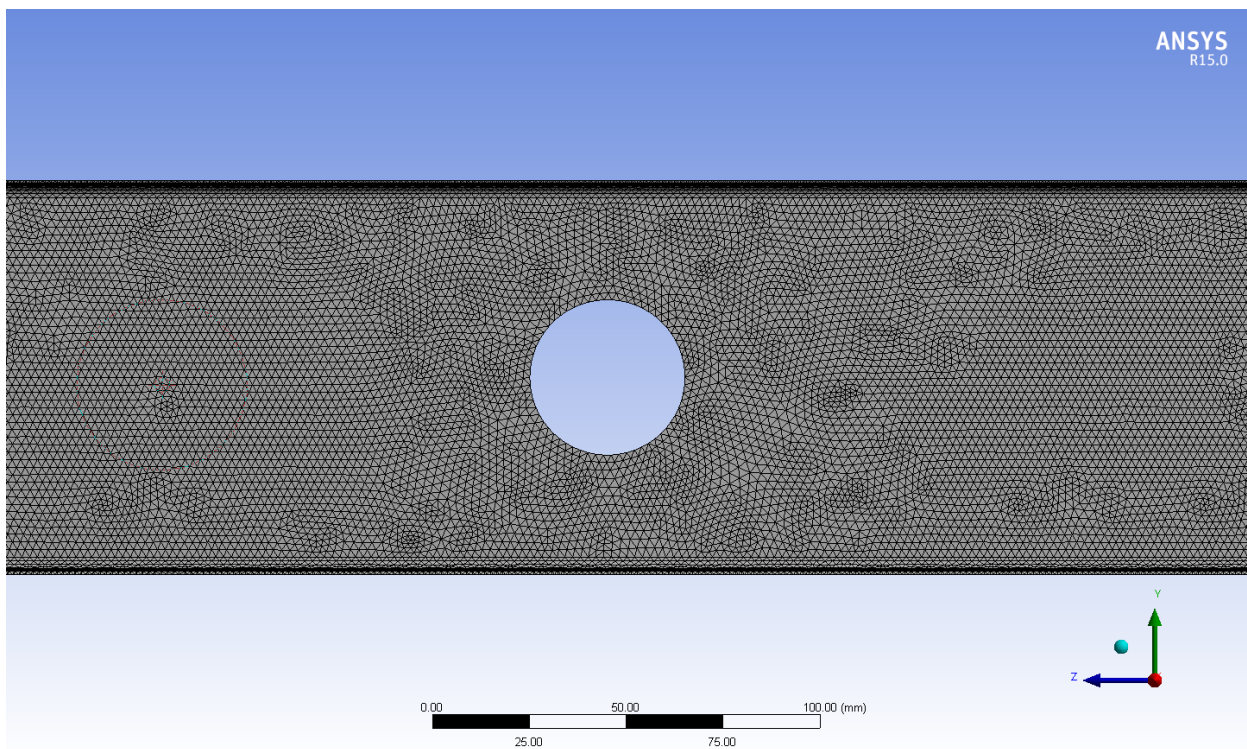


Figure 2.1-Meshing around hole

Optimal mesh density was used for the I-Beam; the aim was to get the mesh as fine as possible in order to obtain more accurate results. This was done by reducing the size of the

elements, the element size was reduced from the default setting (18mm) to 4mm. Smaller element sizes were tried but the ANSYS license would not allow it. The relevance centre was changed to fine which also refined the mesh density.

Total deformation: 6.3197mm

Equivalent stress: 167.52MPa

The higher quality mesh produced worse results than the coarse mesh in terms of deformation and stress, although the results are more accurate. These results are more accurate and thus more reliable than the coarser mesh.

