Jet Engine Bracket FEM Analysis

by

Nikhil Rane

Faculty of Technology and Bionics

Professor: Herr Prof. Dr.-Ing. Henning Schütte

Date of submission :13th March,2021 Kleve, Germany

1. Summary

1.1 Task Description

A Jet Engine Bracket made of 30CrNiMo8 (1.6580) Quality Treated was loaded with static force. The static strength of the part was analysed using the FKM guidelines assuming that the part was manufactured by forging and/or machining. The analysis was ran to find the degree of utilization against onset of plasticity and rupture when the bracket is subjected to a horizontal static load. The point of plastic collapse was also determined.

1.2 Realization

Examined system

The system to be analysed is the jet engine bracket. It has the following interfaces as shown in

Table 1: Interfaces

Interface Number	Description
1	Infinitely stiff Pin of diameter 0.75 inch
2-5	0.375-24 AS3239-26 machine bolt. The bolts are to be considered infinitely stiff.

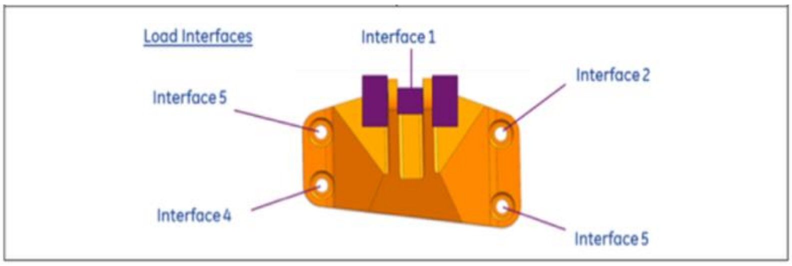

Figure 1: Examined System

1.3 Method of Evaluation

The software ANSYS was chosen to perform a Finite Element Method analysis on the given task. All the evaluations have been carried according to FKM guidelines. The final mesh was converged with an accuracy 3%.

1.4 Results

Factors of safety of 1.5 was considered against yielding and 1.75 was considered against rupture. The material 30CrNiMo8 (1.6580) Quality Treated was found to be adequate for the task as it would neither undergo yielding nor rupture. Further the following degree of utilization was calculated

Table 2: Degree of Utilization

Load case	Degree of utilization (yield a_{pe})	Degree of utilization (Rupture a_f)
2	9.52 %	10.06 %

Table of Contents

1. SUMMARY....2
1.1 Task Description....2
1.2 Realization....2
1.3 Method of Evaluation....3
1.4 Results....3
2. OBJECT OF EXAMINATION....7
2.1 Parts....7
2.2 Environment....7
2.3 Design Loads and Loading Conditions....7
3. SIMULATION ENVIRONMENT....8
3.1 Used Softwares....8
3.2 Units....8
4. SOLVING AND RESULTS....9
4.1 Default Meshing....9
4.2 Boundary Conditions....10
4.3 Material Properties....11
4.4 Load Case....12
4.5 Solving Load Case with Default Meshing....12
4.6 Submodelling Operation....13
4.7 Mesh Refining....15
5. EVALUATION OF RESULTS....18
5.1 Allowable Stress Calculation....18
6. PLASTIC LOADING....19
7. CONCLUSION....22
8. REFERENCES....23

Table of Figures

Figure 1: Examined System....2
Figure 2: Isometric view (left) and Top view (right) of Jet Engine Bracket 7
Figure 3: Default Meshing is applied to the part....9
Figure 4: Mesh Metrics of Default Meshing....9
Figure 5: Frictionless support 10
Figure 6: Factionless Support is applied on the outside circular face of the clevis....11
Figure 7: Frictionless support on the base 11
Figure 8: Remote force was applied in the horizontal direction 12
Figure 9: Averaged and Unaveraged stress using defalt mesh....13
Figure 10: Submodelling of area of interest 13
Figure 11: Frictionless support on the cut surface 14
Figure 12: Average Equivalent stress (left) and Unaveraged Equivalent Stress (right)....14
Figure 13: Cylindrical slicing for body sizing 15
Figure 14: Averaged and Unaveraged Equivalent Stress after body sizing of 0.8 mm....16
Figure 15: Final Element Quality 17
Figure 16: Tabular data to determine the Plastic load limit and the Elastic load limit 19

List of Tables

Table 1: Interfaces 2
Table 2: Degree of Utilization 3
Table 3: Software used..8
Table 4: Units used...8
Table 5: Averaged Equivalent Stress due to default mesh...13
Table 6: Averaged and Unaveraged Equivalent Stress using default mesh 14
Table 7: Difference between averaged and unaveraged stresses after mesh refinement ...15
Table 8: Final averaged and unaveraged stress for mesh refinement 16
Table 9: Plastic load factor 20
Table 10 : Degree of utilization...21
Table 11: Final results of Degree of Utilization 22

2. Object of Examination

2.1 Parts

The part to be analysed is shown in Figure 2.

