Machine Design FEM Report Group 1

Sayoojya Prasad 791364
Nguyen Xuan Binh 887799
Priya Singh 1007086

MEC-E1060 - Machine Design

Espoo 06.10.2023

Aalto University
School of Science

Contents

1 Introduction

FEM analysis is a numerical simulation method used to analyze complex physical systems and calculate their various engineering aspects. It is utilized to understand the behavior of structures under various load and boundary conditions. FEM employs mathematical techniques to solve equations governing the physical phenomenons of the various elements of the body in question. By combining the solutions regarding all elements, FEM can provide a comprehensive understanding of the entire body's behavior.

FEM is extensively used in structural analysis for understanding how structures deform and distribute stresses. It is vital for ensuring safety, optimizing designs, and identifying failure points. In heat transfer analysis, FEM helps study temperature distribution and heat flow within objects. It is used to aid in efficient cooling systems and insulation design. In fluid dynamics, FEM predicts pressure distributions, flow velocities, and turbulence in fluid systems. It is also critical for optimizing aerodynamics and fluid transport.

The process of structural FEM analysis can be divided into the following primary steps. However, the order of these steps can vary depending on the software being used.

Geometry selection and simplification: First, the body to be analyzed is selected. Often, their CAD files are further simplified just for the simulation as certain aspects of the body will only increase the time required for simulation but will add no valuable knowledge for the engineer.

Material selection: Materials to the different parts of the body must be assigned. If the FEM software has no materials library or certain material is not available in the library, then new material properties must be added manually.

Connection between different parts: Connection between different body parts must be assigned.E.g., bounded, frictional, frictionless, etc. Some FEM software creates all the connections as the parts are bounded to each other. Hence, special attention must be paid to ensure there are no mistakes in this step.

Meshing: Meshing is splitting the body into multiple small parts. The finer the meshing, the more accurate the analysis will be. However, meshing that is too fine will also take much longer to solve. The computational power of the machine or limits of the FEM license can also cause the analysis to either not be carried out or take a tremendous amount of time.

Analysis settings and boundary conditions: In this step, critical things such as loads, acceleration, forces, and fixed supports are applied to the bodies. The linearity or non-linearity of the analysis is decided based on load and boundary conditions. This is the most important step of the FEM analysis.

Solution information: Now is the step to ask the software what information we want from our analysis. Total deformation and equivalent stresses are often the most basic pieces of information to be asked to ensure that the body is strong enough not to deform during operation.

2 Pumpjack CAD model update

The 3D CAD model of pumpjack mechanism created for MBS analysis is shown in Figure 1. In this model, there are no parts connecting the pumpjack linkages to each other. The parts are also designed to move while being in direct contact with each other. Such a design will lose energy due to high friction between different components.

