# Group 15 Finite Element Analysis

## Option B

A *relevant optimisation* of a component of your universal design work

**Objective**

The objective of this assignment was to analyse the structure of our mobility aid adaptation. As part of our Universal Design Course, we decided to tackle the issue of reduced strength, dexterity and mobility in the aging population. A lot of our stakeholders cited lifting and moving objects from one place to another as a difficult daily task. We decided to adapt a universal mobility aid (a zimmer-frame) by attaching a tray to it. This could be used for transporting items such as books, food and other small household items.

The weight of our component is a crucial element of the design, as it was to be optimised for those with reduced strength. It was also important to remember that this device is mobile – the user needs to be able to push the frame around with ease. To do so, we needed to minimise the amount of weight of the design, while ensuring it was of adequate strength to withstand a certain weight.

**Approach**

Finite Element Analysis was a crucial tool to help adapt our design. An initial analysis was performed on our CREO assembly based on an applied load of 120N. This provided an overview of the resultant stress, strain, and displacement experienced by the brackets and connected tray.

An auto-generated mesh produced a basic overview of the structural integrity of our component. The mesh is a feature of FEA in which a grid-like net is seen on the surface of the component. Each section of the grid is known as a node. The nodes in the material have a certain density that depends on the stress on the given area. The higher the node density, the higher the stress, and intuitively, the lower the node density, the lower the stress.

It is important to strike a balance between accuracy and computational time when making the mesh. The more nodes there is, the more accurate it will be. However, this also means there will be a long computational time. It is best to refine the mesh in such a way that there is a higher node density in places where there are high stress concentrations, and an average node density on the rest of the element.

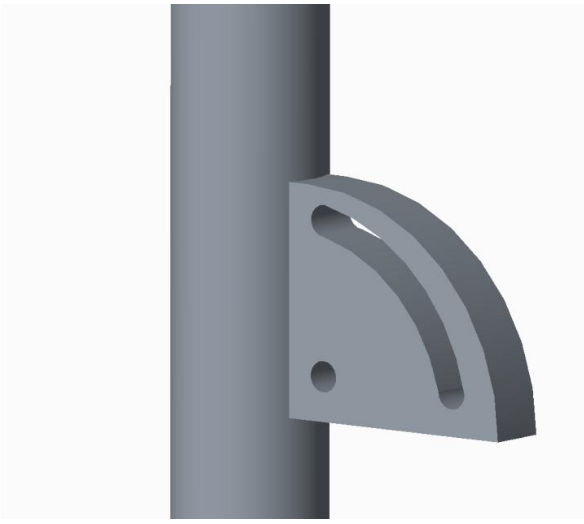
This simple feature is a helpful tool to quickly analyse where areas most prone to stress occur in the component. The mesh can then be refined to increase the amount of nodes in a particular part, providing a more indepth view of the stress, strain or displacement distribution in that area.

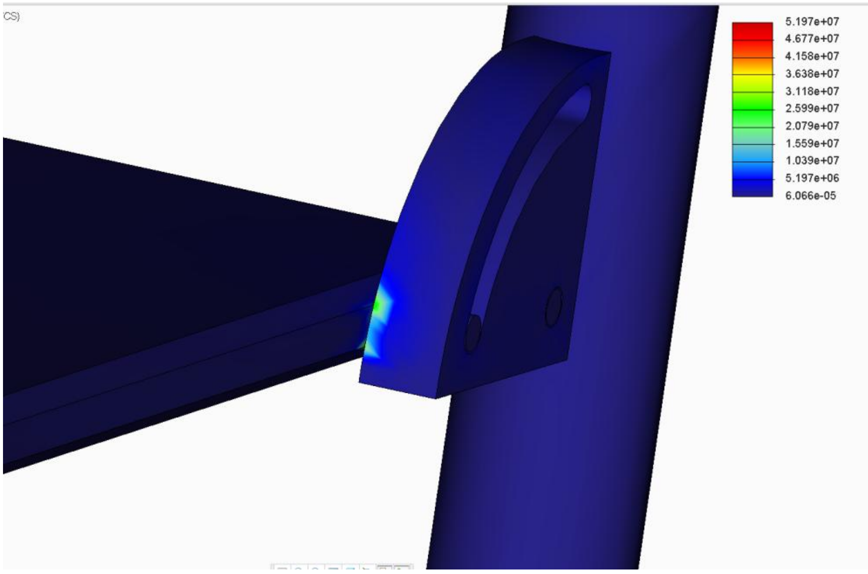
From these actions, conclusions can be drawn as to optimise our component to better match the concept needs. Material can be removed from areas of very low stress concentrations. This contributes to a reduced overall weight of the component, as desired.
If there is an area of stress or strain much higher than desired, additional material may be added to reduce the stress. Naturally, adding material would increase the weight, so it would be important to ensure material added is done so in the correct proportions. Adjusting the mesh to get a clear image of how the stress is concentrated is ideal for ensuring this.