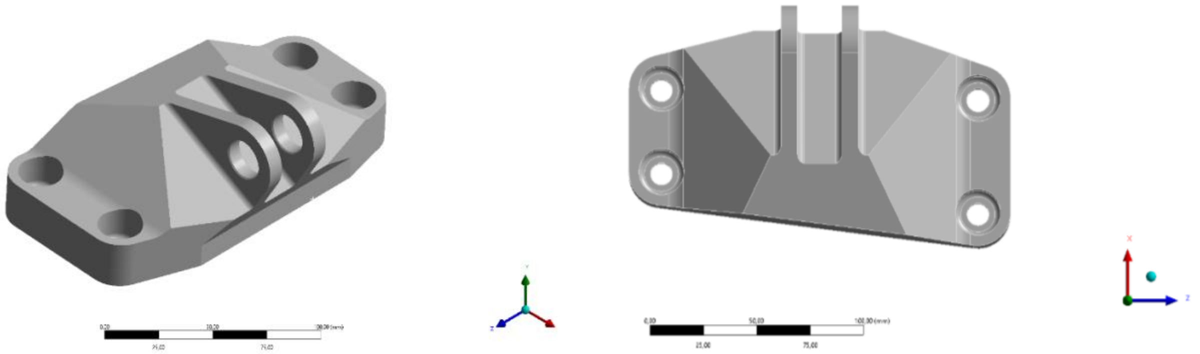

Figure 2: Isometric view (left) and Top view (right) of Jet Engine Bracket

2.2 Environment

The part to be analyzed belongs to a jet engine assembly.

2.3 Design Loads and Loading Conditions

Static horizontal load case was analysed as shown in Figure 8.

3. Simulation Environment

3.1 Used Softwares

Table 3: Software used

Task	Software
Geometry Editing	ANSYS Design modeler 2021R1
Meshing	ANSYS Mechanical 2021R1
Defining Boundary Conditions	ANSYS Mechanical 2021R1
Results	ANSYS Mechanical 2021R1

3.2 Units

The units used are metric in the ANSYS Software. The Table 2 shows the units used in this report

Table 4: Units used

Dimensions	Unit
Length	mm millimeter
Force	N newton
Moment	Nmm newton millimeter
Stress	MPa Mega Pascals
Angle	° Degrees
Time	s seconds

4. Solving and Results

4.1 Default Meshing

The Figure 3 shows default meshing being applied to the analysed part in the mechanical mode. The element size mesh matrix graph as shown in Figure 4 has its peak at approximately 50-60 percent which doesn't not satisfy the criteria of a good mesh. To get a general idea of the point of interest, the default mesh was used.

