Stress Analysis of an Excavator Hook

Course:	Finite Element Method
Professor:	Herr Prof. Dr. -Ing. Henning Schütte
Submitted By:	Syed Moiz Uddin
Matriculation Number:	27426
Date of Submission:	01.03.2022

Table of Contents

2. Summary

2.1 Task Description

The Excavator Hook has to be analyzed under four times the nominal load according to (FKM Guideline, 2012). A suitable material has to be selected considering the analysis and the Degree of Utilization has to be determined as well as for the Bolts according to the (VDI Guideline, 2015) for high-strength bolted connections.

2.2 Examined Part

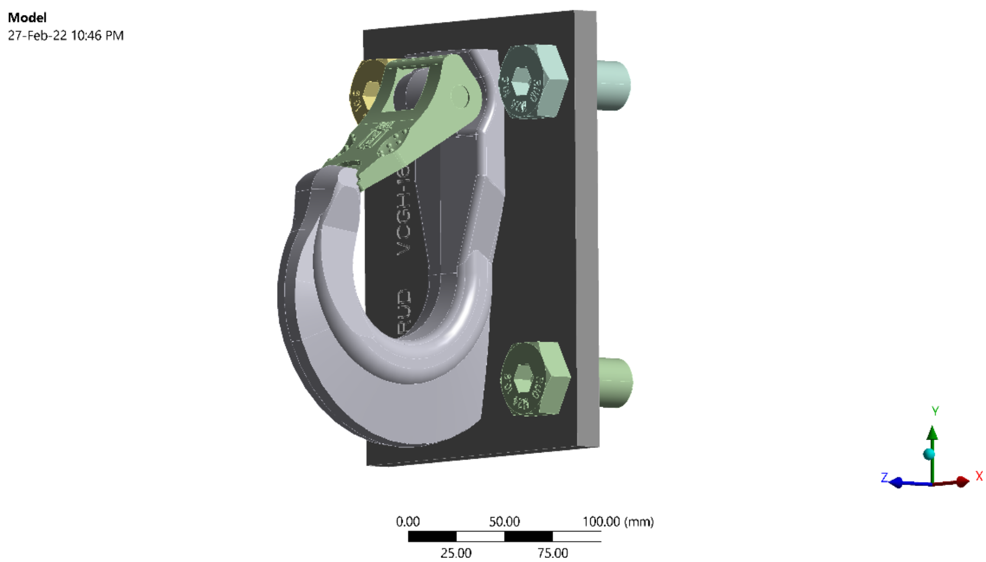


Figure 1: Part to be examined

2.3 Examined Load Case

It is deduced in that a Force of 20 Metric Tonnes is acting on the Excavator Hook. The Force due to symmetry of the part is taken as 10 Metric Tonnes and acts at 90 degrees downwards along the Global Y axis of the part.

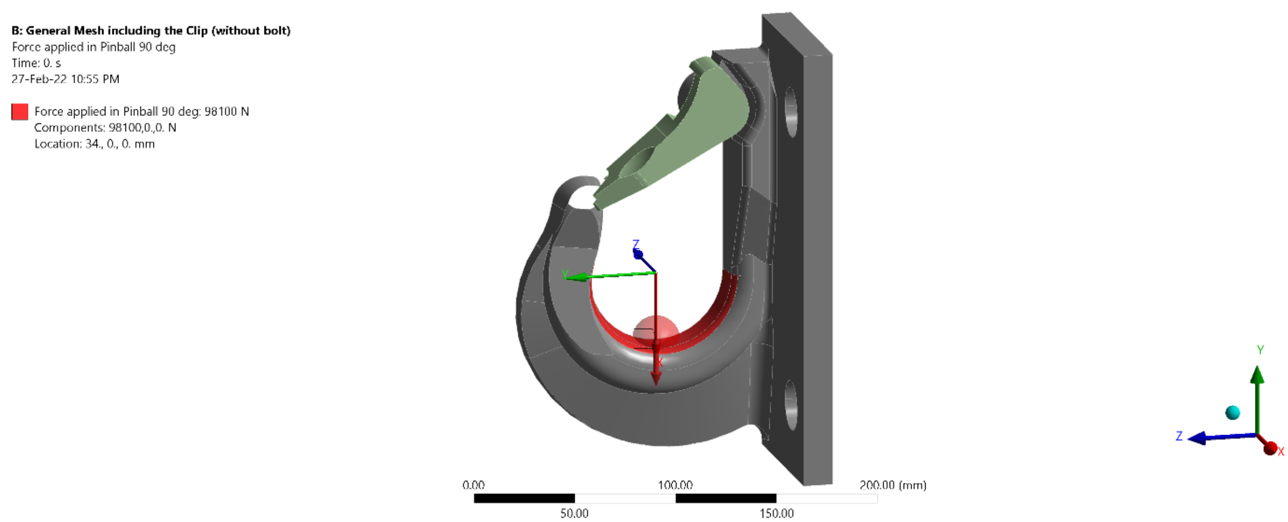


Figure 2: Force Applied at 90 Degrees

2.4 Method of Evaluation

The Excavator Hook is evaluated using Finite Element Method with ANSYS Workbench 2021.

Initially a general mesh is computed to identify regions of possible stress hotspots. The mesh is then refined where the stress hotspots are located in order to get the stresses to converge and therefore are more accurate.

The evaluated stresses using our Load Condition (5.1 Design Load Case) give an idea of a suitable material that could be used and are also used in determining the degree of utilization of the bolts.

2.5 Results

Generating the general mesh, we deduce that there are 2 stress hotpots in the part (explained in more detail in 6.3.1 Stresses with General Mesh). Simulating the stresses using a more refined mesh leads to the stresses in the fillet stress hotspot (Hotspot 2) to converge however the stresses in the Inner Hook stress hotspot (Hotspot 1) do not converge due to an artificial singularity.

The Maximum Averaged Von-Mises Equivalent Stress for the Fillet is recorded to be 1151.6 MPa and the Unaveraged of the same is recorded as 1152 MPa.

The region around the Artificial Singularity Point in the Inner Hook stress hotspot converges (however not the maximum) and the Average and Unaveraged Von-Mises Equivalent Stress is considered to be 1650 MPa which is higher than the fillet therefore has to be considered. Explained in more detail in (7. Stress Evaluation).

Considering the above stresses, the material chosen is 30CrNiMo8 which has Yield Strength of 1050 MPa which is near to the stresses that occur in the Hook's loading condition.

The strain analysis of the part leads to a plastic load factor of 4.48 and using a Safety Factor of 2 using the (RUD Handbook, 2017) for the part, the degree of utilization for the part is evaluated to be 82.5% of the material property.

Simulations for the Bolt Pretension are also carried out and it is determined using the (VDI Guideline, 2015) for high strength bolted connections that the M24 bolts degree of utilization is 8.55%

3. Object of Analysis

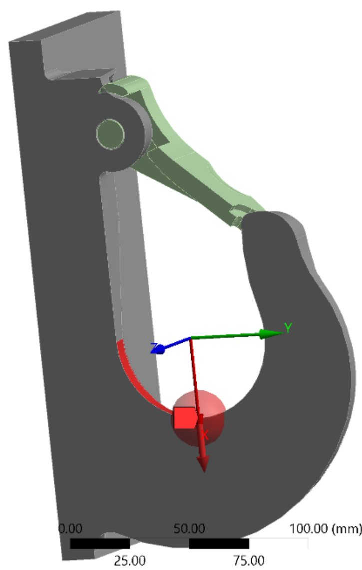


Figure 3: Object of Analysis

4. Simulation Environment

The part in the simulation is considered to be in an isolated environment. The boundary conditions mentioned in 5.2 Boundary Conditions simulate the reactions and loads the Excavator hook experiences in the real world.

4.1 Software

The software used here in the analysis is ANSYS Version 2021R2 Student. It should be noted that the student license only allows for 128,000 elements/nodes to be evaluated.

4.2 System of Units

Metric system of units (mm, kg, N, s, mV, mA) are used for the simulation.

4.3 Coordinate System

The Global Coordinate system and also a local coordinate system is used for the force application. The local coordinate system is a centroid coordinate system using the circular face of the hook in the center in order to apply a force 90 degrees to the Global Z coordinate. The coordinate system can be seen in Figure 2

5. Load Cases

This section deals with how the load condition for the part was set up in order to begin the simulation the Equivalent Stress.

5.1 Design Load Case

The part is to be examined for four times the load of the nominal load. The nominal load of an Excavator Hook is stated to be 98100 N (10 Metric Tonnes). The part is examined therefore for a lifted body of force 392,400 N (40 Metric Tonnes).

Method of lift		G	G	G		G	G		G
Number of legs		1	2	2	2	2	3 and 4	3 and 4	3 and 4
Angle of inclination <ß		90°	90°	0-45°	45-60°	unsymm.	0-45°	45-60°	unsymm.
Factor		1	2	1.4	1	1	2.1	1,5	1
	Type	**WLL in lbs, bolted**							
	VCGH-G 6 *	3300	6600	4620	3300	3300	6930	4950	3300
	VCGH-G 8 *	5500	11000	7700	5500	5500	11550	8250	5500
	VCGH-G 10 *	8800	17600	12300	8800	8800	18500	13200	8800
	VCGH-G 13 *	14300	28600	20000	14300	14300	30000	21450	14300
	VCGH-G 16	22000	44000	30800	22000	22000	46200	33000	22000
	VCGH-G 20	35200	70400	49300	35200	35200	74000	52800	35200
	VCGH-G 22	44000	88000	61600	44000	44000	92400	66000	44000

Figure 4: Load Conditions according to Rulebook (VCGH-G 16)

According to Case 2 in the Lifted Body is lifted with 2 Excavator Hooks with the force acting at 90 degrees therefore it is determined that one hook therefore experiences half the force of the lifted body of 196,200 N (20 Metric Tonnes).

Due to symmetry conditions explained in (6.1 Symmetry). The final applied force on the part is set to 98100 N (10 Metric Tonnes). This force can be seen in Figure 2

5.2 Boundary Conditions

Two boundary conditions of the displacement boundary conditions are used for the simulation.

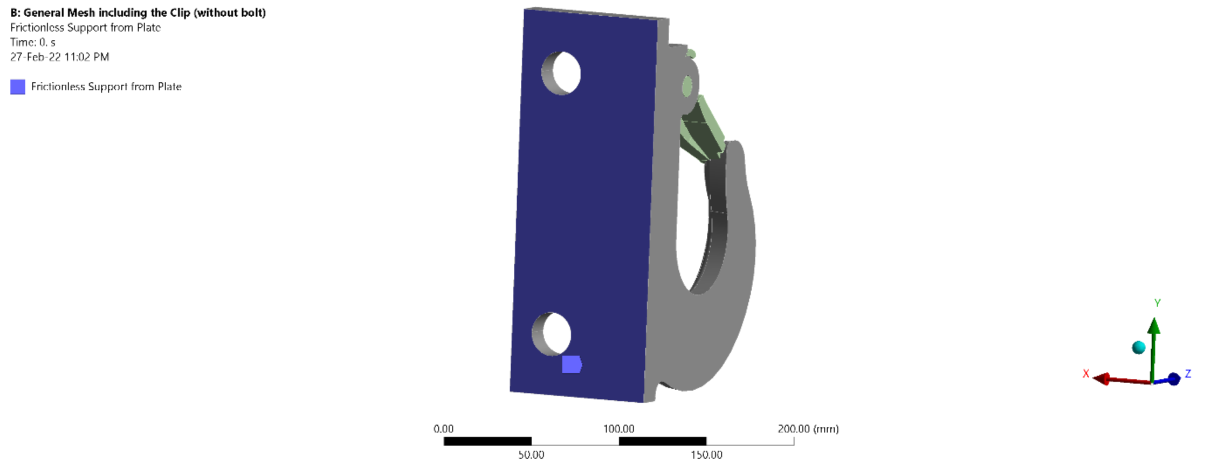


Figure 5: Frictionless Support

A Frictionless support on the base of the part is used to fix all motions of the plate of the hook. A fixed support for the base is not used in this case as it might cause singularities at the sharp edges of the base plate.

The bolts for the initial stress evaluation are suppressed in order to not complicate the geometry and keep the elements under the 128,000 element/nodes limit and are replaced with the boundary condition given below. However, an analysis on the Pretension Load on the bolt is carried out later on in (10. Bolt Pretension)

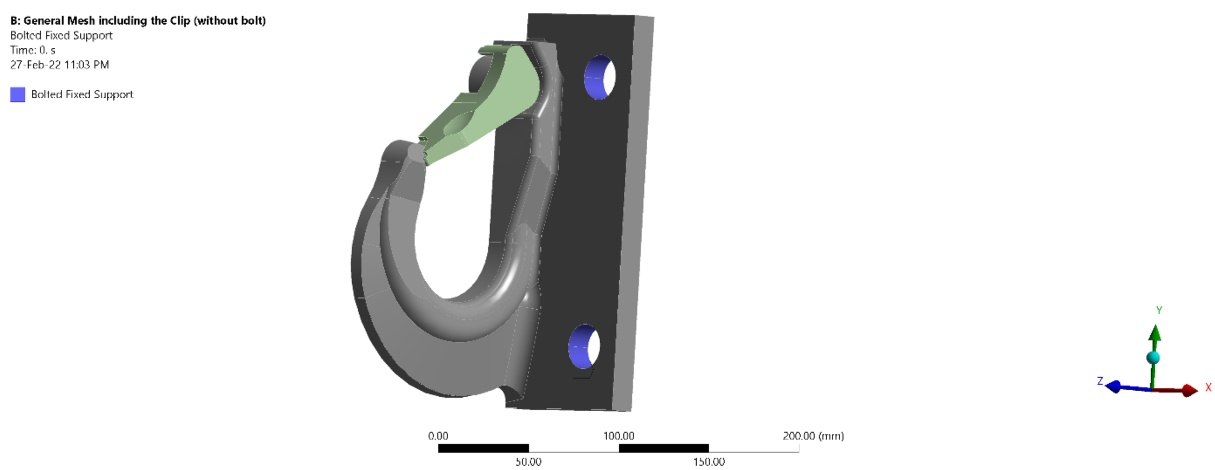


Figure 6: Bolted Fixed Support

A Fixed support on the bolted connections is used to restrict the motion of the plate but more importantly it also restricts the rotation of the part from the positions of the bolts.

6. FEM Model and Simulation

6.1 Symmetry

The part is symmetric about the Global YZ Plane as shown in Figure 7. Therefore, the symmetry command in Design Modeler is used which then automatically evaluates the full part using the mesh of only half of the part as stresses are the same on the other side due to symmetry. This helps in reducing element/node size and makes the evaluation process time faster than it would have been if the entire part was considered.