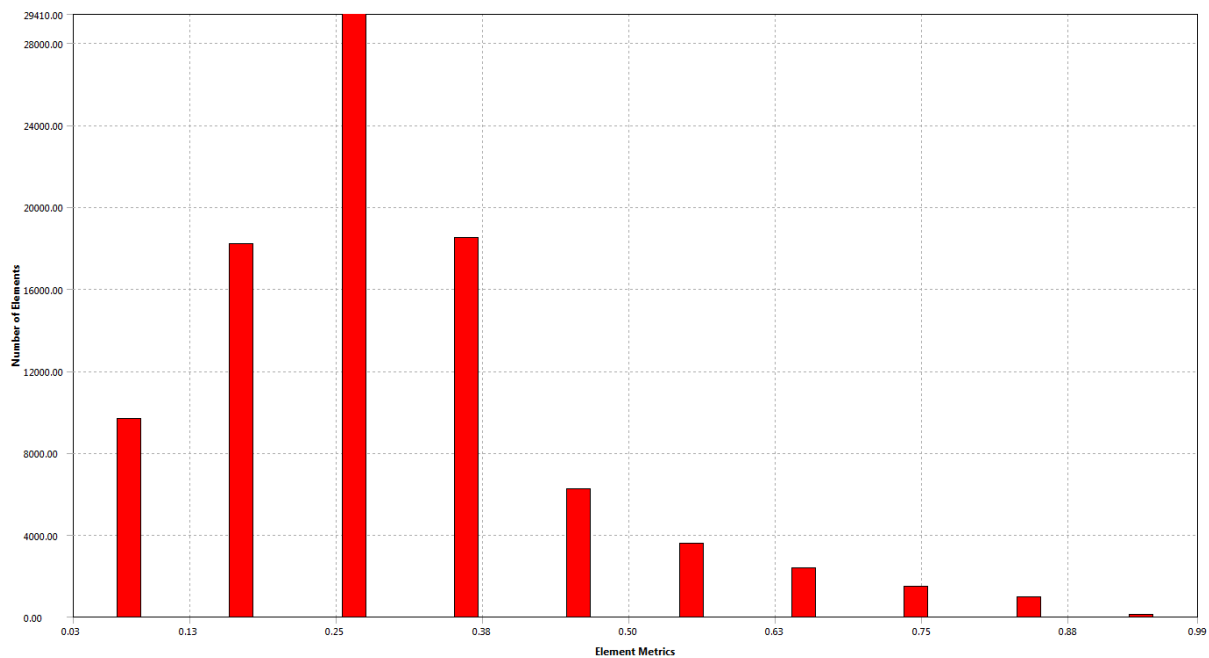


Figure 2.2 - Element Quality Plot Graph for Fine Mesh on I-Beam Re-design

As can be seen from figure 2.2, it is evident that the changes in the mesh settings in order to obtain a fine mesh, produced a high quality mesh in comparison to the coarse mesh, which can be told by the graph which shows more elements situated in the centre of the graph.

Patch conforming method was added to the body of the I-Beam and the method was changed to 'tetrahedrons', this was done because this helps give a higher quality mesh with more complicated geometry. This proved to give a higher quality mesh, especially a finer mesh in the radii and around the holes as seen in figure 2.1.

Total deformation without patch conforming method: 6.3195mm

Equivalent stress without patch conforming method: 182.61MPa

Total Deformation with patch conforming method: 6.3197mm

Equivalent stress with patch conforming method: 168.77MPa

As seen from the results above, using the patch conforming method produced a finer mesh, and thus more accurate results.