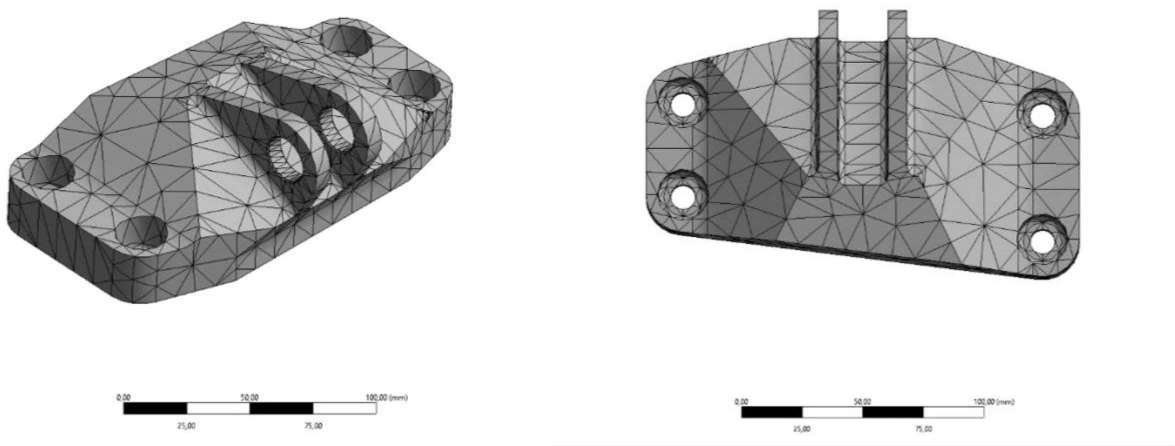

Figure 3: Default Meshing is applied to the part

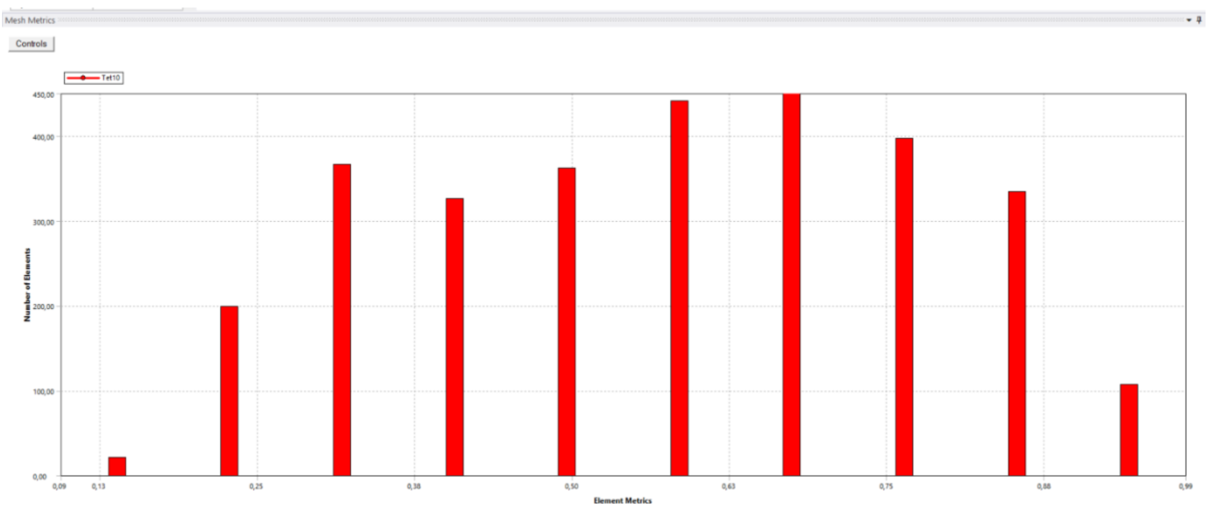

Figure 4: Mesh Metrics of Default Meshing

4.2 Boundary Conditions

The 3 displacement boundary conditions were defined as follows:

- As shown in Figure 5, frictionless support was applied to cylindrical area around all 4 bolts to constrain the bracket.

- An additional fictional support was defined on the outside surface of the clevis due to contact with infinitely stiff pin as shown in Figure 6. This constraint is not necessary as the force is only acting in one direction (horizontal). Factionless Support was applied on the outside circular face of the clevis on both sides

- A fictional support was added on the base of the bracket as shown in Figure 7. to simulate the contact of the bracket and the jet engine assembly.

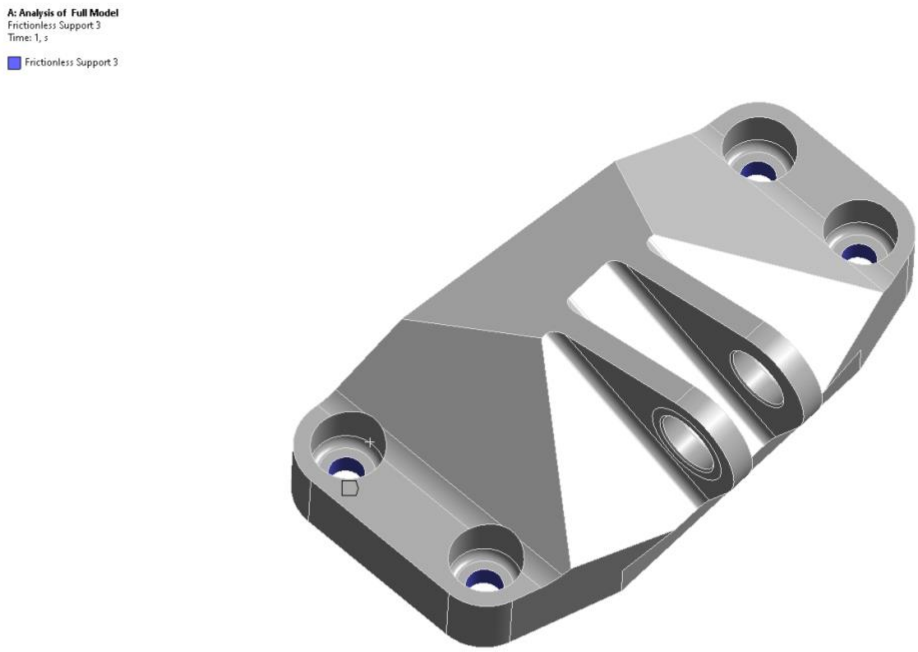

Figure 5: Frictionless support

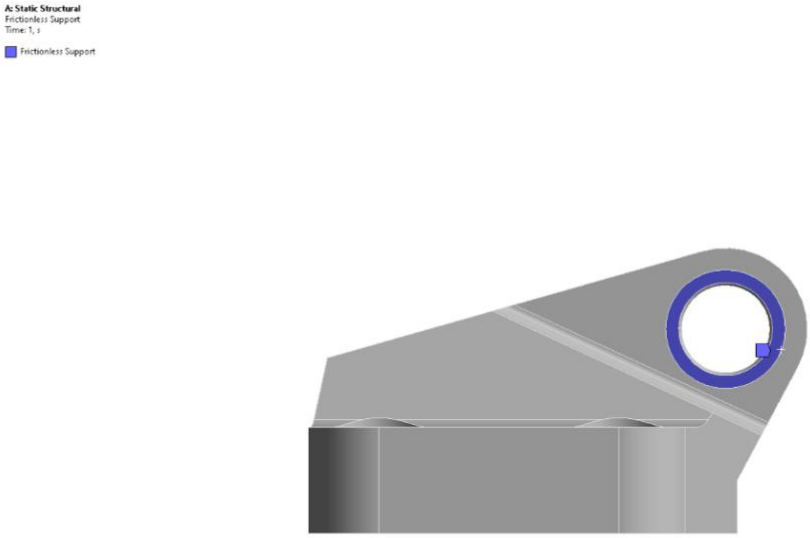

Figure 6: Factionless Support is applied on the outside circular face of the clevis