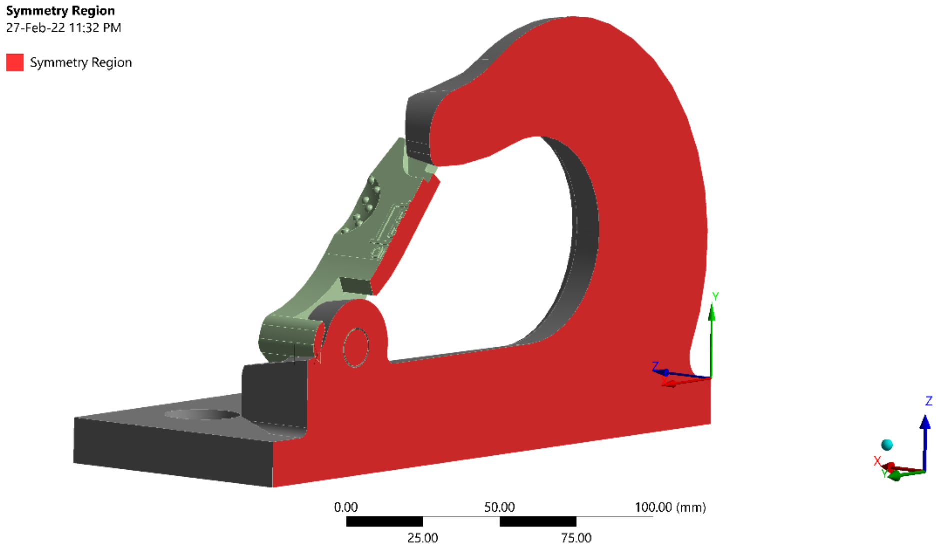


Figure 7: Symmetry Region

6.2 Geometry Preparation

The first thing done in ANSYS is the preparation of the geometry. The Excavator Hook is said to be welded to the base plate and therefore stresses act on it as one part. This is achieved by creating fillets between the Hook and the base plate as shown below in Figure 8

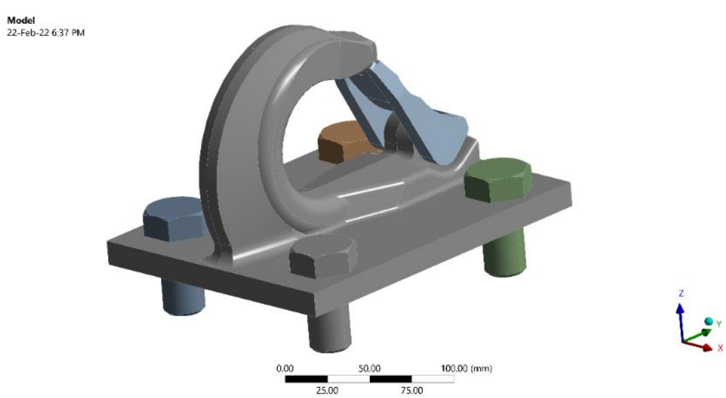


Figure 8: Fillet added to part between Hook and Plate

There still exist however many small sections of faces in the part which are then turned into one seamless face by merging faces together in order to be able to generate a better mesh and not get low quality elements at the extremely small faces. One of the processes is indicated below:

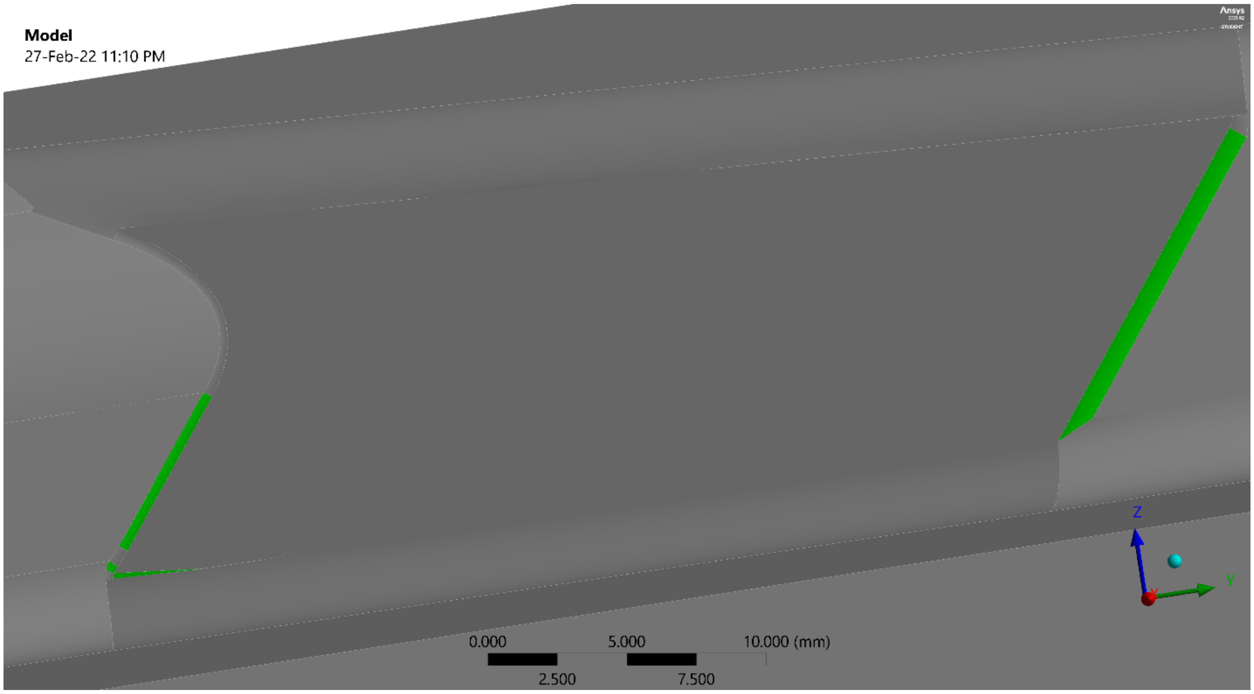


Figure 9: Problematic Faces for Meshing

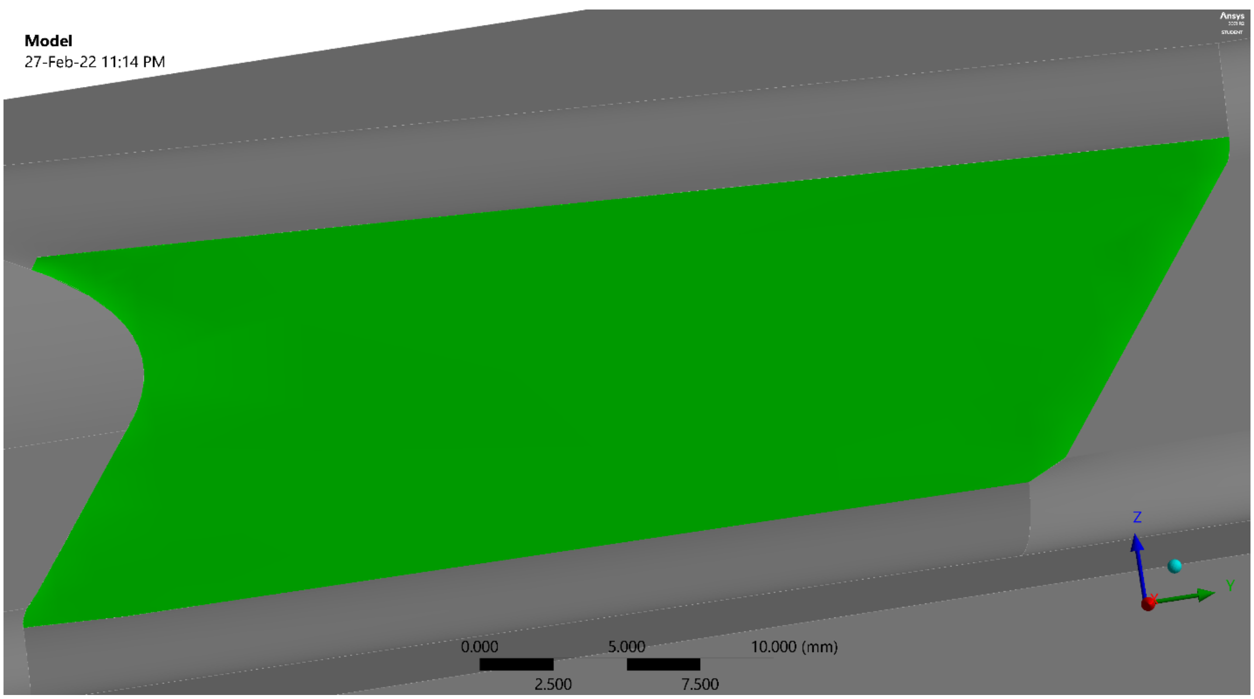


Figure 10: Merged Faces for Meshing

For the mesh to be more accurate and less problematic with higher quality elements in general. Faces of parts not necessary for the geometry are deleted. For example, the circular ball grips on the clip of the hook

Figure 11: Ball Grips on Clip

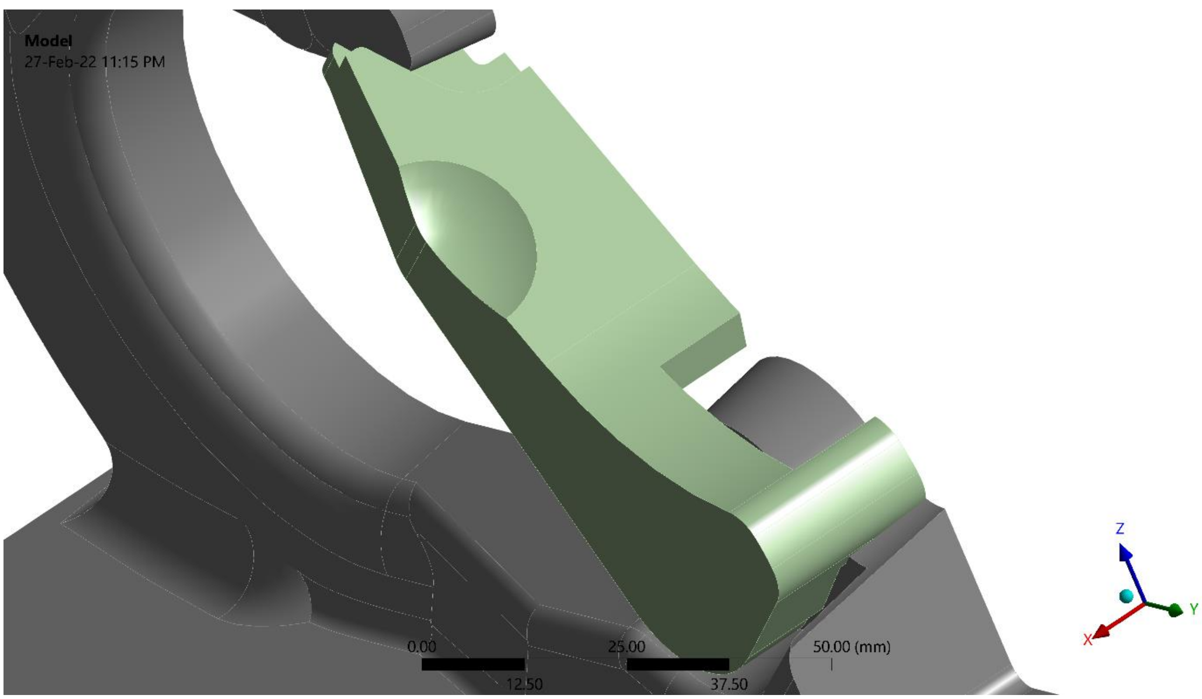


Figure 12: Ball Grips Removed for better meshing

6.3 General Mesh

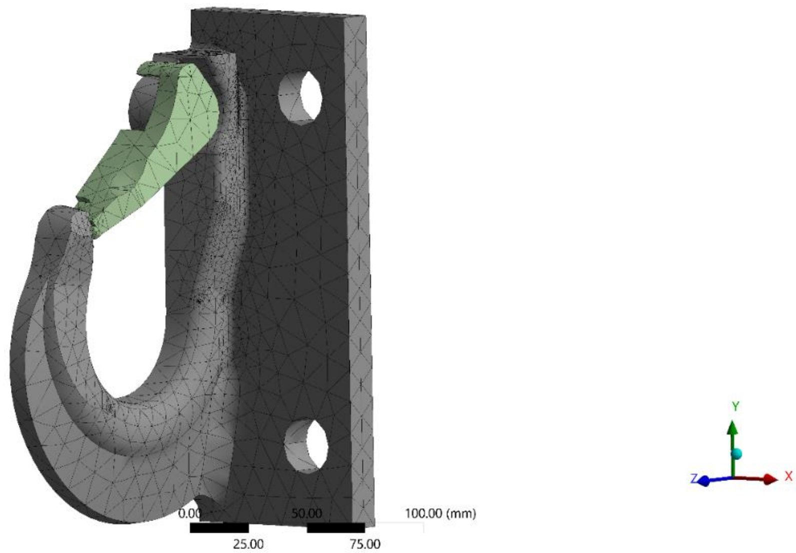


Figure 13: General Mesh with Clip

The part shown above with the clip is initially used to generate a default mesh in order to locate the possible stress hotspots.

However, it is later determined that there almost no viable stresses acting on the clip of the hook therefore it is removed from the geometry for all further simulations in order to make the mesh quality better as the hook had a few problematic faces which could later on be a reason the mesh fails to generate. The stresses on the clip are shown more in detail (6.3.1 Stresses with General Mesh).

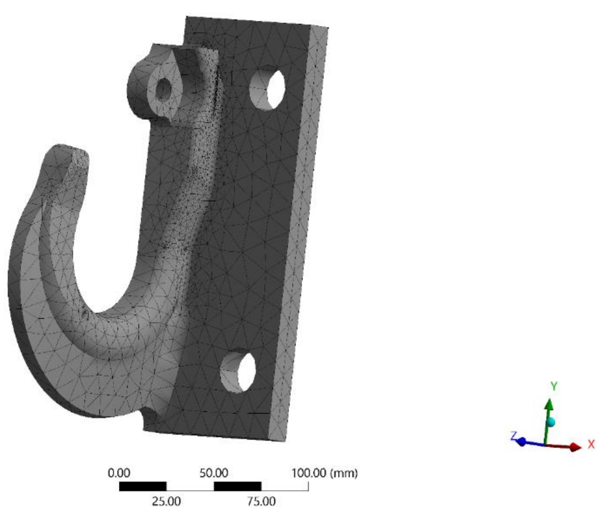


Figure 14: General Mesh without Clip

The above model is therefore used to generate the new general mesh and also used in all subsequent simulations.