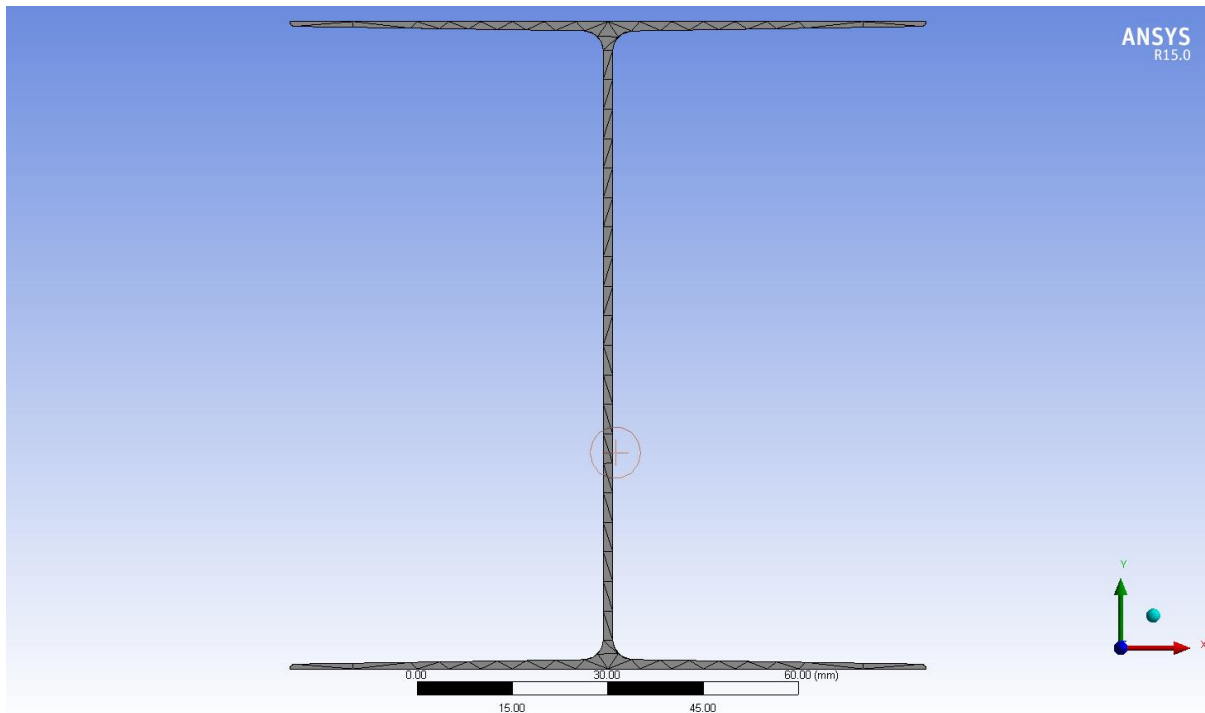


Figure 2.2-Face sizing on fixed support face

As shown in figure 2.2, face sizing was added to the face of the I-Beam, on the fixed support face. The element size was reduced to 10mm. Face sizing was added to create a finer mesh on the face of the fixed support, this proved to give more accurate results.

Total deformation without face sizing: 6.3195mm

Equivalent stress without face sizing: 182.61MPa

Total Deformation with face sizing: 6.3197mm

Equivalent stress with face sizing: 168.77MPa

As seen from the results above, using the face sizing also helped produce more accurate results.

Refinement was also used to help produce a higher quality mesh. Refinement was placed on the web of the beam and helped refine the mesh on the web and especially around the holes. Refinement is used primarily for sensitive areas, thus why it was placed on the web of

the beam that would experience large stress when under load. The difference that refinement made on the results can be seen below:

Total deformation without refinement: 6.3195mm

Equivalent stress without refinement: 182.61MPa

Total Deformation with refinement: 6.3197mm

Equivalent stress with refinement: 194.39MPa

Below is a table showing a comparison of the results from the coarse mesh to the fine mesh.

	Coarse mesh	Fine mesh
Total Deformation (mm)	6.2793	6.3197
Equivalent Stress (MPa)- MAX	120.50	194.39

Figure 2.3-Table showing comparison between coarse and fine mesh

As can be seen from the table above, the fine mesh gives much more accurate results than the coarse mesh and thus was implemented on the final I-Beam in order to obtain more accurate results. Further refinement for the mesh was attempted but due to license restrictions, would not let it mesh any further.