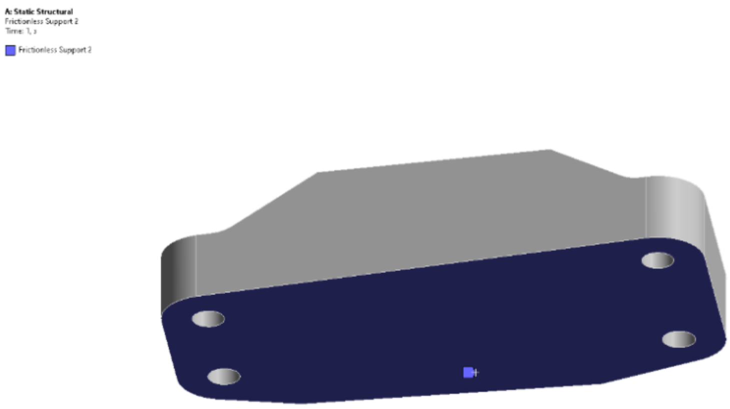

Figure 7: Frictionless support on the base

4.3 Material Properties

The part to be analysed was made of material 30CrNiMo8 (1.6580) quality treated to R_e=1200MPa and R_m=1400MPa, A=12%, Z=40%.

4.4 Load Case

The design load case was determined as per the matriculation number that is Load Case 2. The infinitely stiff pin was subjected to a load of 22341 N in horizontal direction as shown in Figure 8.

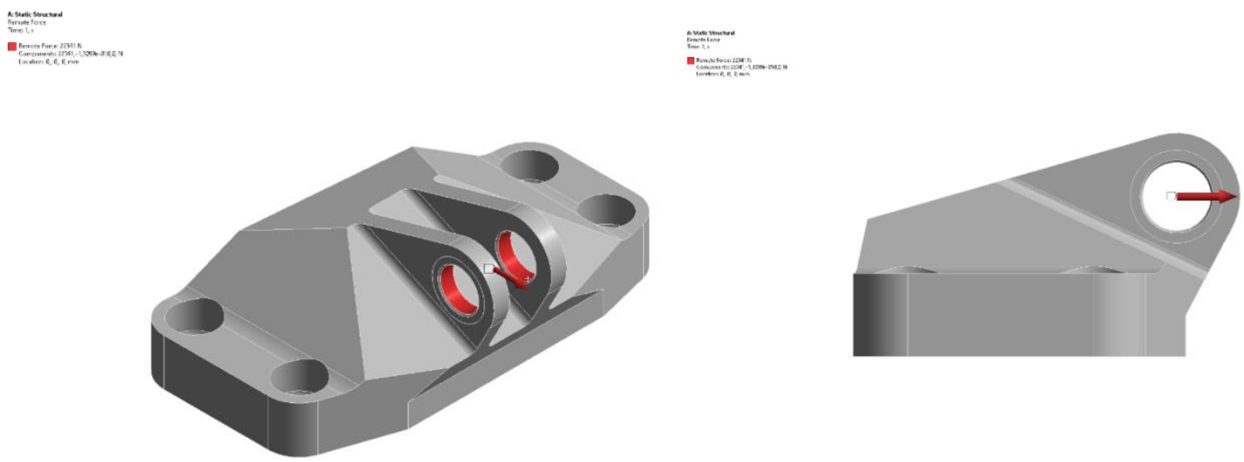

Figure 8: Remote force was applied in the horizontal direction

4.5 Solving Load Case with Default Meshing

According to the simulation, it was seen that the maximum equivalent stress was found on the inner surface of the clevis as shown in Figure 9 . As this was also the area where the force was acting due to the pin, a more detailed study of this part is needed as the meshing was not evenly distributed. The area near the bolts also had stresses but these are not high enough to cause failure.