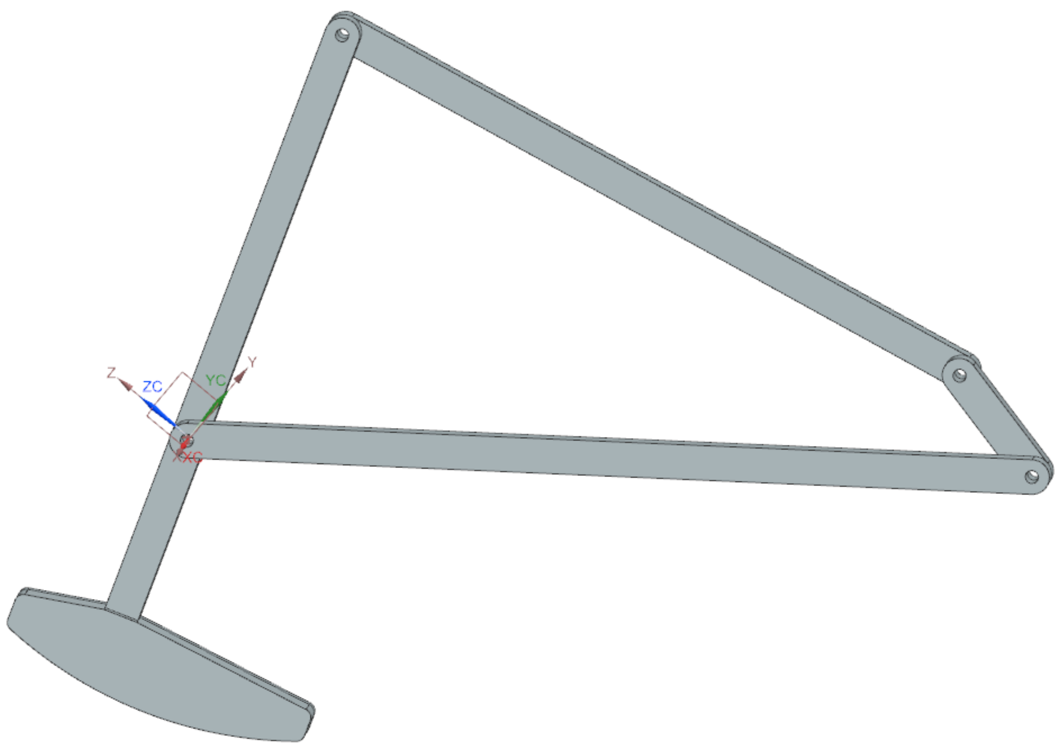


Figure 1: Pumpjack 3D model after MBS analysis

The model shown in Figure 1 is modified with a very minimalist approach to make it suitable for FEM analysis and is shown in Figure 2. Pins are added on different linkages to connect them with each other. In the final pumpjack model, thrust bearings will be used between the pins and holes to reduce friction. However, at this stage, it is unnecessary as the goal is to find a suitable shape to support the loads acting on the pumpjack. Figure 3 shows the back view of the new model, and the pins joining the different linkages can be seen in this figure. Figure 4 shows that none of the parts collide and are only connected by the pins. These pins will allow for the rotation motion between the joints.

