• Abstract

The purpose of this project is to analyze the feasibility of adding holes to the web of universal beam (I-beam), which could reduce weights and be applied to construct the bridge of overhead cranes, by using finite element method. Overhead cranes, or bridge cranes, are widely used in manufacturing plants, building constructions and warehouses because of easily building, stability and little ground space costing. Additionally, the unique parallel runways design provides strong support for the entire bridge, which makes the crane can easily handle heavy loads. As shown in Figure 1, the new bridge design could effectively reduce weights and costs for manufacturing. However, the holes might reduce the carrying capacity of the crane, which could result in safety problems. This project will analyze the feasibility of the new bridge structure by computing yield moment and yield load using finite element methods, and comparing the result with the conventional bridge design.

• Problem Description

The physical model of this problem could be simplified as shown in Figure 2. When the crane is loading, the displacement of two endpoints of the bridge should be zero both in horizontal and vertical direction. Furthermore, the load could be seen as a force applied on the beam. The design of the beam being analyzed in this project is shown in Figure 3. The structural parameters of the universal beam are refer to the BS 4-1:2005[1] and Steel Designers' Manual[2].

• Objectives

The objective of this project is to compare the difference of yield moment and yield load (determined by using von-Mises stress) between the two designs, according to the results from ANSYS, to determine the feasibility of the new bridge design. Also, the strong form and weak form formulation of beam structure will be studied and some solid mechanics analysis will be implemented.

• Background

1. Solid Mechanics

According to the Euler-Bernoulli beam bending theory[3] in solid mechanics, as shown in Figure 4, the maximum tensile stress and maximum compressive stress is on the uppermost edge and lower edge of the beam, respectively. The stress is zero at neutral axis. So adding holes on the web of I-beam is possible for reducing weights, without resulting in the increase of stress significantly.

In addition, as shown in Figure 5, the dangerous section for failure caused by tensile stress and shear stress is different with respect to the different positions the load is applied. In other words, in this project, the position of hook will determine which type of dangerous section should be considered. Figure 5(a) shows the dangerous section where failure will be induced by tensile stress, where the moment is 1.25 times the load force F. Figure 5(b) shows the dangerous section where failure will be induced by shear stress, where the shear force is approximately equals to the load force F. As a result, in this project, the tensile yield stress, measured by von-Mises criterion, is the dominant factor causing failure when comparing to shear stress. Therefore, I will choose the load and moment at which the maximum von-Mises stress reaches the yield level, as yield load and yield moment of this structure. While shear stress is only used to confirm that the yield is induced by the tensile stress but not the shear stress, as a result checking method.

2. FEM Formulation[4]

(a) The strong form of the beam equation is:

$$EI\frac{d^4u_y}{dx^4} - p = 0$$

as shown in Figure 6, u_y denotes the vertical displacement of the midline of the beam and p denotes the vertical loading.

(b) In this project, as shown in Figure 5, fixed supports is applied at the two endpoints of the beam , so the Essential Boundary Conditions are:

$$u_y = 0 \text{ on } \Gamma_u$$

$$\theta = \theta_0 \text{ on } \Gamma_\theta$$

where θ denotes the rotation of the midline.

(c) And the Natural Boundary Conditions are:

$$m = 1.25F \text{ on } \Gamma_m$$

when examining the possible failure induced by moment, as shown in Figure 5(a).

$$s = F \text{ on } \Gamma_s$$

when examining the possible failure induced by shear force, as shown in figure 5(b). Where m denotes the moment applied and s denotes the shear force, respectively.

(d) When obtaining the weak form:

$$\int_\Omega w\left(\frac{d^2m}{dx^2} - p\right)dx = 0$$

then integration by parts and apply $w = 0$ at essential boundaries:

$$\int_\Omega \frac{d^2w}{dx^2} m dx = \int_\Omega wpdx + \left(\frac{dw}{dx}m\right)\bigg|_{\Gamma_m} + (ws)\big|_{\Gamma_s} \qquad \forall\, w \in V$$

$$\int_\Omega \frac{d^2w}{dx^2} EI \frac{d^2u_y}{dx^2} dx = \int_\Omega wpdx + \left(\frac{dw}{dx}m\right)\bigg|_{\Gamma_m} + (ws)\big|_{\Gamma_s} \qquad \forall\, w \in V$$

this is the weak form for beam elements.

(e) When doing FEM formulation, we denote the displacement matrix as:

$$\overline{d^e} = [u_{y1}, \theta_1, u_{y2}, \theta_2]^{\mathbf{T}}$$

and denote the nodal forces as:

$$\overline{f^e} = [f_{y1}, m_1, f_{y2}, m_2]^{\mathbf{T}}$$

We denote the matrix of shape functions as $\mathbf{N}^e$, and denote $\frac{d^2\mathbf{N}^e}{dx^2}$ as $\mathbf{B}^e$.

So the stiffness matrix is:

$$\mathbf{K}^e = \int_{\Omega^e} EI\mathbf{B}^{e\mathbf{T}}\mathbf{B}^e dx$$

and the external forces matrix is:

$$\mathbf{f}^e = \int_{\Omega^e} \mathbf{N}^{e\mathbf{T}} p dx + (\mathbf{N}^{e\mathbf{T}} s)\big|_{\Gamma_s} + \left(\frac{d\mathbf{N}^{e\mathbf{T}}}{dx} m \right)\Bigg|_{\Gamma_m}$$

where $p = 0$ in this project.

