Davi Uliana

Homework 2 Report – Finite Element Modelling of an Aluminium Bracket

Contents

1. Introduction

The finite element method (FEM) is used to analyse the behaviour of structures under various loads. This assignment focuses on FEM for a bracket fixed at points B and C, subjected to a load at point A with horizontal and vertical components. In this study ANSYS software is utilised to model the bracket using beam, shell, and solid approaches, each providing unique insights into stress distribution and deformation characteristics.

This analysis employs a linear static simulation, wherein the material properties and geometry of the bracket remain consistent under the applied loads, and the relationship between stress and strain follows a linear pattern. The simulation assumes small deformations, ensuring that the equilibrium equations are solved without considering any changes in the structure's stiffness. The relevant equations guiding this process include Hooke's law for linear elasticity, equilibrium equations, and compatibility conditions, which together facilitate an accurate prediction of the bracket's response under the specified loading conditions.

The objectives of this analysis are to:

- Compute the displacements, von Mises stress, normal stress distribution, and shear stress distribution using the beam model.
- Perform a sensitivity analysis with the shell model to determine the optimal mesh and plot the stress and displacement distributions.
- Conduct a similar sensitivity analysis with the solid model and plot the stress distributions, including along a specified path DE.
- Compare the results obtained from the three models, explain the differences, and draw conclusions based on the findings.