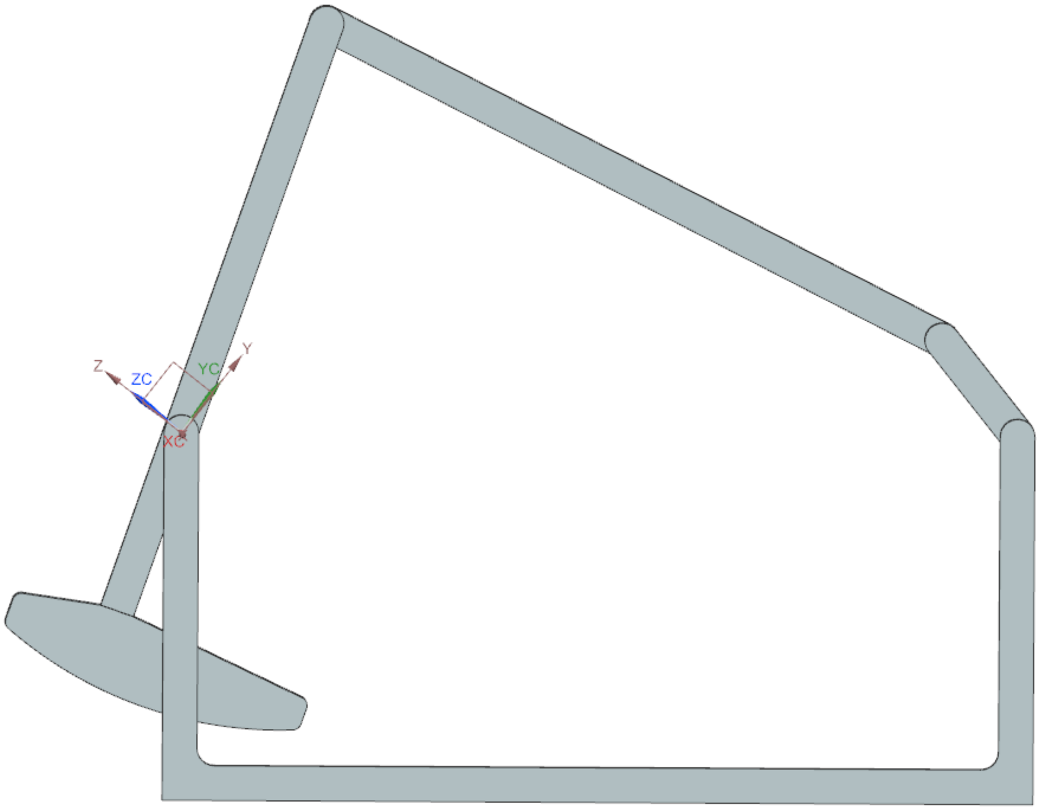


Figure 2: Pumpjack 3D model for FEM analysis front view

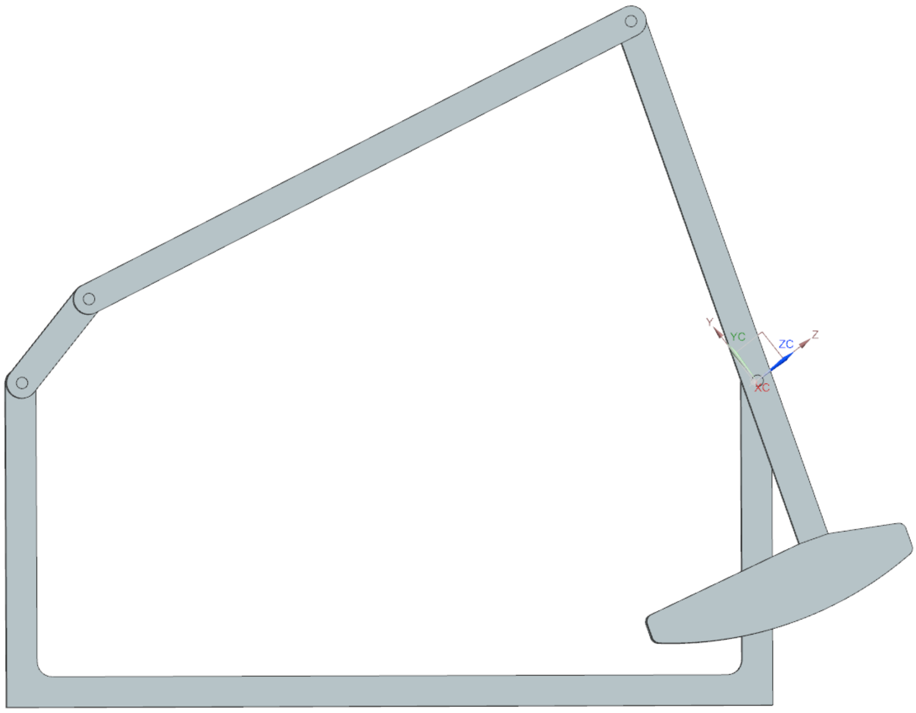


Figure 3: Pumpjack 3D model for FEM analysis back view

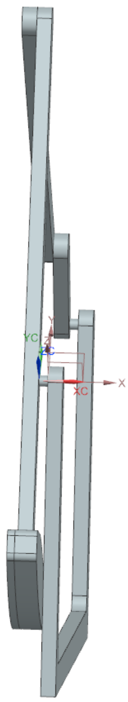


Figure 4: Pumpjack 3D model for FEM analysis side view

3 Material selection

For a pumpjack, the material needs to be of high strength, corrosion resistant, and durable in salt water. Based on these requirements, the material used for the simulations is AISI 410 stainless steel. The material mainly consists of iron, a high percentage of chromium, and other alloying elements like manganese and silicon, costing approximately 2 euros per kilogram (Granta EDUPACK). This grade of steel is primarily martensitic, which is a phase type that contributes significantly to strength and hardness. It can reach an ultimate tensile strength of 500-1400 MPa depending on the different heat treatments used in its production. The material shows good resistance to corrosion in air, water, and some chemicals [1]. This material is suitable for pump jacks, as it is highly durable in saline water, i.e., the material does not degrade even in high exposure to saline water. This material is economical as it is a low-alloy steel and is typically used in bolts, nuts, screws, coal mining equipment, pump parts and shafts, steam, and gas turbine parts, etc.

4 Analysis preparations

Figure 5 shows the names of the pumpjack's linkages and joints. This report will use these names to represent the joints on each linkage. The force acting on each joint was calculated in the MBS report and is shown in Table 1

