ANALYSIS OF RIGGING PLATE

Project 2

Sudarsan Srinivasan

2427453

ME478 Summer 2025

Contents

Abstract

In this project, a rigging plate was analyzed using finite element analysis (FEA) to evaluate its structural performance under rescue loading conditions. The plate was modeled as a 2D plane stress system with a thickness of 1.0 cm, made of steel (E = 190 GPa, v = 0.3, Sy = 290 MPa). The objective was to determine its breaking strength and verify compliance with the certification requirement of 36 kN.

A convergence study was performed with multiple global mesh sizes, and the results showed that a global element size of ~0.8 mm achieved accurate and stable stress predictions. Further refinement around stress-concentrated rope lug holes revealed higher localized stresses, with maximum von Mises stresses exceeding the yield threshold due to stress singularities. To address this, stresses at adjacent nodes were considered more reliable.

The rigging plate was found to have a breaking strength of approximately 40 kN, which exceeds the certification requirement. Based on these results, the current design is sufficient for use, although future improvements to reduce stress concentrations at the holes are recommended.

Introduction

Rigging plates are a life-critical component that is utilized in a variety of applications including industrial rope-access, firefighting, and rescue. Since failure could have severe consequences, these plates must meet strict strength requirements.

Figure 1: Rigging plate application

In this project, a 2D finite element analysis (FEA) model of a steel rigging plate was developed to evaluate stress and deformation under load. FEA is used because it allows us to capture stress concentrations around holes and curved regions that are difficult to analyze accurately with hand calculations. This simulation approach provides a more reliable estimate of the plate's breaking strength and ensures the design can be certified to withstand at least 36 kN.

The plate is 1.0 cm thick. The plate is made of steel with a Young's Modulus of E = 190 GPa, Poisson's Ratio v = 0.3, and a yield strength of Sy = 290 MPa.

Analysis Procedure

The rigging plate seen in Figure _ was modelled as a 2D surface using SolidWorks and exported to IGES format.

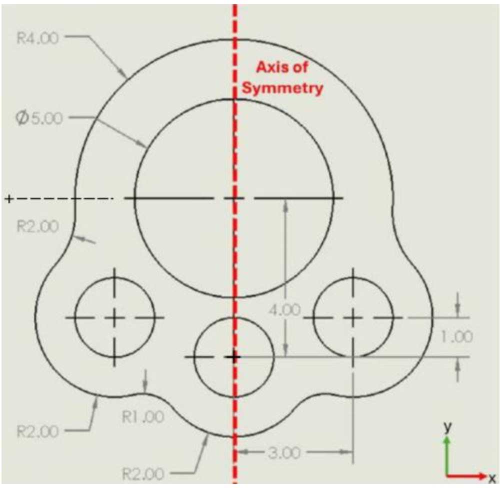

Figure 2: Rigging plate geometry and dimensions (units: cm).

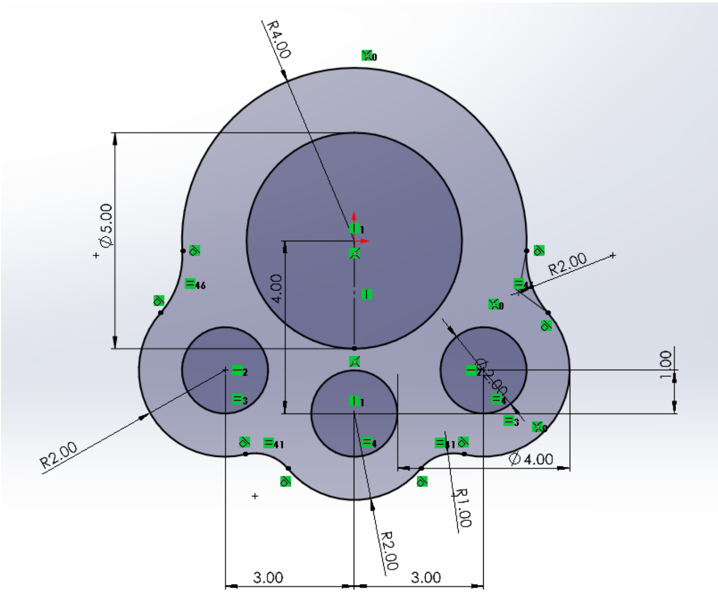

Figure 3: 2D Rigging plate dimensions in SolidWorks

The rigging plate shown in Figure 3 was modeled in SolidWorks with the given 2D dimensions as per Figure 2 and exported as IGES file.