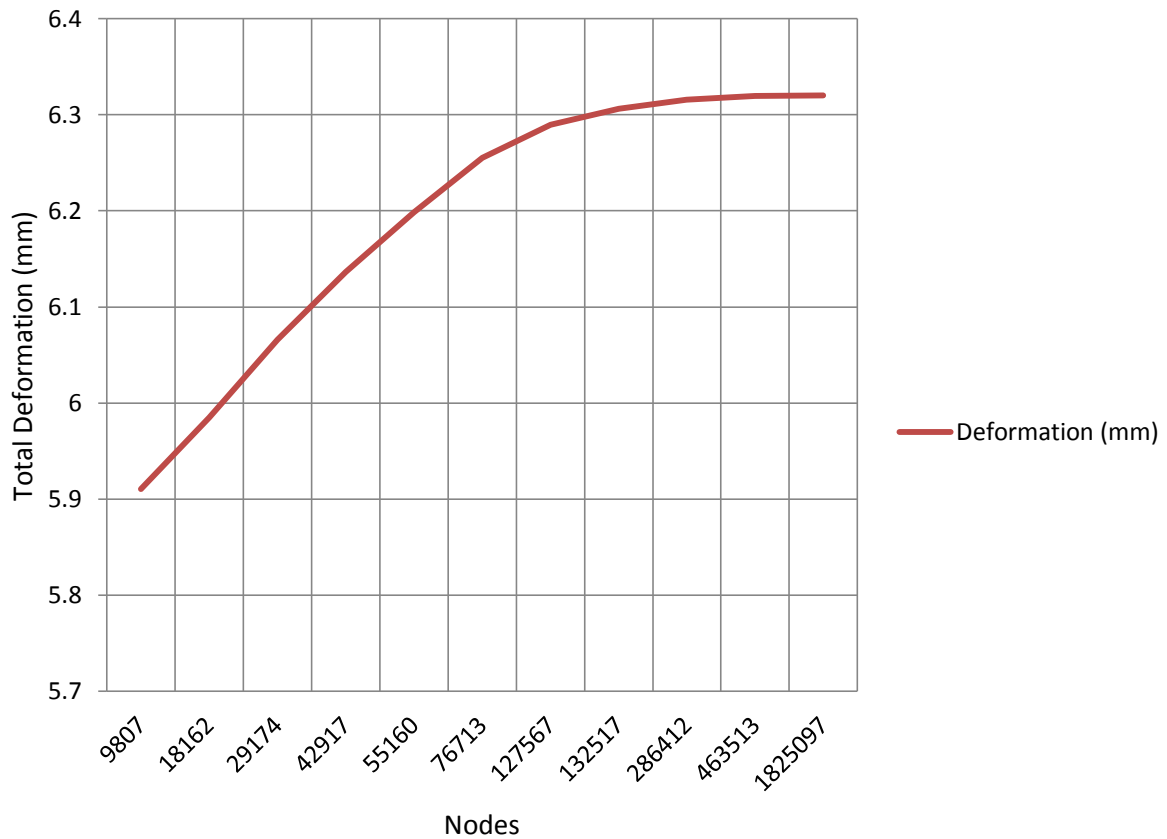


Figure 2.4-Graph checking mesh validity

Figure 2.4 shows a graph of the total deformation of the beam, against the total number of nodes in the mesh of the beam. The graph was used to conduct a convergence study to validate the results for the beam calculation. The results from ANSYS were put into a data sheet in excel and the displacement for the beam was calculated. The graph shows that an increase in the number of nodes for the mesh of the beam, results in a larger displacement, this shows that towards the higher end of the nodes, more accurate results were obtained. The mesh validity was approved through checking this graph, where the nodes were 463513 (the current mesh nodes) on the optimal mesh for the I-Beam, the deformation, checked on ANSYS shows 6.3197mm, which when checked against the graph, shows that the value is correct.

4.0 - Conclusion

In conclusion, the results from the I-Beam compared to the original T-Beam were better, the total deformation and equivalent stress were less than the T-beam and the weight was reduced by 1.42Kg. Therefore the beam was optimized. Goal driven optimization was run on the beam to produce the best results with the parameters set, and the final result showed the best results. The total deformation was reduced by 2.849mm and the equivalent stress by 85.57MPa. The Mass of the beam was also reduced by 1.413Kg and the safety factor was increased by 0.599. The table below shows the comparative results.

	T-beam (Original)	I-beam (Re-design)
Mass (Kg)	5.822	4.439
Total deformation (mm)	9.168	6.319
Equivalent stress (MPa)	279.96	194.39
Safety factor	0.89298	1.4923

It was also found that although the coarse mesh for the I-Beam gave better results in terms of deformation and stress, they were less accurate; therefore a higher quality mesh was used in order to obtain more accurate results. (See Figure 2.2)

Using the various different mesh settings it was possible to produce a beam which had an equivalent stress below the structural steels tensile yield strength of 250MPa. This optimized design ensured that the beam would no longer plastically deform under 2KN of force and would therefore be suitable in real world environments.

In order to optimize the beam further, the holes would be removed from the structure and the web and flange thicknesses would be increased, although this would increase the overall mass of the beam it would make it a lot stronger. The mesh would also be optimized by making it finer, depending on license restrictions. Even finer mesh would be added in particular to the places that experience high stress levels i.e. the top and bottom faces of the beam.