• Approach

1. Assumptions

First of all, this problem is linearity and static. Also, I assume that the beam is originally straight and slender, and any taper is slight. And the material is isotropic, linear elastic, and homogeneous across any cross section. In addition, because of using solid mechanics theory in the analysis part, I also assume that only small deflections are considered and inertia should be neglected.

Secondly, because the strong form used in FEM formulation is derived from Euler-Bernoulli beam equation, following assumptions has been made[5]:

(a) The beam is initially straight with a cross section that is constant throughout the beam length.

(b) The beam has an axis of symmetry in the plane of bending.

(c) The proportions of the beam are such that it would fail by bending rather than by crushing, wrinkling or sideways buckling.

(d) Cross-sections of the beam remain plane during bending.

2. Geometry and parameter set

Both the 3D geometry model of conventional beam and beam with holes was developed in ANSYS as shown in Figure 7. The structural parameters of the universal beam are refer to the BS 4-1:2005[1] and Steel Designers' Manual[2]. The material used is the default structural steel provided by ANSYS and the materials properties obtained from 1998 ASME BPV Code, Section 8, Div 2, Table 5-110.1. The detailed materials properties and geometry parameters are outlined in Table 1. The detailed beam design is shown in Figure 3.

3. Element types and convergence studies

In this project, I studied the convergence when using Q4, Q8, T3 and T6 elements and the results are shown in Table 2. These types of mesh are generated by “Hex Dominant” method in ANSYS and element size are constrained by using “Body Sizing Control” feature in ANSYS. Element size as 100 mm, 50 mm, 25 mm and 12.5 mm has been studied and the change rate is calculated by the relative difference of von-Mises stress:

$$e = \frac{|\sigma - \sigma'|}{\sigma} \times 100\%$$

The result shows that Q8 element has the fastest convergence rate and Q8 is the only element type whose relative difference converges to 2% relative difference at the 12.5 mm element size level. So I choose Q8 for meshing and the final mesh is shown in Figure 7.

• Results and Discussion

As aforementioned, Q8 element will be selected for meshing because of the good performance. For boundary condition, for the sake of constraining the displacement at A and B, as shown in Figure 5, I add “fixed support” at the two end faces of the beam. The load force is applied at the middle of the beam and I use the “Direct Optimization” feature in ANSYS to obtain the maximum load at which resulting in the yield stress. At last, I applied the load obtained in the former step at a position near one end face, as shown in Figure 5(b), to calculate the maximum shear stress and maximum von-Mises stress in the structure and verify that the yield is not caused by shear stress.

The result is shown in Table 3, Figure 8 and Figure 9. For the newly designed beam, a 35950 N load will causes the von-Mises stress reaches 249.88 MPa and causes the beam to yield. Accordingly, the moment applied on the bean is 44937.5 N·m. However, for the conventional designed beam, the load cause the beam to yield is 37350 N, where the von-Mises stress is 249.5 MPa and the moment is 46687.5 N·m. When applying the loads obtained above at a position near one end face, the shear stress of these two designs are 33.095MPa and 34.188 MPa, respectively. Both shear stresses are far more less than their shear yield strength, which is 145 MPa. So the tensile stresses are responsible for the yield. In addition, the difference of maximum load inducing these two beams to yield is only 3.89%, so it is safely to conclude that adding holes on the web of I-beam will not reduce the performance of the beam significantly.

• Summary and Conclusions

This project use Euler-Bernoulli beam theory and finite element method to analyze the maximum load and moment of two different designed beams. As expected, all the yield failures are caused by tensile stresses other than shear stress. And as a consequence, the newly designed beam could be applied to construct the bridge of overhead cranes with advantages including light weight and providing enough strength.

• References

1. "BS 4-1:2005, Structural steel sections", 2005.
2. "Steel Designers' Manual" 7th edition, 2012. SCI, B. Davison and G.W. Owens.
3. "Advanced mechanics of materials", 1993. A. Boresi, R.J. Schmidt and O.M. Sidebottom.
4. "A First Course in Finite Elements", 2007. J. Fish and T. Belytschko.
5. "Mechanical Engineering Design", 1986. J. Shigley.

• Tables and Figures

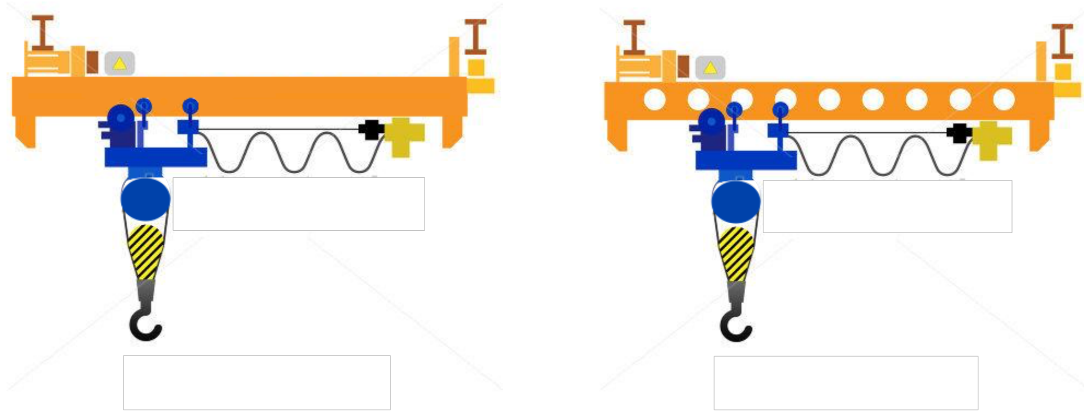


Figure 1. The conventional bridge (left) and the newly designed bridge (right)