For our concept, the tray is only supported at one end, so deflection of the tray would be a key aspect to consider. The FEA can give an approximation as to how the tray will react under the 120N load. If the deflection experienced by the tray is greater than desired, alternative support methods or alterations to the current support method may need to be considered. By altering the bracket dimensions, the stress distributions would also change. This would need to be monitored to ensure the stress experienced by the bracket is within the desired limits. If it exceeds these limits, the analysis and compensation methods described above may need to be utilised once more.

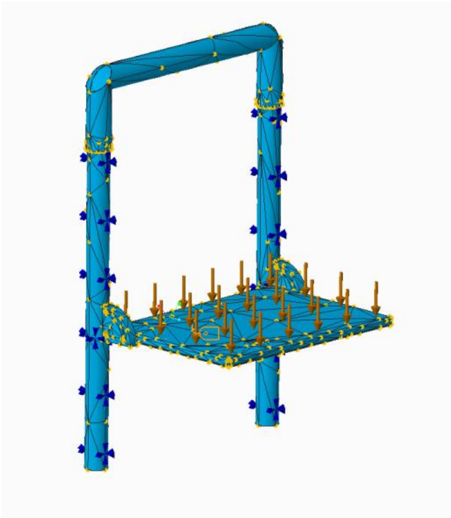
**Results**

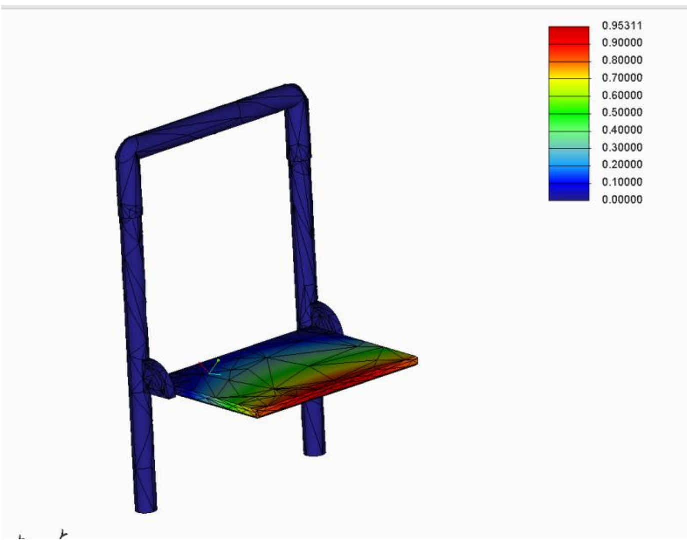
**(i) Original bracket under 120N load**



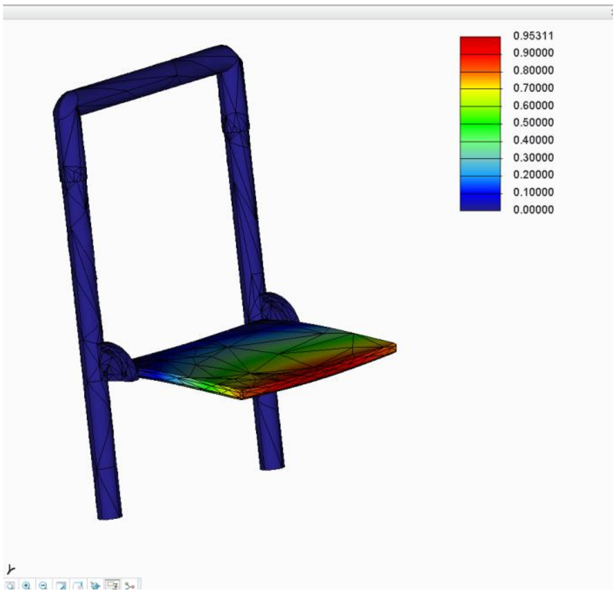


5.197e+07
4.677e+07
4.158e+07
3.638e+07
3.118e+07
2.599e+07
2.079e+07
1.559e+07
1.039e+07
5.197e+06
6.066e-05

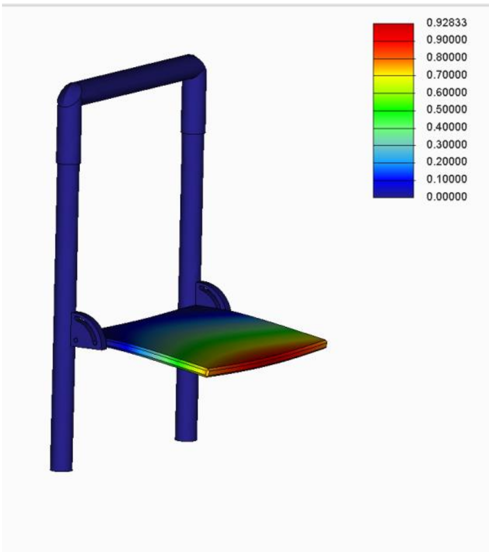





0.95311
0.90000
0.80000
0.70000
0.60000
0.50000
0.40000
0.30000
0.20000
0.10000
0.00000