6.3.1 Stresses with General Mesh

First as mentioned before, the initial general mesh with the clip is used to evaluate the Averaged and Unaveraged Maximum Average Von-Mises Stress. This mesh helps to determine the possible stress hotspots, these are labelled as the Inner Hook Stress Hotspot and the Fillet Stress Hotspot respectively as shown in Figure 15 below

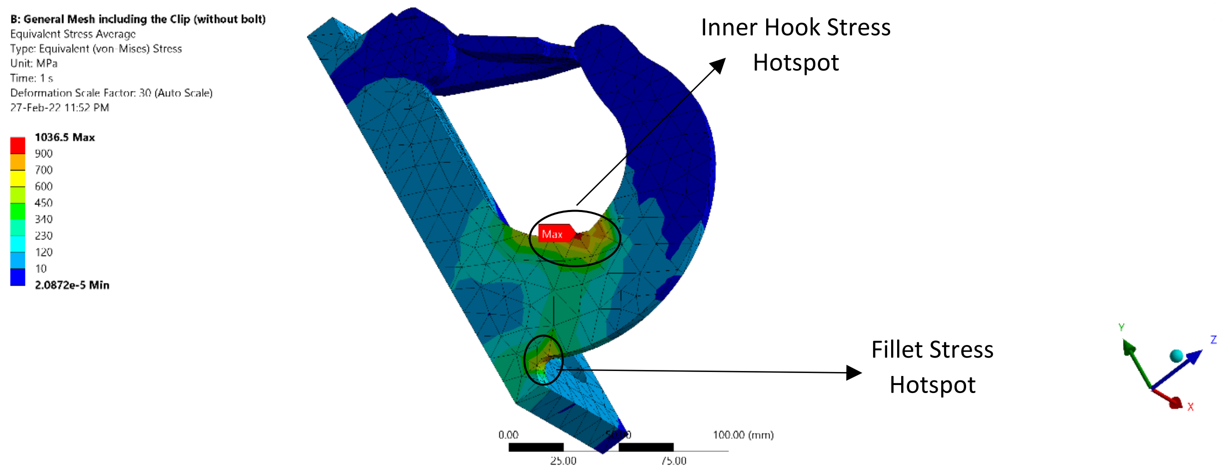


Figure 15: General Mesh stresses with clip

It is determined that there are almost no stresses acting on the clip of the hook as shown below in Figure 16 therefore the clip is then removed from the geometry in order to simplify the mesh.

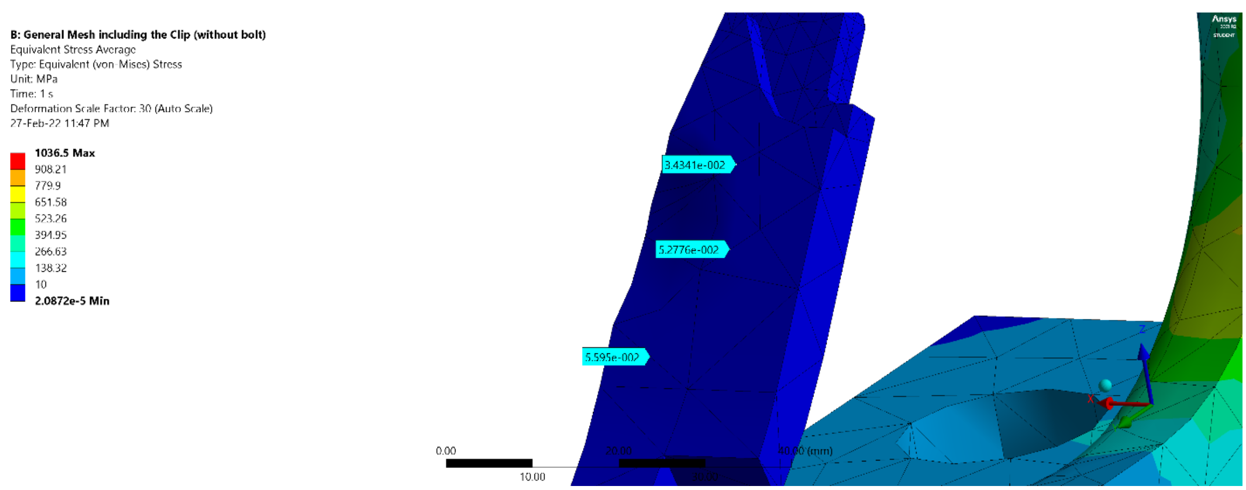


Figure 16: Stresses in Clip

The Average Maximum Von-Mises Stress is recorded as 1036.5 MPa for the general mesh located at the Inner Hook stress Hotspot (Hotspot 1).

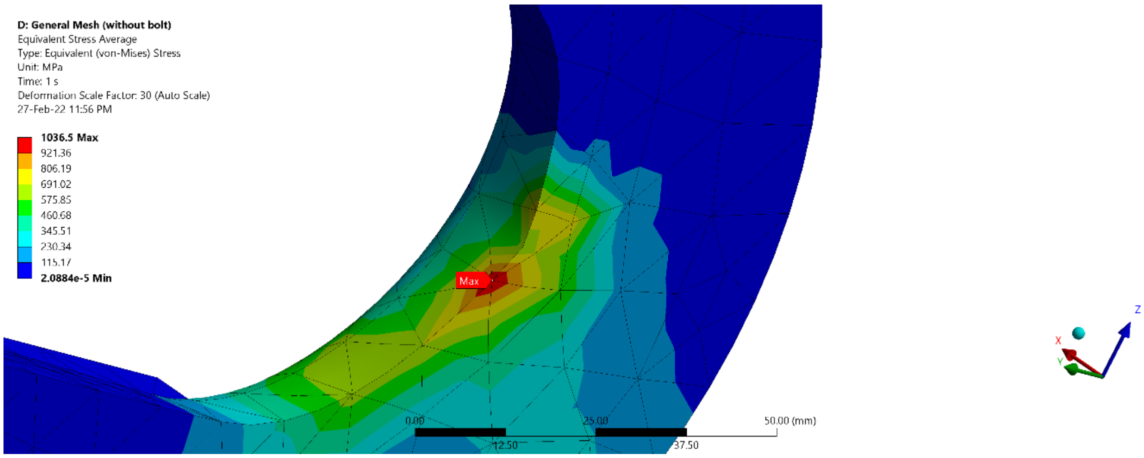


Figure 17: Inner Hook Averaged Von-Mises Stress

The Unaveraged Maximum Von-Mises Stress is recorded as 3132.6 MPa for the general mesh located at the Inner Hook stress Hotspot (Hotspot 1).

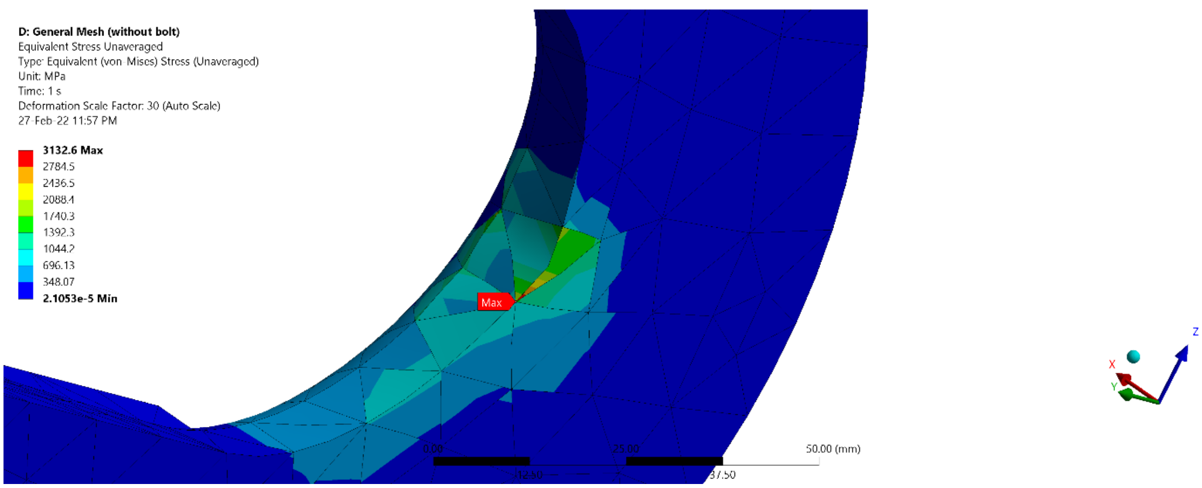


Figure 18: Inner Hook Unaveraged Von-Mises Stress (general mesh)

A second stress hotspot is also seen at the fillet of the hook. With the use of the probe feature it can be seen the Averaged Von-Mises Stress in the region is 905.63 MPa.

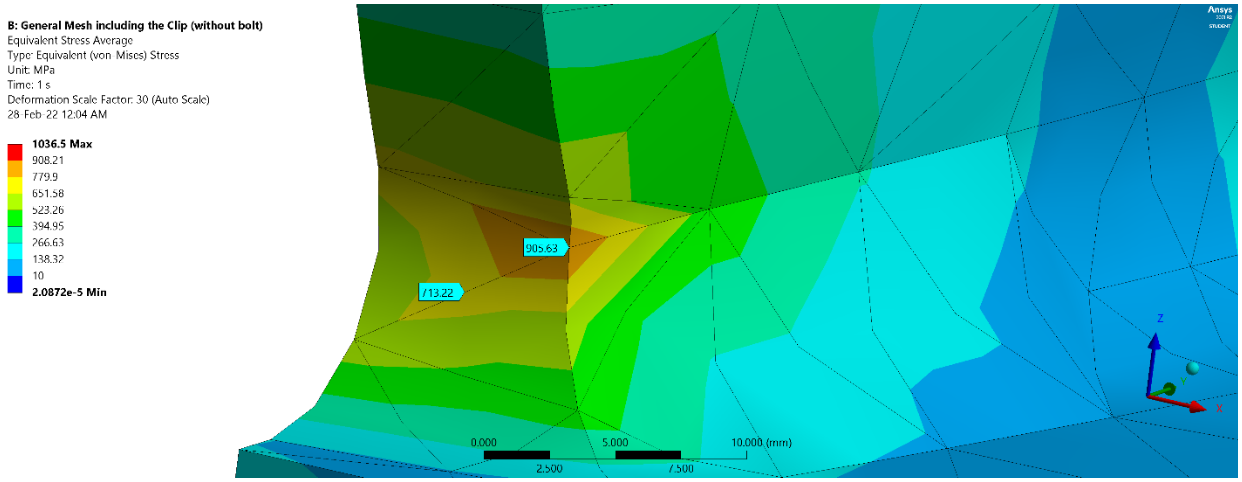


Figure 19: Fillet Averaged Von-Mises Stress (general mesh)

In the fillet stress hotspot (Hotspot 2) using the probe the Unaveraged Von-Mises Stress in the region is 937.91 MPa.

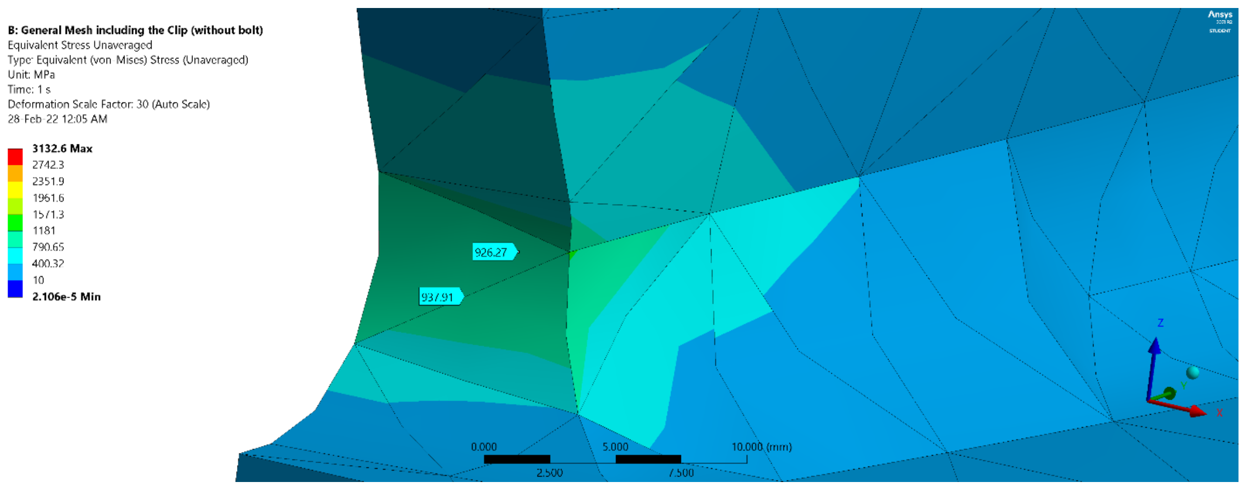


Figure 20: Fillet Unaveraged Von-Mises Stress (general mesh)

The difference in the Average and Unaverage Von-Mises Stress should converge to around less than 3% for it to be determined as an accurate simulation of the stress. In the general mesh this is clearly not the case and therefore a further refinement step is required.

6.4 Preliminary Mesh

It is determined from the general mesh that there are two stress hotspots present. One is at the Inner Hook and the other is located at the fillet.

In order to make the mesh more accurate with higher number of elements at the hotspots, initially a face sizing is applied to both the stress hotspots.