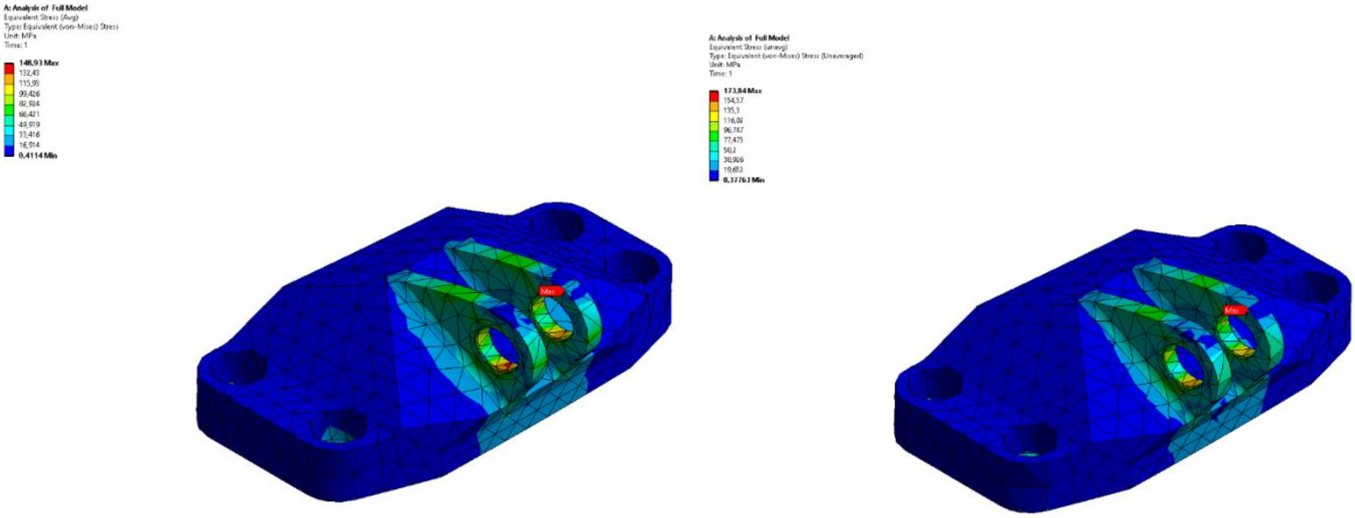

Figure 9: Averaged and Unaveraged stress using defalt mesh

Table 5: Averaged Equivalent Stress due to default mesh

Maximum Averaged Equivalent stress	Unaveraged Maximum Equivalent stress
148.93 MPa	173.84 MPa

4.6 Submodelling Operation

To get a better view of the area of interest and to save computation time and load, submodelling was used. A sphere of 60 mm was made for studying the area of interest in more dept.

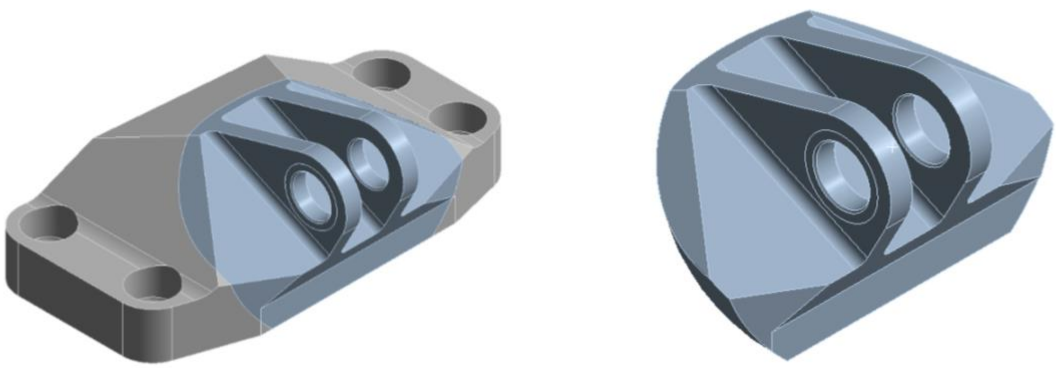

Figure 10: Submodelling of area of interest

A frictionless support was applied on the cut surface to simulate behaviour of the cut-off material surface.1

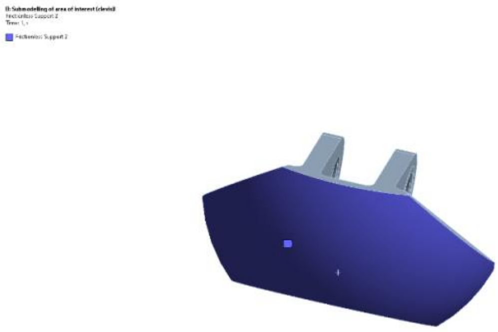

Figure 11: Frictionless support on the cut surface

Default mesh was applied on this submodel to check for the hotspots. It was seen that maximum averaged and unaveraged equivalent stresses values differ greatly as shown in Figure 12. This indicates that more mesh refining is needed to get accurate results.