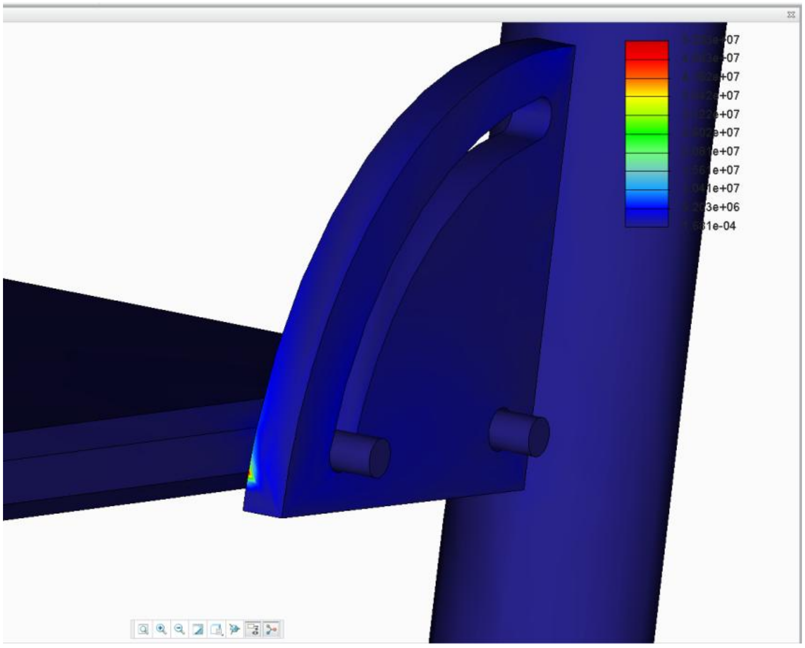

Stress with original bracket in place



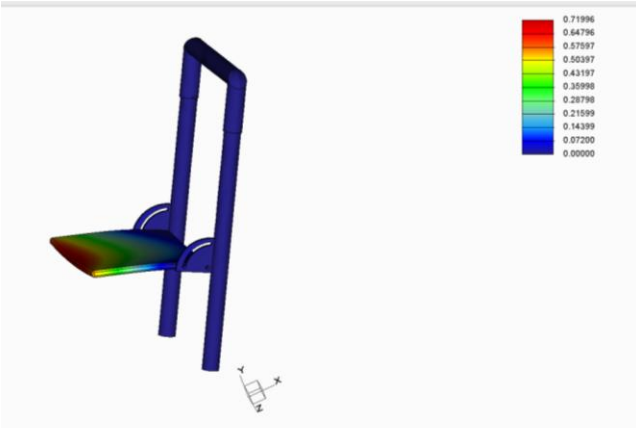

Displacement with original bracket in place

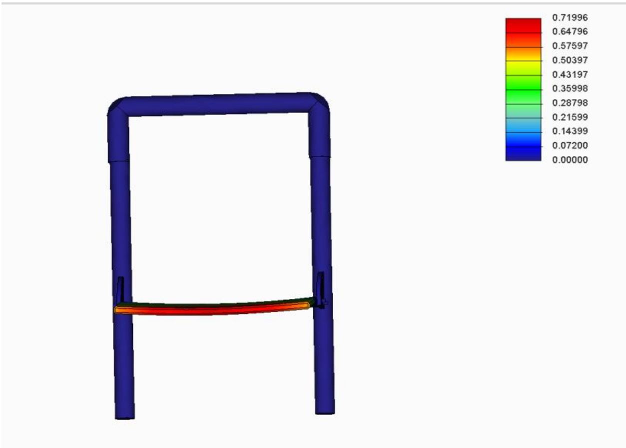
### (ii) Modified bracket under 120N load

The decision was made to optimise the bracket by reducing material and increasing radius. The thickness of the component was reduced by 10mm while the radius was increased by 20mm in order to provide better support and reduce deflection.

