UNIVERSITY OF HULL

University of Hull
Hull, HU6 7RX
United Kingdom
T: +44 (0)1482 346311
w: www.hull.ac.uk

Department of Engineering

Designing and analysing a bracket by changing its parameters

Forename/s: Ema

Family Name: Shek

Year of Submission: 2022

Module: 601065_A22_T1

Degree Programme: Biomedical Engineering

For the attention of Dr Peter Watson

I. Abstract

This report aimed to determine the accuracy and validity of two models: the 2000 and 3000 models. It was determined that both models are accurate as both deflection and stress converge around 1400 elements, at an h refinement of 4. It was also determined by comparing the theoretical results to the calculated ones, that if the load and other geometrical constraints are the same, the computed values match what was obtained by the theoretical answer further proving accuracy. When determining which model would be able to hold the weight, a safety factor of 1.5 was used and it was determined that both models are likely to fail, however, with a score of 0.91, the 2000 model is near a safety factor of 1 and might be able to hold the load for a limited amount of time before failure. It was also determined that due to the 2000 model having less maximum deflection and less stress in the Y direction, it would be a more suitable choice. Finally, when interpreting the validity of the model and comparing it to real life, it was discussed that the model does not accurately represent fixation to the wall, tolerances that are not included in the FEA model can cause an impact in real-life during manufacturing and in real life, a point load does not exist. To fix these issues, a series of changes in the FEA model have been advised to improve validity.

II. Table of Content

III. Table of Figures

IV. Table of Tables

V. Table of equations

1. Introduction

Just Hanging Ltd has two bracket models that need to be analysed through FEA (finite element analysis) modelling to determine the most adequate for their clients, these two brackets are the SupportMaster2000 and SupportMaster3000. Both brackets must hold a force of 100N, and the company is enquiring if the two models can hold a force of 100N by examining maximum deflection and stress in the Y direction at a specific point. In this report, both models will be analysed to determine the most suitable.

2. Model construction

In this section key steps to model the bracket will be explained. To construct the same model, it is important to know that the bracket model has very specific geometric constraints, which are shown in table 1.

Table 1: all parameters needed to be able to design the model are shown in this table. On the left, the values for the 2000 model are listed and on the right for the 3000.

Parameters	Values for 2000	Values for 3000
R1 – the radius of the internal arc	12mm	15mm
D1 – diameter of bolt holes	3mm	2mm
THICKNESS t	2mm	2mm
Young's Modulus	200GPa	200GPA
Poisson's ratio	0.3	0.3
Loading on bracket (on lower bolt holes)	100N	100N
Element type	Shell 281	Shell 281
Width	40mm	40mm
Radius	R1	R1
Length of the bracket top section	50mm	50mm
Radius of curvature	20mm	20mm
Distance between the top of the bracket and the hollow section horizontally	2xD1	2xD1
Distance between the top of the bracket and the hollow section vertically	10mm	10mm

Table 1: parameters and their corresponding value for the SupportMaster 2000 and SupportMaster 3000 models of the bracket along with Young's Modulus variable, geometry constraints, type of element needed and Poisson's ratio.

It is easier to start designing only one-quarter of the complete model and then mirror the area in the plane of symmetry XY before merging all entities and applying a force of 12.5 N instead of a 100N load at the bottom hole. It is important to define the parameters shown in table 1, which are R1=12mm, D1=3mm and t1=2mm (for the SupportMaster 2000 model): R1 is the radius of the hollow section at the middle of the bracket, D1 the radius of the smaller holes where the bolt goes through and t1, the thickness of the bracket. The width of the bracket is 40mm, and the length of half the section is 50mm, the distance between the top of the bracket and the hollow section is 10mm, and the horizontal distance between the side of the bracket and the start of the hollow section is 2 times the value of D1, as shown in figure 1.

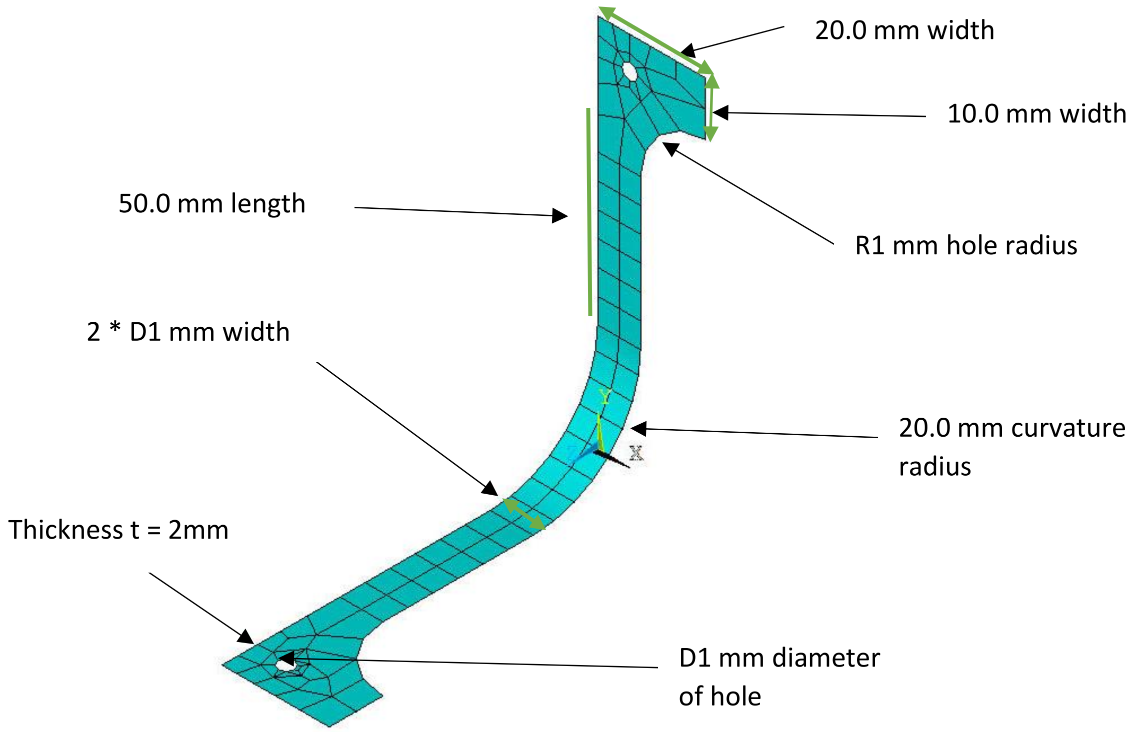

Figure 1 showing geometry construction for 2000 model

It is also important to determine material properties and choose Shell281, along with a young's modulus of 200GPa, which determine the elastic properties of the material when undergoing tension lengthwise in one direction, and Poisson's ratio of 0.3, which determines the ratio of change in width per unit width due to strain of the material. Shell 281 has been chosen as it is ideal for curved shapes of thin or medium thickness and the material properties have been defined with those values so that the model is solved as A36 steel.

Now that material and material properties are defined, as shown in table 1, it is important to determine the numbers of elements which will then be increased until they converge. The elements must be created so that there are 2 elements in 8 lines, four of which are around the hole, 4 elements in 3 lines, 6 in 2, and 5 in 1 with a spacing of 0.5 and 2, as shown in figure 2.

