UNIVERSITY OF SHEFFIELD

# DESIGN OF STRUCTURES, MACHINES AND SYSTEMS MEC209 FINITE ELEMENT INDIVIDUAL REPORT

*Author*
A.V. MYTHEN

*Registration Number*
170154415

April 1, 2019

**TABLE OF CONTENT**

## 1. INTRODUCTION

This is a report to show both analytical and computational methods in the design and optimisation of a plywood structure capable of supporting an 18kg weight, with dimensions shown in figure 1. The structure must also allow a block of the same dimensions to slide underneath the structure as shown in figure 2 (before loading). The structure itself must be made using 3mm thick plywood and its mass to be kept to a minimum.

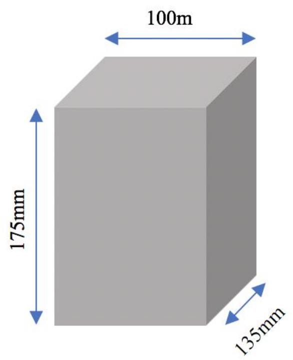



**Figure 1**: 18kg Test Block Dimensions

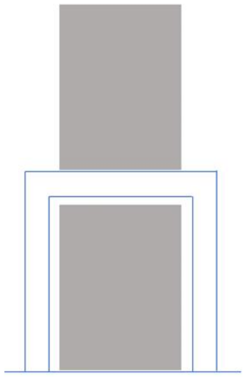


**Figure 2**: Geometrical Arrangement

In order to test different designs without having to construct each one, therefore saving lots of time and resources, Fusion 360 and Finite Element Analysis (FEA) is used to computationally analyse the effects the load has on the structure. FEA shows where in the model there are higher and lower levels of stress and displacement. This then allows optimisation of the design as they can be altered accordingly i.e. building up parts with high stress levels and reducing the mass where there are low stress levels. First a safe design was produced which could confidently support the structure. This was then altered to produce a risky/lightweight design.

## 2. METHODS

In order to use FEA the physical properties of the material being analysed must be inputted into the Fusion 360. As birch plywood was the material being used in this project, the strength and mechanical properties of this material had to be found. The properties of interest are shown and quantified in table 1. Tensile tests [1] were conducted along and across the grain on 3 pieces of plywood for each test, all of the same dimensions. Although all the pieces of plywood tested were of the same species and same dimension the tests showed variation in the material, which shows that plywood, like many other materials have varying mechanical properties therefore any values we collect will have some margin of error and can't be relied on fully. This was taken into account when determining safety factors for both safe and risky designs.

The results from the tensile tests were tabulated and manipulated in order to find stress and strain. With this data, stress-strain curves were produced and used to find yield strength, tensile strength and Young's modulus. Tensile strength is the point at which the material fails, so was found where the graph suddenly drops off. An average value was taken of the three different tests. Yield strength is where the material stops behaving linearly, therefore can be found on the graph where line begins to curve (shown in figure 3). Youngs modulus was found by using the gradient of the linear lines below the yield point (shown in figure 3).

From these results it could be seen that the material had a higher yield strength, tensile strength and young's modulus along the grain rather than across it. Therefore, the values for along the grain were used, assuming the plywood would be laser cut in the orientation that means the load is applied along the grain. Shear modulus, Y, was found using the equation $Y = \frac{E}{2(1+v)}$ (1), where E is the Young's modulus and v is Poisson's ratio which was found online. Density, $\rho$, was calculated using the equation $\rho = \frac{m}{V}$ (2), where mass, m, and volume, V, were measured from a strip of plywood.