The model is imported into ANSYS mechanical and the following properties applied:

Analysis type: 2D

Material Properties: Steel with a Young's Modulus of E = 190 GPa, Poisson's Ratio v = 0.3, and a yield strength of Sy = 290 MPa, with isotropic behavior.

Plate thickness: 10 mm

Mesh Method: Quadrilateral dominant method with quadratic element order. Because 8-node quads capture bending and stress gradients at holes/fillets much better than linear.

Mesh configuration 1

Mesh size: 1 mm Body sizing mesh

Number of nodes: 14473

Number of elements: 4694

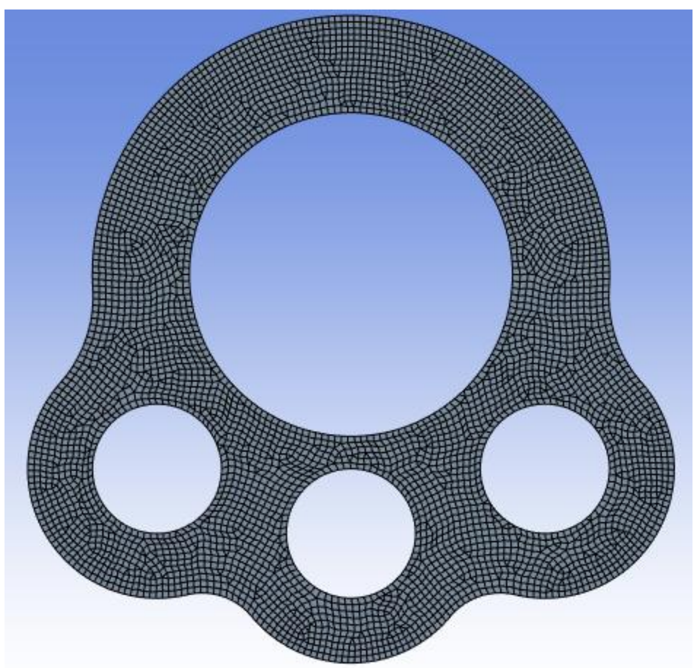

Figure 4a: Mesh configuration 1

Mesh configuration 2

Mesh size: 0.8 mm Body sizing mesh.

Number of nodes: 21922

Number of elements: 7157

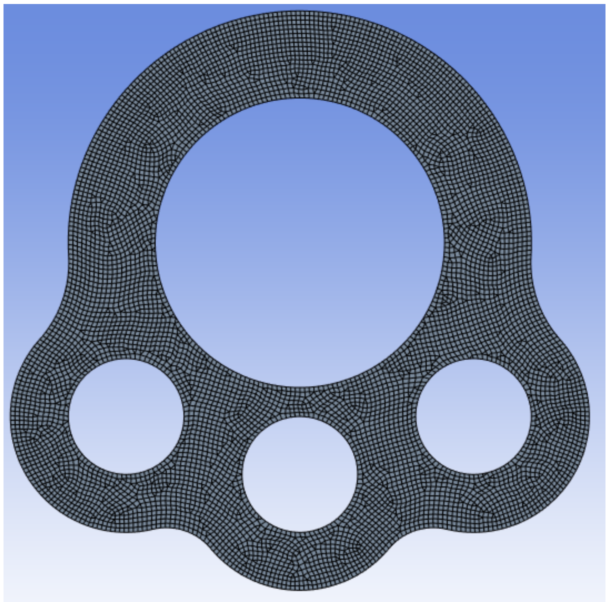

Figure 4b: Mesh configuration 2

Mesh configuration 3

Mesh size: 0.8 mm Body sizing mesh with level 3 Edge refinement at all the Dia 2 cm rope lugs.