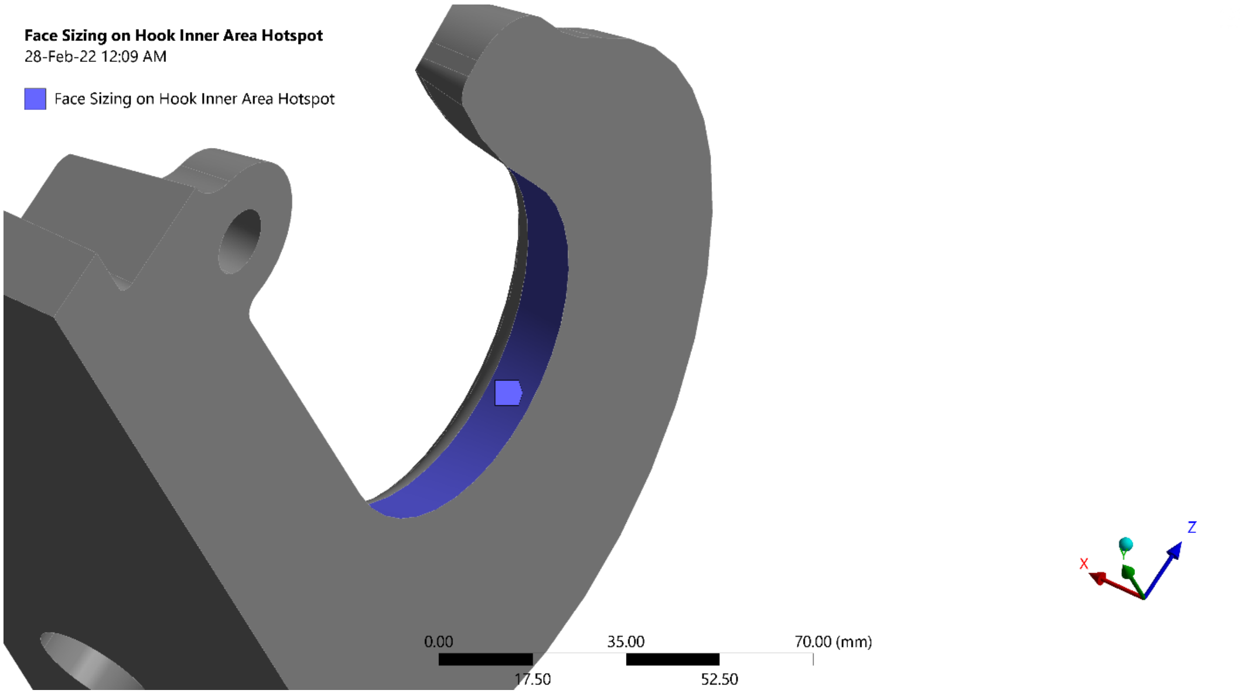


Figure 21: Face Sizing for Inner Hook Stress Hotspot

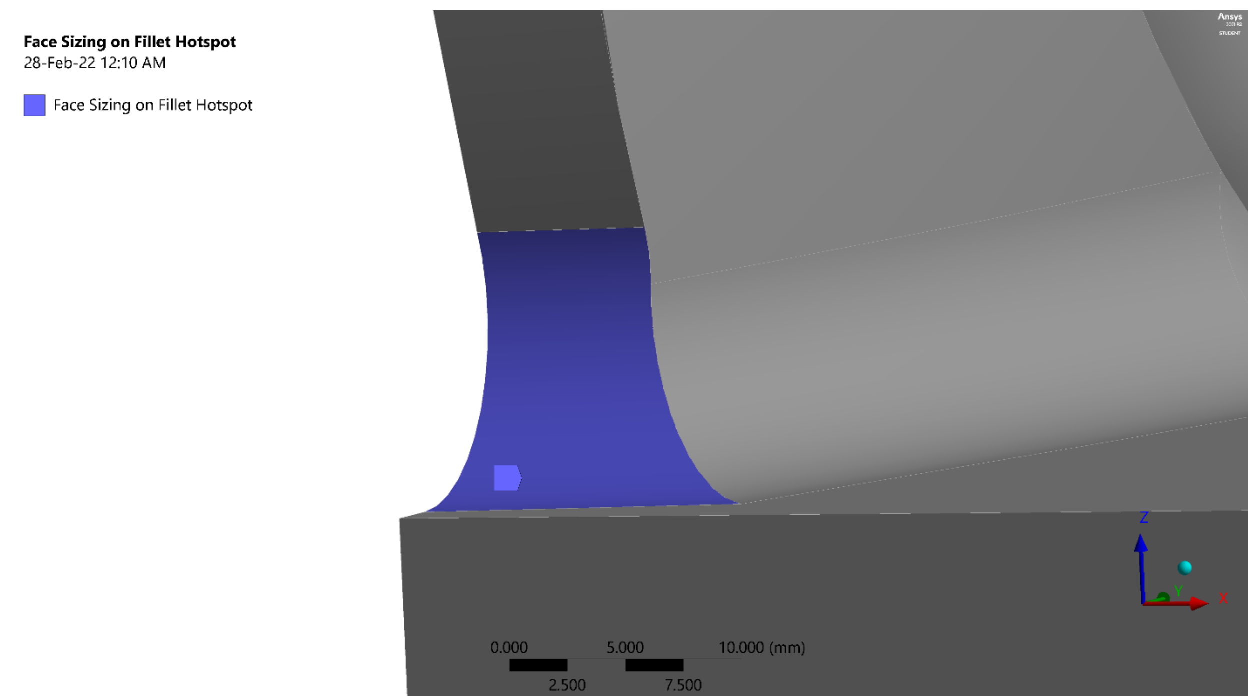


Figure 22: Face Sizing for Fillet Stress Hotspot

Both stress hotspot faces are applied with a face sizing of 3.0 mm. Element size is also changed to 12.0 mm in order higher number of elements overall in the mesh. Adaptive sizing is turned off and a Growth Rate of 1.5 is applied so that elements do not grow extremely rapidly when the mesh transfers from the face sizing at the stress hotspots to the overall mesh of the body.

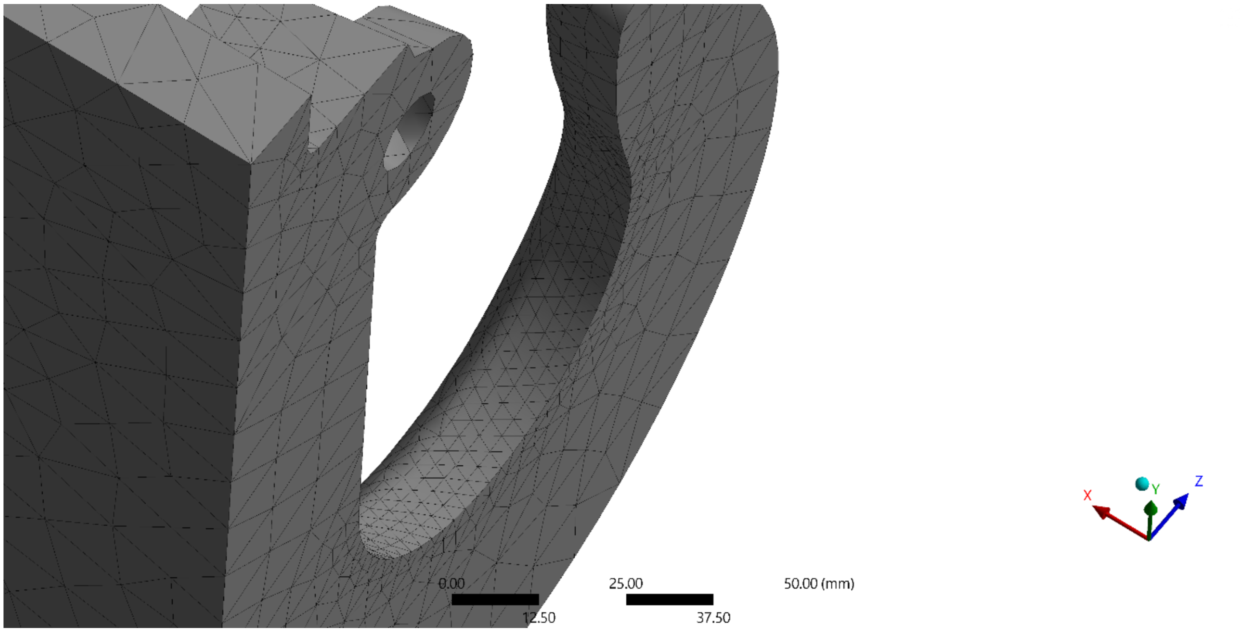


Figure 23: Inner Hook Stress Hotspot Preliminary Mesh

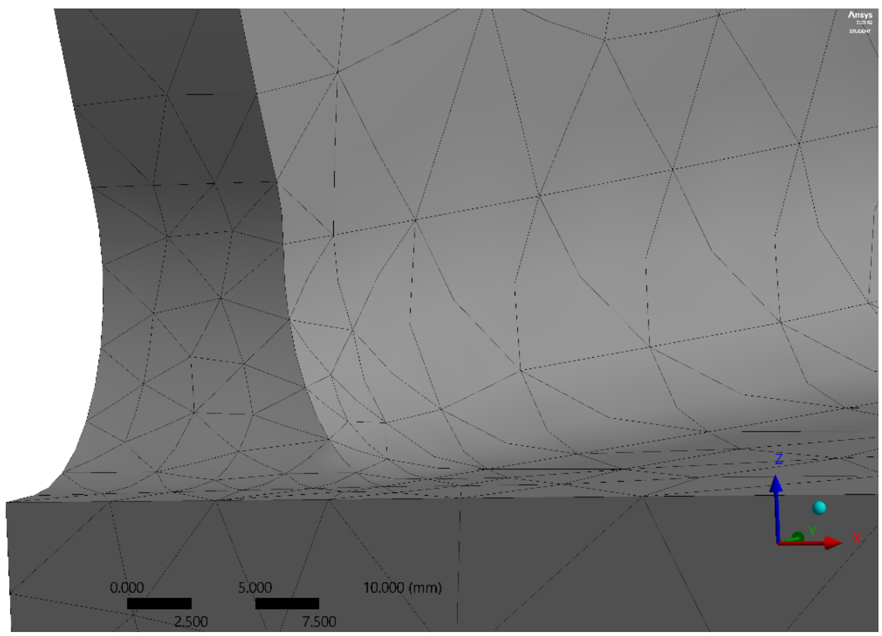


Figure 24: Fillet Stress Hotspot Preliminary Mesh

6.4.1 Stresses with Preliminary Mesh

In order to evaluate the stresses more accurately on the respective stress hotspots' maximums individually, instead of a probe, a slicing in the geometry is performed. Two slices are made at the stress hotspots as shown below:

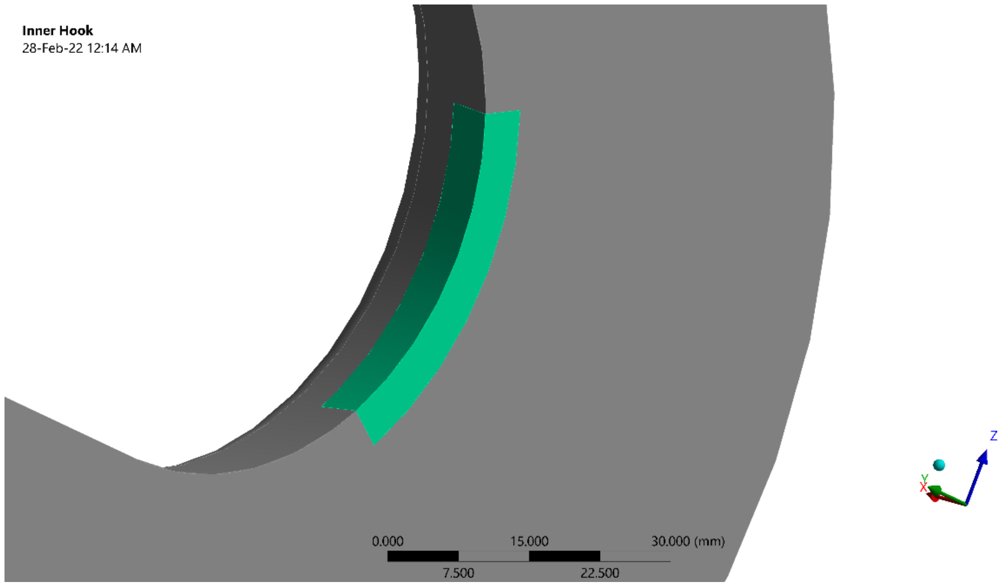


Figure 25: Slice for Inner Hook Stress Hotspot

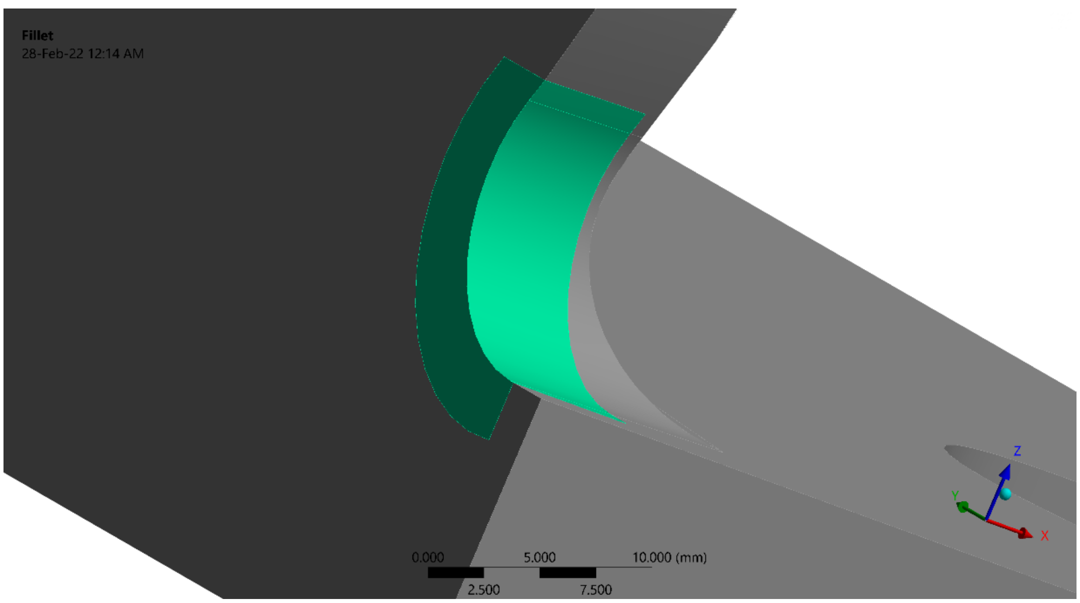


Figure 26: Slice for Fillet Stress Hotspot

These slices later on also help us compare the stresses between the preliminary mesh and refined mesh (6.5.2 Stresses with Refined Mesh).

The Average Maximum Von-Mises Stress is recorded as 1533.4 MPa for the preliminary mesh located at the Inner Hook stress Hotspot (Hotspot 1).

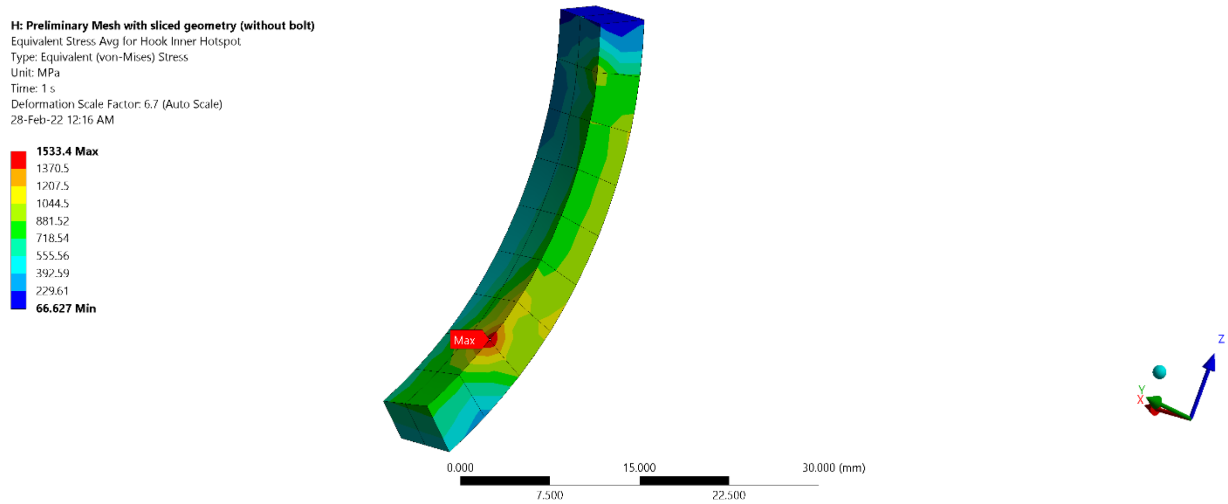


Figure 27: Inner Hook Averaged Von-Mises Stress (preliminary mesh)

The Unaveraged Maximum Von-Mises Stress is recorded as 1849.6 MPa for the preliminary mesh located at the Inner Hook stress Hotspot (Hotspot 1).