**Table 1:** Calculate and Researched Properties of Birch Plywood

| **Properties of Birch Plywood:** | |
|---|---|
| Youngs Modulus | 5.196 GPa |
| Poisson's Ratio | 0.43 |
| Shear Modulus | 1.817 GPa |
| Density | 0.68 g/$cm^3$ |
| Yield Strength | 53.00 MPa |
| Tensile Strength | 73.39 MPa |

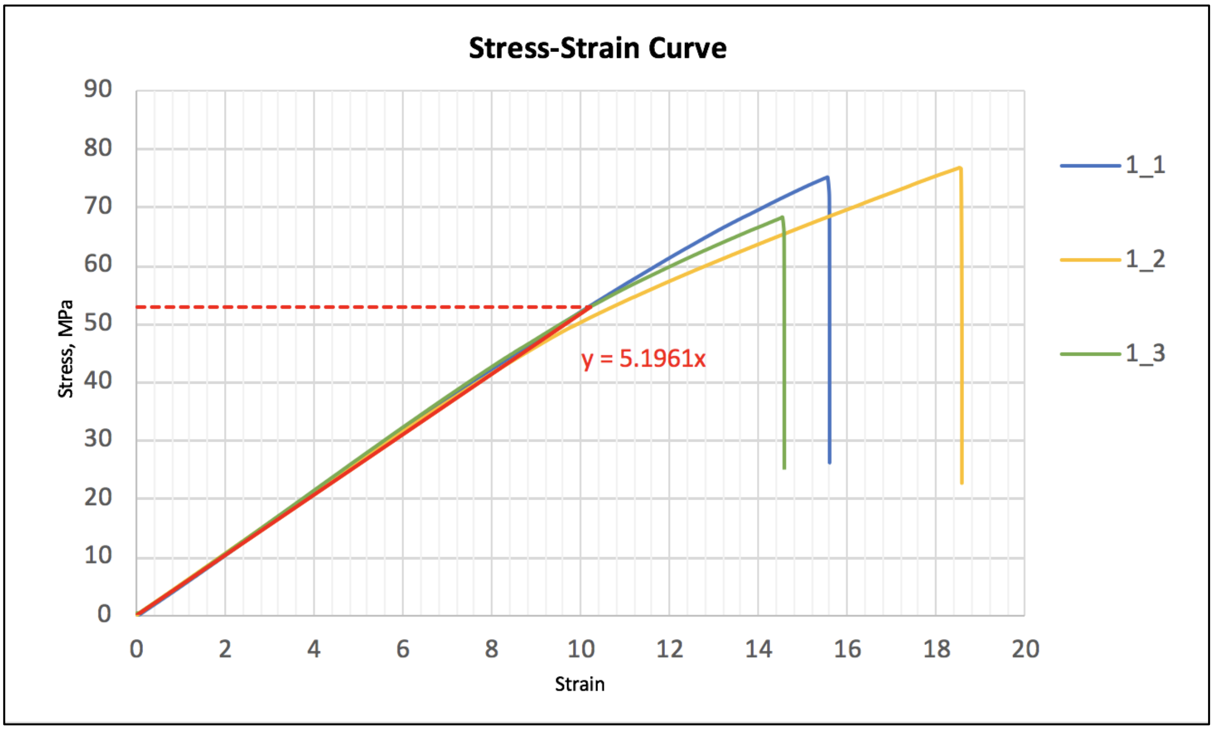



**Figure 3:** Stress-Strain graph of 3 tensile tests

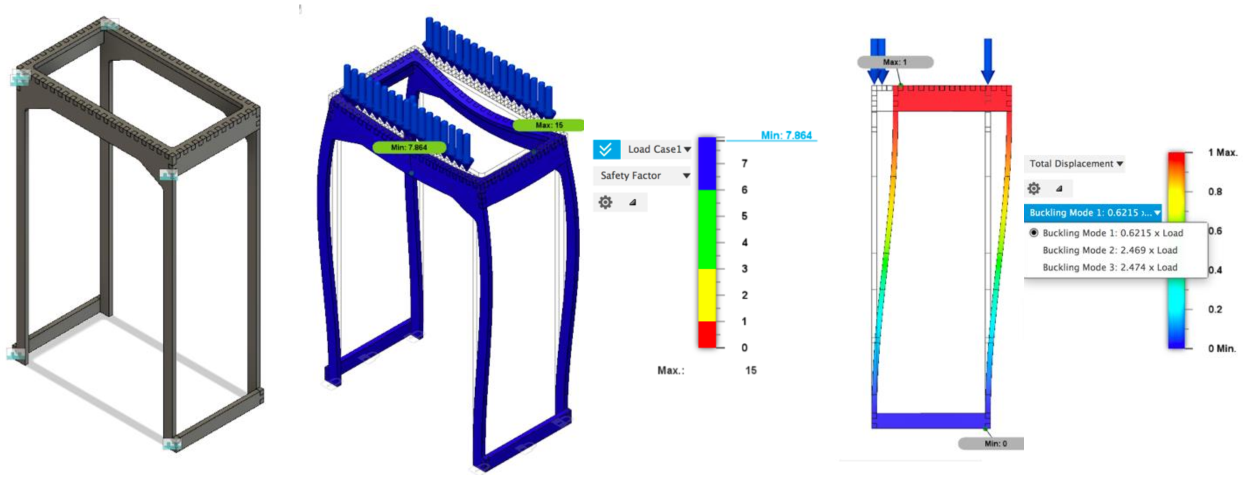
As mentioned earlier the physical properties of plywood can vary even within a species. This can be due to manufacturing processes, and therefore means a margin of error must be considered when using FEA to model designs. Because of this, a factor of safety above 3 will be what is aimed for while designing a safe model, as this should account for both the possible difference in theoretical material properties and actual material properties, and also the limitations in Fusion 360. A factor of safety of 1.5 will be aimed for while designing the risky model, while mass should be kept as low as possible.