Number of nodes: 39067; Number of elements: 12638

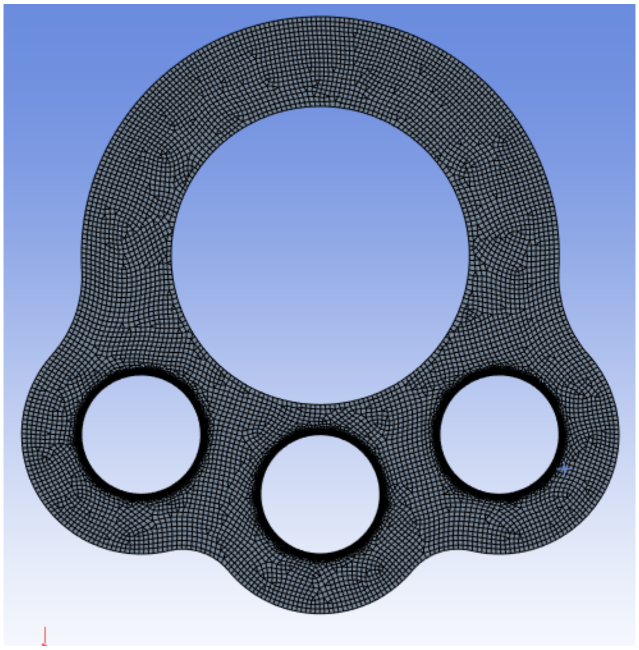

Figure 4c: Mesh configuration 3

Boundary conditions

Common to all the mesh configurations.

- Fixed support at edge A
- Forces applied at:
  - Load B: Fx = - 8485.28 N; Fy = - 8485.28 N
  - Load C: Fx = 0 N; Fy = - 12000 N
  - Load D: Fx = 8485.28 N; Fy = - 8485.28 N

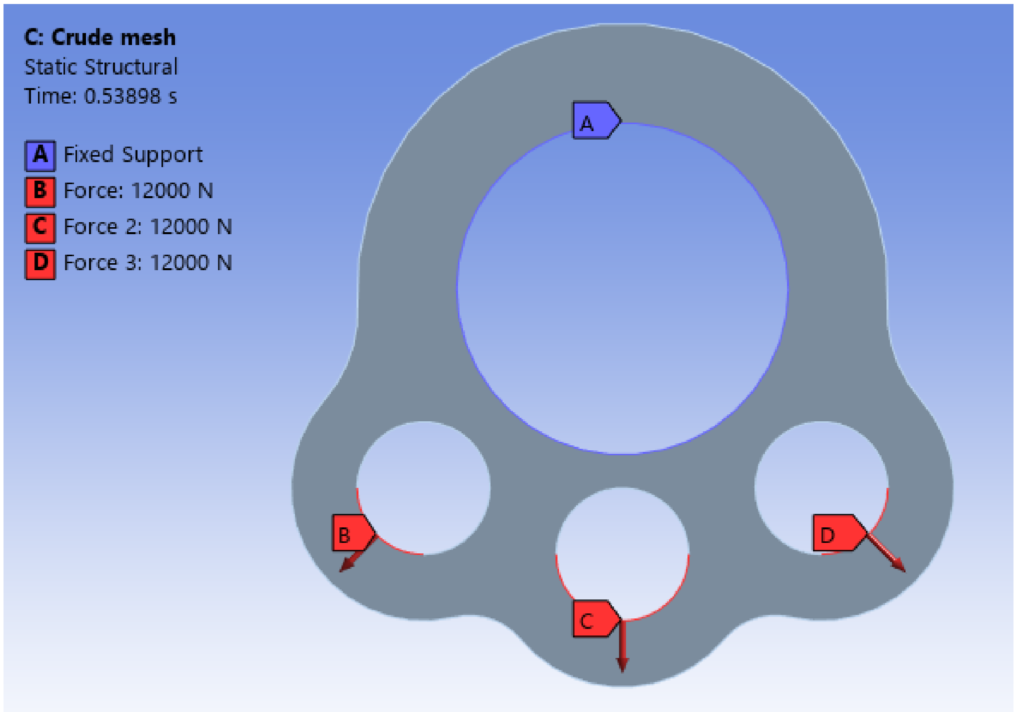

Figure 5: Boundary conditions and loads on the rigging plate

Results

(1) Results from Mesh configuration 1: 1 mm body mesh

A maximum deformation of 0.022 mm was observed as seen on figure 9a and a maximum stress of 261.17 MPa was observed as seen on figure 9b which is shown on *350x scale*.