2. Geometry and Material Properties

2.1. Geometry

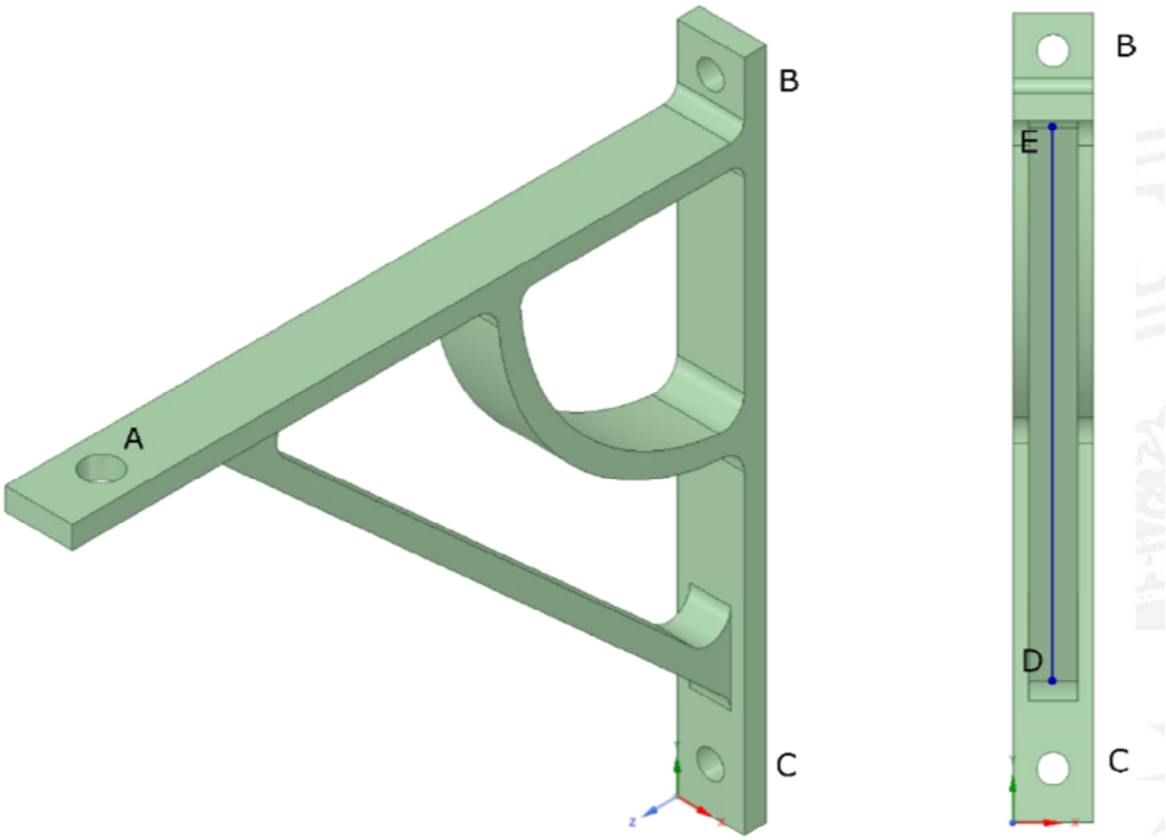

Figure 2.1.1 Bracket Geometry

2.2. Material Properties of Aluminium

Property	*Value*	*Unit*
Density	2770	Kg m^3
Young's Modulus	71	GPa
Poisson's Ratio	0.33	N/A
Tensile Yield Strength	0.28	GPa
Compressive Yield Strength	0.28	GPa
Shear Modulus	26.992	GPa

3. Model Setup

3.1. Beam Model

3.1.1. Assumptions and Simplifications

The bracket was converted from a 3D model to a simplified sketch, with cross section profiles applied to each line. The holes were represented as points to ensure precise positioning of forces and supports.

3.1.2. Cross Sections

Two cross sections have been applied to the model as per the problem statement. To obtain these, the model was generated in Ansys Discovery and subsequently imported into Ansys Mechanical. However, to obtain the solution for Stress and Shear the cross sections must be generated with Ansys Mechanical.

3.1.3. Boundary Conditions (BC)

The bracket is supported on both holes, B and C as Figure 2.1.1 indicates. To represent the indicated support a fixed support was used.
In ANSYS, applying a Cylindrical Support to a vertex is not supported (ANSYS, 2025) due to the nature of the BC and the potential for unrealistic stress concentrations, hence a fixed support was used for simulation.

3.1.4. Load Application

Similarly to the BC above, the load is applied as a force with individual components: X, Y and Z.
The Figure 3.1.1 below shows the BCs and Load applied to the Beam Model in Ansys Mechanical.

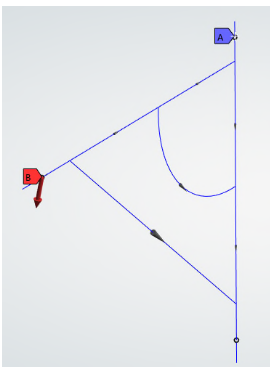

Figure 3.1.1 Beam Element Line Model

3.2. Shell Model

3.2.1. Assumptions and Simplifications

The geometry was simplified from a 3D model to surfaces. This enable for a shell analysis to be performed.
This simulation can include mode details which are not possible with line bodies used in the Beam Analysis, i.e., the holes.

3.2.2. Element Orientation

Shell Models utilise the Solution Coordinate system to plot Normal Stress and Shear Stress. Therefore, it is essential to add element orientation to the geometry in order to obtain accurate results.

3.2.3. Boundary Conditions

The BCs are similar to the Beam Model, with fixed support at holes B and C. Shell elements, although using similar BCs to beam elements, allow for a more accurate representation of thin-walled structures and complex geometries. This is crucial for models with features like holes that impact structural response significantly.
In addition to setting the BCs it is essential to also apply the necessary connections between each surface and edge. For this a bonded connection was applied to each surface and edge interaction as shown by Figure 3.2.2 below.

3.2.4. Load Application

Similarly to the BC above, the load is applied as a force with individual components: X, Y and Z.
Figure 3.2.1 below shows the BCs (blue tag) and Load (red tag) applied to the Shell Model. Please note that only one blue tag is shown, however the BC is applied to both holes at B and C position