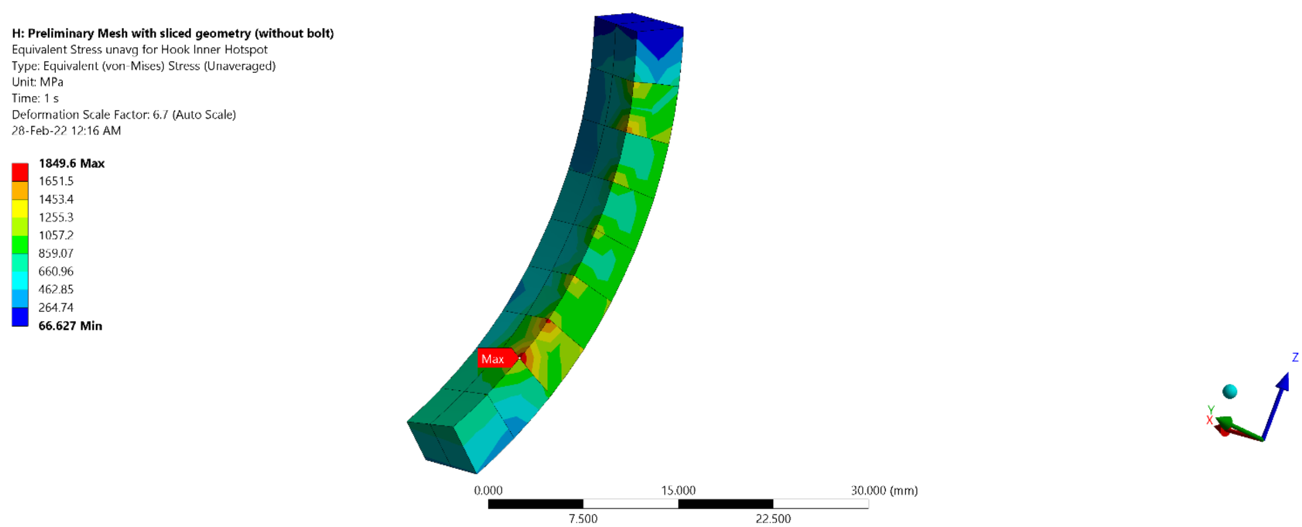


Figure 28: Inner Hook Unaveraged Von-Mises Stress (preliminary mesh)

The Average Maximum Von-Mises Stress is recorded as 1159.8 MPa for the preliminary mesh located at the Fillet stress Hotspot (Hotspot 2).

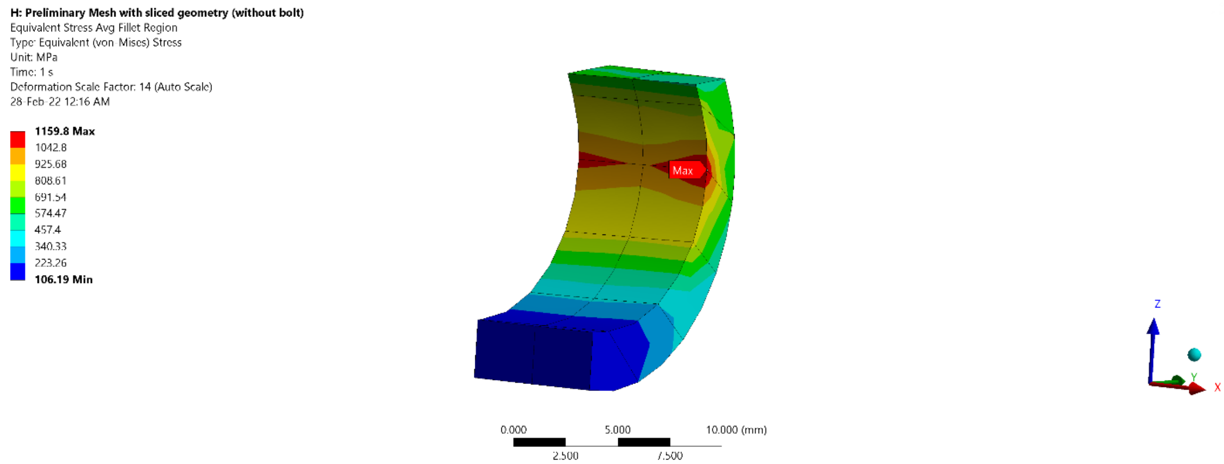


Figure 29: Fillet Hotspot Averaged Von-Mises Stress (preliminary mesh)

The Unaveraged Maximum Von-Mises Stress is recorded as 1172.4 MPa for the preliminary mesh located at the Fillet stress Hotspot (Hotspot 2).

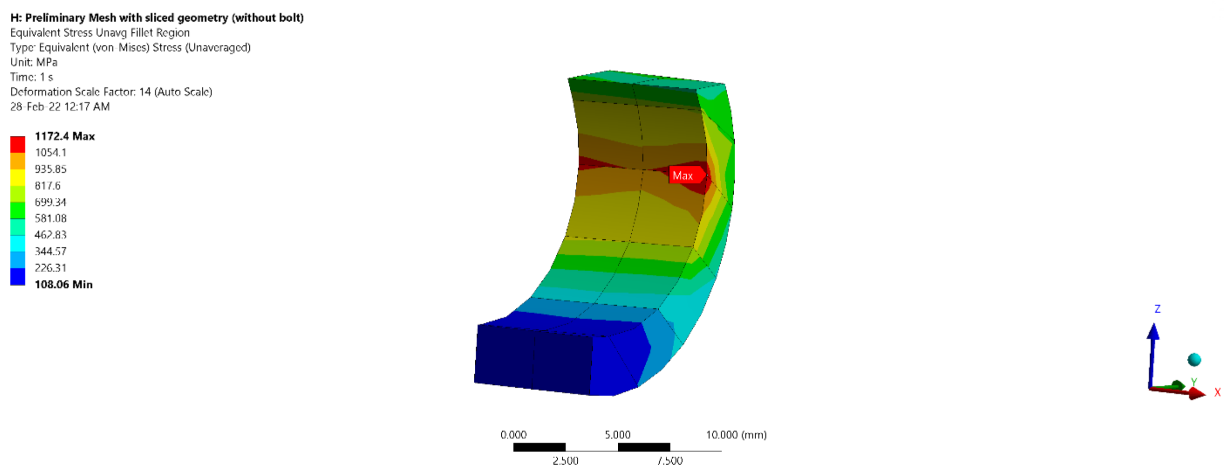


Figure 30: Fillet Hotspot Unaveraged Von-Mises Stress (preliminary mesh)

The stresses do seem to converge especially for the fillet stress hotspot however inner hook stress hotspot still not within the error margin of less than 3% therefore another refining step required.

6.5 Refined Mesh

In order to furthermore refine the mesh, body sizing and custom meshing methods are used on the sliced bodies of the two stress hotspots to obtain better quality elements.

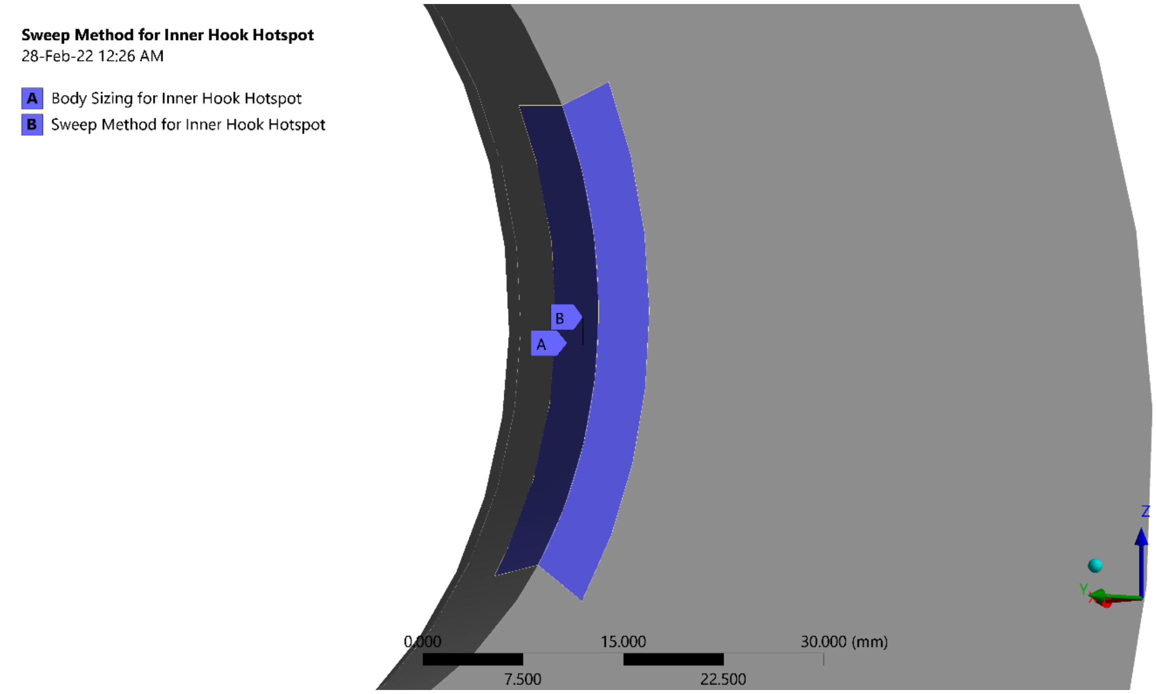


Figure 31: Body Sizing and Sweep Method for Inner Hook slice

Body sizing of 1.0 mm and sweep method is used on the Inner Hook stress hotspot (Hotspot 1).

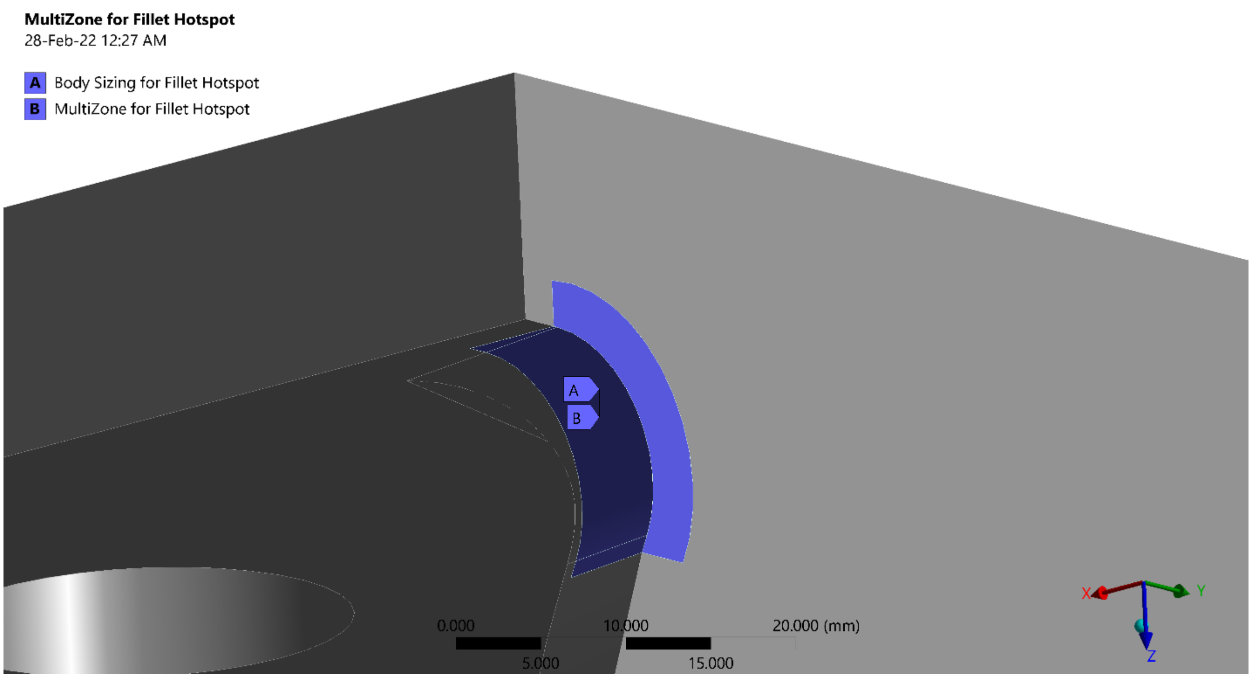


Figure 32: Body Sizing and Multizone Method for Fillet Slice

Body sizing of 2.0 mm and Multizone method is used on the Fillet stress hotspot (Hotspot 2).

6.5.1 Mesh Quality Goals

A high mesh quality is required in the stress hotspots in order to obtain accurate results for the stresses. For all Hex elements a quality of above 50% is sufficient.