Table 6: Averaged and Unaveraged Equivalent Stress using default mesh

	Average stress
Averaged	158.92 MPa
Unaveraged	193.07 MPa

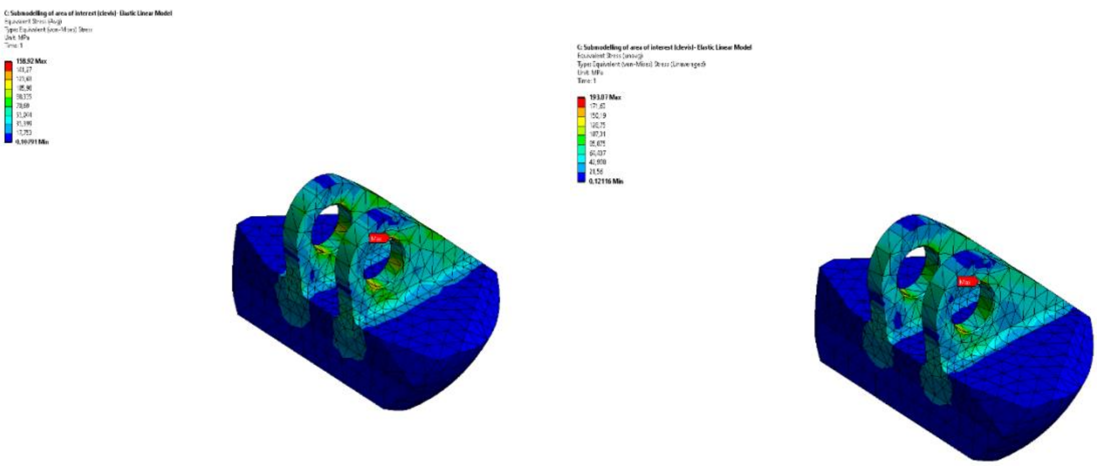

Figure 12: Average Equivalent stress (left) and Unaveraged Equivalent Stress (right)

4.7 Mesh Refining

To get an even distribution of unaveraged and averaged stress, the clevis was sliced into small cylindrical parts as shown in **Error! Reference source not found.** so that body sizing with hex dominant elements could be done. Body sizing was performed as stress maxima was found in this region using default mesh. A reasonable theory could be that the inner surface of the clevis was under bending stress which was causing the stress concentration. Further, the particular area of interest would be upper and lower area of the inner clevis surface as this area would get stretched due to the static load which could lead to rupture eventually. The table below shows the face sizing operation

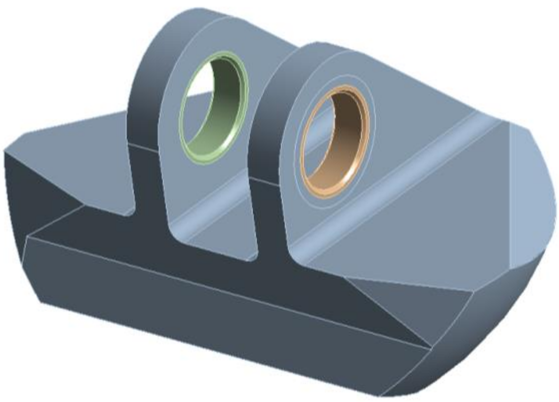

Figure 13: Cylindrical slicing for body sizing

The following table shows the body sizing operation

Table 7: Difference between averaged and unaveraged stresses after mesh refinement

Body sizing	Averaged	Unaveraged	Percent Difference
3mm	156,27 MPa	162,51 MPa	3.83
1mm	160,22 MPa	160,41 MPa	0.12
0.8mm	160,43 MPa	160.54 MPa	0.06

Since the difference between the unaveraged stress of 1mm and 0.8mm was less than 1% hence, a mesh with body sizing of element size 1mm was chosen. Along with this, it can also be seen that averaged and unaveraged stress are within an accuracy of 3%, and they act at same spot and the element quality was more than 70% in the area of interest as shown in Figure 14 and Figure 15. This indicates convergence.

Higher value of unaveraged Equivalent and Principal stress was considered for the further calculations.

Table 8: Final averaged and unaveraged stress for mesh refinement

	Maximum Principal stress	Equivalent stress
Averaged	160,78 MPa	160,22 MPa
Unaveraged	160,9 MPa	160,41 MPa
Difference	0.07 %	0.12 %