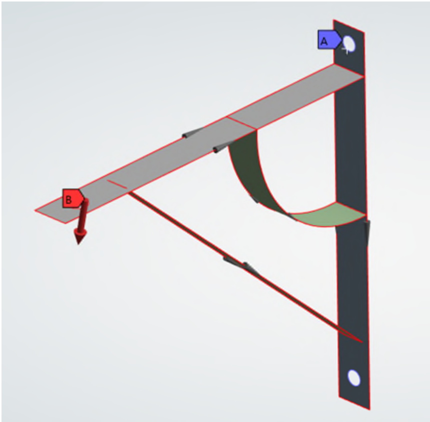

Figure 3.2.1 Shell Model

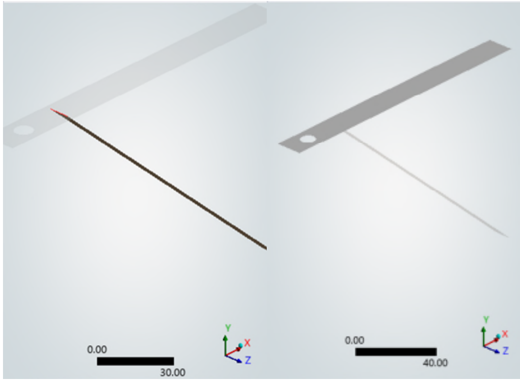

Figure 3.2.2 Bonded Contact

3.3. Solid Model

3.3.1. Assumptions and Simplifications

It has been assumed that the backed is solid one piece of metal, no welds had been considered. It also has been assumed that the holes are constraint in radial and axial directions.
No additional simplification has been done.

3.3.2. Boundary Conditions

As per the assumptions above, a cylindrical support was used as it allows for a better representation of radial and axial constrain, whereas a fixed constraint here would potentially over constrain the bracket leading to inaccurate results.

3.3.3. Load Application

Similarly to the BC above, the load is applied as a force with individual components: X, Y and Z.

4. Mesh Description and Sensitivity Analysis

4.1. Beam Model Mesh

The model was meshed using program-controlled mesh and default element sizes.
Following the first run, a parametric approach was taken to take a sweep on different element sizes. The graph Figure 4.1.1 Mesh Convergence for Beam Model below shows the Von Mises Stress vs Element Count converging at datapoint 5 (red circles) which is the 2mm element size.

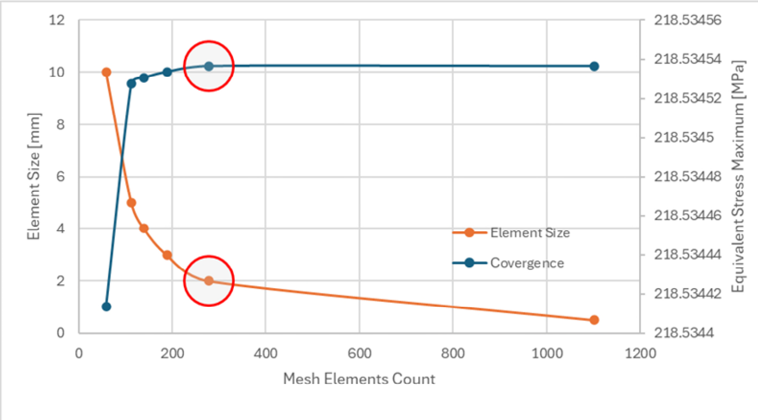

Figure 4.1.1 Mesh Convergence for Beam Model

4.2. Shell Model Mesh

To determine the optimum mesh size, a sensitivity analysis was conducted using various element sizes. Figure 4.2.1 Mesh Convergence for Shell Element presents the analysis results. It can be observed that with a 1mm element size, the stresses begin to converge, and no significant benefit is achieved by further reducing the element size.

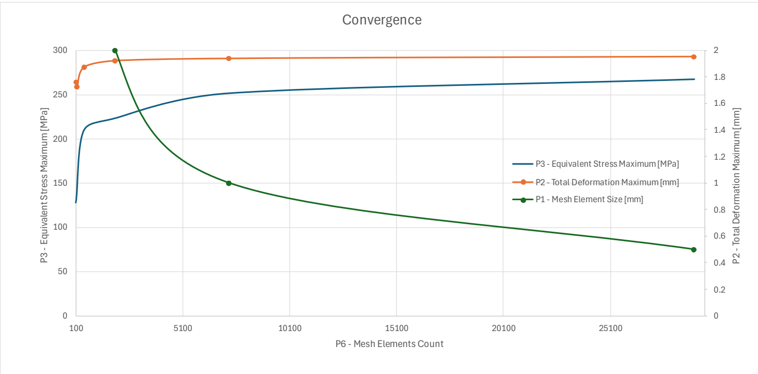

Figure 4.2.1 Mesh Convergence for Shell Element

4.3. Solid Model Mesh

The model was meshed using program-controlled mesh (quadratic) and default element sizes