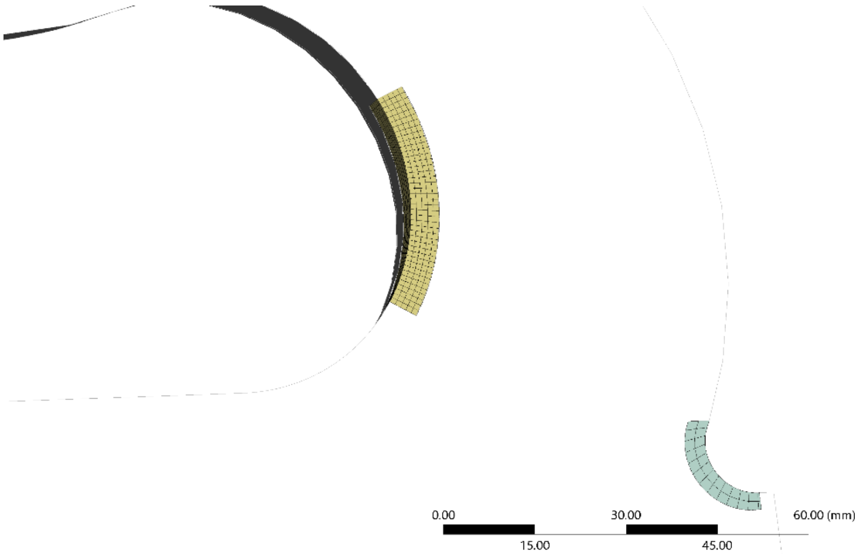


Figure 33: Mesh generated for sliced bodies

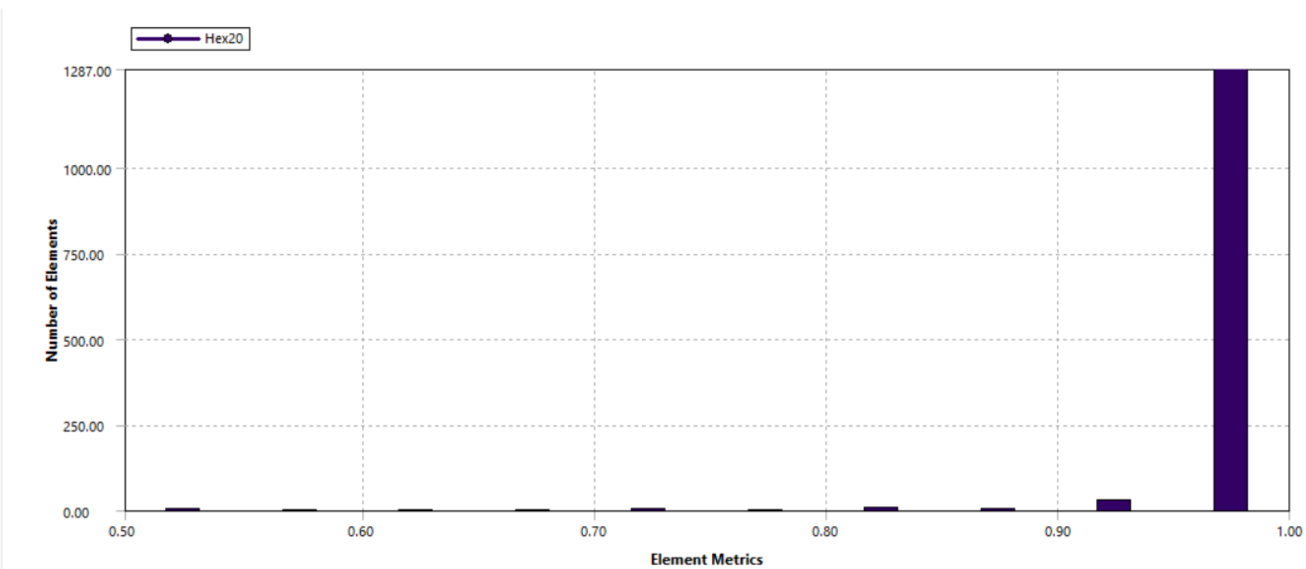


Figure 34: Element Quality > 50% for mesh on sliced bodies

As shown from Figure 33 and Figure 34, only generating meshes for the hotspots the quality is covered by 50% and greater elements therefore the mesh is considered as refined.

6.5.2 Stresses with Refined Mesh

The Average Maximum Von-Mises Stress is recorded as 1943 MPa for the refined mesh located at the Inner Hook stress Hotspot (Hotspot 1).

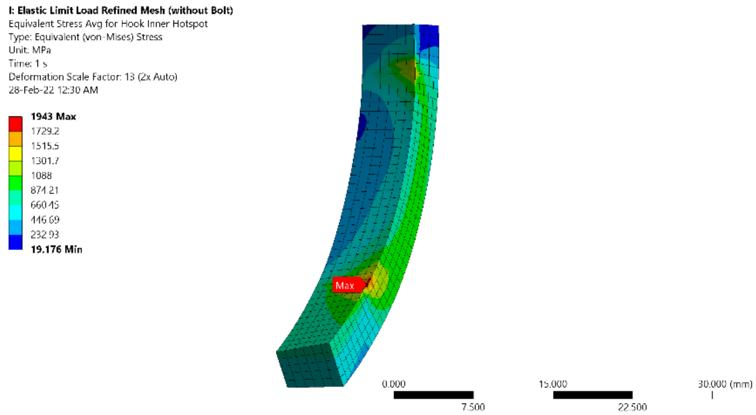


Figure 35: Inner Hook Averaged Von-Mises Stress (refined mesh)

The Unaveraged Maximum Von-Mises Stress is recorded as 2070.7 MPa for the refined mesh located at the Inner Hook stress Hotspot (Hotspot 1).

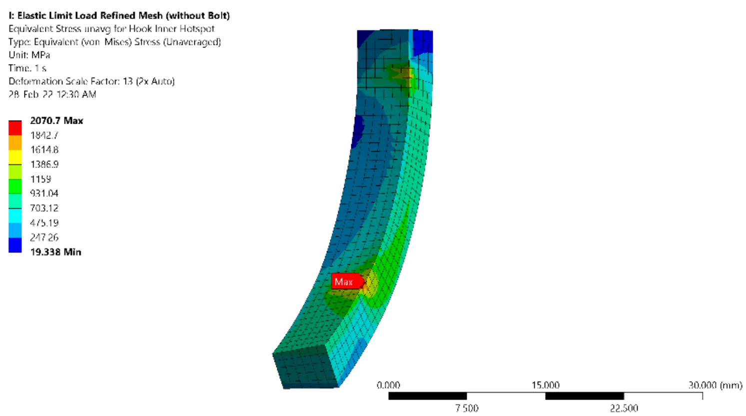


Figure 36: Inner Hook Unaveraged Von-Mises Stress (refined mesh)

The Average Maximum Von-Mises Stress is recorded as 1151.6 MPa for the refined mesh located at the Fillet stress Hotspot (Hotspot 2).

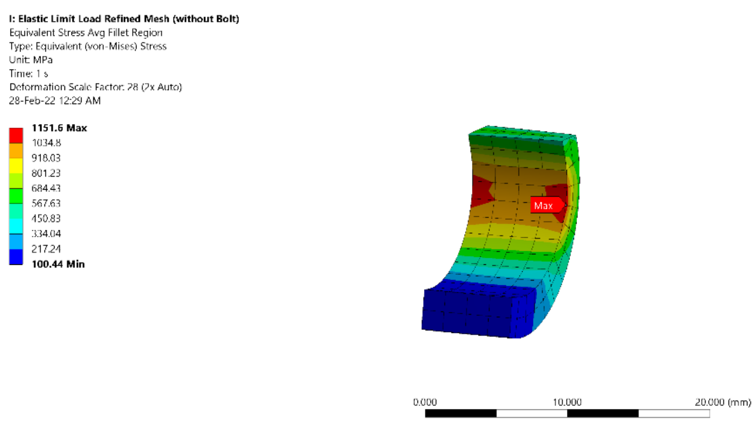


Figure 37: Fillet Averaged Von-Mises Stress (refined mesh)

The Unaveraged Maximum Von-Mises Stress is recorded as 1152 MPa for the refined mesh located at the Fillet stress Hotspot (Hotspot 2).

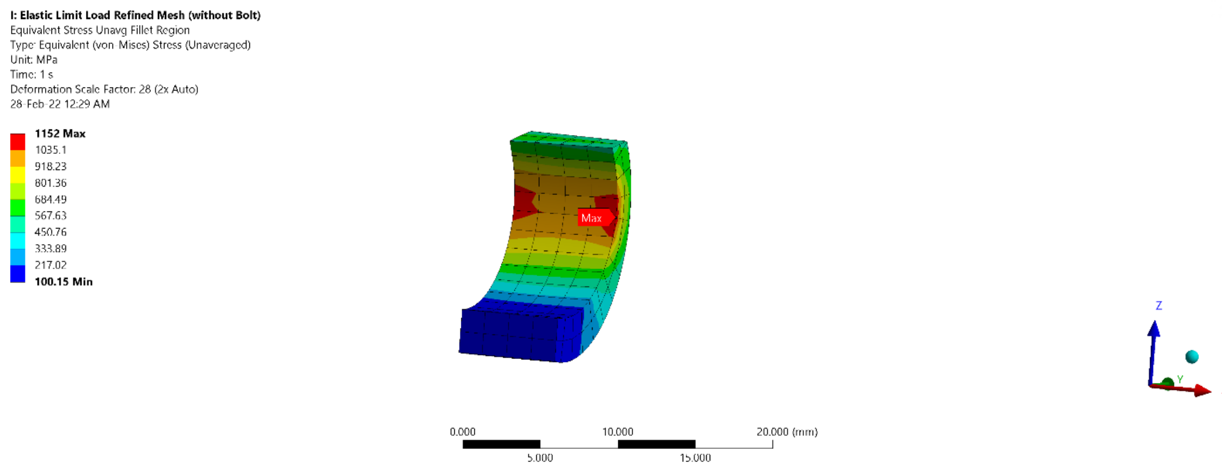


Figure 38: Fillet Unaveraged Von-Mises Stress (refined mesh)

7. Stress Evaluation

All the stresses compared at each mesh are shown in the Table 1 below:

	Nodes	*Averaged Von Mises Stress (Hotspot 1)*	*Unaveraged Von Mises Stress (Hotspot 1)*	*Difference Between Avg and Unavg (Hotspot 1)*	*Averaged Von Mises Stress (Hotspot 2)*	*Unaveraged Von Mises Stress (Hotspot 2)*	*Difference Between Avg and Unavg (Hotspot 2)*
General Mesh	17177	1036.5	3132.6	66.9%	905.63	937.91	3.44%
Preliminary Mesh	41169	1533.4	1849.6	17.1%	1159.8	1172.4	1.07%
Refined Mesh	54573	1943	2070.7	6.2%	1151.6	1152	0.03%

Table 1: Stress Convergence comparing all meshes

As seen from the table the Fillet stress Hotspot already converges in the Preliminary Mesh. For the Hook Inner Stress Hotspot another final refinement is carried out by changing body size from 1mm to 0.75mm however the % Difference between the stresses only reduces to 5.3%.

Therefore, it is deduced as the Mesh around the Hook Inner Stress Hotspot is already covered with all above 90% Mesh Quality Elements, it therefore does not converge and is an artificial singularity.

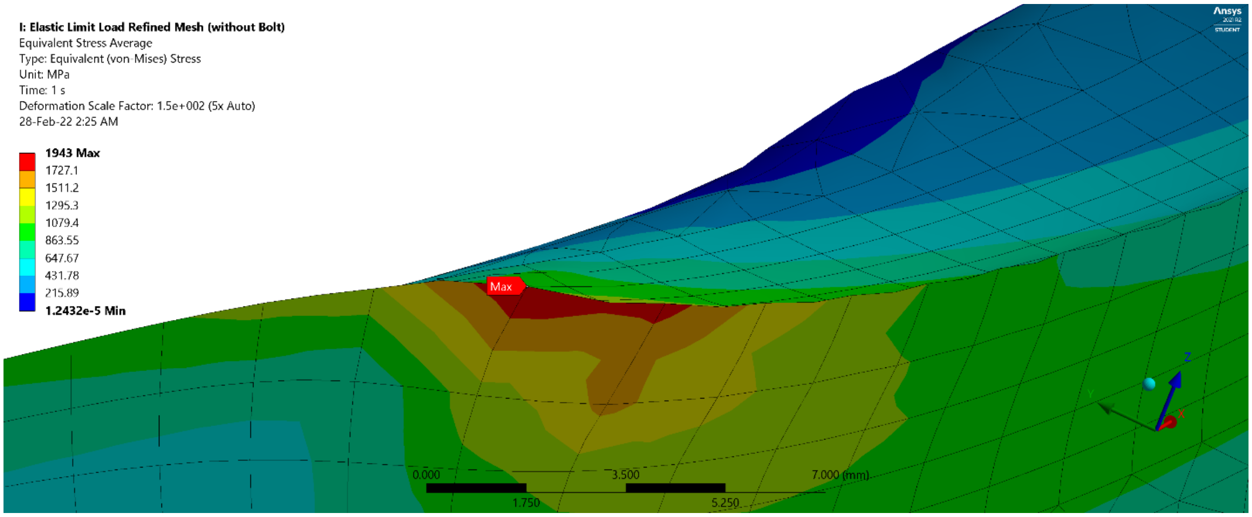


Figure 39: Artificial Singularity at Inner Hook Stress Hotspot

Even though the Stress of the Hook Inner Stress Hotspot does not converge, the region around it converges as shown from the probe feature in Figure 40 and Figure 41 , The stress from probes in the Averaged and Unaveraged Von-Mises Stress is compared. The stress acting in this region is approximated to be **1650 MPa**. As this stress is higher than the fillet stress hotspot it is important and will be considered for the Material Selection.

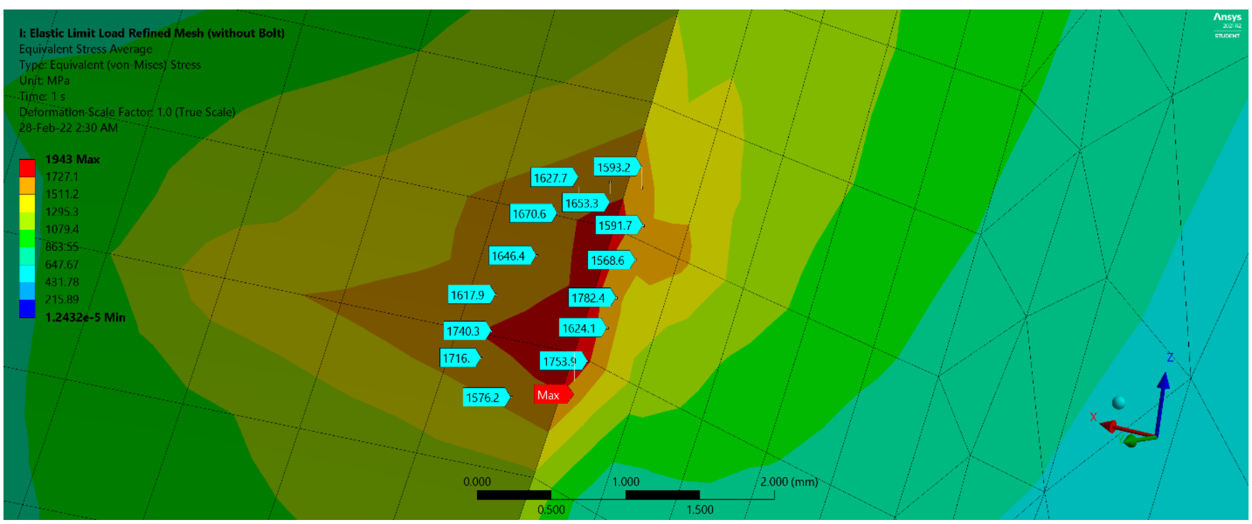


Figure 40: Probes used to approximate stress in Inner Hook Hotspot (Averaged)