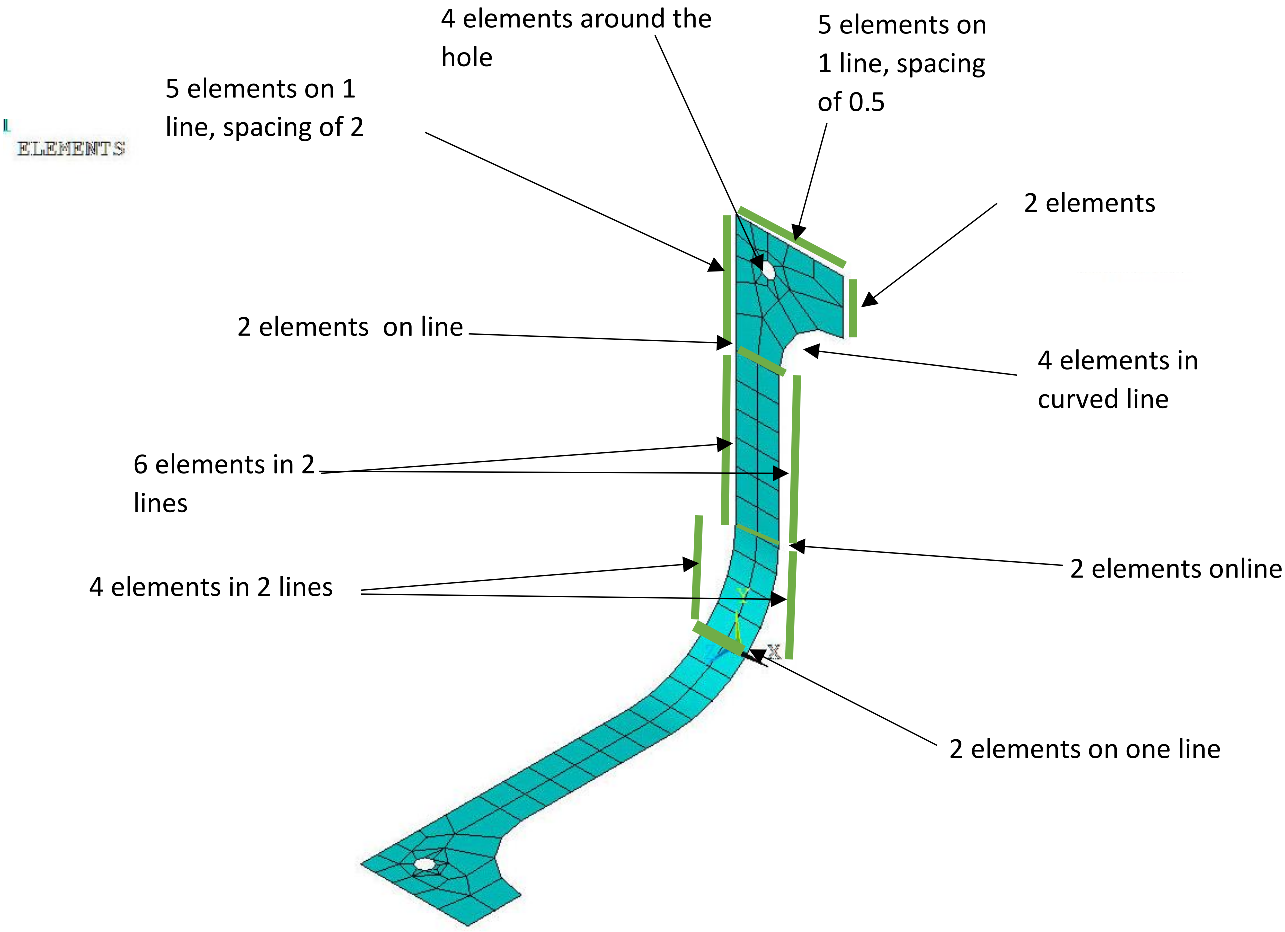

Figure 2 showing the number of elements in each line to create the geometry needed.

This is where the end of the construction approaches and all the areas have meshed, as shown in figure 3. In the end, as shown in table 1, a force of 12.5N, as it is ¼ of the full geometry (12.5 N on each side of the geometry), is applied to the 4 key points at the bottom hole after constraining all the key points to the top hole (as this would be attached to the wall), at the end the workplace is changed back to the global cartesian and solved for what is required to be found.

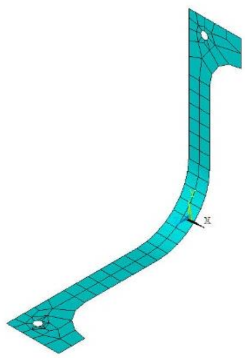

Figure 3: the meshing of half of the bracket for the the SupportMaster 2000 model showing 43 elements

3. Validity of the model

In this section, the validity of the 2000 model is discussed, along with the position of the key node and where the model has converged.

3.1 Node location, maximum deflection and stress in the Y direction at the given node

The validity of the model for the SupportMaster2000 is shown in this sub-section. The chosen node had to be in the middle of the top geometry, this is why node 43 was chosen, which had the coordinates of (-16.000,14.000,-20.000), this particular node was chosen as the maximum value changes as the mesh increases. As shown in figure 3, without starting h refinement increments, the maximum deflection is 16.55mm, the stress in the Y direction for that node is of 603.14MPa and the number of elements is 84.

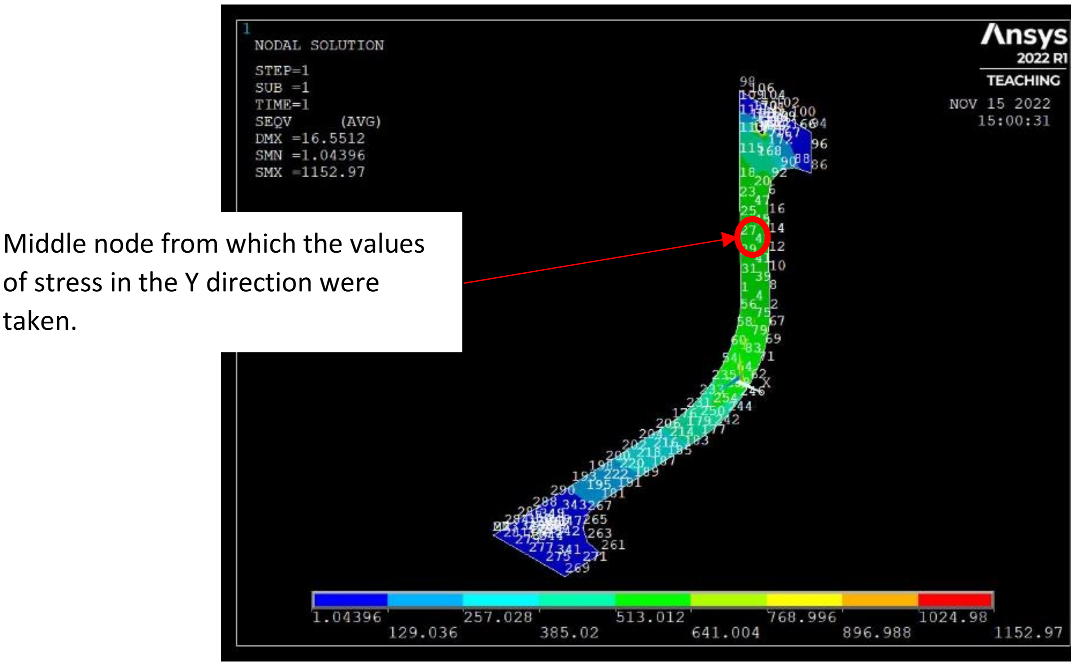

Figure 4 shows the location of the chosen node from which stress in the Y direction is taken in the SupportMaster 2000.