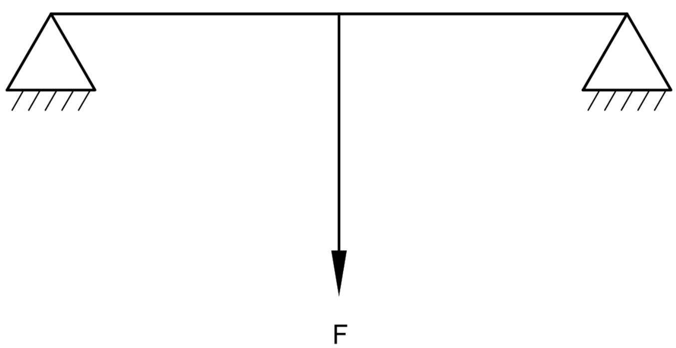


Figure 2. The Physical Model

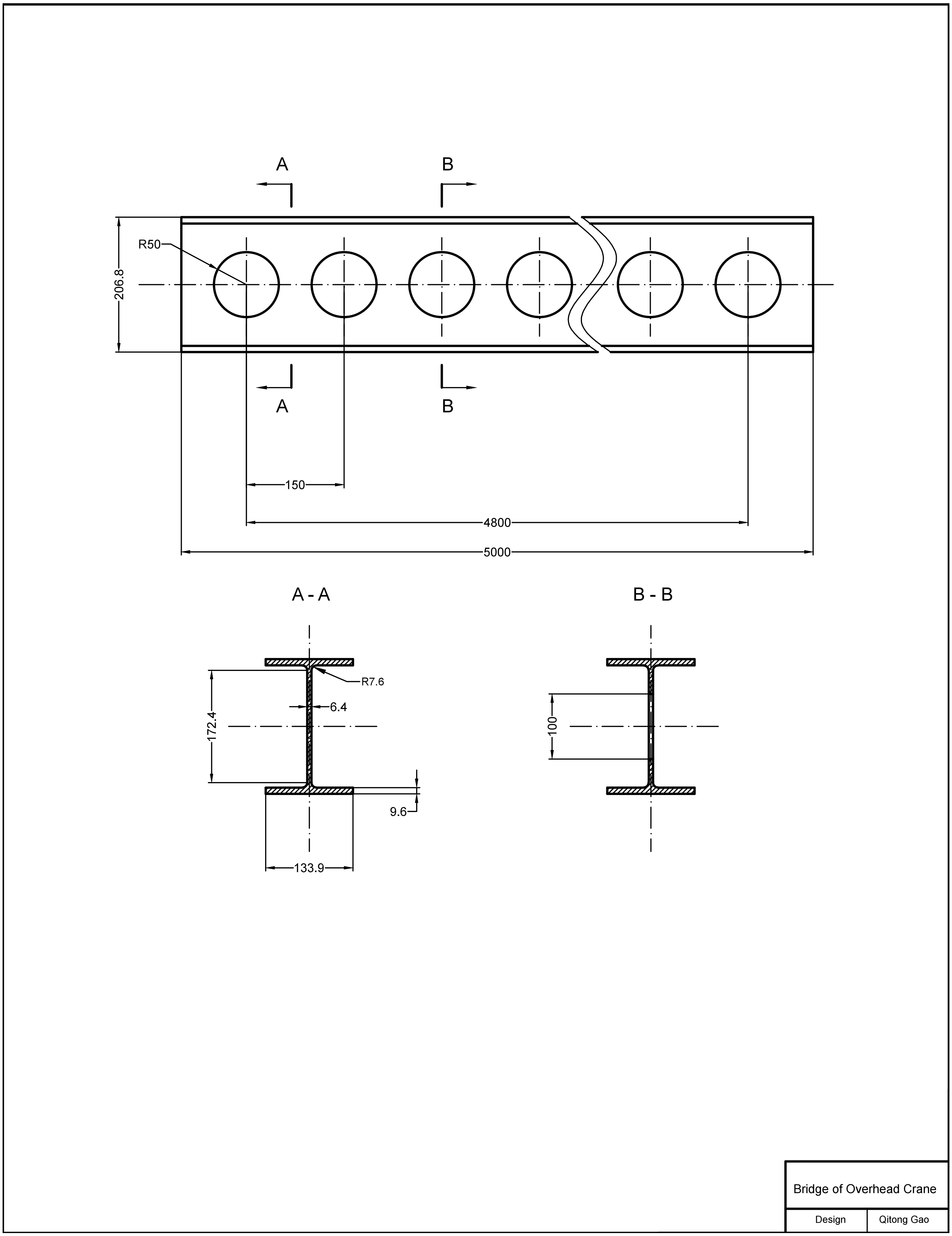


Figure 3. Detailed Beam

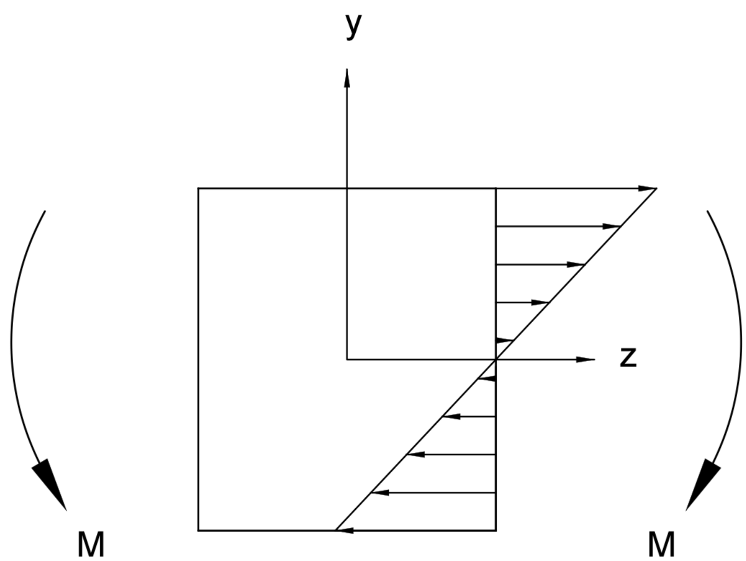
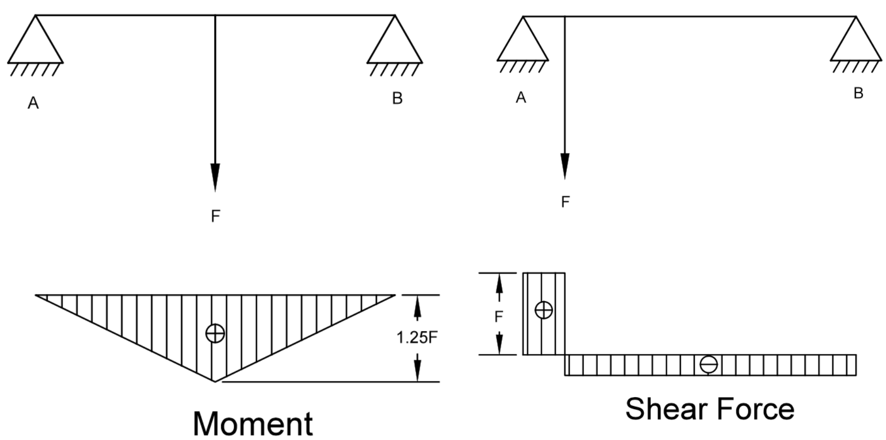


Figure 4. Stress distribution of a bent beam



(a) (b)