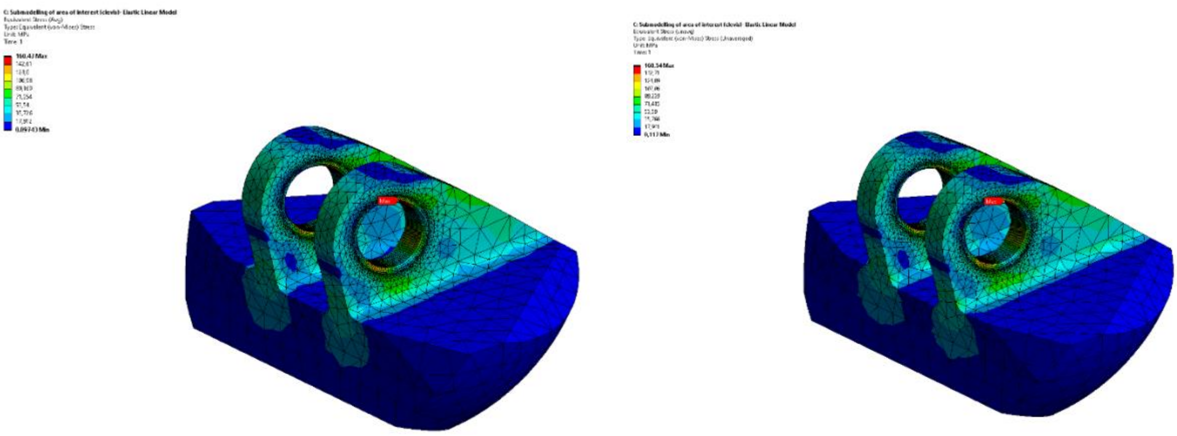

Figure 14: Averaged and Unaveraged Equivalent Stress after body sizing of 0.8 mm

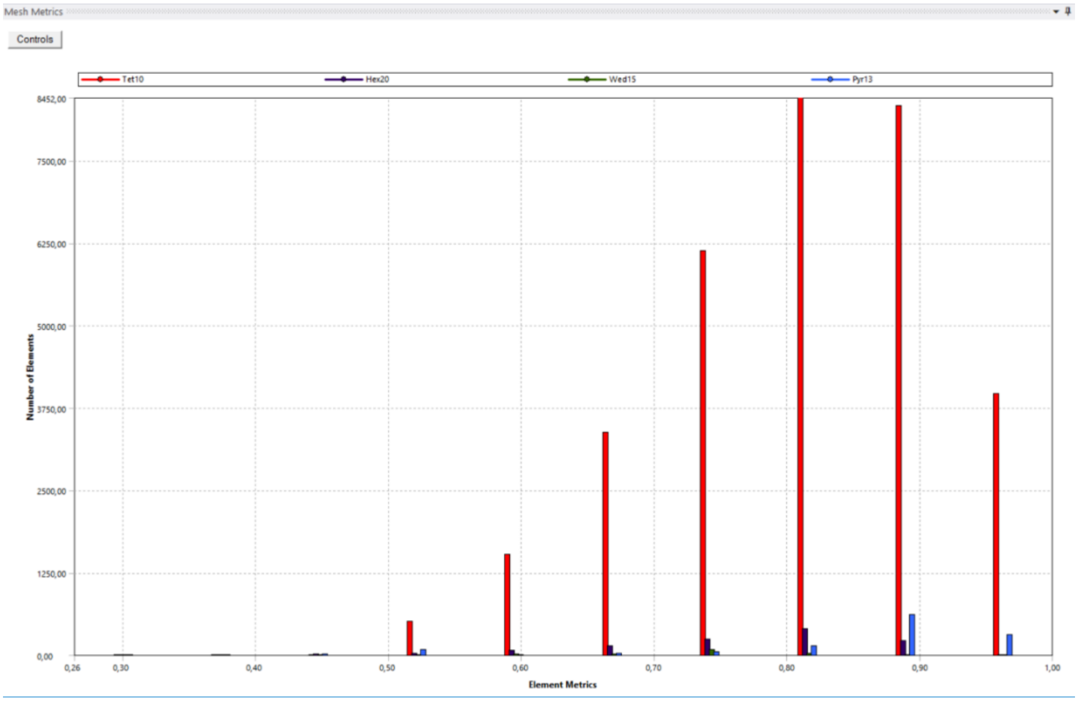

Figure 15: Final Element Quality

5. Evaluation of Results

5.1 Allowable Stress Calculation

The yield strength of 30CrNiMo8 (1.6580) is 1200 MPa and the ultimate tensile strength is 1400 MPa. A safety factor of 1.5 was considered against yield strength and a factor of safety of 1.75 was considered against tensile strength.

Thus the allowable stress is:

$$\sigma_{yield,allowable} = \frac{1200\ MPa}{1.5} = 800\ MPa \qquad \text{...eq i}$$

$$\sigma_{rupture,allowable} = \frac{1400\ MPa}{1.75} = 800\ \text{MPa} \qquad \text{...eq ii}$$

Where,

$$\sigma_{yield,allowable} = allowable\ yield\ stress$$

$$\sigma_{rupture,allowable} = allowable\ rupture\ stress$$

It can be seen that the stress found in the analysis for the horizontal static load case was well below $\sigma_{yield,allowable}$ and $\sigma_{rupture,allowable}$. This illustrates that material 30CrNiMo8 (1.6580) is appropriate for the desired design load.

To determine the yielding and rupture point, further analysis must be done using a much higher load than the design load.

6. Plastic Loading

A plastic simulation has been performed for the load case using Isotropic Bilinear Hardening with a yield Strength of 160 MPa and Tangent Modulus of 1 MPa. The tangent modulus is considered to be 1 MPa since the hardening is not being considered (elastic – ideally plastic). The force value was set to 450 kN so that plastic deformation could be seen.

$$K_p = \frac{Plastic\ Load\ limit}{Elastic\ Load\ Limit} \quad \text{...eq iii}$$

The whole process was divided into 3 substeps (1 second each) each of 150 kN. The yielding occurred at 1.3 seconds meaning at 195 kN force and the part failed at approximately 2.73 seconds that is at 410 kN force as the maximum elongation 30CrNiMo8 (1.6580) can undergo is 12%. The Figure 16 shows the yielding point and rupture point of the part in tabular data form.