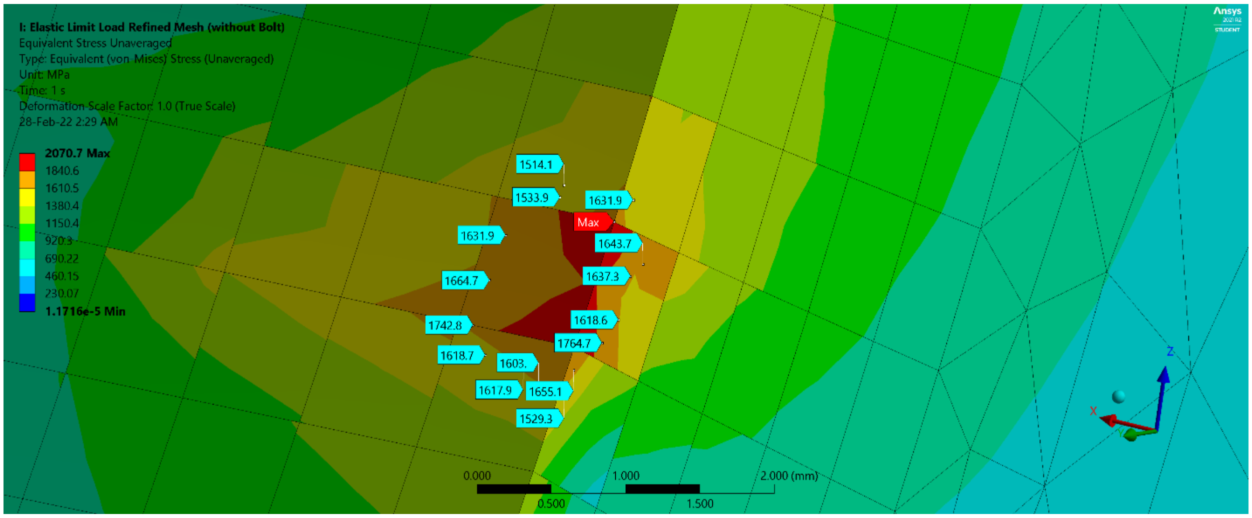


Figure 41: Probes used in order to approximate stress in Inner Hook Hotspot (Unaveraged)

8. Material Selection

The maximum stress considered to be acting on the part in our given load case is 1650 MPa and so the material 30CrNiMo8 with proof strength 1050 MPa is selected according to the (FKM Guideline, 2012). The material for the determined stress will yield after 1050 MPa however it should not fracture for our load case. As the load is 4 x Nominal Load allowed for the Hook, it can be assumed that these stresses will not be experienced frequently so allowing the part to yield at the stress is acceptable. The yielding will be determined in detail in

9. Elastic Ideally Plastic Simulation. The outline in Figure 42 shows the original position of the hook. As seen from below the yielding for an extreme load condition seems to be reasonable.

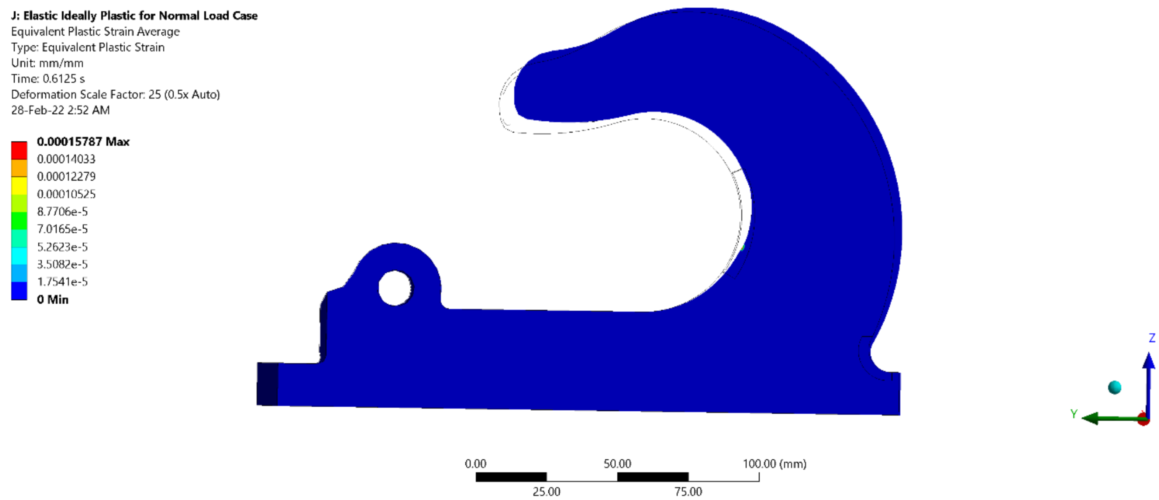


Figure 42: Strain Analysis for Given Load Case

8.1 Material Properties for Strain Analysis

For the strain analysis the Material Properties for the Elastic Ideally Plastic material has to be changed so that the part plastically deforms according to the material selected.

As the material selected is 30CrNiMo8, its properties of Young's Modulus = 210 GPa, Poisson's Ratio = 0.3 are chosen. The behavior is considered to be bilinear so that the simulation runs smoothly which means the material has one section which behaves linearly elastic and then after the elastic limit behaves linearly plastic. For this the proof strength of 1050 MPa has to be entered into ANSYS and a Tangent Modulus of $\frac{1}{1000} * 210000\ MPa$ = 210 MPa is used for the strain analysis.

Properties of Outline Row 3: 30CrNiMo8 elastic ideally plastic

	A	B	C
1	Property	Value	Unit
2	Material Field Variables	Table	
3	Isotropic Elasticity		
4	Derive from	Young's Modulus and Poisso...	
5	Young's Modulus	2.1E+05	MPa
6	Poisson's Ratio	0.3	
7	Bulk Modulus	1.75E+05	MPa
8	Shear Modulus	80769	MPa
9	Bilinear Isotropic Hardening		
10	Yield Strength	1050	MPa
11	Tangent Modulus	210	MPa

Table 2: Material Properties in ANSYS for Elastic Ideally Plastic 30CrNiMo8

9. Elastic Ideally Plastic Simulation

A Strain Analysis is carried out on the part with the refined mesh by creating a duplicate system in ANSYS in order to obtain the Plastic Load Factor. For the Plastic Load Factor, the values of the time-step for the Collapse Load and time step for the onset yielding is required. To get the part to collapse in the simulation a load of 500000 N is applied.

Using Table 3 shown below obtained from the Strain Analysis, a suitable time step, where the part collapses, is selected and the time step where the first deflection occurs is determined to be the onset yielding point.

	Time [s]	Minimum [mm/mm]	Maximum [mm/mm]	Average [mm/mm]
1	5.00E-02	0	0	0
2	0.1	0	0	0
3	0.15	0	2.29E-03	4.41E-07
4	0.2	0	9.84E-03	6.68E-06
5	0.25	0	2.45E-02	5.87E-05
6	0.3	0	6.67E-02	4.09E-04
7	0.31664	0	9.67E-02	6.75E-04
8	0.33327	0	0.16706	1.12E-03
9	0.34327	0	0.25797	1.56E-03
10	0.35327	0	0.41922	2.32E-03
11	0.36327	0	0.6459	3.74E-03
12	0.37327	0	0.94587	6.76E-03
13	0.38327	0	1.3076	1.27E-02
14	0.39327	0	1.6881	2.11E-02
15	0.40327	0	2.1168	3.09E-02
16	0.41327	0	2.5292	4.17E-02
17	0.42327	0	2.9253	5.35E-02
18	0.43327	0	3.3042	6.59E-02
19	0.44327	0	3.6771	7.94E-02
20	0.45327	0	4.0356	9.33E-02
21	0.46327	0	4.3925	0.10817
22	0.47327	0	4.7411	0.12356
23	0.48327	0	5.0925	0.13996
24	0.49327	0	5.4396	0.15693
25	0.50327	0	5.7931	0.175
26	0.51327	0	6.1485	0.19391
27	0.52327	0	6.5094	0.21389
28	0.53327	0	6.8769	0.23511
29	0.54327	0	7.2504	0.25779
30	0.55327	0	7.6278	0.28208
31	0.56327	0	8.0121	0.30822
32	0.57327	0	8.429	0.33624
33	0.58327	0	8.8537	0.36602
34	0.59327	0	9.2887	0.39763
35	0.60327	0	9.7336	0.43111
36	0.61327	0	10.186	0.46652
37	0.62327	0	10.643	0.50397
38	0.63327	0	11.1	0.54338
39	0.64327	0	11.551	0.58464
40	0.65327	0	11.993	0.62744
41	0.66327	0	12.421	0.67145
42	0.67327	0	12.834	0.71632

Table 3: Yielding for Collapse Load

Time step determined for collapse limit: 0.67327 s

Time step determined for onset plastic yielding: 0.15 s

9.1 Calculation of Plastic Load Factor

The Plastic Load Factor, k_p , is given as follows:

$$k_p = \frac{Time\ Step\ Collapse\ Limit}{Time\ Step\ Onset\ Plastic\ Yielding}$$

$$k_p = \frac{0.67327\ s}{0.15\ s}$$

$$k_p = 4.49$$

9.2 Degree of Utilization of the Excavator Hook

The Degree of Utilization for the part, A_{Hook}, is calculated using the following formula:

$A_{Hook} = \frac{\sigma_{eqv}}{Re_N/SF} / \, k_p$ where,

σ_{eqv} *is the equivalent stress*

Re_N is the proof strength of the material

SF is the Safety factor to be used

k_p is the Plastic Load Factor

The Safety Factor to be used is 2 which is used from the (RUD Handbook, 2017) of the Excavator Hook (see Figure 4)

The above formula is entered into ANSYS using the user defined result and the maximum Degree of Utilization is determined to be 82.5% as shown in the Figure 43 below

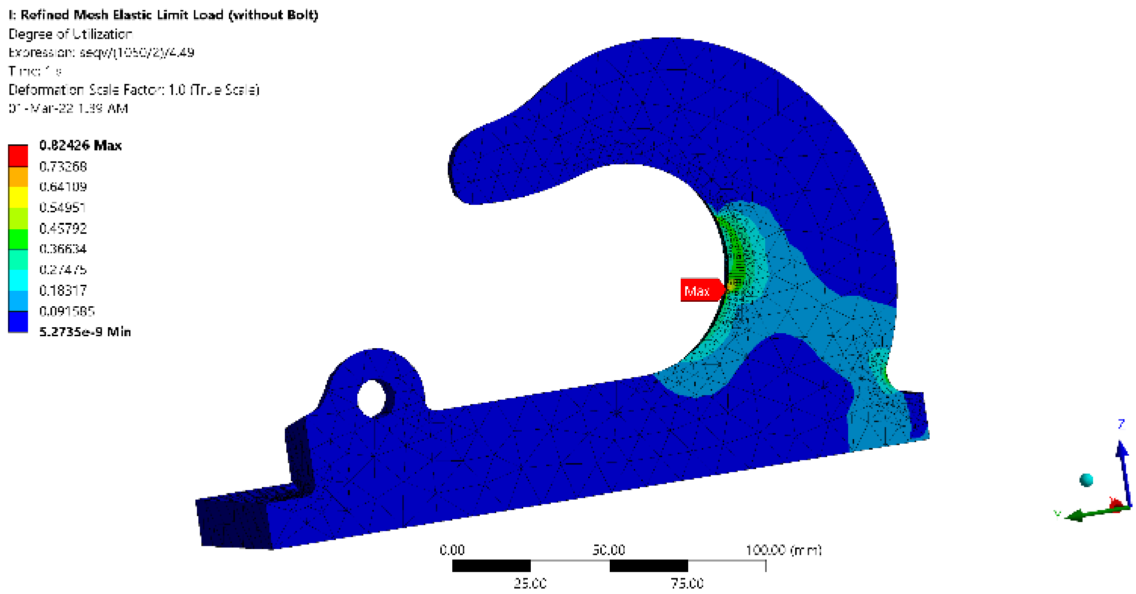


Figure 43: Degree of Utilization of the Part

10. Bolt Pretension

10.1 Geometry of the Bolt

In the part two of the bolts are unsuppressed from the initial geometry in order to evaluate the bolts. It is to be noted that anything to be applied throughout this section applied to Bolt 1 is also applied to Bolt 4 as they both are the same.

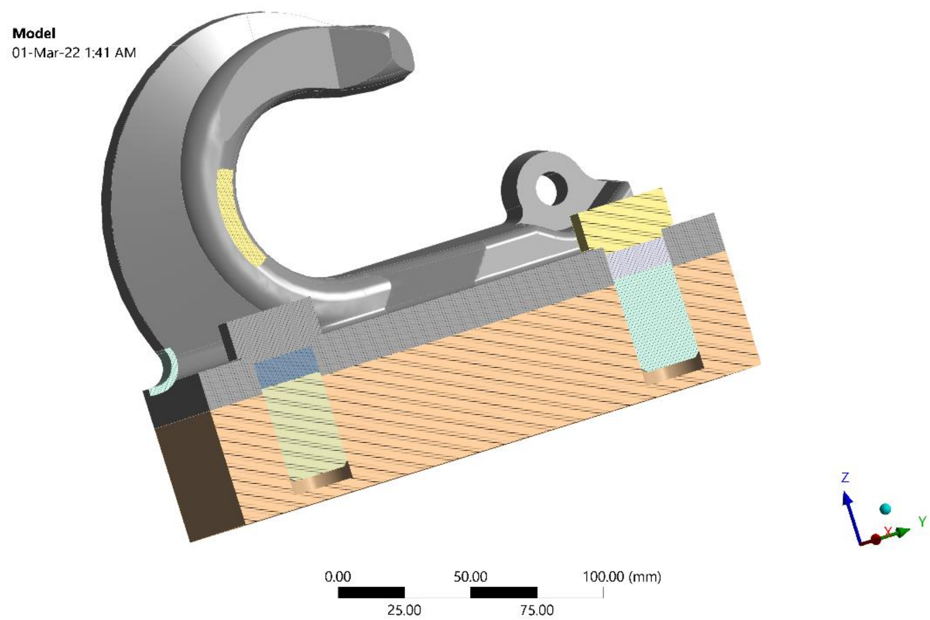


Figure 44: Bolt Geometry

The bolts are sliced into three sections in order to properly simulate the pretension in ANSYS. The three parts contain the Bolt Heads, Bolt Midsections and the Threads respectively.

10.2 Contacts for the Bolts and the Excavator Hook

The simulation consists of 9 contacts in total. 4 each for the two bolts and 1 contact from Hook to Lifted Body.