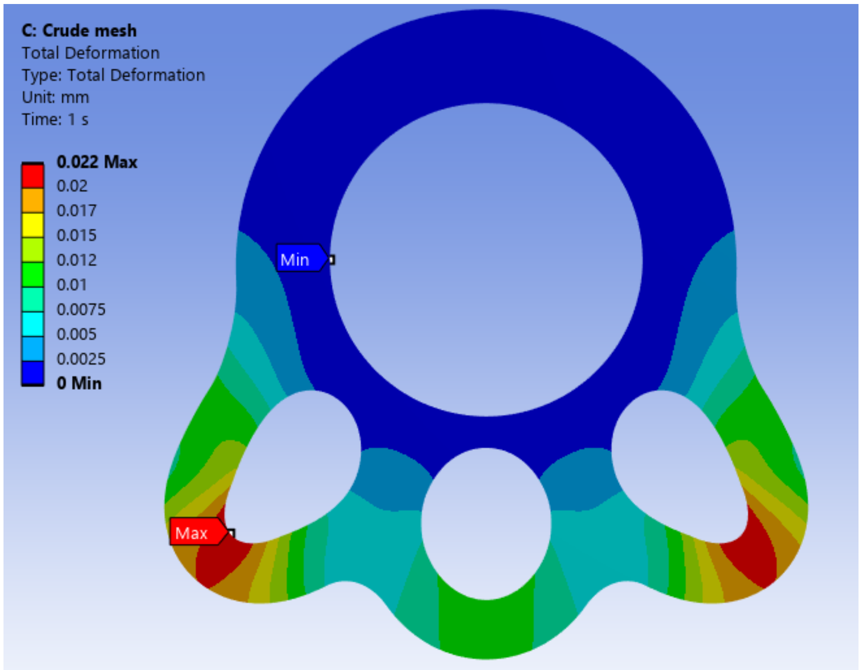

Figure 9a: Total deformation of rigging plate

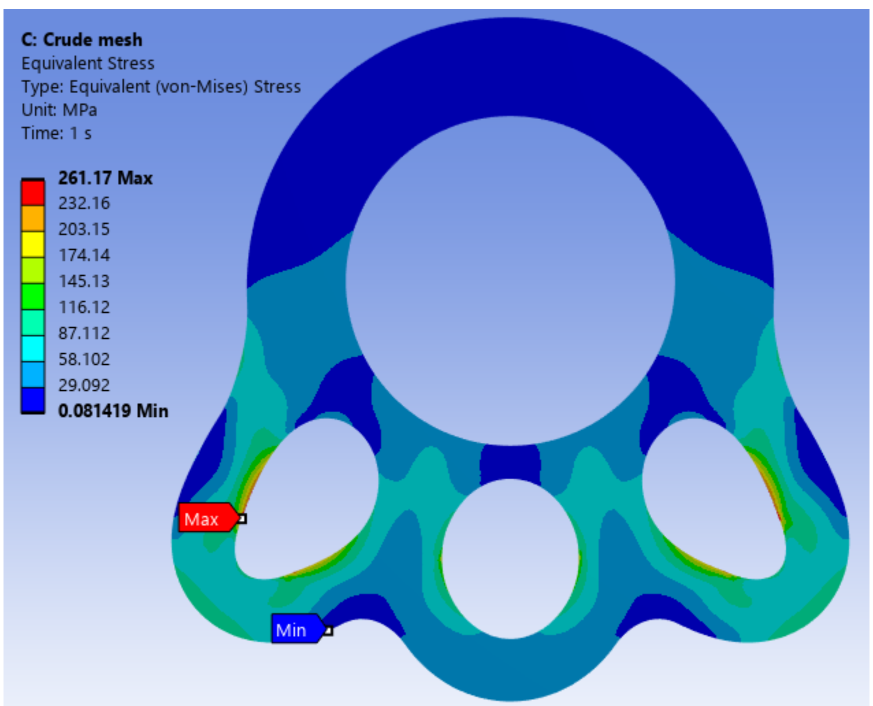

Figure 9b: Equivalent von-mises stress of rigging plate

Since the properties are linear, the maximum force that can be applied before the yield strength of 290 MPa is reached can be interpolated and is seen on Table 1. The max force represents the sum of the total forces applied and was calculated to be 39.98 kN.