Figure 5. Dangerous sections

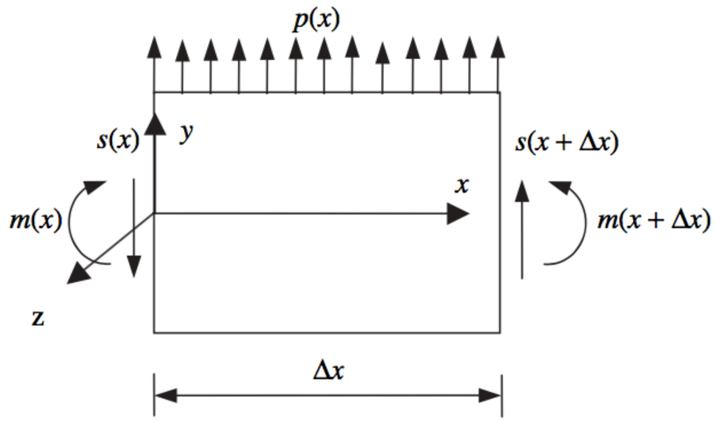


Figure 6. FEM analysis with beam elements

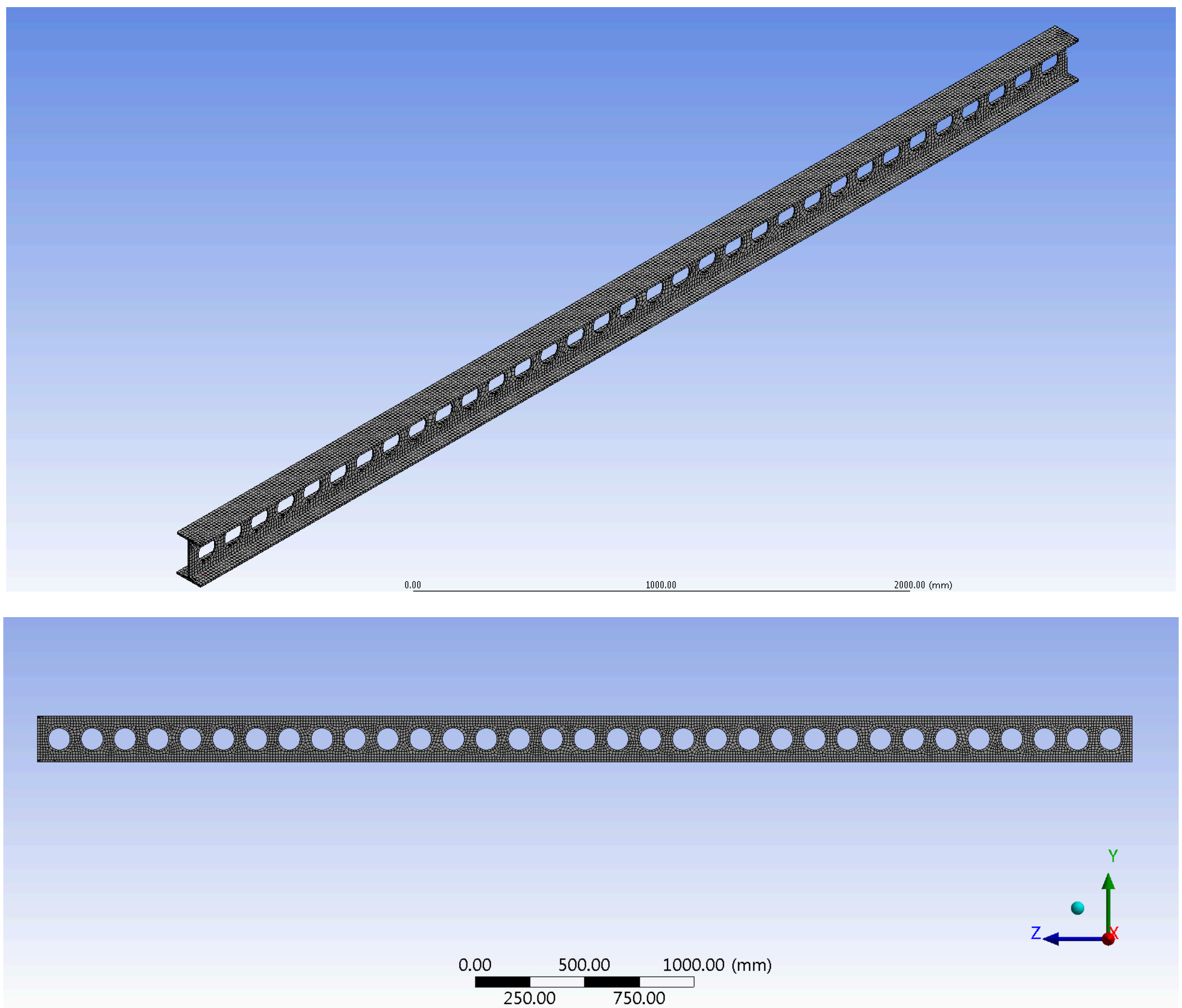


Figure 7. Geometry and Mesh

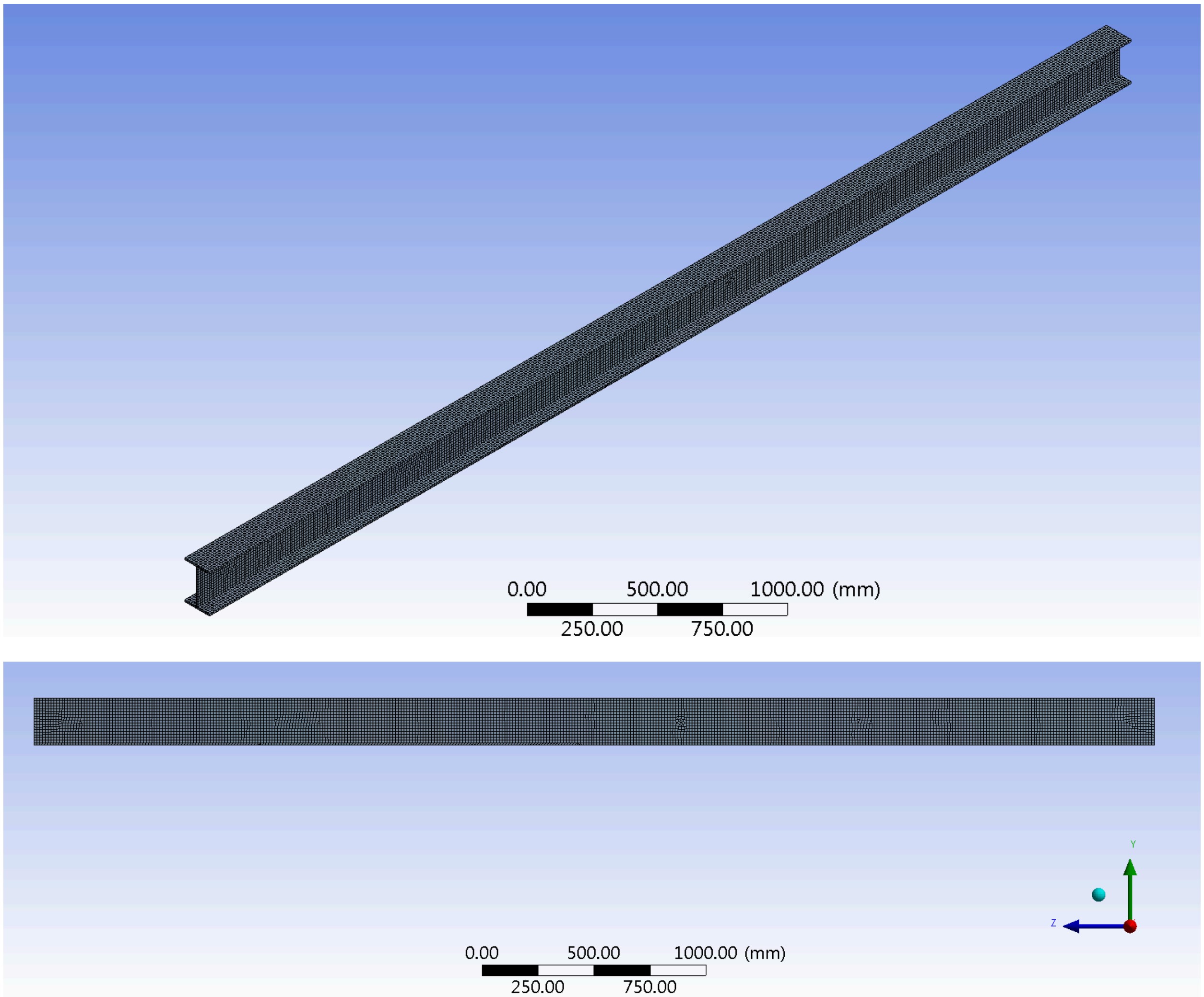


Figure 7. Geometry and Mesh

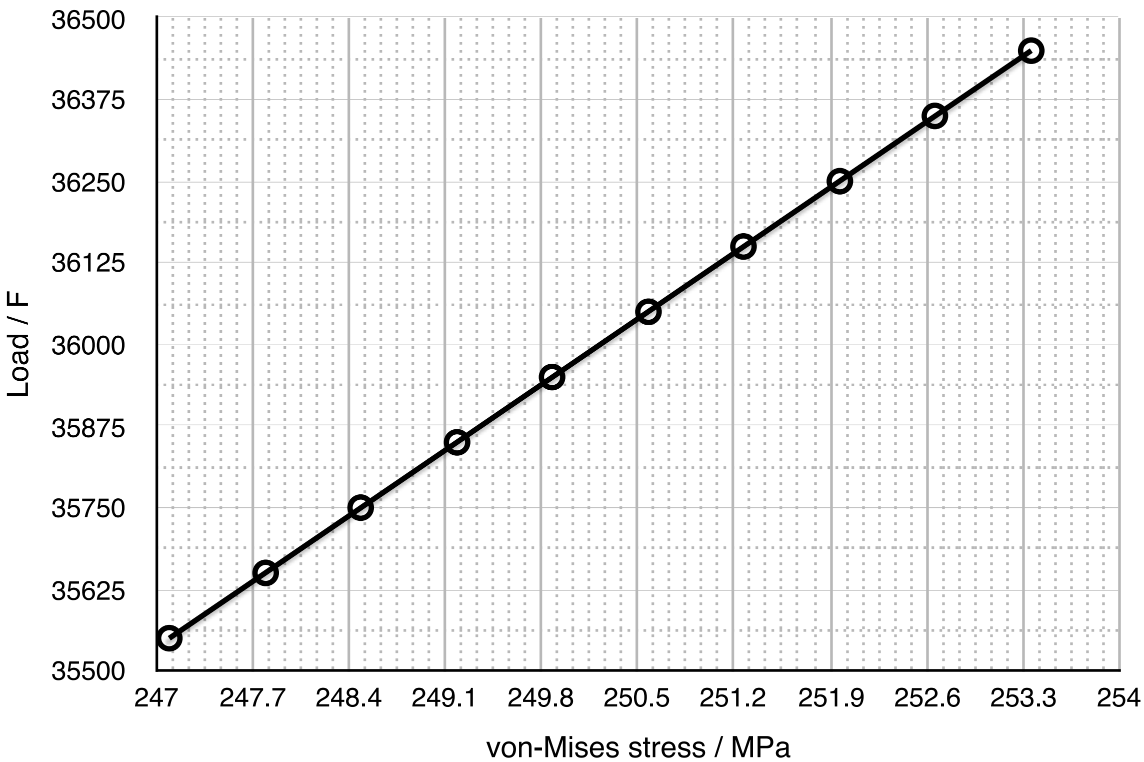


Figure 8. New designed beam

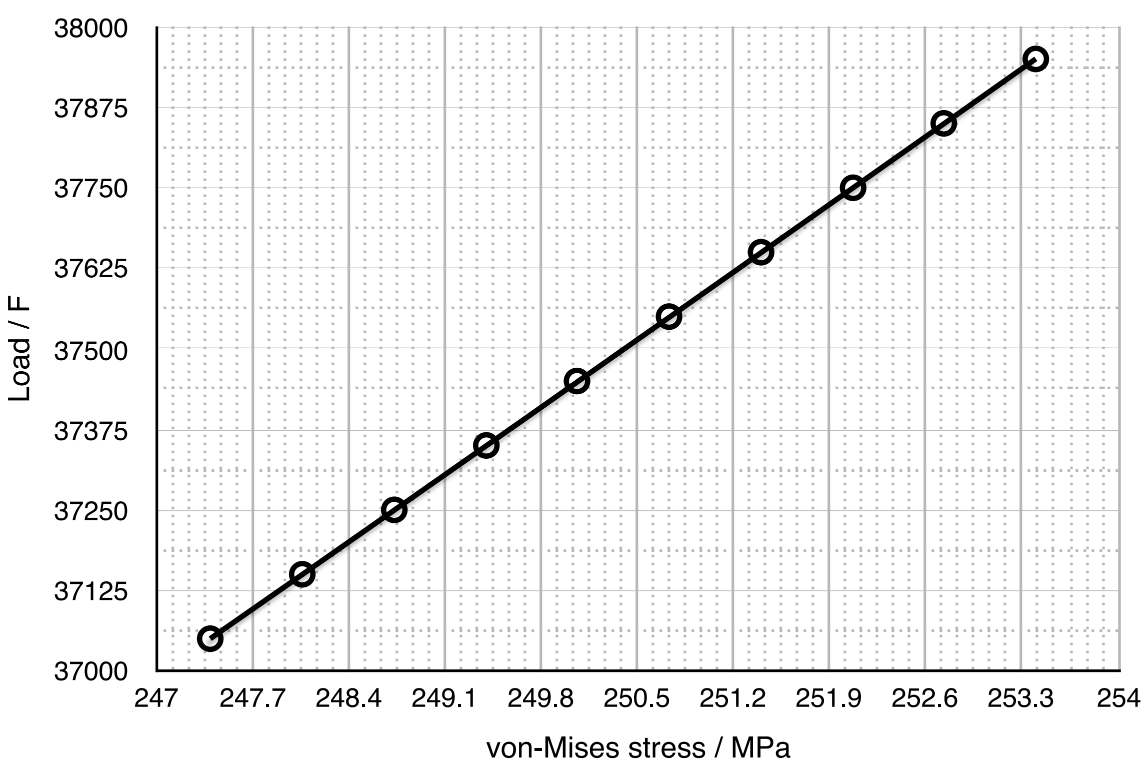
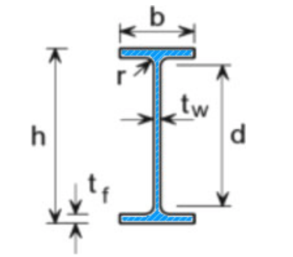