3.2 H-refinement and convergence

H refinement is a method used to achieve more accurate finite elements solutions that improve the mesh, giving a more accurate answer; it works by moving the nodes and decreasing the length of the elements (known as h) by dividing it into more elements. Its name, h refinement, comes from mathematics and indicates the step size in a numeric integration. Even though h refinement increases the number of elements, it does not change the type of element or its properties (Kuo and Cleghorn, 2007). This means that by increasing the number of elements and creating a finer mesh, the answer should be more accurate too. If there is a large difference between two solutions, another mesh in between must be done, very small h refinement increments have been done to find the accuracy and validity of this mesh. By multiplying the number of elements by 2 uniformly each time, using h refinement, it was found that after timing the h refinement by 4, the stress in the y direction does not change and the maximum displacement changes get smaller as the h refinement increases, as shown in Table 2.

Table 2: This table helps to localise the coordinates of the same section in each mesh by looking at the node in each refinement at a specific location, then it shows how much the elements were multiplied by (h-refinement increments of 2), the total number of elements per each mesh, the maximum

displacement for that mesh and the stress in the Y direction for the chosen coordinates (16.000, -14.000,20.000).

Node number in each model for coordinates of (-16.000,14.000,-20.000)	H refinement increments	Number of elements	Maximum displacement (mm)	Stress in the Y direction (MPa)
43	1	86	16.55	603.14
121	2	374	16.61	603.11
385	4	1476	16.72	603.11
1345	8	5850	16.81	603.11
4993	16	22376	16.90	603.11
91148	32	91148	16.99	603.11

Table 2: table showing how more elements were added for the Support Master 2000. It indicates the location of the node, h refinement increments, and the number of elements along with maximum displacement and stress in the Y direction at the given coordinates.

The data in table 2 has been plotted in table 3 showing the maximum deflection on the left axis, the stress in the Y direction on the right y-axis and the number of elements on the horizontal axis. It can be seen that in between increments of 8 and 16, the trendline and the values tend to coincide for the maximum deflection showing it is proportionally increasing in small quantities. The deflection values as the number of elements increases fluctuate as it can be noticed that between 86 and 374 elements, the maximum deflection changes by 0.0612, then 0.1087, 0.0854, 0.0973, 0.0924mm, therefore it can be observed that the difference between values diminishes at the first conversion, then increases and then diminishes again, however, it is a very small change. From looking at table 3, it can be deduced that the maximum deflection has converged but it will keep fluctuating in small quantities even after 91148 elements while the stress in the Y direction converged at 374 elements and does not change. This fluctuation of values is due to two reasons: the first one is that what is calculated is the maximum deflection and not the deflection at a specific point, therefore it will fluctuate as it is an average. The second reason is that Ansys makes small deflection approximations, and these can impact the results slightly. The deflection is shown as a straight line because of the very small range of values shown in the graph, a better view of them is shown in table 5.

Table 3: maximum deflection has been plotted against the number of elements on the left side and stress in the Y direction against the number of elements on the right. The point where stress seems to converge has been highlighted in red.

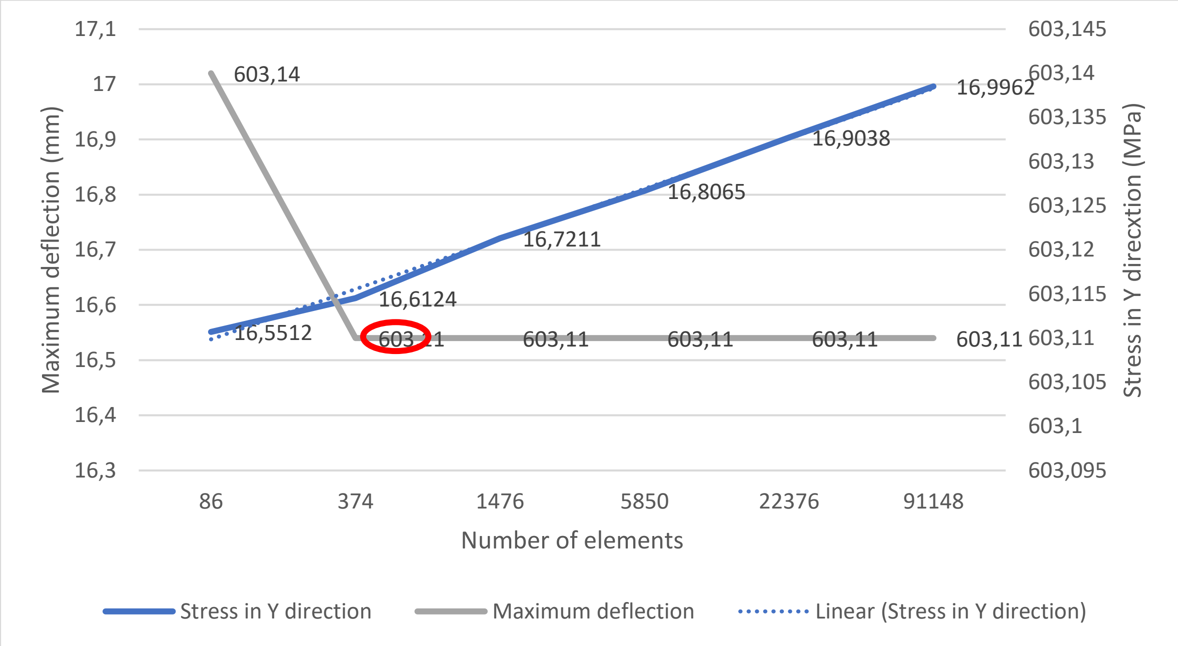

Table 3: in the y-left axis, values for maximum deflection are shown in mm, while in the left y=axis, values for stress in the Y direction (MPa) are shown. Both values are plotted against the number of elements. Highlighted and circled in red is the point where the model has converged.

It can be seen, in table 4, that the deflection converged even though values fluctuate in small amounts as the difference between each mesh in the maximum deflection is less than 0.65%. All the maximum deflection decimals available have been used to obtain a more accurate answer.

Table 4: method to find the part where the deflection has converged by finding where the error percentage is lower in the 2000 model.

Range of number of elements for section analysed	Difference percentage between deflections of section (%)
86-374	0.37
374-1476	0.65
1476-5850	0.51
5850-22376	0.57
32376-91148	0.54

Table 4 shows the range of elements for each section and the percentage difference. Deflection for the 2000 model was found to be at its smallest between 385 and 1354 elements.

To be able to say that the error difference is acceptable, it is important to find the tolerable error for this model. A tolerable error is defined as the maximum acceptable error when examining a population and helps to determine the accuracy when evaluating the computed model's maximum deflection. An error of or below 1% usually indicates that the value obtained is near the correct value (Thakur, 2019).

If we look at table 4, it can be seen that the difference between 16.7211 mm and 16.8065 mm is 0.51%, which is the smallest difference present in maximum deflection, therefore it can be said that the model has converged at 1476 elements at an h refinement increment of 4 where stress in the Y direction is of 603.11MPa and deflection of 16.7211 mm (outlined in green in table 5), as it falls within an acceptable difference of values.

Table 5: deflection by itself is plotted against the number of elements. Highlighted in green is the region where the deflection is converged which has an h refinement of 4.

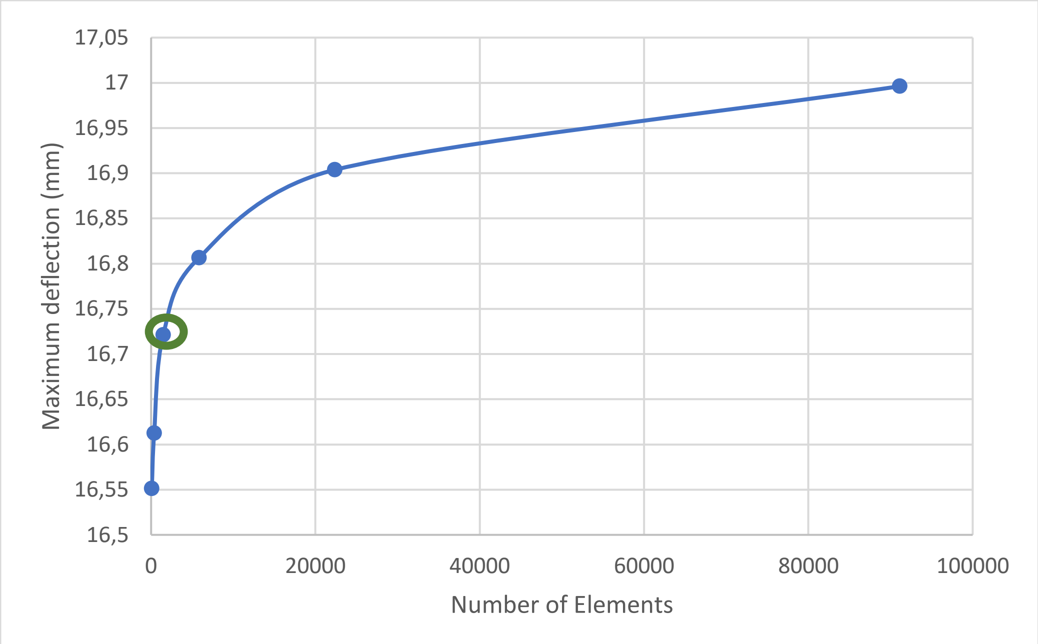

Table 5 shows the deflection (mm) against the number of elements for the 2000 model and highlights in green the exact location of the deflection conversion.

Therefore, the model has converged at 1476 elements, as shown in the mesh in figure 5, and this will be the number of elements used in further modelling to find where the model has converged.

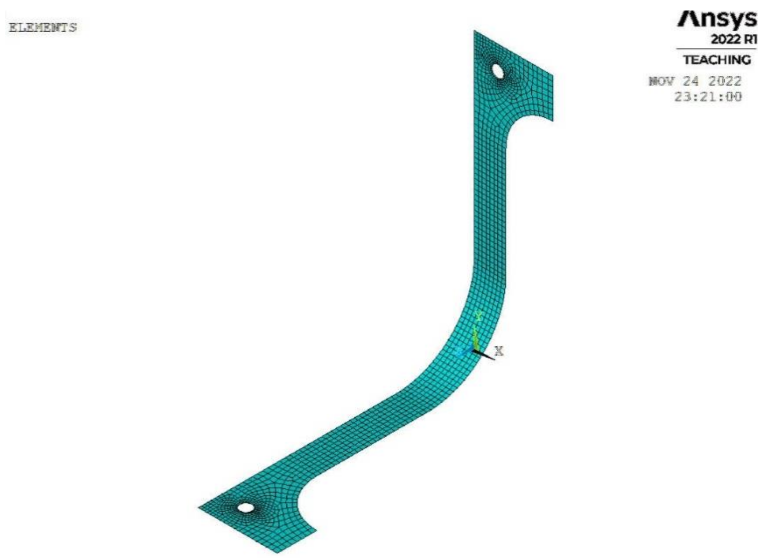

Figure 5: mesh of the Support master 2000 at 1476 elements where it has converged.

3.3 Comparison with theoretical values and further modelling

In this sub-section comparison between theoretical and computed values will be discussed.

3.31 deflection results comparison

The computed answer for stress in the Y direction is of 603.11MPa and for maximum deflection is 16.72 mm. To make sure that the computed answer is accurate, it is important to compare it with the theoretical values, which are 603.13Mpa for maximum stress in the Y direction and 11.88mm for maximum deflection. The answers obtained in Ansys are likely to be accurate: it is important to notice that there are going to be small differences between the theoretical values and the ones obtained through the model as the model uses the finite element method to obtain the answer for the given boundary conditions and makes approximations, rather than an equation, which uses small displacements assumptions. When comparing the theoretical and computed values, it can be noticed that absolute error values are 0.00331% for stress in the Y direction and 4.84mm (40.70% increase) difference for the maximum deflection. If we compare how the theoretical and computed answer is obtained, a few things can be noticed by looking at figure 6: L3=30mm in the theoretical values used for the distance between the start of the curvature and the part where the bracket has been constrained (line DC) to the wall as shown above point D. However, in the computed values this distance is of 45.5mm [50mm – (2*D1)+(D1/2)], where D1 =3.

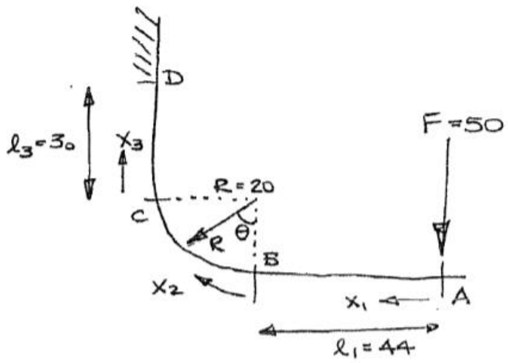

Figure 6 shows how the theoretical values for displacement were calculated and what values for lengths, R1 and load were used and where (Raimes, 2022).

In the theoretical calculation, the force F has been applied at point A, at 44mm from the curvature (as shown in section AB of figure 6), however for accuracy, in the computed value, it has been applied to the lower holes key point which is at the same distance as the point where the model has been constrained, therefore AB and CD are both 45.5mm, as shown in figure 7, due to the model being created based on symmetry.

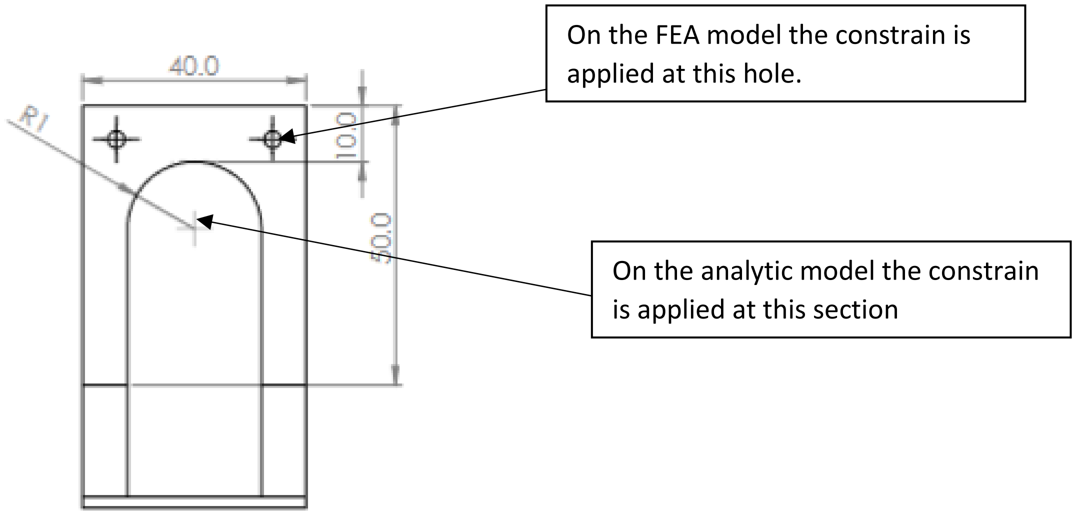

Figure 7 shows how the model was designed on Ansys and the main difference between the theoretical and computed deflection.

If the calculations are repeated but with l_1 and l_3 as L= 45.5, while keeping F=50N, R=20N, b=8mm and d=2mm while E = 200$* \ 10^9$, the maximum deflection would be 15.76 mm as shown in equation 1, created with the use of equation 2.

$$Deflection\ (mm) = \frac{1}{EI}\left(\left(\frac{FL^3}{3}\right) + \left(\frac{FRL^2\pi}{2} + \ 2LR^2F + \ \frac{FR^3\pi}{4}\right) + FL(L+R)^2\right) = 15.76 * 10^{-3} \ \pm 0.01$$

m

Equation 1 shows how to find the theoretical deflection (Adapted from Raimes, 2022).

$$I = \frac{bd^3}{12}$$

Equation 2 shows how to find the second moment of area (Raimes, 2022).

If we compare these theoretical values to the one computed which is 16.72mm, it can be observed that there is a difference of only 5.740% which is way less than the previous difference. This shows how important is to apply the geometric constraints at the same location in both analytical and computed answers. Differences between theoretical and calculated values for displacement could be also due to using the Euler beam to find the theoretical values and therefore a linear model rather than a nonlinear model in FEA, a better comparison would have been given by using an element that follows Euler's beam theory. As the number of elements increases, the FEA model gets more accurate but as it measures the maximum deflection of the beam and not at a specific point the values will keep fluctuating by very small amounts, which can be considered negligible as it is less than 0.65% error difference between the deflection values obtained.

Errors in the FEA model also diminish with small gradual element increments due to errors from the approximation decreasing. It can also be noticed, by looking at table 5, that the deflection values follow the trendline in a straight line and then change to a curve showing a plastic deformation region. A more defined mesh could have slightly improved the solution given for deflection, but further meshing has been avoided as the refined mesh would have not given enough change, it would have been negligible. The transverse shear stress is negligible due to the model being thin. In further modelling, elements above 1476 should be enough to show where the convergence happened, therefore an h refinement increments of 4 (timing each element line by 4). As shell 281 is for thin to thick-moderate shell structures (8-node quadrilateral finite strain shell), it seems to be the most adequate in this case.

3.32 Stress results comparison

It is also important to compare the stress computed with the calculations. The theoretical calculations were found by assuming that the force is applied at the centre of the bottom node of the bottom section of the bracket, as shown in figure 8, this causes two reaction forces at point A. The first one is the momentum, the second one defines if the force is compressive or tensile. Euler-Bernoulli beam theory was applied to find these reaction forces. For both theoretical and computed models, t=2, R1=12 and D=3 and the length, d, of the model is 44mm and the furthest distance from the centroid of 1mm, these values can be used as shown in equation 3:

$$\frac{My}{I} + \frac{F}{A} = 603.13\text{Mpa.}$$

Equation 3 showing how the theoretical stress for the 2000 model has been calculated

Therefore, as the stress for both analytical and computed values have been calculated with the same values and constraints, as shown in figure 8, their answer should be the same, this not only confirms the accuracy of the theoretical calculations but further proves the validity of the model due to a difference between the theoretical and computed values of only 0.003%.

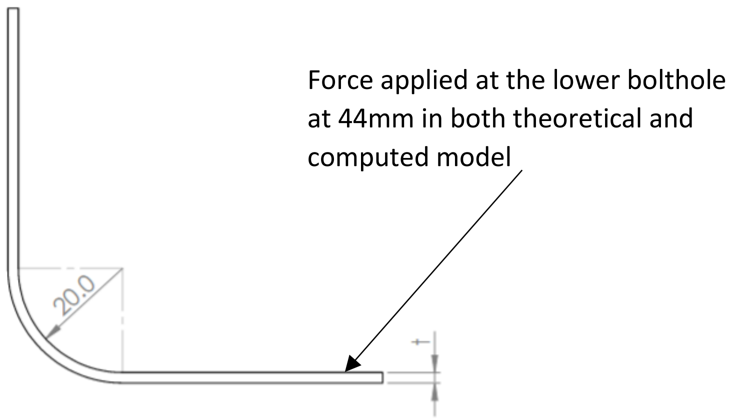

Figure 8 shows there are no differences between the way the theoretical and computed geometry, load application etc

4. Performance of the SupportMaster3000

In this model, after h refinement of 4, as previously advised, displacement is 26.923mm and stress in the y direction for the same location as for the SupportMaster2000 is 995.00 MPa (node number 385, which has the same position as for the one chosen for the 2000 model). Values for stress and deflection are very different to the support 2000 as R1 values are increased (to 15mm) and D1 values decreased (to 2mm). While the 2000 model has values of 603.13MPa, this one has an increment of 64.00% for stress in the Y direction and a 61.00% increment in deflection. The contour plot parameters of figure 5 have been adjusted to show a greater colour scale and visualise the maximum and minimum stress in the model, as shown in figure 9.

Figure 9: nodal solution for stress in the Y direction for the Support Master 3000, this model also shows the value of the maximum deflection obtained at h refinement of 4.

The mesh was applied until the recommended number of elements was reached, in this case, is 1450; very near the value advised of 1476 elements. This was done by h refinement increments of 4 as shown in the mesh in figure 10.

Figure 10: mesh at h refinement increments of 4 showing model converged and with 1450 elements for the SupportMaster 3000.

5. Details of a design variant

A design variant was created by changing R1 and thickness, T1, values where R1 = 12.5 and T1=2.5, this new variant was created to have a deflection smaller than 11.88mm. In this case, it was 9.12mm, as shown in figure 11. This shows that when R1 and T1 are increased, the deflection decreases and vice versa. R1 and T1 increased by only 1mm from the 2000 model while D1 was kept the same (value of 3), however, the new deflection compared to the 2000 model is 45.45% smaller and 66.13% smaller than the 3000 model.

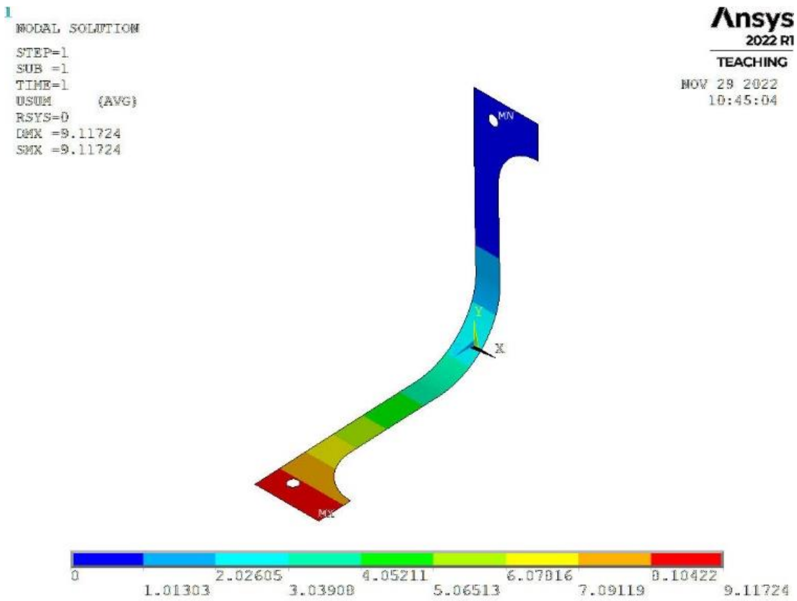

Figure 11: nodal solution for stress in the Y direction showing the range of stresses and maximum displacement for the new variant created by changing parameters R1 and t1.

This new variant was also meshed after doing h refinement increments of 4 to make sure it has converged, as shown in figure 12, the number of elements was 1502, near the number of elements previously advised of 1476.

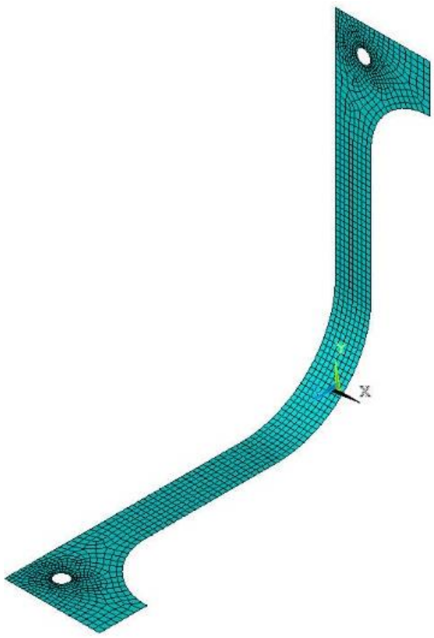

Figure 12: meshed and converged model of the new variant showing 1502 elements

6. Recommendations

In this section results analysed previously will be broken down a bit more in debt and recommendations for further models of the bracket will be made.

6.1 Support Master 2000 and 3000 ability to carry the load.

When trying to determine the most adequate bracket model, it is fundamental to clarify that the most adequate one is the one with the lowest deflection. Deflection refers to the movement of a node from its original position due to forces being applied to it at a specific location (Liang, 2019), in this case, we have a cantilever beam and the force is applied at the end of one extremity, the force applied to the model designed is of 50N, however, the finished version of the bracket would have a force of 100N. The force causes the beam to deflect from its original position due to bending, as it causes plastic deformation. The lower the value of deflection, the lower the risk that the beam has cracks due to the force being applied, causing excessive deflection.

Low deflection promotes the durability of the bracket as it does not create cracks that can compromise the structural integrity of the model, efficiency or its appearance. Very high deflection can cause the model to fail due to crack propagation which can mean catastrophic failure (sudden and not expected failure), failure also helps to identify if the beam has been overloaded or is under fatigue (van Aken et al., 2008). Furthermore, very high deflection values indicate that the stiffness and rigidity of the model need to be increased. To find if the 2000 and 3000 models can hold the weight, it is important to find the maximum allowable beam deflection for a cantilever beam. For cantilever steel beams, the general deflection limit, created to prevent excessive cracking of a material that could lead to failure, is L/180 (CVL engineers, 2020), which in this case is 28.00mm and higher than the deflection obtained of 16.72 mm for the 2000 and 26.93mm for the 3000 model and shows that both models can hold the force without cracks that leading to immediate failure.

The 3000 model however is very near these values showing that it will likely fail and will not be able to hold the force. This does not mean that they are not likely to have cracks that can lead to fatigue and then failure as the model is already within a plastic deformation region, but that the risk of catastrophic failure is very low. Another way to determine if both models can hold the force is to find the factor of safety. The factor of safety is given by the ultimate load strength divided by the allowable load stress. The model has been created with A36 steel and its ultimate tensile strength, which is the maximum stress a material can withstand while stretching before breaking, 400-550 MPa (MatWeb, 2022) and the computed stresses in the Y direction at node (-16,14,-20) is 603.11 MPa and 995.0 MPa respectively for the 2000 and 3000 model. A factor of safety of 1.3 to 1.5 should be used when the material is highly reliable (Engineering Toolbox, 2010), loading and environmental conditions are not severe and weight has to be considered. To find if the load before failure is higher than 1 and therefore reliable or lower and therefore not reliable, equation 4 has been used (Engineering Toolbox, 2010).

$$Load\ before\ failure = \frac{Ultimate\ tensile\ strenght}{\text{Stress created} * \text{safety factor}} - 1$$

Equation 4 showing how to calculate the load before failure

The cross-sectional area is of 88 * 10^-3 (due to being the distance between the force and the start of the beam), and the chosen safety factor of 1.5, therefore the load before failure is -0.39 for the 2000, which is less than 1 and therefore likely to fail, and -0.56 for the 3000 showing that both models would fail if the material had an ultimate tensile strength of 550MPa. This shows that the ultimate tensile strength needed is much higher for the 3000 model and therefore, it is not adequate. The 2000 model is not adequate either but improvements can be made to make it reach a safety factor of 1 (currently for the 2000 the safety factor is 0.91, and 3000 model 0.55), which is the minimum to avoid large cracks.

6.2 Explanation of which model is more adequate

Even though it has been explained why both models cannot be used due to them being likely to fail when the load is applied, it is still important to examine the main differences between the two models if the ultimate tensile strength was higher due to the material being changed. The main difference between the 2000 and 3000 models is the hole diameter, when decreasing the diameter of the holes, the bracket will not be constrained as efficiently to the wall and will be weaker and therefore the deflection will be larger due to more stress concentration being applied there and smaller bolts holding the bracket. It is also important to consider how much R1 influences the deformation of the model, in fact as that increases, the deflection increases too. Now to find out which model is better, it is important to consider two factors, deflection and stress. It was already explained that the lower the deflection, the more durable the structure and the less is likely to crack or reach catastrophic failure etc, but it is also important to define what low stress indicates. Stress is the ratio of the external forces applied to a material against its cross-sectional area and when the stress exceeds some of the material limits, it will cause permanent deformations to the structure, such as plastic flow, fracture, impact the atomical structure of the material, causing dislocations, it is known that ductile materials can sustain high plastic deformation before failure (Holmes, 2013). The 2000 model as it has a 16.72mm deflection compared to the 26.92mm of the 3000 model, therefore its deflection is 61.00% less than the 3000 model, it is a better option. As half of the section has been used, the stress of 603.11 MPa has been created when a load of 50N is applied, this is less than the theoretically calculated stress of the 2000 model of 603.13MPa but within its range showing that the 2000 model is behaving as expected, the second model in comparison is not as suitable due to the higher deflection and stress of 995.00MPa on the same middle node at the top section of the bracket but, as proved before, both cannot hold the force of 100N and the 2000 is just more suitable due to lower deflection and stress.

6.3 Validity of the model

Two things are important to verify: confirm the model has been structured correctly and verify that the results are reasonable.

6.31 Model Construction

To make sure the model has been constructed correctly, it was important to check a few things before solving it. These things are shown in this section. JustHanging.Ltd wants to check if the two bracket models would be able to hold a force of 100N, even though it has been shown that they cannot, it is important to check the validity of the model to make sure the accuracy and therefore statements made are correct. The load must be applied to the bottom loads in the 4 key points around the lower hole equally as the weight needs to be distributed, as shown in figure 13. This shows that the load has been applied correctly on the 4 lower key points, which are 23,24,25 and 26.

```
LIST SELECTED POINT LOADS ON ALL SELECTED KEYPOINTS
CURRENTLY SELECTED LOAD SET= FX   FY   FZ   MX   MY   MZ

 KEYPOINT      LOAD LAB        VALUE(S)
     23       FY              -12.500          0.0000
     24       FY              -12.500          0.0000
     25       FY              -12.500          0.0000
     26       FY              -12.500          0.0000
```

Figure 13 showing in which 4 key points the load of 12.5N has been applied, it can be noticed it has been applied uniformly, and that it has been applied in the bottom bracket nodes and that is pointing down

The model had also to be constrained at the top 4 key points around the hole, this was done to replicate the bracket attached to the wall, and therefore displacement = 0. These constraints were set so that the 4 key points would have a displacement of 0 in all directions (x,y,z). These key points were 2,5,8 and 9 as shown in figure 14. This also shows the model has sufficient constraints.

LIST SELECTED DOF CONSTRAINTS ON ALL SELECTED KEYPOINTS
CURRENTLY SELECTED DOF SET= UX UY UZ ROTX ROTY ROTZ

KEYPOINT	LOAD LABEL	VALUE(S)		EXP KEY
2	UX	0.0000	0.0000	0
2	UY	0.0000	0.0000	0
2	UZ	0.0000	0.0000	0
2	ROTX	0.0000	0.0000	0
2	ROTY	0.0000	0.0000	0
2	ROTZ	0.0000	0.0000	0
5	UX	0.0000	0.0000	0
5	UY	0.0000	0.0000	0
5	UZ	0.0000	0.0000	0
5	ROTX	0.0000	0.0000	0
5	ROTY	0.0000	0.0000	0
5	ROTZ	0.0000	0.0000	0
8	UX	0.0000	0.0000	0
8	UY	0.0000	0.0000	0
8	UZ	0.0000	0.0000	0
8	ROTX	0.0000	0.0000	0
8	ROTY	0.0000	0.0000	0
8	ROTZ	0.0000	0.0000	0
9	UX	0.0000	0.0000	0
9	UY	0.0000	0.0000	0
KEYPOINT	LOAD LABEL	VALUE(S)		EXP KEY
9	UZ	0.0000	0.0000	0
9	ROTX	0.0000	0.0000	0
9	ROTY	0.0000	0.0000	0
9	ROTZ	0.0000	0.0000	0

Figure 14 showing which 4 key points were constrained to simulate real life constraints of the bracket

After setting the material to be Shell281, it was also important to set its Young's Modulus to 200Gpa and Poisson's ratio to 0.3, as shown in figure 15. This helps the software to define the material properties of A36 steel and solve the model as if it was made of A36 steel.

```
EVALUATE MATERIAL PROPERTIES FOR MATERIALS      1 TO      1 IN INCREMENTS OF      1

MATERIAL NUMBER =      1 EVALUATED AT TEMPERATURE OF   0.0000
 EX   =  0.20000E+06
 NUXY =  0.30000
 PRXY =  0.30000
```

Figure 15 showing values of Poisson ratio and Young's Modulus for the model designed

It was already mentioned in section 3, the accuracy of the mesh through h refinement, however, it is also important to check it from a visual side. As three parts have been glued together, it is important to check the mesh within that region and make sure it is a smooth transition. When the model was created, it was divided into three sections that were glued together: these were the top rectangular section, the bottom and the longer one, as shown in figure 16.

Figure 16 showing the model divided into the three glued sections

If we look at figure 17, we can see it has meshed correctly as we can see a smooth transition of the mesh, it is hard to see where the three geometries were glued together just by looking at the mesh, it can also be noticed by knowing that if elements are not joined efficiently, such as not being joined at the middle side node, there would have been a gap in between nodes due to elements deforming differently from how they should have. Also, it can be seen geometrical simplifications are microscopical, for example from looking at the curvature, it is hard to tell if it is a series of quadratic elements joined together to form a curve. Quadratic elements carry an error smaller than 1% due to the angle of the curve being less than 45%, making this error negligible.

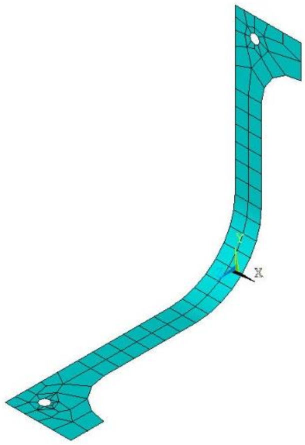

Figure 17 showing the mesh after glueing all the components together

It is also important to notice that if any of these parts created during the pre-processing phase were not made correctly, an error would have popped out in most cases and its absence further proves the validity of the model.

6.32 Evaluation of solution

When evaluating the solution, a few things can be checked. As the model's top 4 key points are constrained to the wall and a force is applied to the bottom 4 key points hole, it is expected to see the highest stress in the Y direction around the 4 top key points, due to the bolt passing through (stress tends to concentrate around holes due to abrupt change in geometry) and on the top middle section of the bracket due to the material in that section and the reaction forces trying to hold the weight of the force applied around the lower key point and the weight of the lower bracket section, therefore being the first section to 'feel' the weight of the lower section, this will cause it to show the stress concentration in that section, as it is also shown in figure 18 proving the model is behaving as expected.

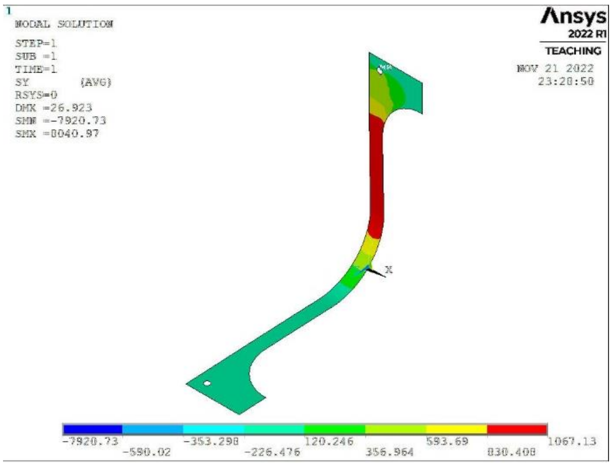

Figure 18 showing the stress distribution and concentration in the Y direction

For cantilever beams, such as the one modelled for this report, the maximum deflection will occur where the load is applied at the end of the beam, in this case, is around the lower key point, as shown in figure 19. It is important to notice that deflection is caused due to two loads, the weight of the beam which in this case is counted as applied evenly distributed load which causes maximum deflection at the end of the beam, and the applied load. When plotting graphs for stress and deflection, these statements are supported by showing that they have been modelled correctly.

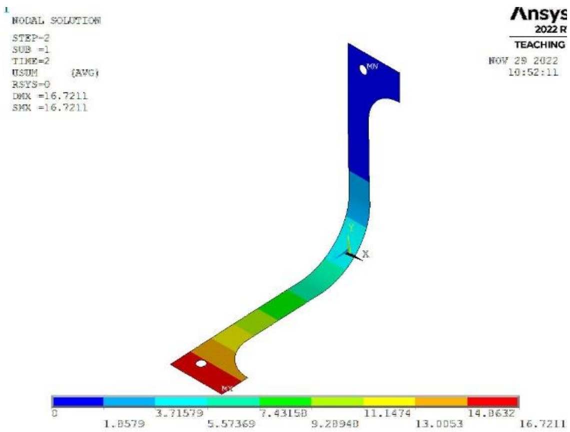

Figure 19 showing the deflection distribution and concentration

Another method to verify if the model has been created correctly is by comparing it to the theoretical values. Theoretical values for stress for the 2000 model were of 603.13MPa, and for the 3000 model were calculated to be of 994.1MPa. The 3000 model stress was calculated by using equation 5:

$$\frac{\mathrm{My}}{\mathrm{I}} + \frac{\mathrm{F}}{\mathrm{A}} = 994.1\text{Mpa.}$$

Equation 5: 3000 model stress equation

Where M is the moment, I the second moment of area, F is the force applied and A is the cross-sectional area, the main difference from the 2000 model is that in the 3000 model: d=2 and R1=15. These two parameters increase the momentum and decrease the second moment of area and the cross-sectional area causing the stress in the Y direction to increase. However, it can be noticed that differences between analytical and computed stress in the Y direction are 0.09% for the 3000 model and 0.003% for the 2000 model. When examining deflection, small differences have been found between computed and analytical values as to where the constraint and load were applied. These differences, as shown in section 3.3, where found to be that the bracket was not symmetrical, the length of the top of the bracket to the top bolt hole was smaller than the one for the bottom part of the bracket and the model was constrained, at the centre of R1 rather than the key point hole. This was rectified by making both bracket distances equal as they were modelled passing on symmetry and constraining the top bracket to the wall at the bolt hole rather than the centre of R1 (the curvature of the top section). After rectifying them, as shown in equations 6-7-8, it was found that the theoretical deflection for the 2000 model is $15.76 * 10^{-3}$, and for the 3000 model by using the same formula with the different parameters shown before is of:

$$Deflection\ (mm) = \frac{1}{EI}(\left(\frac{FL^3}{3}\right) + \left(\frac{FRL^2\pi}{2} + 2LR^2F + \frac{FR^3\pi}{4}\right) + FL(L+R)^2)$$

Equation 6: deflection equation to find the 3000 model

$$I = \frac{bd^3}{12}$$

Equation 7: second moment of inertia

$$Deflection\ (mm) = \frac{1}{EI}\left((1.57 * 10^6) + (5.39 * 10^6) + 9.76x10^6\right) = 25.10 * 10^{-3} \pm 0.01 \text{ m}$$

Equation 8: deflection equation to find the 3000 model

The increment in stress for the 3000 model is caused by the hole being smaller and therefore concentrating more stress, and the width of the bracket being smaller causing stress to be distributed at higher amounts per section as the cross-sectional area is smaller. This also causes deflection to be higher as the same load is applied to a smaller cross-sectional area. Now if we compare these theoretical values to the computed ones for both the 2000 and 3000 models, we will obtain a difference of 5.74% and 7.2 % respectively showing both the validity and accuracy of the results. These modelled deflection values are within a reasonable value as was previously mentioned in section 3.3, and they are not higher than 0.28mm (L/180), which indicates that the cracks are not going to be leading to failure but also shows that there will be some cracking in the model and it is weak. Due to calculating the theoretical values by using linear equations, the FEA model, which uses nonlinear equations and deformations, will have slight differences from the theoretical values, however, due to the bracket being thin, these changes should not excessively impact the overall design and transverse shear strain can be ignored too. The mesh converging in section 3.2, as shown in table 3, proves that the model has been plotted correctly. The mesh is shown to be converged because, as previously explained, values for stress in the Y direction are stable and values for deflection fluctuate by such small amounts it is negligible.

6.4 Comparison with the real world and FEA improvements

When manufacturing the bracket, it is also important to notice that the thickness t is very small, the tolerances are ± 1mm for linear and ± 3 degrees for angular. These tolerances are very small and it will be hard to manufacture something of this size in a workshop. It would be very hard to create holes of 2 or 3mm with tolerances of only 1mm. But also, affects the validity of the model.
Microscopic simplifications have been made when designing the quadratic element and simplifications such as creating half of the bracket based on symmetry, however, they do not affect the accuracy of the model as previously shown in section 3, however, they do slightly affect its validity.

Another simplification made in the model is that the force is applied as a point load rather than a distributed load, in real life the force could have been evenly distributed or not depending on the shape, weight and size of the load as in real life point load does not exist. For example, if the force is evenly distributed, local high-stress distribution around the lower constraint would diminish and be more distributed along with the deflection and the stress in the Y direction. Another adequate model would have been to use truss elements to model the relationship between the bracket and the load, this would have allowed us to consider all directions in which the load could have deformed and consider the transmitted tensile and compressive stresses in the region.

A36 when only 2mm thick is very hard to machine and very malleable, therefore in real life it would deform very easily. A different material or the same one but thicker would have guaranteed more durability. This has not been taken into account with shell 281 as it represents general steel and therefore ignores material properties, etc of the actual material causing validity issues.

Instead of constraining all hole key points so that they do not move in any direction, the model central node, where the constrain has to be applied, has to be constrained so that it does not move in all DOF, while the other nodes should have been constrained vertically while leaving them to deform in the horizontally and let the width of the bracket reduce as the load was applied. In better terms, the FEA model, due to fixing the top hole's key points, is constrained and therefore the bracket has been designed so that it does not move horizontally to replicate real-life scenarios. Ansys does not accurately model the fixation to the wall as it does not take account of the bolt type and quality, and in real life, the bracket is likely to shrink horizontally when applying a load of 100N to a bracket screwed to the wall depending on the material of the wall and inserts of the wall too, which are not included in the FEA analysis. To further improve validity, modelling the bolt, its size and shape, and constraining the bracket to it would have increased validity. It is also important to notice that due to the bolt passing through the keyhole, there will be friction contact between the two and between the bolt and the wall and this has to be modelled on Ansys as well to increase validity. Modelling the wall as a fixed support would have increased the validity, and the bolts could have been exported from CAD to Ansys.

The ideal bracket would need to be made out of a ductile material, as these materials are very strong in tension, even though are weaker in shear. Most brackets are made of stainless steel, brass, aluminium, and steel but all these materials have a lower or equal ultimate tensile strength than A36 steel therefore it seems like the design of the bracket needs changing rather than its material properties. Impurities in the steel from the manufacturing process can also cause differences between the FEA model and the real world due to changes in material properties.

7. Conclusion

The 2000 and 3000 models, due to having a safety factor below 1, would not be able to hold the force, however, if used, the 2000 model would be better than the 3000 due to lower stress in the Y direction and deflection. It was shown that both models converged at around 1400 elements, h refinement of 4. It was demonstrated that theoretical and calculated values match the computed values when adjustments are made. It was also shown that the validity of the model could be improved by changing the way it is constrained, in fact by constraining only the middle node at all DOF and keeping the others deforming vertically, the model would have represented real-life situations better. To further improve validity, the bolts and the attachment to the wall could have been modelled. The point load does not represent how the load would have been distributed, evenly distributing the load would have been better in terms of comparison with real-life scenarios.

8. References

CVL engineers (2020). *Maximum Span of Cantilever Beams - CVL Engineers Inc.* [online] cvl-eng.ca. Available at: https://cvl-eng.ca/max-span-of-cantilever-beams/#:~:text=The%20deflection%20limit%20for%20cantilever [Accessed 28 Nov. 2022].

Engineering Toolbox (2010). *Factors of Safety*. [online] Engineeringtoolbox.com. Available at: https://www.engineeringtoolbox.com/factors-safety-fos-d_1624.html.

Holmes (2013). *Mechanics of Materials: Stress» Mechanics of Slender Structures | Boston University*. [online] www.bu.edu. Available at: https://www.bu.edu/moss/mechanics-of-materials-stress/#:~:text=Stress%20is%20the%20a%20measure.

Kuo, Y.L. and Cleghorn, W.L. (2007). The *h*-, *p*-, and *r*-refinements of Finite Element Analysis of Flexible Slider Crank Mechanism. *Journal of Vibration and Control*, 13(4), pp.415–435. doi:10.1177/1077546307076901.

Liang, Z. (2019). *What is Deflection? | SkyCiv Cloud Structural Analysis Software*. [online] Skyciv.com. Available at: https://skyciv.com/docs/tutorials/beam-tutorials/what-is-deflection/.

MatWeb (2022). *MatWeb - The Online Materials Information Resource*. [online] www.matweb.com. Available at: https://www.matweb.com/search/DataSheet.aspx?MatGUID=afc003f4fb40465fa3df05129f0e88e6&ckck=1 [Accessed 28 Nov. 2022].

Raimes, M. (2022). *Deflection Analysis of Bracket*.

Thakur (2019). *Margin of Error Formula | Calculator (Excel Template)*. [online] EDUCBA. Available at: https://www.educba.com/margin-of-error-formula/.

van Aken, C.A.J.M., Pallav, P., Kleverlaan, C.J., Kuitert, R.B., Prahl-Andersen, B. and Feilzer, A.J. (2008). Effect of long-term repeated deflections on fatigue of preloaded superelastic nickel-titanium archwires. *American Journal of Orthodontics and Dentofacial Orthopedics: Official Publication of the American Association of Orthodontists, Its Constituent Societies, and the American Board of Orthodontics*, [online] 133(2), pp.269–276. doi:10.1016/j.ajodo.2005.10.030.