Force [N]	Max von mises stress (MPa)	# of elements	# of nodes
36,000	261.17	4694	14473
39,972	289.99		

Table 1: Breaking strength of the rigging plate

(2) Mesh convergence study

A convergence test was performed using the mesh type from the original run (same one as used for the results in Figure 5). Five additional meshes were tested of different element sizes for the same load of 36 kN and boundary conditions, until convergence (difference < 2-3%) was reached at 0.8 mm global mesh element size. The results can be seen in Table 3 below.

Sl. No	Global mesh size (mm)	# of nodes	# of elements	Max von mises stress (Mpa)	% Change
1	Default - 8.22	354	97	265.59	0.00
2	6	567	162	181.68	-31.59
3	4	1114	329	221.1	21.70
4	2	3973	1244	236.3	6.87
5	1.5	6745	2146	241.08	2.02
6	1.25	9506	3059	244.15	1.27
7	1	14473	4694	261.16	10.52
8	0.9	17675	5750	258.09	-1.18
9	0.8	21922	7157	257.85	-0.09

Table 2: Global mesh convergence study

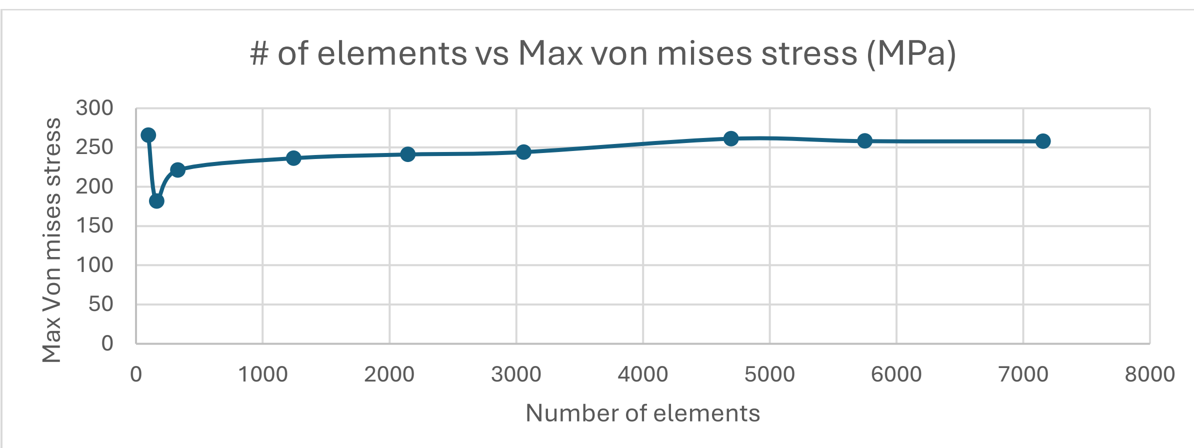

Figure 10: Equivalent von-mises stress of rigging plate

(3) Mesh convergence study using Local Mesh refinement

Now, a convergence test was again performed using the optimum mesh size of 0.8 mm that we found from the last step using Mesh refinement for 3 levels, the results of which can be found in the below table 3. Couldn't move to 4th level of refinement as the limit was 3.

Sl. No	Refinement level	# of nodes	# of elements	Max von mises stress (MPa)	% Change
1	lv 1	26265	8526	282.42	-
2	lv 2	31019	10054	298.11	5.6
3	lv 3	39067	12638	307.14	3.0