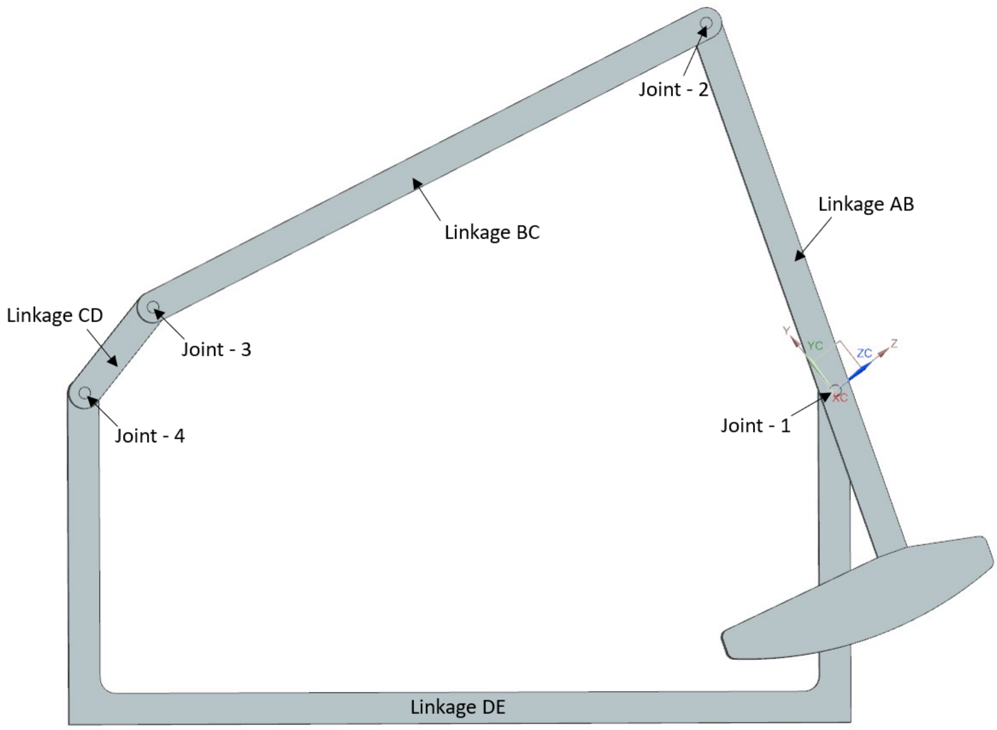


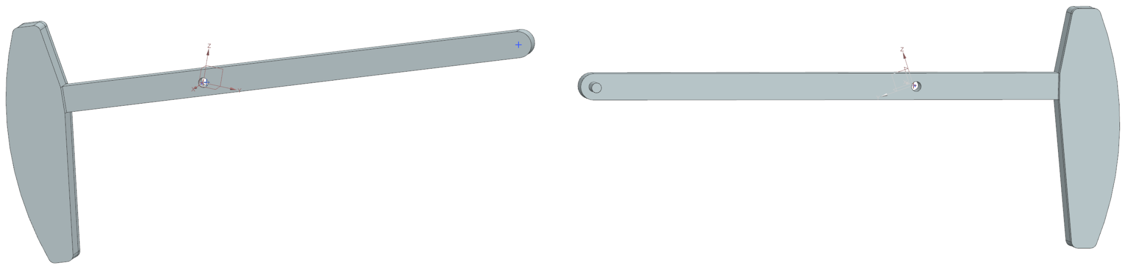
Figure 5: Pumpjack linkages' and joints' names

Table 1: Pumpjack loads from MBS analysis

joint no.	Max. absolute force (kN)
Joint 1	377.1
Joint 2	120.6
Joint 3	-62.8
Joint 4	-52.1

4.1 Pumpjack mechanism parts

Figures 6, 7, 8, and 9 show each linkage of the pumpjack mechanism individually. In the following sections these components will be analyzed under the load conditions shown in Table 1.



(a) Front-View

(b) Back-View

Figure 6: Linkage AB front and back views

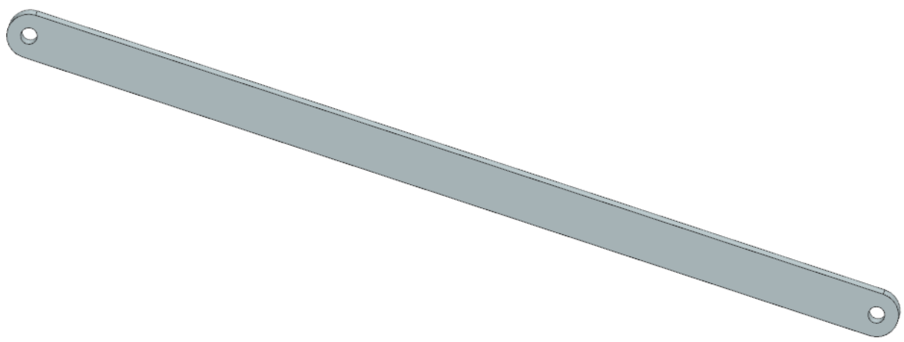
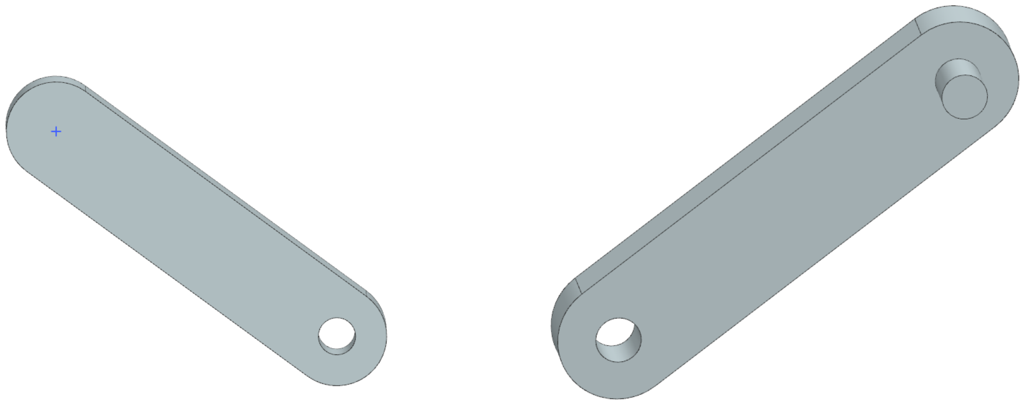
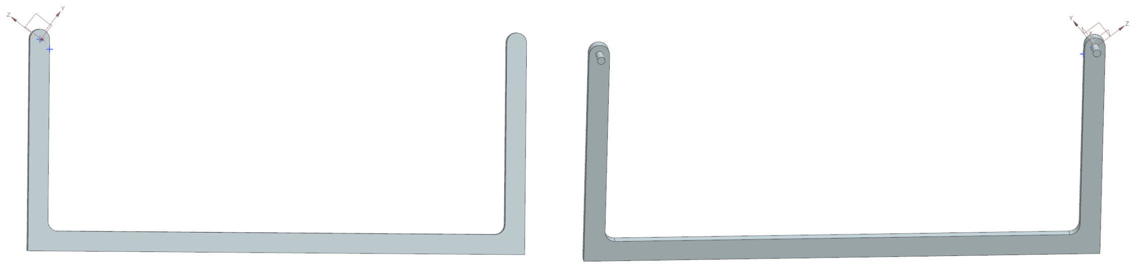


Figure 7: Linkage BC



(a) Front-View (b) Back-View

Figure 8: Linkage CD front and back views



(a) Front-View (b) Back-View

Figure 9: Linkage DE front and back views

4.2 Meshing

Meshing conditions for the NX simulator are shown in Figure 10. A 3D tetrahedral mesh of an element size of 50 mm was chosen. Figure 12 show each component's meshed state.

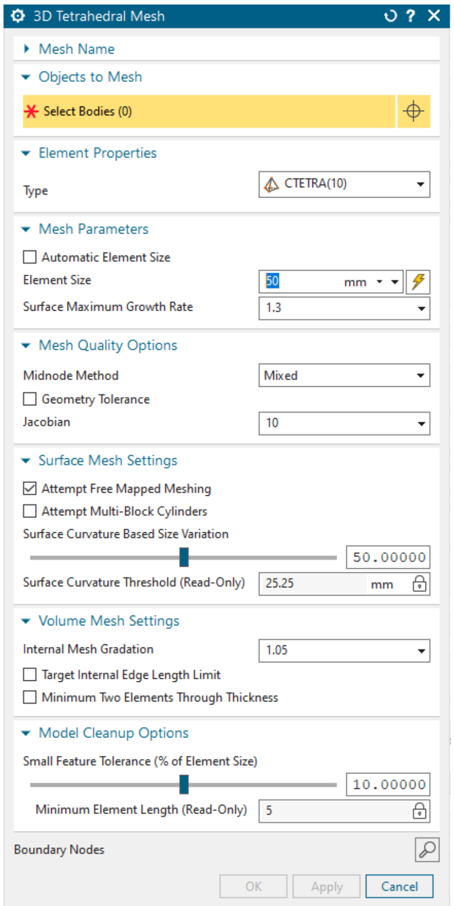
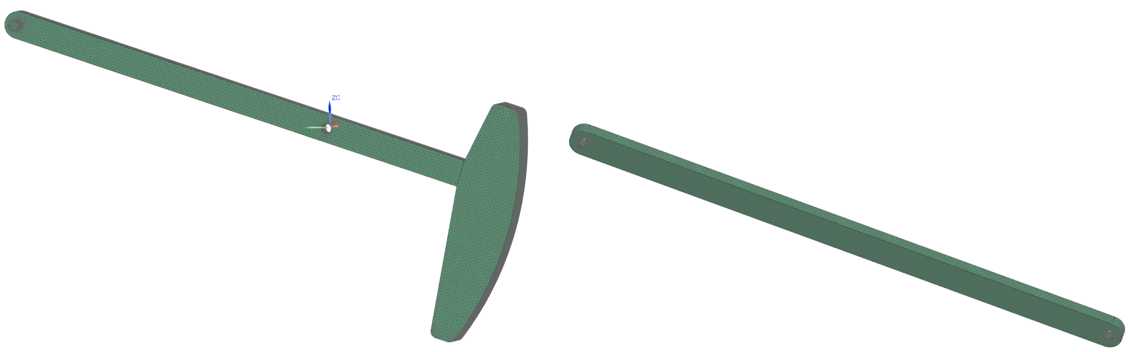
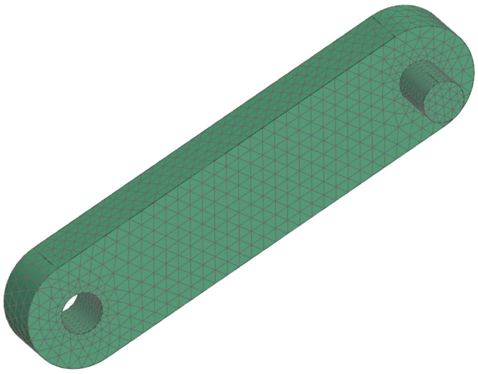


Figure 10: Meshing condition in NX simulator

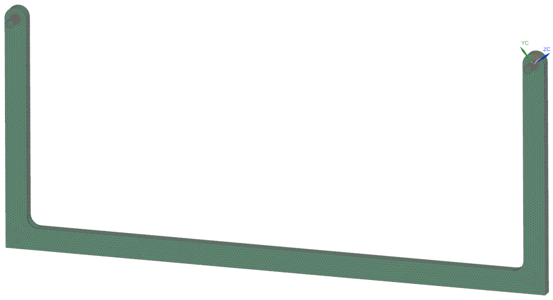


(a) Linkage AB

(b) Linkage BC



(a) Linkage CD



(b) Linkage DE

Figure 12: Meshing of linkages

4.3 Loading and boundry condition

As each linkage has two joints, two analyses were done for each linkage. For each analysis, one joint was fixed, and force was applied to the other. This was done to understand the displacements and deformations of each joint individually. Figures 13, 14, 15, 16, 17, 18, and 19 show the load and boundary condition of each joint on each linkage. Fixed constraints are shown by blue lines, and red arrows show the direction of applied force.

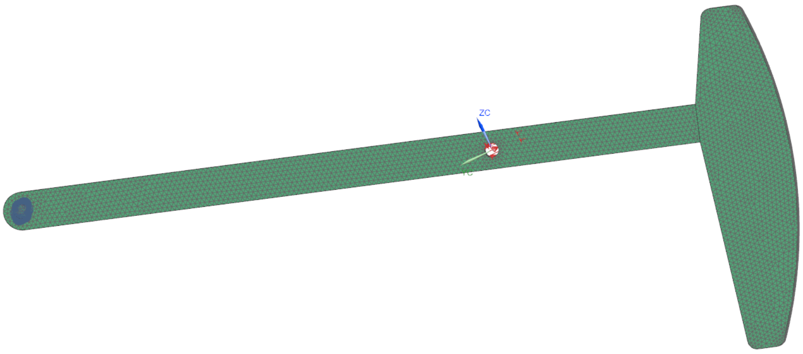


Figure 13: Load and constrain applied of joint 1 on linkage AB

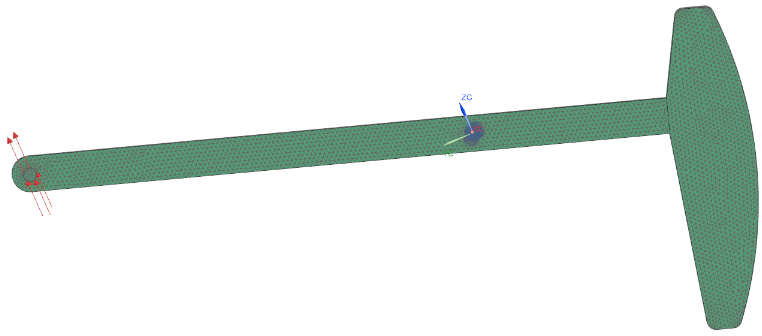


Figure 14: Load and constrain applied of joint 2 on linkage AB

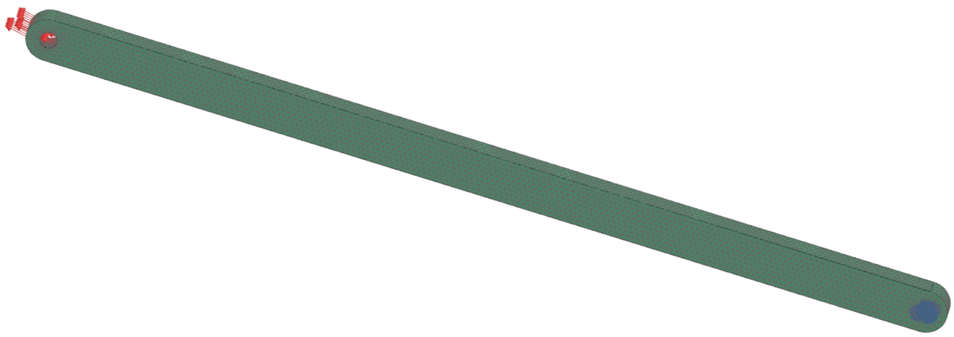


Figure 15: Load and constrain applied of joint 2 on linkage BC

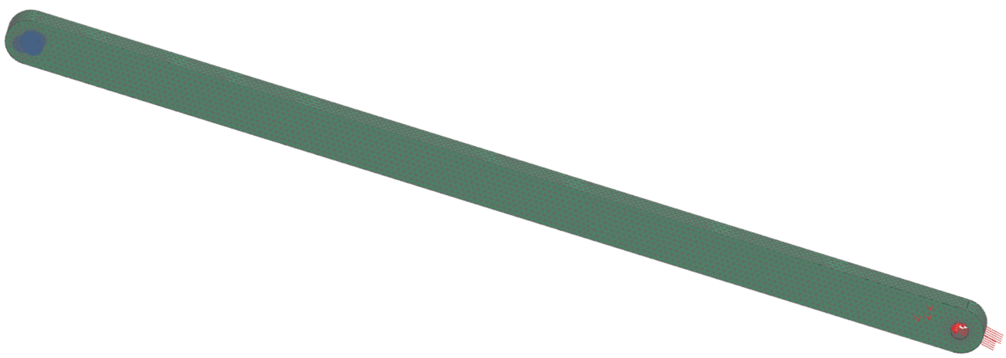


Figure 16: Load and constrain applied of joint 3 on linkage BC

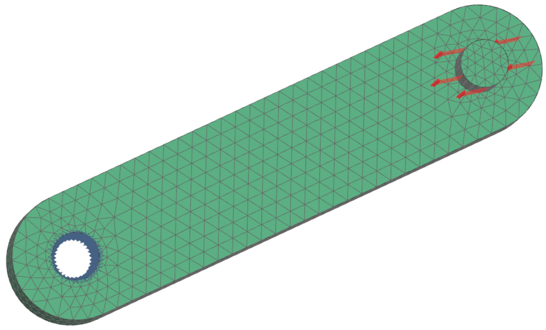


Figure 17: Load and constrain applied of joint 3 on linkage CD

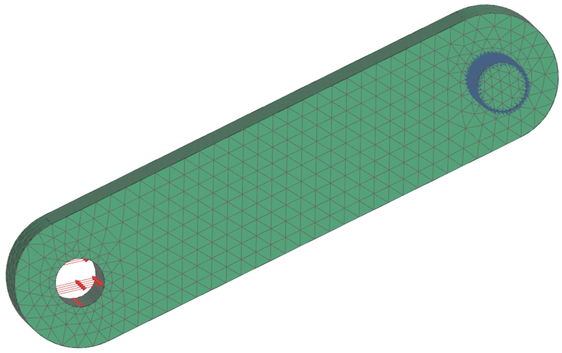


Figure 18: Load and constrain applied of joint 4 on linkage CD

Figure 19: Load and constrain applied of joint 3 and 4 on linkage DE

5 Results of FEM analysis

This section shows the results of FEM analysis for each joint on each linkage. The maximum values of stress and displacement are summarized in table 2. According to the NX data sheet, the ultimate tensile strength of AISI 410 stainless steel is 520 MPa. The safety factor of all joints is calculated and listed in table 2.

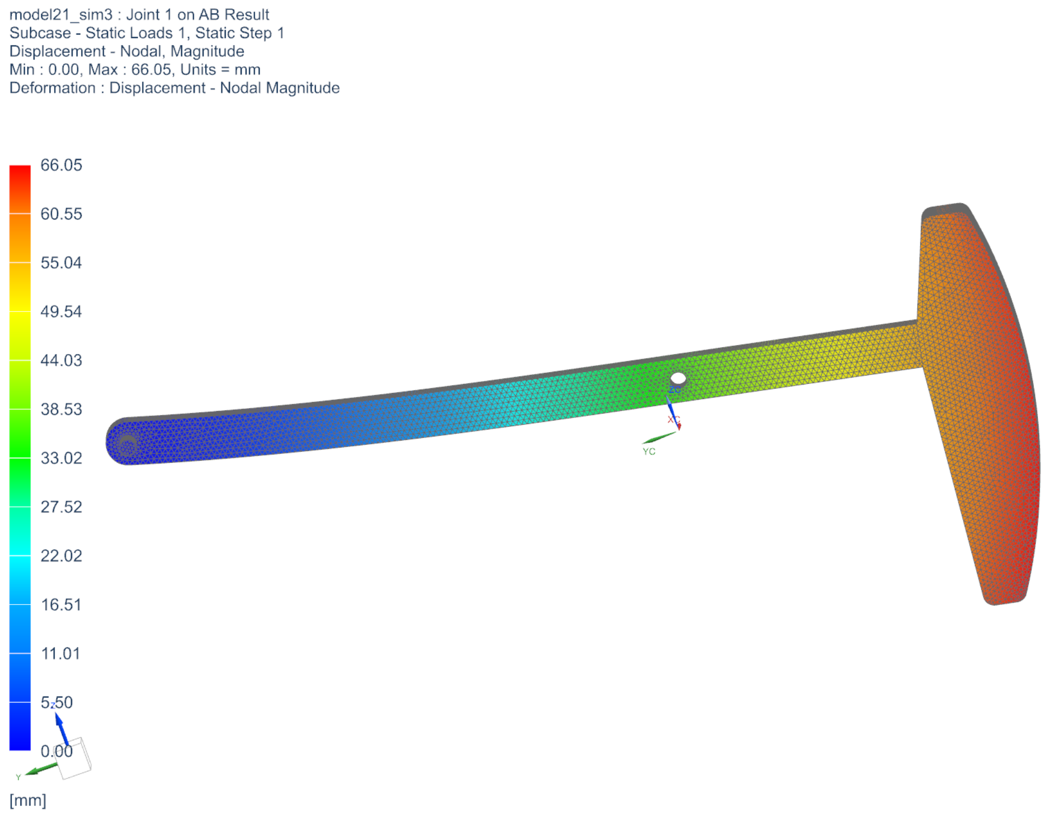


Figure 20: Displacement of joint 1 on linkage AB

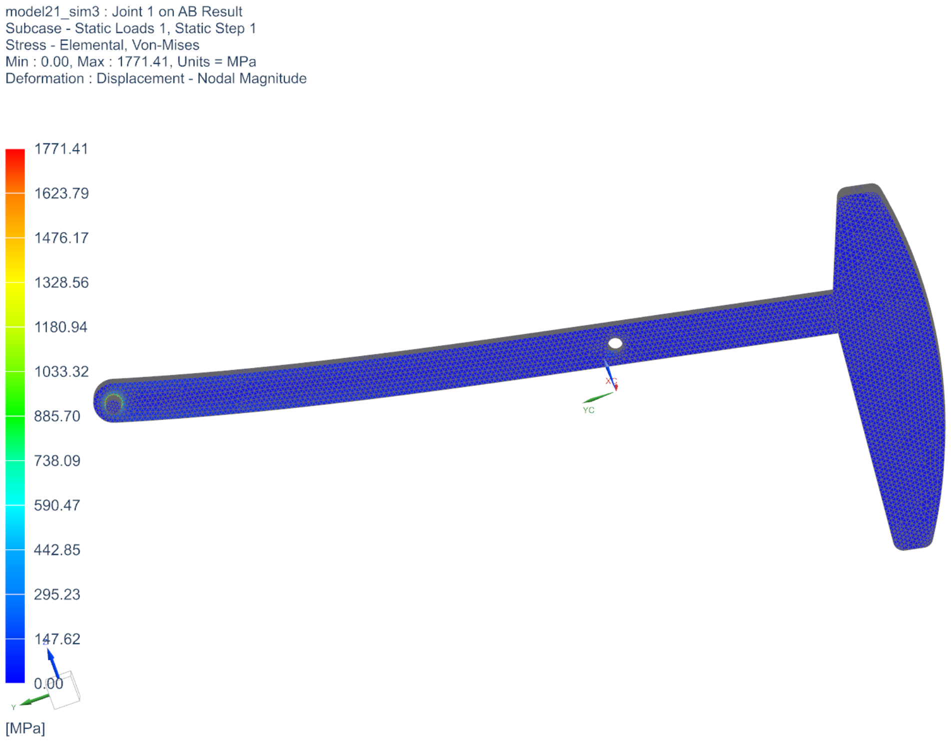