Figure 9. Conventional beam

Table 1. Materials properties and geometry parameters

Parameter	Value	Unit
Young's Modulus	200	GPa
Poisson's Ratio	0.3	-
Tensile Yield Strength	250	Mpa
Shear Yield Strength	145	Mpa



Parameter	Symbol	Value	Unit
Depth of section	h	206.8	mm
Width of section	b	133.9	mm
Web thickness	t_w	6.4	mm
Flange thickness	t_f	9.6	mm

Table 2. Convergence of elements

Element type	Element size /mm	Relative difference %
Q8	100	-
	50	56.36%
	25	34.10%
	12.5	1.49%

Element type	Element size /mm	Relative difference %
Q4	100	-
	50	58.05%
	25	45.26%
	12.5	31.96%

Element type	Element size /mm	Relative difference %
T6	100	-
	50	62.17%
	25	29.49%
	12.5	22.28%

Element type	Element size /mm	Relative difference %
T3	100	-
	50	66.32%
	25	41.85%
	12.5	37.32%

Table 3 Yield load force of new designed beam

Maximum von-Mises stress /Mpa	Load /N
247.1	35550
247.8	35650
248.49	35750
249.19	35850
249.88	35950
250.58	36050
251.27	36150
251.97	36250
252.66	36350
253.36	36450

Table 4 Yield load force of conventional beam

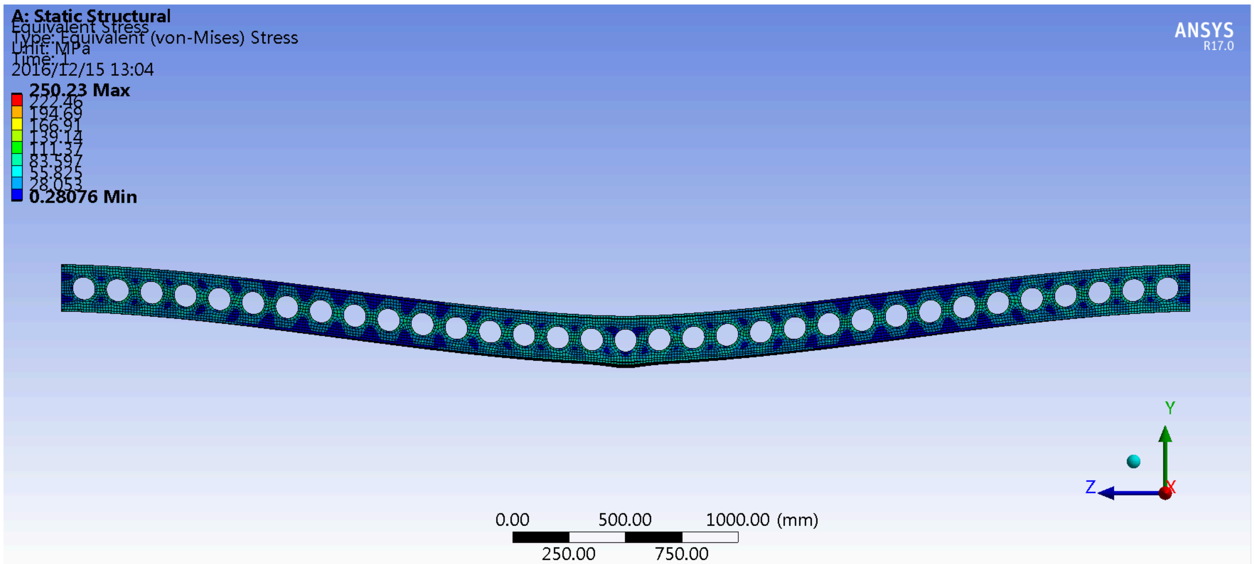
Maximum von-Mises stress /Mpa	Load /N
247.39	37050
248.06	37150
248.73	37250
249.4	37350
250.06	37450
250.73	37550
251.4	37650
252.07	37750
252.73	37850
253.4	37950