Similar to the Shell model, a sensitivity analysis helped determine the optimal mesh size. Figure 4.3.1 below shows the convergence plot for von-Mises stress, indicating a stress singularity that prevents achieving convergence.

To address this, a sensitivity analysis can be performed (Figure 5.2.1). Rather than recording the maximum stress, it is recommended to average the stress in the nodes near the peak. Additionally localised mesh sizing controls were added to help the issue.

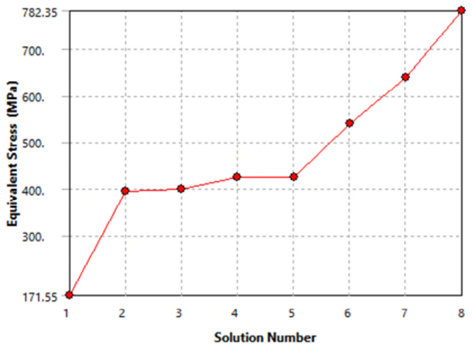

Figure 4.3.1 Solid Model Convergence Graph

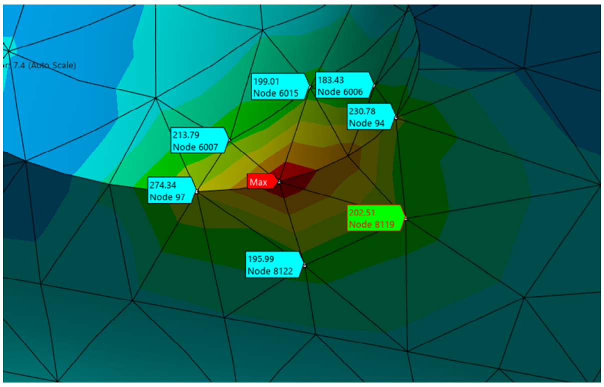

Figure 4.3.2 Solid Model near peak stress Averaging

5. Results and Discussion

5.1. Discussion

The analysis shows that resolution improves progressively from Beam to Shell and Shell to Solid. This is expected, as Beams and Shells capture lines and surfaces, while Solids capture volumes, significantly increasing the number of elements and processing time. The significant increase in element size leads to longer computing times.

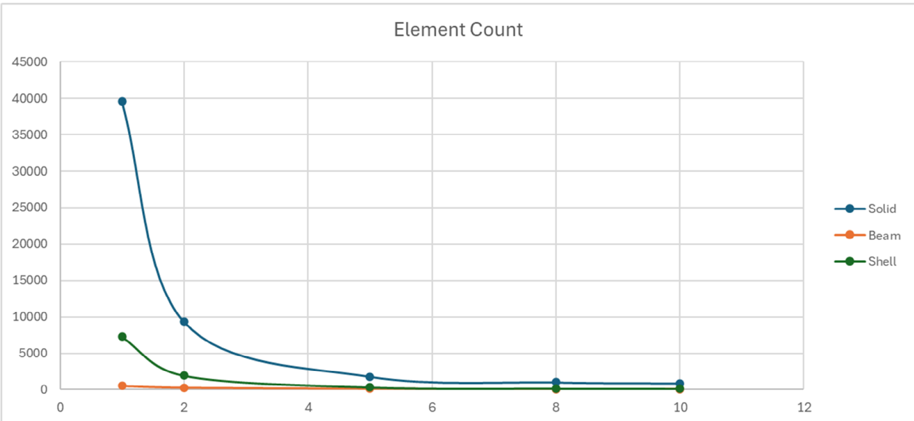

Figure 5.1.1 Element Count after Sensitivity Analysis

Assuming the Solid Model is most accurate, there is a 14% improvement from Beam to Shell, and another 14% from Shell to Solid using a 1mm element size as Table 5.1.1 Results Comparison shows.