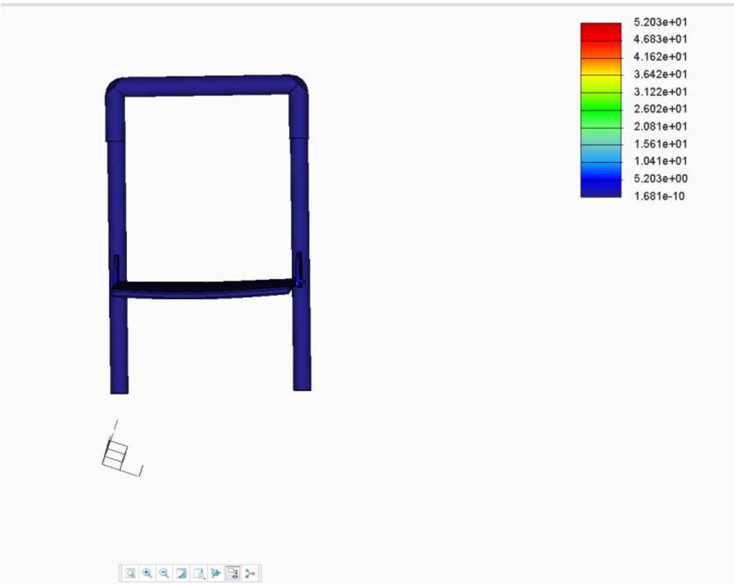


Stress concentration in optimised bracket



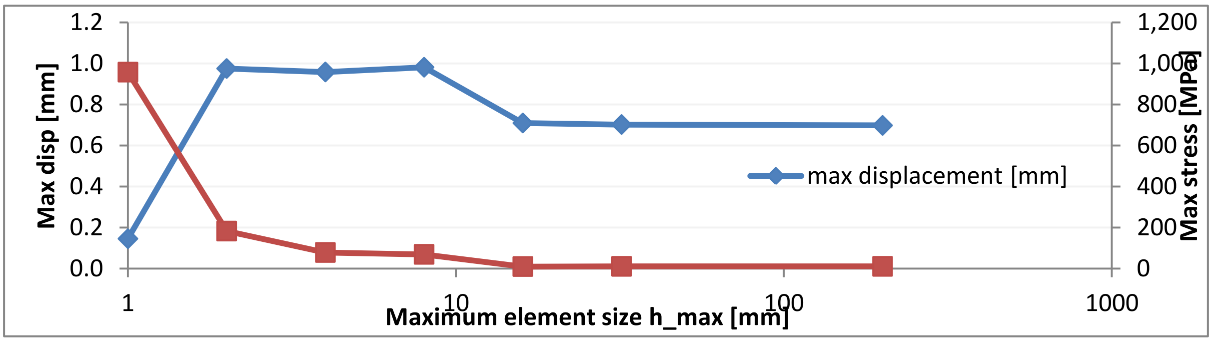



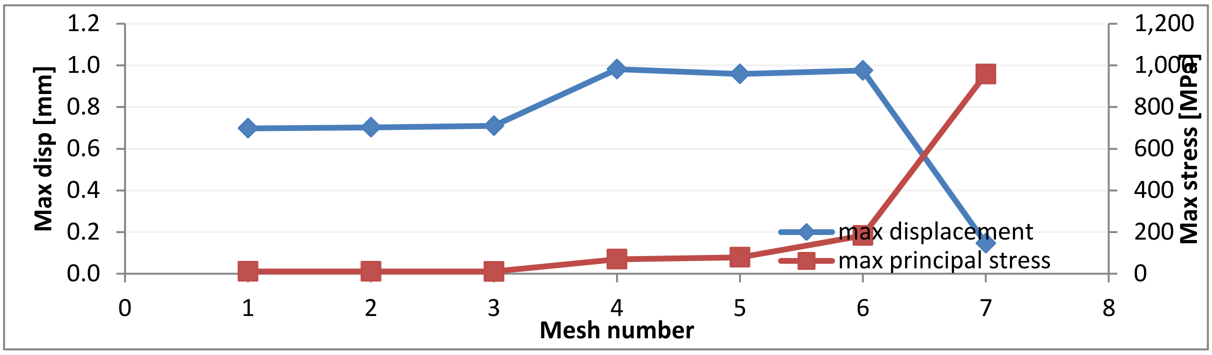

Displacement with modified bracket




Stress with modified bracket

### (iii) Progressive Mesh Refinement




**Conclusion**

The bracket was successfully optimized through the following two methods:

- Material used was reduced. This was achieved by halving the thickness of the bracket from 20mm to 10mm. This did not have negative effects on the overall project as the piece was excessively large for the load it needed to support.
- The radius of the bracket was increased. This served to increase the support offered and decrease deflection.

These optimisations were successfully analysed in CREO. As the mesh is refined, it becomes clear that the results of the FEA are mesh independant. An excel sheet provided in Demo 2 was modified with the values corresponding to our model.

<u>Contributions</u>
Ciara: FEA, CREO Report & Excel
Ben: CREO, Excel, FEA & Report
Libby: CREO & Report
James: CREO & Report