• Appendix

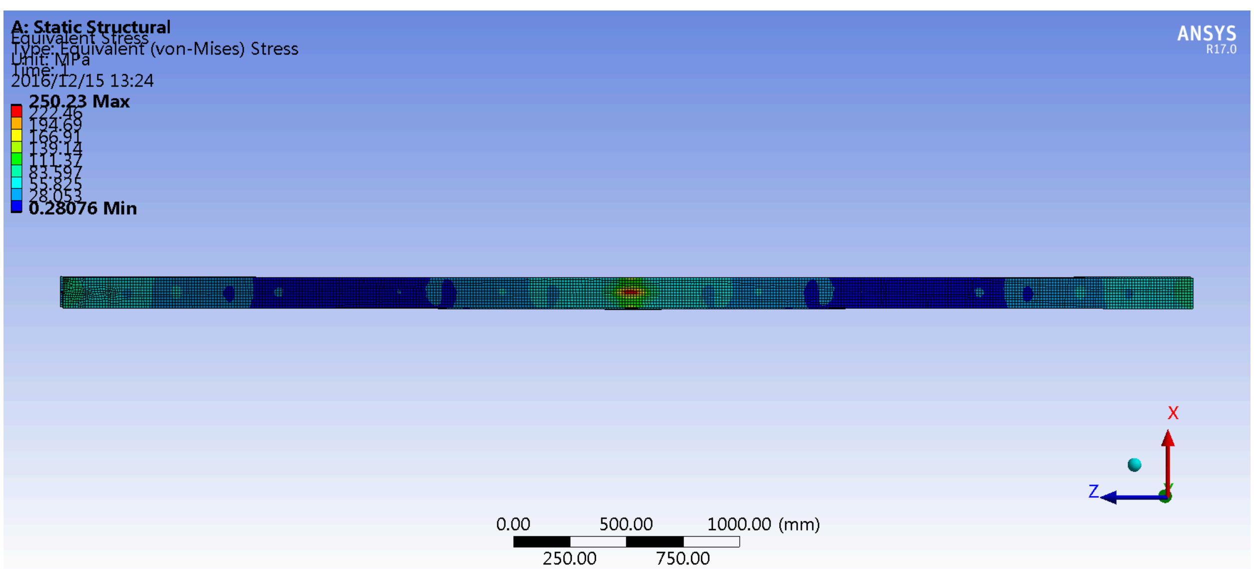
Distribution of von-Mises stress:

- New design

Front view:

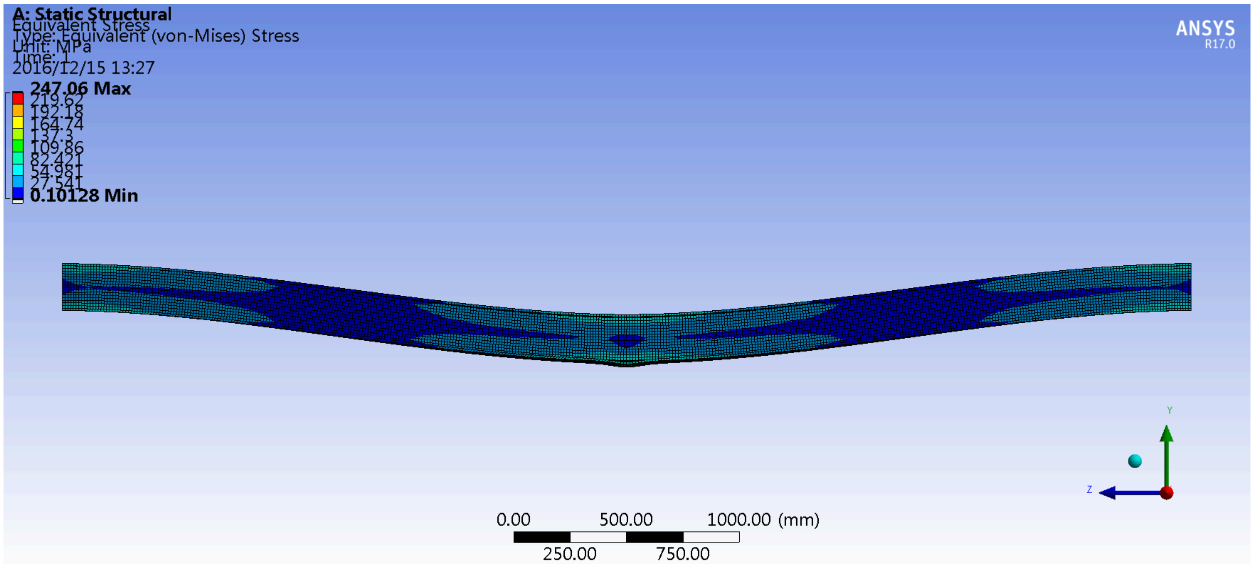


Bottom up view:



- Conventional design

Front view:



Bottom up view: