EMS602U – Simulation Tools in Engineering Analysis and Design

Finite Element Method (FEM) Lab Report

Name:	Student ID:
JATINDER DHALIWAL	220451019

Abstract

This report studies the use of Finite Element Method (FEM) to approximate the real-world reactions of a structure in response to applied physical effects. An experiment was conducted to simulate the effect of a shelf applying a uniform, distributed load on the surface of a shelf bracket by using Abaqus CAE, a FEM software that simulates structures through meshing. Results were obtained for the Von Mises stress and displacement values of four different mesh types – quadrilateral linear, triangular linear, quadrilateral quadratic and triangular quadratic. Out of these mesh types, the one that exhibited the best convergence, and therefore the most accurate simulation, was the quadrilateral linear mesh type. The use of FEM software and meshes to test and improve design are frequently used in industrial applications, highlighting the importance of the processes outlined in the report.

Introduction

The Finite Element Method (FEM) is a computational method which can be used to investigate how a structure reacts to real-world physical effects such as forces and applied loads. A FEM software works by breaking down a structure, which is under a specific load, into many finite elements containing nodes, in a process known as meshing [1]. The software can then mathematically determine the behaviour of each element by using the linear basis functions ($N_1(x), N_2(x)$ ...) that are associated with each node of an element. Theoretically, the formula that is used in FEM to approximate the behaviour of a singular element is:

$$u(x) = \sum_{j=1}^{n} u_j N_j(x)$$

In this application of FEM, the software that will be used is Abaqus CAE to analyse the behaviour of a shelf bracket, which is under a distributed load, in a 2D simulation. As the bracket is considered fixed onto a wall, there is an applied boundary condition on the vertical surface of the bracket to model this system. The load applied to the top surface of the bracket is a pressure load of 2 MPa, which is uniformly distributed along the surface. Typically, a shelf bracket is made of stainless steel, due to its ability to withstand the mechanical stresses of a shelf without significant deformation. Therefore, the properties of stainless steel will be computed into Abaqus CAE. Images of the relevant dimensions, forces, boundary conditions and material properties can be seen in Figures 1 and 2, and Table 1 below.

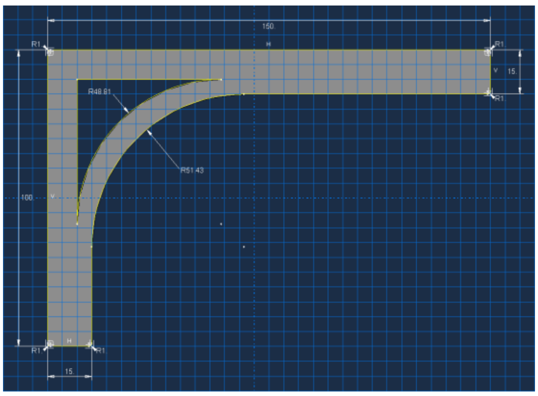

Figure 1 – Dimensions of the shelf bracket.

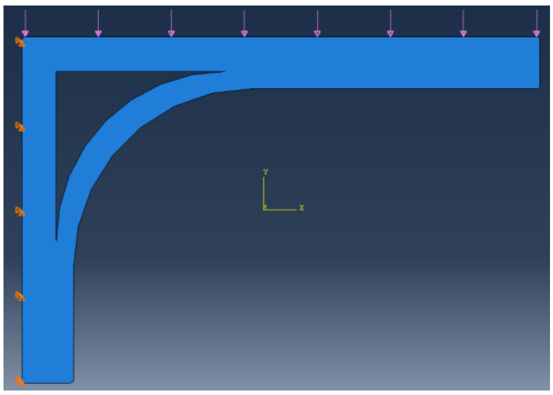

Figure 2 – Applied load and boundary condition.

Material Property	Value
Young's Modulus	200 GPa
Poisson's Ratio	0.3

Table 1 – Material Properties of Stainless Steel [2,3].

Abaqus CAE is able to create triangular and quadrilateral meshes, which can each affect the final prediction for the behaviour of the bracket. This is because the structure is broken up differently in both types, meaning that the individual behaviours of each element will vary. Therefore, the results of using both of these mesh types will be compared to evaluate the most effective measure of the behaviour of the shelf bracket. The method to evaluate this would be to first calculate the global seed size of the mesh using the following equation:

$$GSS = \frac{2\pi r}{4} \div 2$$

Once the global mesh size is obtained and the bracket has been meshed using the triangular/quadrilateral shape, the stress and displacement values, subsequent from the distributed load, can be found. After the data is recorded, the global mesh size is halved, which increases the number of elements and provides a more accurate result for the stress and displacement due to a finer mesh. This process is repeated until a suitable number of data points are collected. The Abaqus CAE license has a limit of 250,000 nodes in a structure, highlighting that there will be a threshold for how far the mesh size can be reduced. Graphs of stress convergence and displacement convergence are then plotted using these recorded values. Both graphs should exhibit a convergence as the mesh gets finer, since there is a smaller decrease in the global mesh size after each iteration, so the output values would become closer after each reduction in mesh size, and eventually converge to a certain value. The convergences of the triangular and quadrilateral mesh types can be compared to each other, and the result with the most prominent convergence can be deemed the more suitable mesh type to use.

However, another parameter can be implemented; the polynomial order of the mesh types. Linear order elements have nodes only on the corners of each element, whereas quadratic order elements also have nodes halfway along each edge of an element. This implies that fewer quadratic order elements are required to achieve accurate results due to more variation in behaviours when a force is applied [4]. Though this is generally the case, linear order elements can represent certain structures more effectively, depending on the forces and the shape of the structure that is being modelled. Therefore, these are also critical parameters that will be analysed in the results.

Results and Analysis

The global seed size was initially calculated using the equation mentioned in the Introduction. However, the resulting mesh provided extremely unreliable results that did not converge for stress or displacement. This may have been due to the complexity of the structure, with a varying range of radii at different points on the bracket, meaning Abaqus CAE could not provide a suitable mesh at this seed size for this specific model. Consequently, the initial global seed size provided automatically by Abaqus CAE, which was 7.4, was used to resolve this issue. Figures 3 and 4, shown below, represent the mesh using the seed size of 7.4, for both quadrilateral and triangular mesh types.

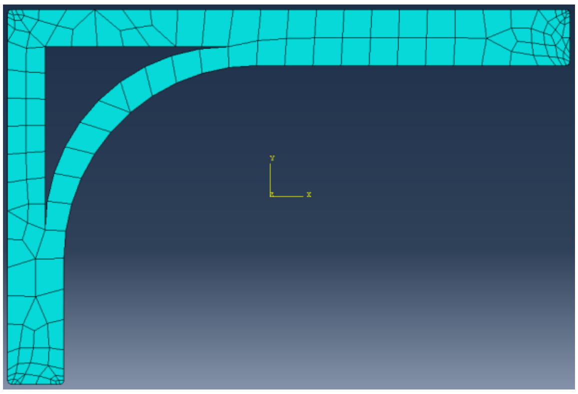

Figure 3 – Quadrilateral Mesh Type.

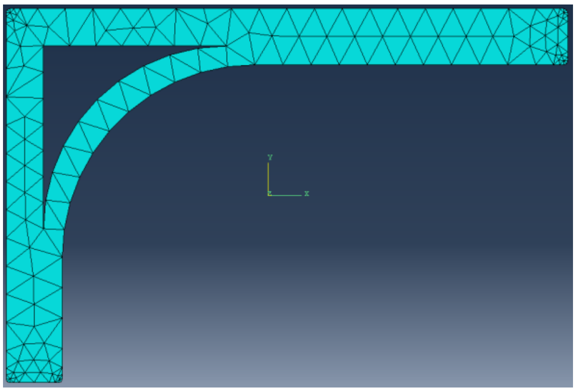

Figure 4 – Triangular Mesh Type.

Once the global seed size was determined, the maximum values of Von Mises stress and displacement were recorded. The seed size was then halved, and the stress and displacement values were recorded again. This was repeated until the seed size could not be reduced any further. Due to Abaqus CAE's node limit, the lowest seed size allowed was 0.23125, meaning that five data points of Von Mises stress and displacement were collected for each mesh type, in total. The Tables showing all of the recorded data can be found in the Appendix.

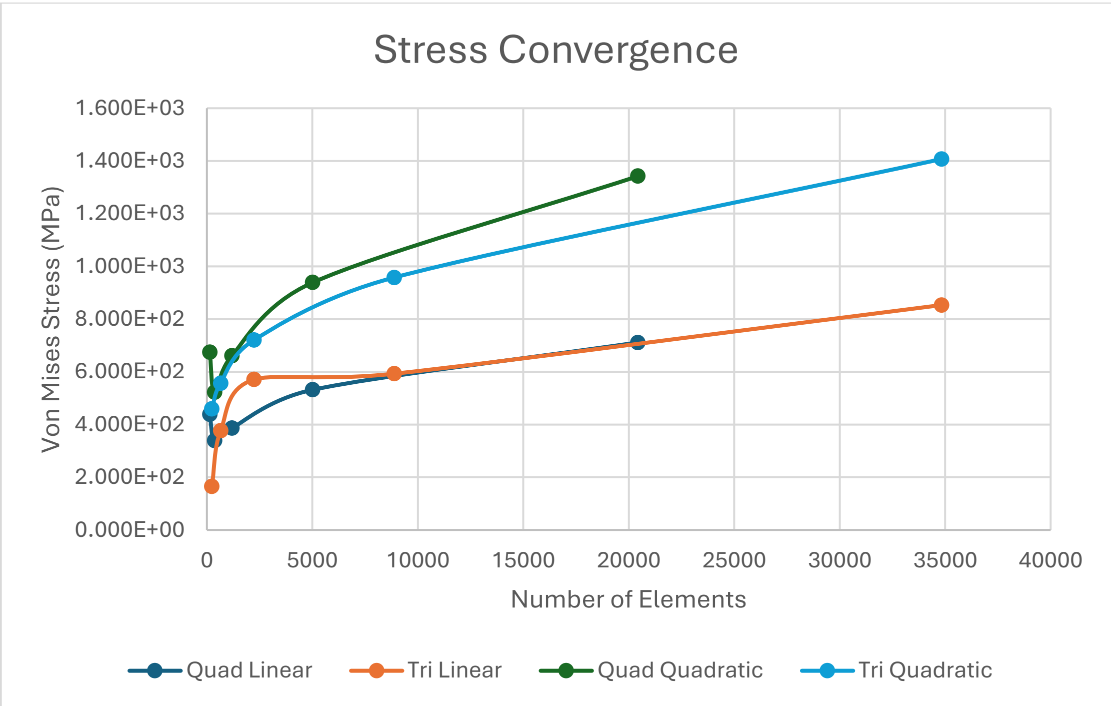

Figure 5 – Graph of Von Mises Stress Convergence for all mesh types.

Figure 5 shows the convergence of the Von Mises stress for all the mesh types. Observing the initial points on the graph, for both triangular mesh types, the line shows a smooth curve of Von Mises stress increasing from data point 1. However, for both quadrilateral mesh types, there appears to be an initial drop in stress from 438.5 MPa and 674.4 MPa (data points 1) to 338.5 MPa and 522.1 MPa (data points 2) for linear and quadratic, respectively. These are followed by a smooth curve which then eventually converges. Typically, a curve that follows a consistent pattern would be preferable as there is higher clarity of a trend in the data. Therefore, it could be argued that the triangular mesh type is more suitable for this application. Though this is the case at the beginning of the graph, for the triangular linear mesh type, there is a sudden increase in the Von Mises stress from 592.8 MPa (data point 4) to 853.1 MPa (data point 5), which does not follow the ideal convergence pattern. This could be a result of the drastic increase in the number of elements when the seed size was halved, which may have deviated the stress value from the expected convergence. This occurrence does not show in the triangular quadratic mesh type, so this value of Von Mises stress may have resulted from an error when sketching the shelf bracket in Abaqus CAE.

When comparing the overall trend of the quadrilateral and triangular mesh types, it can be seen that in both instances that the quadrilateral mesh type was used, the value for Von Mises stress was projected to converge a lot quicker than triangular-shaped mesh, where the values are expected to converge at a higher number of elements. Alongside this, the number of elements created by the quadrilateral mesh at each seed size is significantly less than the number of elements created by the triangular mesh, meaning that the distance between data points is reduced, implying a more reliable convergence. Based on this perspective, the quadrilateral mesh type is more favourable for approximating the shelf bracket.

Focusing on the comparison between the polynomial orders of the mesh types, the graph shows much higher values for Von Mises stress for the quadratic mesh type, in both quadrilateral- and triangular-shaped meshes. This alone does not underline the preference of one polynomial order over the other. However, looking at the quadrilateral meshes alone, the linear polynomial order is projected to converge at a lower number of elements than the quadratic polynomial order, as indicated by the shape of each line, where the linear line plateaus sooner than the quadratic line. Alternatively, the comparison between both types of triangular meshes shows that the quadratic polynomial order is more reliable, as the curve is fairly uniform for the triangular quadratic mesh type, in contrast to the triangular linear mesh type, which had the aforementioned sudden increase from data point 4 to data point 5. Therefore, both polynomial orders are advantageous depending on the shape of the mesh used.

Weighing all the comparisons, the mesh type that is most suitable for simulating the Von Mises stress on the shelf bracket is the quadrilateral linear mesh type. This is because the convergence of this line is projected to occur at a lower number of elements relative to the other mesh types. Though there is an initial drop in stress unlike the triangular-shaped meshes, it is trivial as the initial global seed size used is not as accurate of a model as the following, smaller seed sizes. There is a uniform curve for the remainder of the graph, so the quadrilateral linear mesh type can be considered the most accurate mesh for the Von Mises stress in this application.

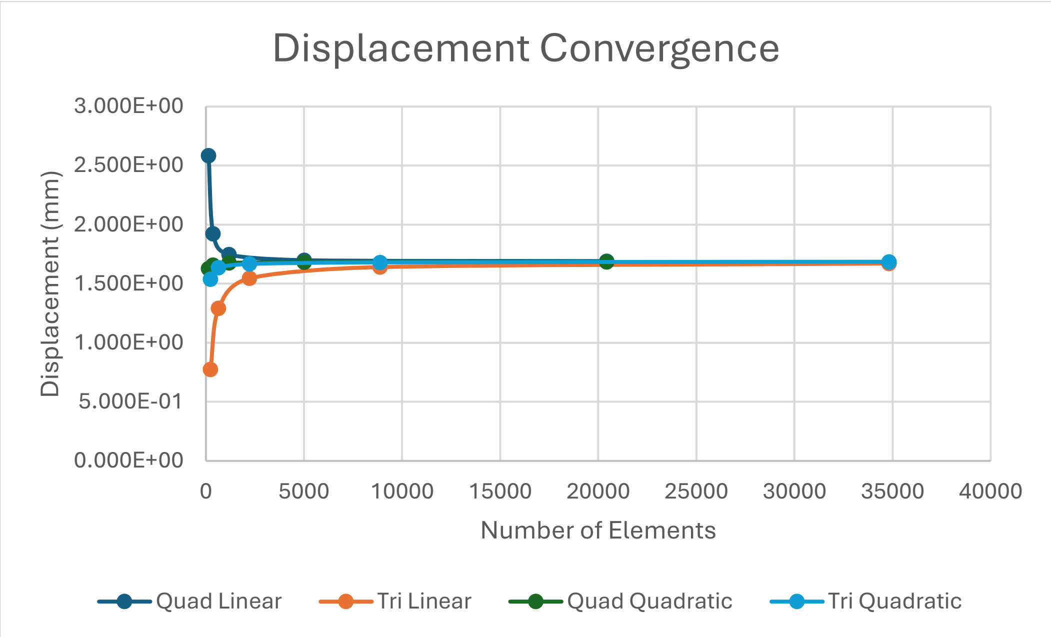

Figure 6 – Graph of Displacement Convergence for all mesh types

Figure 6 highlights the convergence of the displacement, due to the applied load on the shelf bracket, for all the mesh types. In terms of the trend across each line, there are no anomalies or instances where the expected convergence pattern has differed. However, there is a clear difference between the quadrilateral and triangular meshes, in that the quadrilateral meshes begins at a higher approximation of displacement, and converges to a lower value, whereas the triangular meshes converge from a lower initial value to a higher value of displacement. Despite this contrast, it does not distinctly determine which is the better mesh type to use.

The other major difference between the two types of mesh shapes is in the number of elements at which each of the displacement values converge. For the quadrilateral meshes, the displacement converges at 20430 elements, and for the triangular meshes, the displacement converges at 34823 elements. Again, this is due to the elements created when the seed size is halved. The quicker the convergence and the smaller the gap between each data point, the more accurately the mesh type is able to model the displacement of the shelf bracket under the applied load. Since the quadrilateral meshes are superior in both of these criteria, it can be considered the more suitable shape for the mesh in this application.

There is insignificant comparison to be made between the polynomial orders for the mesh, as the only difference between quadratic and linear is that the initial values start closer to each other for the quadrilateral quadratic and triangular quadratic meshes, than the quadrilateral linear and triangular linear meshes. Regardless, this does not determine the more accurate mesh type, as the initial data point has little impact on the rest of the graph.

Overall, the most suitable mesh type for the modelling of the displacement of the shelf bracket is tied between quadrilateral linear and quadrilateral quadratic, primarily because both graphs converge at the same point, and at a lower number of elements than the triangular mesh types.

Conclusion

This experiment aimed to simulate the real-world effect of a shelf applying a distributed load on the surface of a shelf bracket using the Finite Element Method (FEM) software, Abaqus CAE. A 2D simulation of a shelf bracket was created by studying the geometry of an average shelf bracket and computing it into Abaqus CAE. The distributed load was determined by researching the dimensions and mass of a shelf, and using these parameters to calculate the pressure of an average shelf on a shelf bracket, which came to 2 MPa. The boundary condition set in place was one that restricted any movement of the side surface of the bracket, with the intention of replicating how a bracket is fixed to a wall in a real-life application. When exploring the most common materials for shelf brackets, stainless steel was a prevalent choice due to its sufficient mechanical properties in this application. With a Young's Modulus of 200 GPa and a Poisson's Ratio of 0.3, it is able to withstand the stresses that an average shelf would apply without exhibiting significant deformation.

Abaqus CAE allows for four different mesh types in total – quadrilateral linear, triangular linear, quadrilateral quadratic and triangular quadratic. Generally, the quadratic polynomial order is ideal as a lower number of elements is required for a more accurate simulation of a real-world structure. However, in some cases, the linear polynomial order is preferable. This experiment investigated the convergence of the Von Mises stress and displacement for all the mesh types, by starting at a calculated global seed size, and halving the seed size after each time the stress and displacement values were recorded. The convergence of these values indicates an accurate model, as theoretically, the finer the mesh becomes, the closer to the true values of Von Mises stress and displacement the graph reaches. A graph that converges to these true values with the lowest number of elements was considered the most suitable mesh type for the simulation of the shelf bracket.

This particular application of a shelf bracket was best simulated with a quadrilateral linear mesh type. In the comparison of Von Mises stress for each mesh type, there was a clear convergence pattern for the quadrilateral linear mesh type, unlike the triangular linear mesh type, which displayed a sudden increase in the latter portion of the graph, indicating a less accurate convergence. Furthermore, the quadrilateral linear mesh type was projected to converge the quickest out of the four mesh types. This evidence makes it clear that the quadrilateral linear mesh type modelled the Von Mises stress with the highest accuracy. When comparing the displacement convergences, it was tied between the quadrilateral linear and quadrilateral quadratic mesh types. The core reason for this was because the quadrilateral-shaped meshes produced a convergence at a lower number of elements than the triangular-shaped meshes, and the distance between data points was smaller, which indicates a more accurate convergence pattern. In both cases of Von Mises stress and displacement, the difference in polynomial order was relatively insignificant. As a result, the most prominent differences were seen when contrasting the shape of the mesh.

Since there was a limit on the number of nodes and, in turn, the number of elements due to the academic licensing, the recorded data may not be as accurate as the use of other FEM software or with a higher number of elements available. Therefore, provided that there is a higher amount of elements available for use in Abaqus CAE, this experiment could be performed with more accuracy in the future.

In conclusion, the use of FEM and Abaqus CAE to model the behaviour of a shelf bracket under a distributed load is best simulated by using a quadrilateral linear mesh type.

References

[1] AutoDesk (2023). Finite Element Analysis (FEA) Software | Autodesk. [online] Autodesk.com. Available at: https://www.autodesk.com/solutions/simulation/finite-element-analysis.

[2] Anon, (n.d.). Comparison of structural design in stainless steel and carbon steel – British Stainless Steel Association. [online] Available at: https://bssa.org.uk/bssa_articles/comparison-of-structural-design-in-stainless-steel-and-carbon-steel/.

[3] Instron.com. (2024). Poisson’s Ratio. [online] Available at: https://www.instron.com/en/resources/glossary/poissons-ratio.

[4] A Brief Guide to Meshing. (n.d.). Available at: https://www.lusas.com/user_area/documentation/1025_Brief%20Guide%20to%20Meshing.pdf.

Appendix

Seed Size	No. of Elements	S, Mises	Displacement
7.4	138	4.385E+02	2.585E+00
3.7	371	3.385E+02	1.924E+00
1.85	1186	3.859E+02	1.747E+00
0.925	5017	5.318E+02	1.699E+00
0.4625	20430	7.113E+02	1.691E+00

Appendix 1 – Collected Data for Quadrilateral Linear Mesh Type

Seed Size	No. of Elements	S, Mises	Displacement
7.4	235	1.648E+02	7.735E-01
3.7	652	3.763E+02	1.291E+00
1.85	2233	5.710E+02	1.546E+00
0.925	8882	5.928E+02	1.641E+00
0.4625	34823	8.531E+02	1.671E+00

Appendix 2 – Collected Data for Triangular Linear Mesh Type

Seed Size	No. of Elements	S, Mises	Displacement
7.4	138	6.744E+02	1.628E+00
3.7	371	5.221E+02	1.657E+00
1.85	1186	6.604E+02	1.676E+00
0.925	5017	9.387E+02	1.682E+00
0.4625	20430	1.342E+03	1.686E+00

Appendix 3 – Collected Data for Quadrilateral Quadratic Mesh Type

Seed Size	No. of Elements	S, Mises	Displacement
7.4	235	4.591E+02	1.539E+00
3.7	652	5.569E+02	1.637E+00
1.85	2233	7.204E+02	1.668E+00
0.925	8882	9.578E+02	1.681E+00
0.4625	34823	1.407E+03	1.686E+00

Appendix 4 – Collected Data for Triangular Quadratic Mesh Type