Table 3: Local mesh convergence study

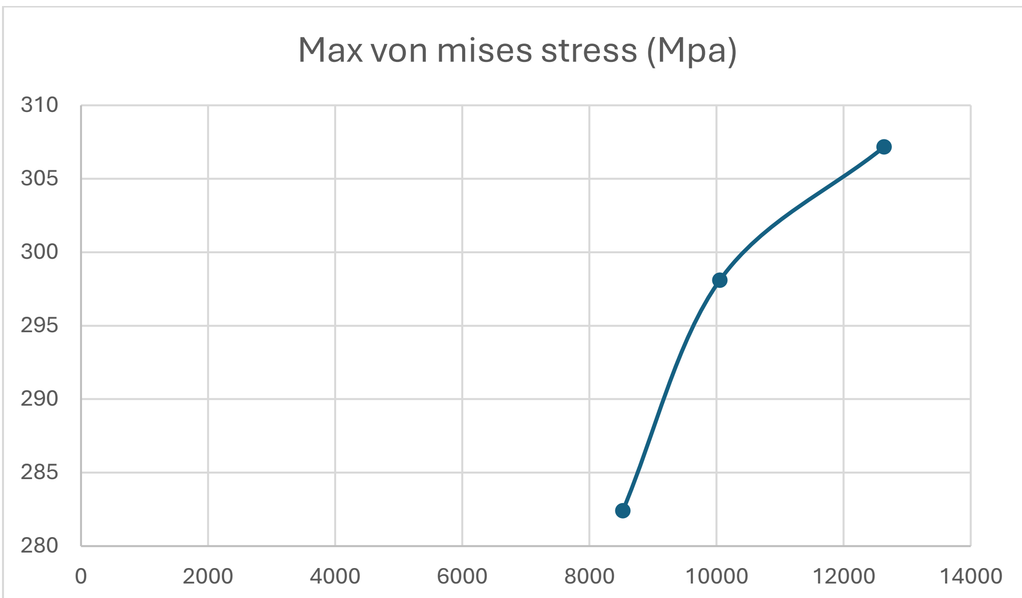

Figure 11: Equivalent von-mises stress of rigging plate

(4) Results from Global mesh and Local mesh refinement

Equivalent von-mises stress:

The overall trend of the Equivalent von-mises stress plots are similar in both cases and the max stresses are notes as 257 MPa and 307 MPa in configs 2 and 3 respectively as seen in Figure 12.

If we zoom up further, we can see the stress on adjacent nodes as 196 MPa and 253.22 MPa on configs 2 and 3 respectively as seen on Fig 14.