Von-Mises stresses rise notably between models due to stress singularities around sharp edges. To accurately assess the stress, the writer averaged values from nearby nodes, resulting in a 1% variation across all three simulations.

	Mesh Size	**Normal Stress**	**Von-Mises**	Displacement	Von-Mises (averaging around singularity)
Beam to Shell	1mm	16%	15%	14	1
Shell to Solid	1mm	13%	60%	14	1

Table 5.1.1 Results Comparison

5.2. Sensitivity Analysis Results

Conducting sensitivity analyses on both Shell and Solid models provides a more accurate representation of stresses and displacements. Figure 5.2.1 below illustrates the improvements in stress results due to the sensitivity analysis, demonstrating the maximum Von-Mises stress at the singularity and the averaged value on the surrounding nodes, thereby offering a more realistic depiction.

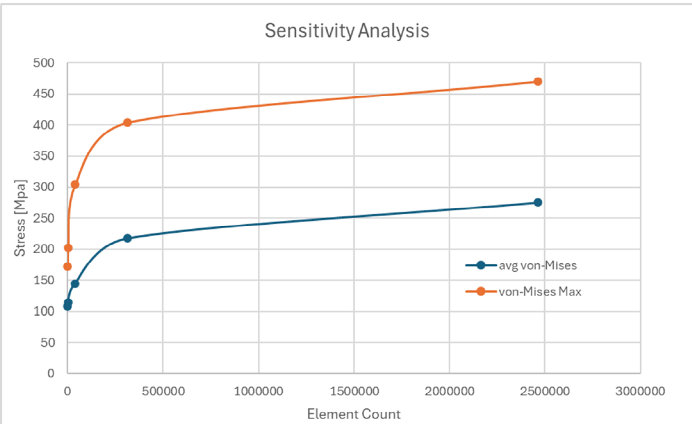

Figure 5.2.1 Sensitivity Analysis for Solid Model and Comparison between Max Von-Mises vs averaged Von-Mises Stresses

6. Comparison of Models

6.1. Comparison of Beam, Shell, and Solid Models

The main differences between each model lie in the computational approach used for stress and displacement analysis. Beam Analysis applies

1D line elements suitable for long structures with significant length to cross section ration, incorporating 6 degrees of freedom (DOF) per node—3 translational and 3 rotational. Shell Analysis, on the other hand, utilizes 2D surface elements designed for thin-walled structures, also with 6 DOF per node—3 translational and 3 rotational. Solid Analysis employs 3D volumetric elements for comprehensive volume analysis, offering 3 DOF per node (translational only).

Because the bracket has significant larger lengths compared to cross section both bean and shell analysis perform well. Shell analysis can also help with shear locking, as the elements can account for bending. However the use of quadratic approximation with solid elements was applied, which also helps.

7. Conclusions

7.1. Summary of Findings and Recommendations

All tree analysis shows that the bracket can withstand the applied load safely, except in areas with stress singularities.

After conducting a stress-error analysis on the model, it was observed that four locations indicate areas where the stress values may be inaccurate. These areas include: the two holes where the bracket is supported, and the connection points between the supporting arm and the top and bottom arms. To address this issue, localised mesh controls were applied to improve the definition in these areas. However, the supporting holes still displayed a singularity.

The bracket twists left under load, with maximum stress occurring in the supporting holes and the radii of the support arm (path DE).

The analysis did not show major concerns with the design. However, shall the user wanted to improve the design, an optimisation analysis could be perform to improve the design based on the applied load.

8. References

[1] ANSYS Inc. (2025). ***Independent Cylinder Support in ANSYS Workbench***. Retrieved from https://ansyshelp.ansys.com/public/account/secured?returnurl=//////Views/Secured/corp/v242/en/wb_sim/ds_Independent_Cylinder.html