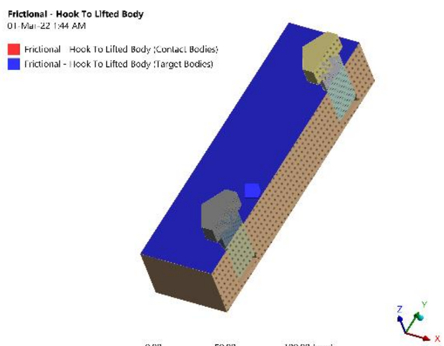


Figure 45: Hook to Lifted Body Contact (Lifted Body)

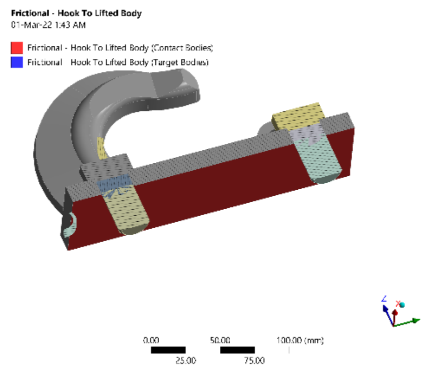


Figure 46: Hook to Lifted Body Contact (Hook)

The contact between the Hook and the lifted body is considered as frictional with coefficient of friction of 0.2 as they are two rigid bodies.

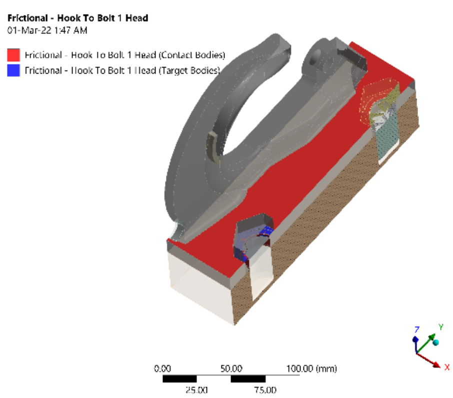


Figure 47: Bolt Head to Hook Contact (Hook)

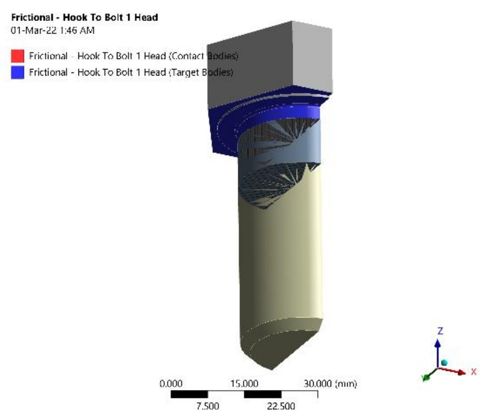


Figure 48: Bolt Head to Hook Contact (Bolt Head)

The contact between the Hook and the Bolt Heads is considered as frictional with coefficient of friction of 0.2 as they are two rigid bodies. (For both Bolts 1 and 4).

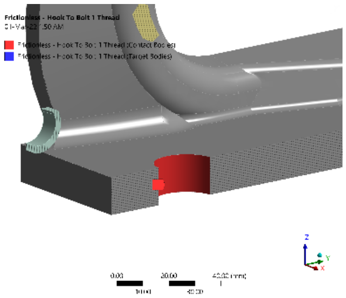


Figure 49: Bolt Midsection to Hook Contact (Hook)

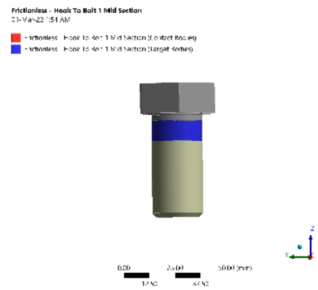


Figure 50: Bolt Midsection to Hook Contact (Bolt)

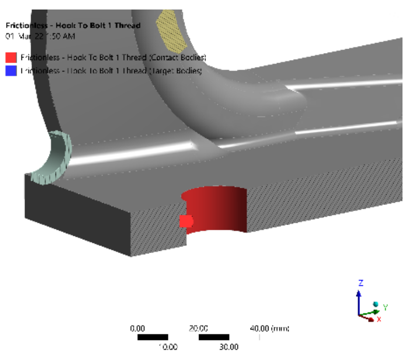


Figure 51: Hook the Bolt Thread Contact (Hook)

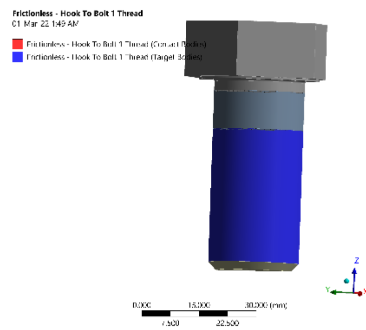


Figure 52: Hook to Bolt Thread Contact (Bolt)

Both these contacts for Bolts 1 and 4 are considered to be frictionless as normally there would be a small gap between the bolts and non-threaded hole of the hook.

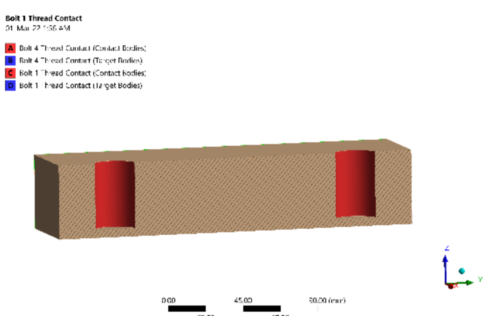


Figure 53: Thread Contact (Lifted Body)

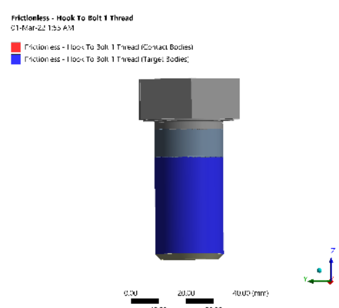


Figure 54: Thread Contact (Bolt)

The contact is selected to be an MPC bonded contact for both Bolts 1 and 4 as it is a threaded connection.

10.3 Meshing of the Bolts

The Mesh on the bolts is improved in order to accurately determine the pretension loads on the bolts.

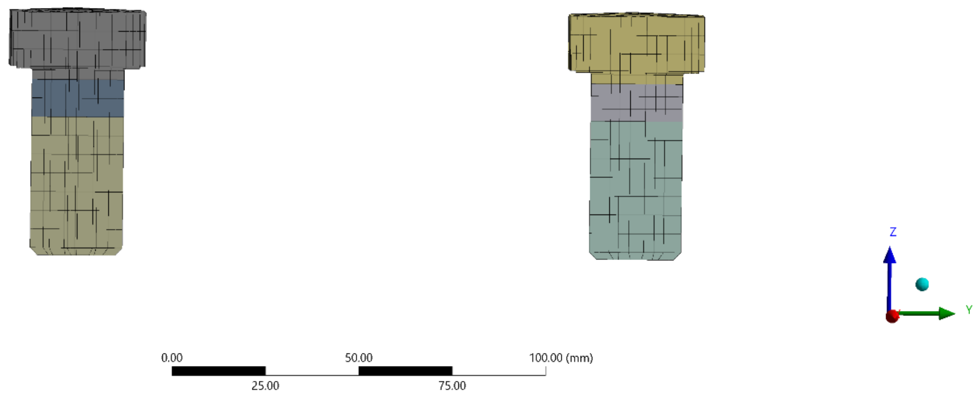
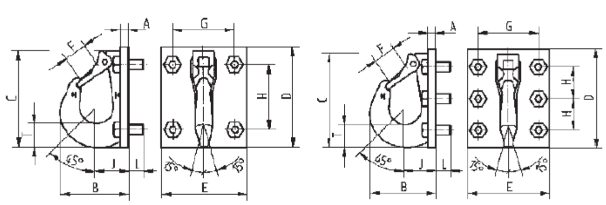


Figure 55: Bolt Mesh

Sweep Method is used on the Bolt Mid-sections and Multizone on the Bolt Head and Thread. Body Sizing for all 3 sliced bodies of the are chosen to be 5 mm. The Mesh quality is also quite high with hex dominant above 50% except some elements at the Bolt Head base which are low quality probably due to the sharp change in edges.

10.4 Degree of the Utilization of the Bolt

VCGH-G 6 up to 16 VCGH-G 20 and 22



Type	WLL t	weight kg	A	B	C	D	E	F	G	H	J	L	M	T	torque Nm	**ref-no.**	safety latch
VCGH-G 6 *	1.5	0.9	10	68	92	100	100	25	70	60	37	15	4xM12	20	100	*	7100299
VCGH-G 8 *	2.5	1.7	10	84	125	135	110	30	80	75	41	15	4xM12	28	100	*	7100300
VCGH-G 10 *	4	2.4	12	106	148	155	130	35	95	100	50	23	4xM16	36	240	*	7100301
VCGH-G 13 *	6.5	4.4	15	120	170	185	160	40	110	120	60	25	4xM20	38	450	*	7100302
VCGH-G 16	10	6.4	15	141	200	220	170	48	120	150	70	35	4xM24	50	800	**7984048**	7100303
VCGH-G 20	16	10.4	20	187	272	288	210	63	150	110	87	30	6xM24	70	800	**7984311**	7101604
VCGH-G 22	20	17.5	20	196	276	292	240	63	150	110	92	30	6xM24	75	950	**7984312**	7101604

Table 4: (RUD Handbook, 2017) Values for Bolt Torque (VCGH-G 16)

Using the torque defined as 800 Nm in the rulebook for a 10.9 M24 Bolt the (VDI Guideline, 2015) of High Strength Bolted Connections gives a value of 187 kN as the pretension load.

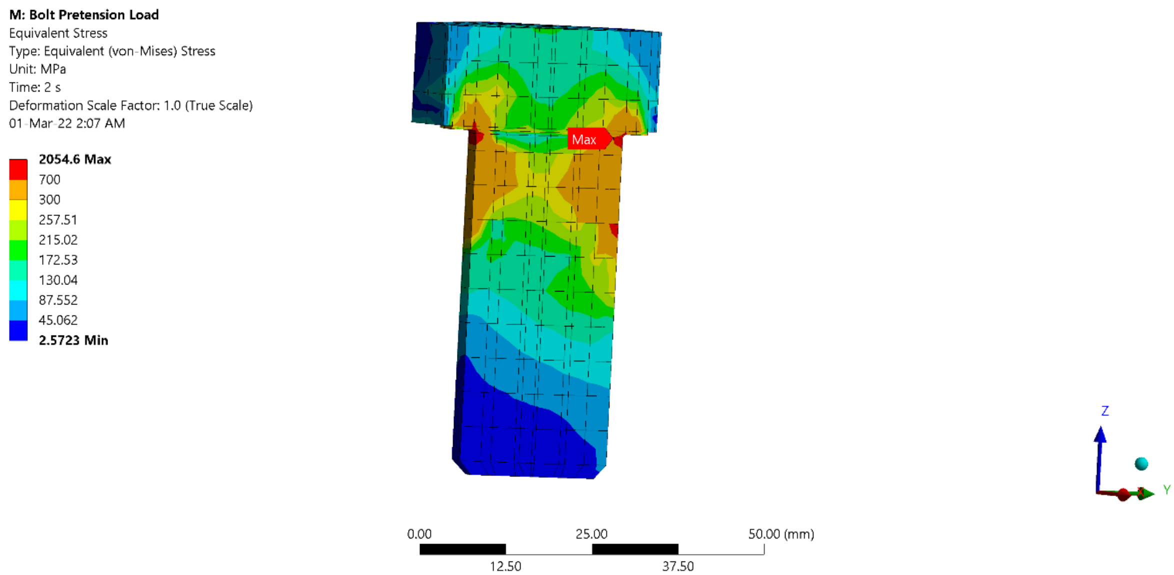


Figure 56: Von-Mises Stress for Pretension Load

It can be seen that the maximum Von-Mises Stress is noted as 2054.6 MPa however this is only due to the sharp edge in the geometry and the contact stresses that, an artificial singularity occurs. It can be seen from the scaling of the stress that the stress hardly goes above 700 MPa (Red Region)

	Time [s]	Minimum [N]	Maximum [N]
1	1.e-002	0.	0.
2	2.e-002	0.	0.
3	3.5e-002	0.	0.
4	5.75e-002	0.	0.
5	9.125e-00:	0.	0.
6	0.14188	0.	0.
7	0.21781	0.	0.
8	0.33172	0.	0.
9	0.50258	0.	0.
10	0.75887	0.	0.
11	1.	0.	0.
12	1.01	1.8698e+005	1.8702e+005
13	1.02	1.8696e+005	1.8705e+005
14	1.035	1.8692e+005	1.8708e+005
15	1.0575	1.8688e+005	1.8706e+005
16	1.0912	1.868e+005	1.8703e+005
17	1.1419	1.867e+005	1.8699e+005
18	1.2178	1.8655e+005	1.8698e+005
19	1.3317	1.8637e+005	1.8705e+005
20	1.5026	1.8614e+005	1.8736e+005
21	1.7589	1.8582e+005	1.8815e+005
22	2.	1.8581e+005	1.8965e+005

Table 5: Working Load Table for Pretension Load

The working load for the bolt is analyzed in ANSYS and then the Degree of Utilization of the bolt, A_{Bolt}, is calculated as follows:

F0: 190000 N

F1: 187000 N

$Df = F0 - F1$

$A_s = 353\ mm^2$ (Area of M24 Bolt)

d = 24 mm

$$\sigma_a = \frac{Df}{2 * A_s}$$

$$\sigma_{asv} = 0.85(\frac{150}{d} + 45)$$

$$A_{Bolt} = \frac{\sigma_a}{\sigma_{asv}}$$

The degree of utilization, A_{Bolt}, in this case is calculated to be 8.55%.

11. Conclusion

The Material is selected considering the maximum stress that the part experiences for the given Load Condition (5.1 Design Load Case). It is determined from the results shown above that the material 30CrNiMo8, although does have a lower proof strength than the stress it experiences, does not collapse for our load condition, this can be seen in the plastic strain analysis. The degree of utilization of the part using a safety factor of 2 is determined to be 82.5% therefore it is acceptable.

The degree of utilization of the bolt from the given pretension load of 187kN is calculated to be 8.55% for a 10.9 M24 Bolt.

Therefore, it can be concluded from these results that the examined Excavator Hook will not break while lifting even 4 x Nominal Load given that the material is 30CrNiMo8 neither does the Bolt fail under the given load.

12. Appendix

12.1 Table of Figures

12.2 Table of Tables

12.3 Bibliography

FKM Guideline. (2012). *Analytical Strength Assessment of Components: Made of Steel, Cast Iron and Aluminum Materials in Mechanical Engineering.* Frankfurt/Main: Forschungskuratorium Machienenbau (FKM).

RUD Handbook. (2017). *Excavator Hook VCGH-G for bolting Safety Instructions.* Aalen: Rieger & Dietz GmbH u. Co. KG.

VDI Guideline. (2015). *Systematic calculation of highly stressed bolted joints with one cylindrical bolt.* Düsseldorf: Verein Deutscher Ingenieure.