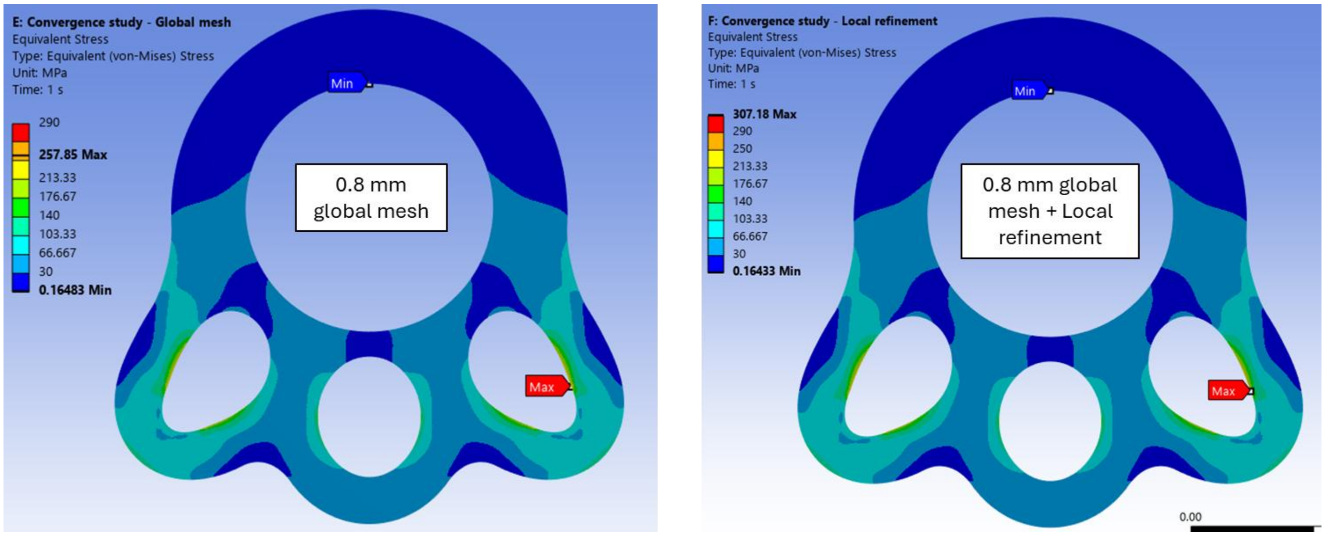

Figure 12: Equivalent von-mises stress comparison

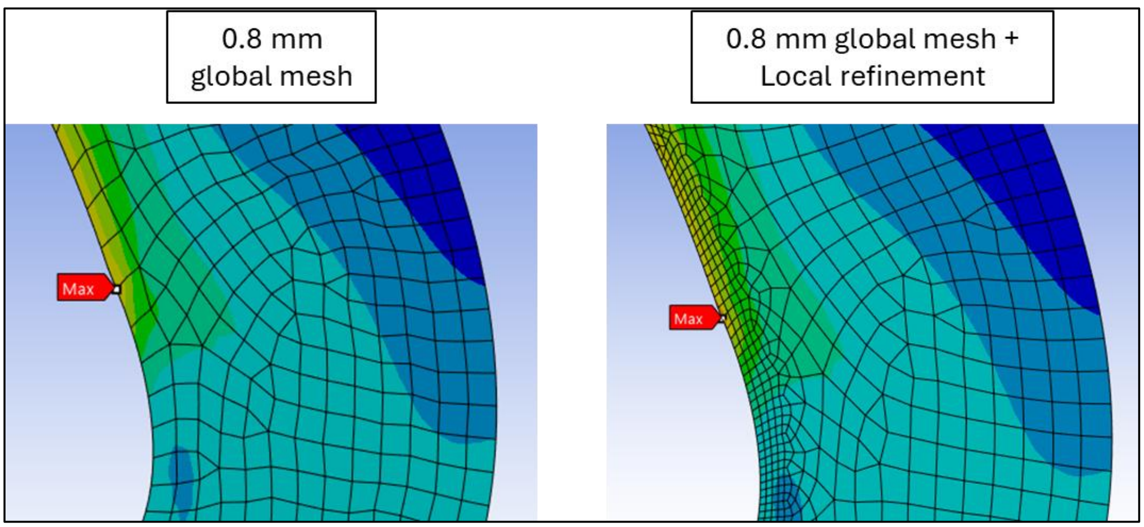

Figure 13: Equivalent von-mises stress comparison – point of stress

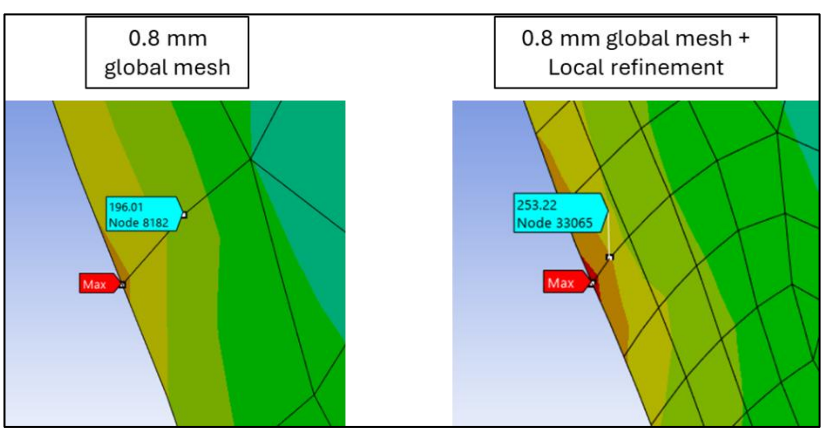

Figure 14: Equivalent von-mises stress comparison – adjacent node

Total deformation

Both configurations show the same maximum deformation of 0.022 mm at the same location as can be seen in Figure 15

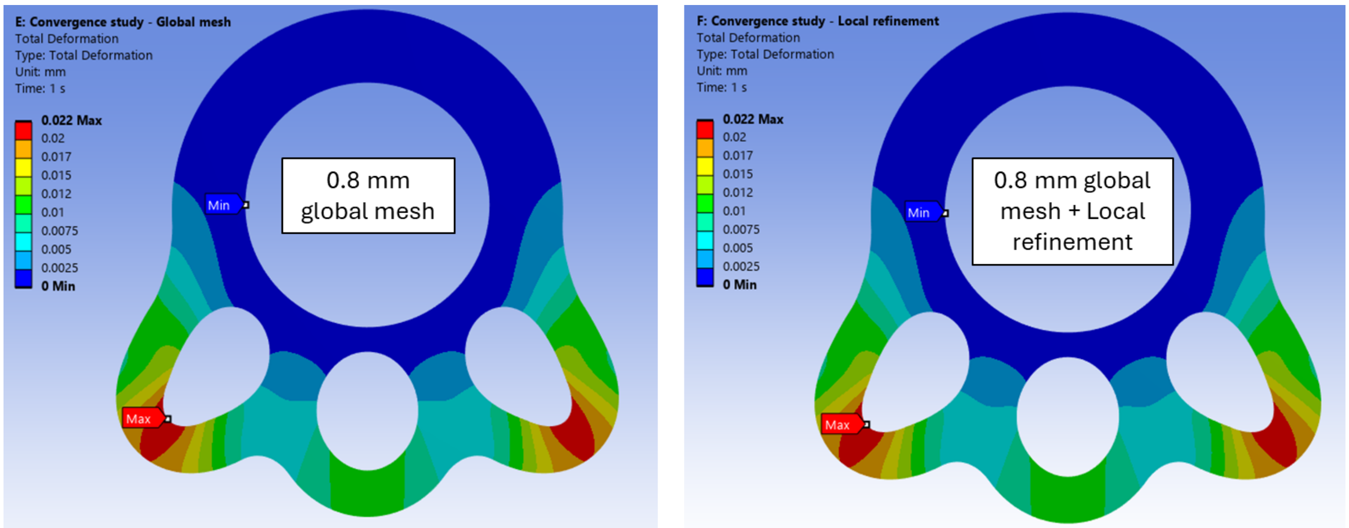

Figure 15: Total deformation comparison

Discussion

(1) Stress, deformation and breaking strength of the design

The rigging plate exhibited a maximum von Mises stress of approximately 261 MPa under the applied 36 kN load using the 1 mm mesh configuration. Since the yield strength of the material is 290 MPa, the plate has a calculated breaking strength of ~39.98 kN, which exceeds the required certification strength of 36 kN. The maximum total deformation observed was very small (0.022 mm), indicating that the plate is structurally stiff under the given loading conditions.

(2) Convergence study for optimal global mesh element size

The global mesh refinement study demonstrated convergence of maximum stress values around the 0.8 mm element size, with changes between successive refinements dropping below 1%. This confirms that a mesh size of ~0.8 mm is sufficient to balance computational cost and accuracy for this model.

(3) Convergence study for optimal local mesh refinement

Local refinement at the rope lug holes revealed higher stress concentrations, with maximum stresses increasing from 282 MPa (level 1 refinement) to 307 MPa (level 3 refinement). This indicates that local mesh refinement is critical in capturing stress gradients at geometric discontinuities. However, deformation results were largely unaffected by refinement.

(4) Optimal mesh

The comparison shows that while global refinement alone provides a good approximation (257 MPa at 0.8 mm mesh), local refinement at stress-concentrated regions improves accuracy, predicting stresses above the yield threshold. This indicates that the global mesh captures the overall behavior, but local refinement is essential to properly resolve stress gradients near the rope lugs.

Since stress singularities occur at sharp corners and hole edges, the reported maximum von Mises stress continues to increase with successive refinement. Therefore, it is good practice to reference the stress value at an adjacent node rather than the singular node itself. Based on this, the optimal meshing strategy is to use a global element size of ~0.8 mm with local refinement at the rope lug holes, while considering adjacent-node stresses for a more realistic assessment.

Conclusions

This analysis evaluated the performance of a rigging plate under rescue loading conditions using finite element analysis. The global mesh convergence study confirmed that an element size of ~0.8 mm yields consistent results. Local refinement at stress-concentrated rope lug holes revealed higher stress levels, with peak values approaching and slightly exceeding the material's yield strength.

The breaking strength of the plate was estimated at ~40 kN, which exceeds the certification requirement of 36 kN, suggesting the current design is adequate. However, the localized stresses observed under refined meshing indicate that fatigue or long-term durability could be concerns in service.

It is recommended that the design be certified for rescue use at 36 kN, with consideration of further design optimization (such as fillet radius adjustments at the rope holes) to reduce local stress concentrations and improve safety margins.