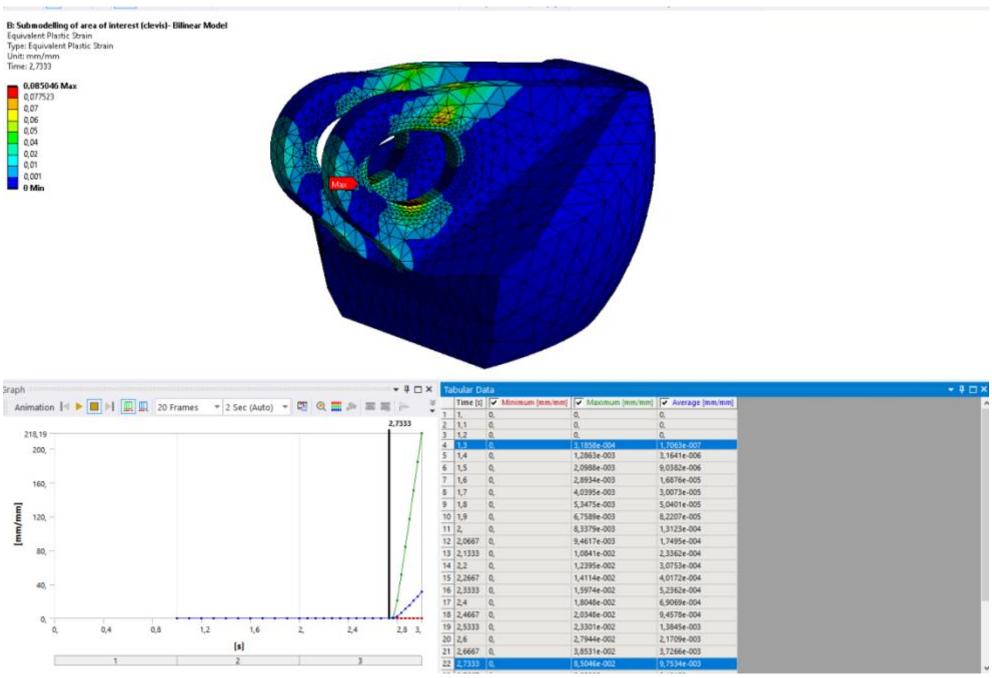

Figure 16: Tabular data to determine the Plastic load limit and the Elastic load limit

Table 9: Plastic load factor

Plastic Load Limit	Elastic Load Limit	K_p Factor
195 kN	410 kN	2.10

According to FKM guideline, section factor is given by the following equation,

$$n_{pl} = MIN(\sqrt{E \cdot \frac{\epsilon_{ertr}}{R_p}}; K_p)$$ ...eq iv

Where,

$R_p = yeild\ strength$

$\in_{ertr} = Critical\ value\ of\ total\ starin$

$E = Young's\ Modulus$

For 30CrNiMo8 (1.6580),

$\in_{ertr} =$ 0.05 and $E = 210\ GPa$

$$n_{pl} = MIN(\,2.958\,; 2.10)$$

Further,

$$\sigma_{SK} = f_\sigma \cdot \frac{R_m}{K_{SK}}$$ ...eq v

For 30CrNiMo8 (1.6580), we can take

$\sigma_{SK} = \frac{R_p}{K_{SK}}$ and $K_{SK} = \frac{1}{n_{pl,\sigma}}$

Where,

$\sigma_{SK} = related\ component\ strength$

$n_{pl,\sigma} = plastic\ section\ factor$

So,

$$\sigma_{SK} = R_p \cdot n_{pl,\sigma}$$ ...eq vi

Degree of utilization against yield:

$$a_{pe} = \frac{Eqvivalent\ stress \cdot 100\%}{allowable\ yield\ stress \cdot K_p}$$ ...eq vii

Degree of Utilization against rupture (ultimate tensile strength)

$$a_f = \frac{Maximum\ principal\ stress \cdot 100\%}{allowable\ rupture\ strength \cdot K_p}$$ ...eq viii

And,

$$\sigma_{allowed,elastic} = \frac{R_p \cdot n_{pl}}{S_{yield}}$$ ...eq ix

Degree of utilization was found using ANSYS by defining user defined result as shown in

Table 10 : Degree of utilization

Maximum Equivalent stress	Maximum Principal stress	Degree of utilization (yield a_{pe})	Degree of utilization (Rupture a_f)	$\sigma_{allowed,elastic}$
160,41 MPa	160,9 MPa	9.52 %	10.06 %	1680 MPa

7. Conclusion

When the Jet Engine Bracket was loaded with the horizontal load case, a structural mechanical failure of the part is not expected.

Table 11: Final results of Degree of Utilization

Degree of utilization (yield a_{pe})	Degree of utilization (Rupture a_f)	$\sigma_{allowed,elastic}$
9.52 %	10.06 %	1680 MPa

As seen in Table 11, the degree of utilizations was found to be considerably low thus, modifications could be done to utilize the material more optimally

8. References

1. Hsrw logo - https://www.hochschule-rhein-waal.de/en