While modelling designs in Fusion 360 several assumptions are made by both the software and the user. While simulating the load onto the structure, Fusion 360 assumes the load is symmetrical and therefore evenly distributed throughout the selected surfaces. As the constraints on the base of the structures are fixed, this therefore assumes that the friction from the bench is great enough to prevent any slipping. Also, by using fixed joints to join the model together, the assumption is made that the wood glue used to join the structure in real-life, will not fail. Finally, there is the assumption that the plywood behaves linearly, in order to simplify the simulation.

## 3. RESULTS

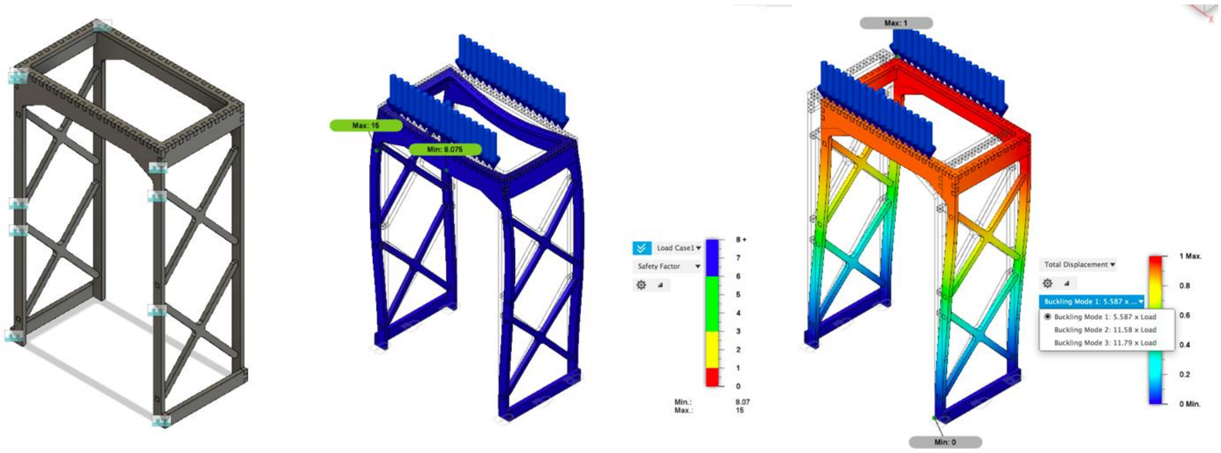
### 3.1 Safe Design

As the most important aspect of the structure is that it can support an 18kg block which could also pass under it, this was the main focus of the initial design, and the reduction of mass came later. Figure 4.a shows a basic initial design. After running a static load simulation, it showed that it could support the load by a safety factor of 7.864, however buckling load simulation shows that it would fail in buckling mode 1, as shown in figure 4.c. This therefore led to 2 cross structures being added to each side, as shown in figure 5, which increased the safety factor in buckling mode 1 to 5.587 and in buckling mode 2 and 3, safety factors both just over 11.




(a) (b) (c)

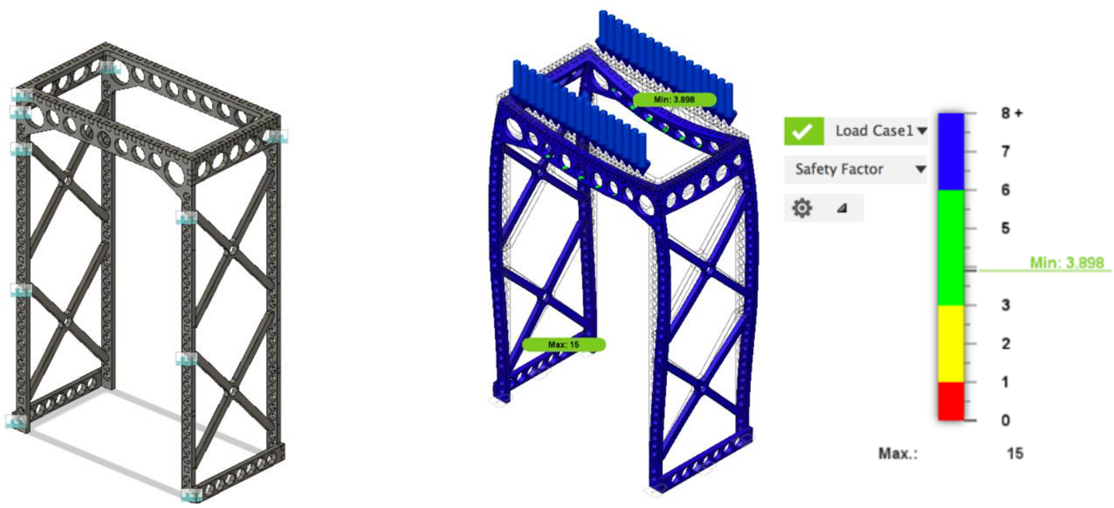
**Figure 4:** Initial Design with Finite Element Analysis (Mass: 27.488g)




(a) (b) (c)

**Figure 5:** Optimised Initial Design with Finite Element Analysis (Mass: 33.718g)

Now the structure has safety factors in both static and buckling greater than 3, the mass can be reduced. In FEA the different colours represent the different stress concentrations for the static tests and levels of displacement for the buckling tests. The dark blue on the static tests show where there is minimal stress, therefore mass can be reduced in these sections by cutting out parts of the material. To reduce a build up of stress concentration is corners, circular holes or ellipses are used, as shown in figure 10. These holes reduced the weight by nearly 10g, making it a total of 23.943g, while all factors of safety are still above 3 (shown in figures 6 and 7). Therefore, the safe design now meets the design criteria and the desired safety factor as discussed in the method, as the mass is much lower than the initial design.




(a) (b)

**Figure 6:** Final Safe Design and Static FEA (Mass: 23.943g)

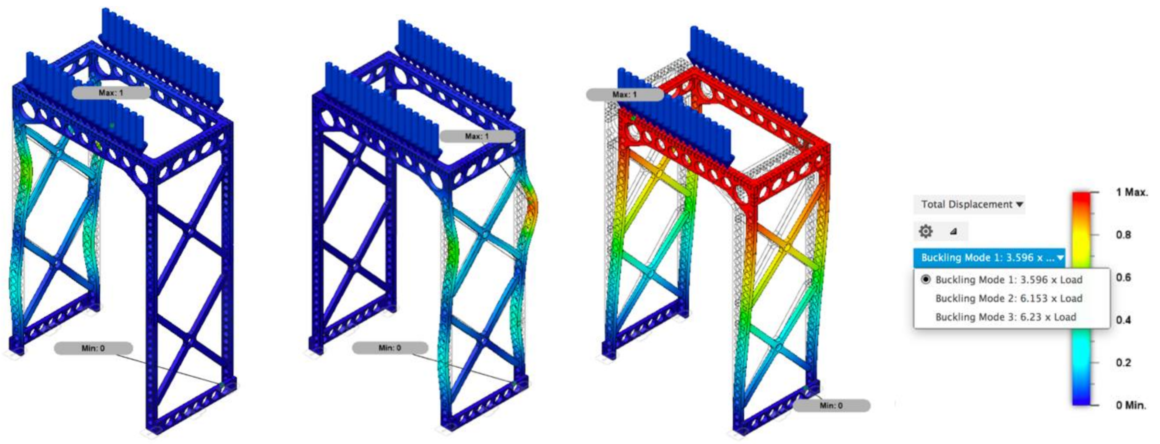



**Figure 7:** Structural Buckling Results of Safe Design FEA (Mass: 23.943g)

## 3.2 Risky Design

Once the safe design was finished, it was further developed to produce a lighter design, which was allowed a lower factor of safety. As there were already many holes cut out of the material, the next thing to look at was reducing the number of parts. The top of the safe design includes a rectangular frame and 2 sides (shown in figure 8), weighing a total of 5.165g. Therefore, to reduce weight, these were replaced by 2 beams (with holes in them) slotted into the top of the design (shown in figure 9), weighing 1.082g together.

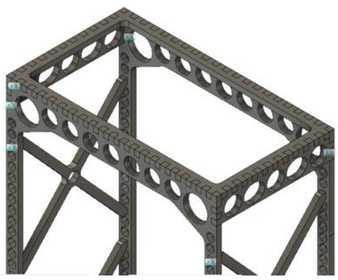
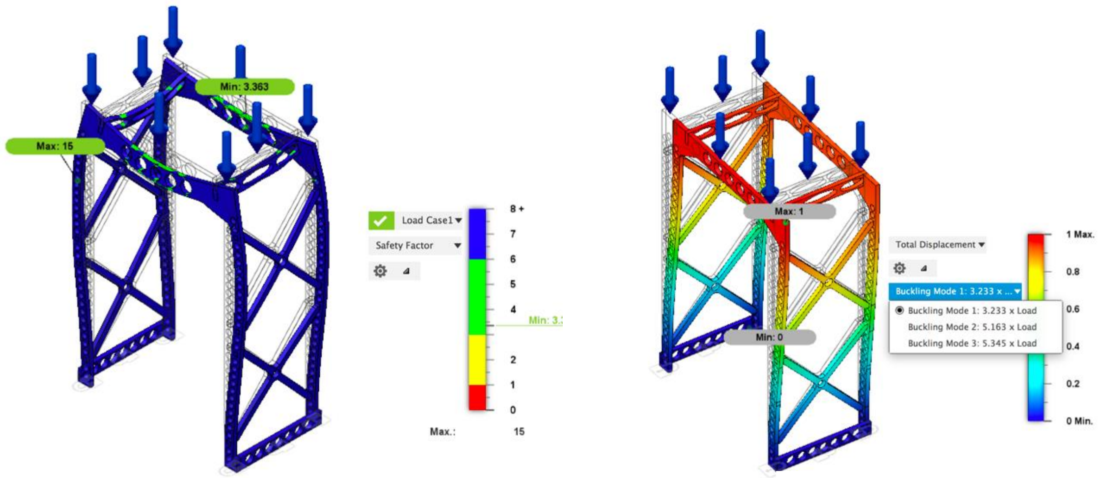


**Figure 8:** Top of Safe Design

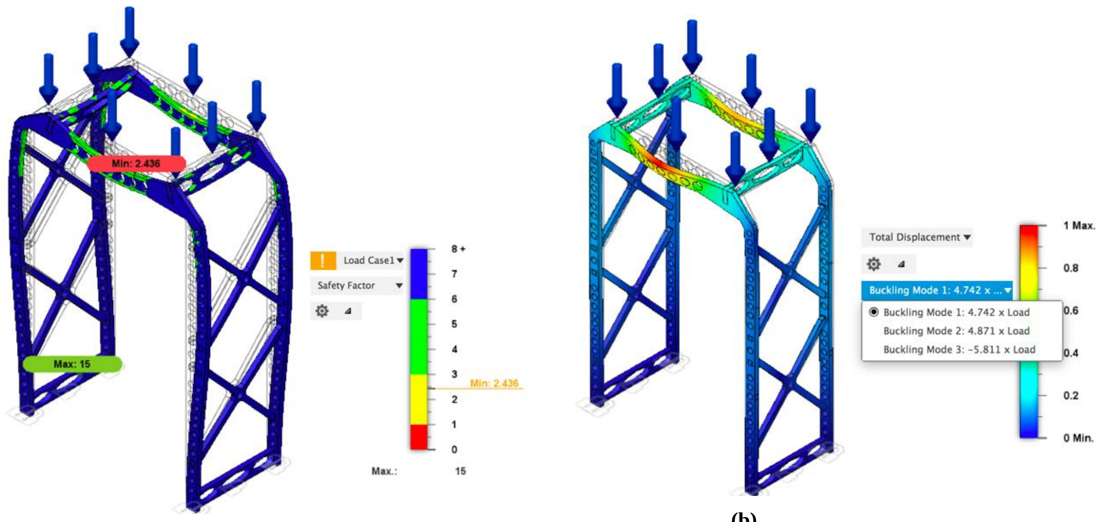
**Figure 9:** Top of Optimised Design




**(a)** **(b)**

**Figure 10:** Static and Buckling FEA Results of Optimised Design (Mass: 19.86)

These beams not only help hold the front and the back of the structure together like the 2 sides did in the safe design, but also add support for the block, like the rectangular frame in the safe design, as they will be directly under the mass. Also, tests show that materials in this orientation are more resistance to bending than when they are lying horizontally. As shown in figure 10.a this alteration only reduces the static safety of factor by about 0.5 and buckling safety factors by less than 1, while reducing the mass by 4.083g. These bars used were also lighter than the ones used for the feet of the safe design, therefore replaced here too, and slotted into the base, like they were at the top, so teeth weren't need to be added to join them together.




**(a)** **(b)**

**Figure 11:** Static and Buckling FEA Results of Optimised Design (Mass: 19.86)

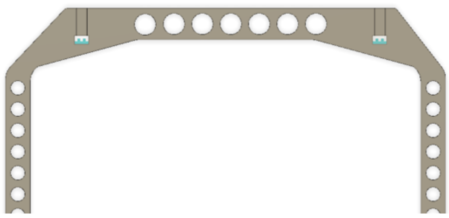
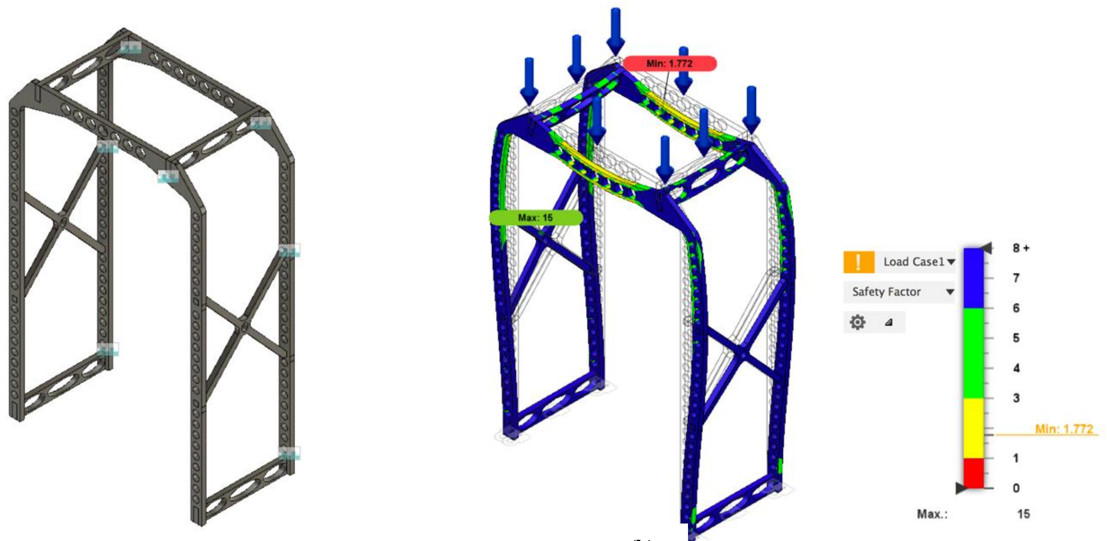


**Figure 12:** Top of Optimised Design with Edges removed

As seen in figure 10.a the corners at the top of the structure are dark blue therefore have minimal stress acting on them, so to reduce weight, were removed (shown in figure 12). This reduced the static load safety factor to 2.436, while the buckling safety factors in all 3 modes were above 3. So, the final stage of reducing the weight was removing a cross from each side and moving the remaining ones more centrally, which as could be expected reduced the buckling safety factor. As seen in figures 13 and 14, the final risky design has safety factors all greater than the desired safety factor of 1.5 and weighs 15.45g, therefore still meets the design criteria however is 8.463g lighter therefore more greatly satisfying the lightweight aspect of the brief.




(a) (b)

**Figure 13:** Final Risky Design and Static FEA (Mass: 15.45g)

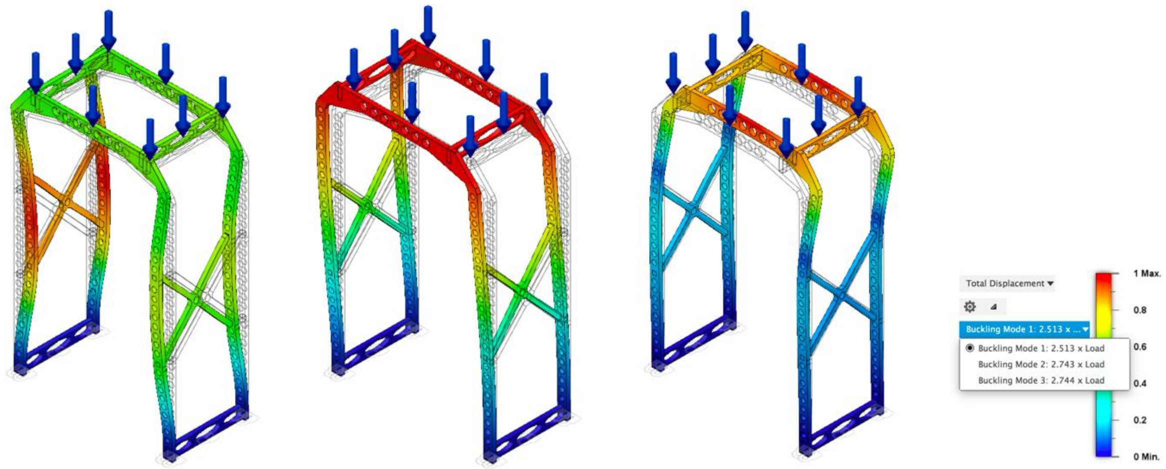



**Figure 14:** Structural Buckling Results of Risky Design FEA

## 4. DISCUSSION

As discussed in the method, when modelling the designs above, fixed constraints were used on the base of the structure and therefore the FEA can't show failure due to slipping however this is a possibility in the real-life scenario. Frictionless constraints are available in Fusion 360 however this wouldn't be accurate either because there would be friction between the bench and structure. This therefore is a limitation in the software. One way to improve the FEA results would be to fix one of the corners of the legs and have frictionless constraints on the others therefore it could show if the legs would spread apart, which would be a more accurate representation of what might happen, and may be a cause of failure in the model.

Another assumption which potentially isn't very realistic and therefore could lead to inaccuracy's in the FEA results was that the load was applied perfectly symmetrical and evenly distributed. In real life this may not be the case, so a way to improve this could be to model the 18kg block in fusion 360 on top of the structure. The software also doesn't show if the structure twists, therefore cannot be determined as a cause of failure although is an extremely likely cause.

Although FEA is a very useful piece of software for modelling designs without having to use resources, it has many limitations and requires the user to make several assumptions for themselves which won't always be very realistic. This therefore means while using the software users need to be aware that their results will have some margin of error, and will vary depending on the assumptions made. One major thing Fusion 360 and FEA are really good for however, is comparing designs to each other. Although there may be inaccuracies in the results, they will be the same for all the models. Using this software can therefore quickly and cheaply find the best model out of group without having to manufacture and test them in real life.

## 5. REFERENCES

[1] University of Sheffield, *Tensile Tests Along and Across the grain conducted 5 March 2019.* [Reviewed March 2019]