Figure 21: Stress on joint 1 on linkage AB

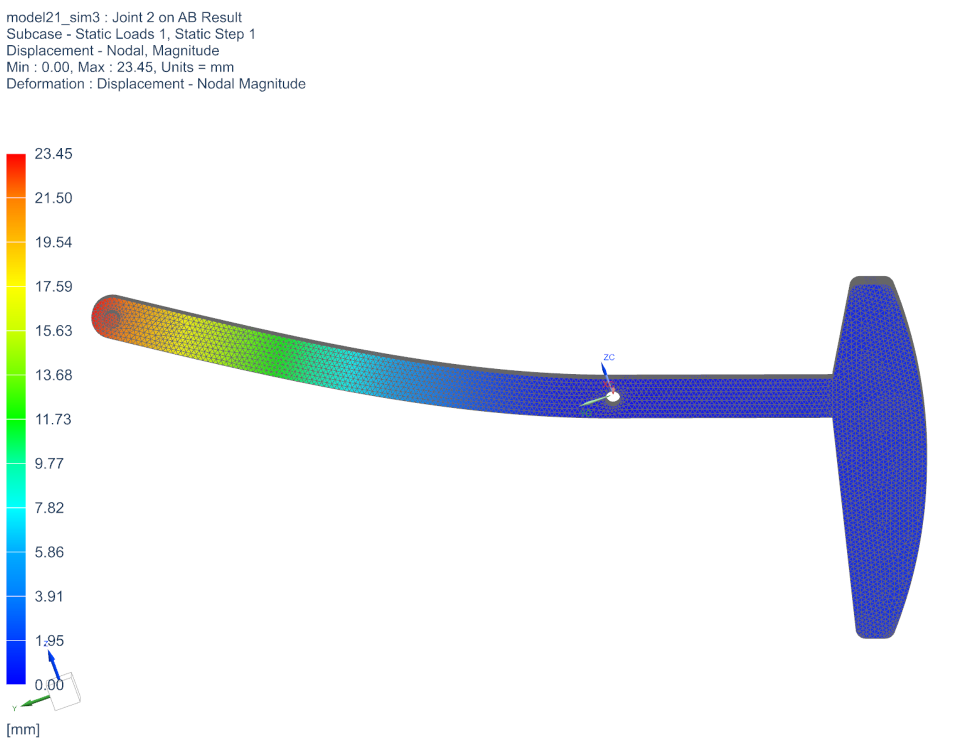


Figure 22: Displacement of joint 2 on linkage AB

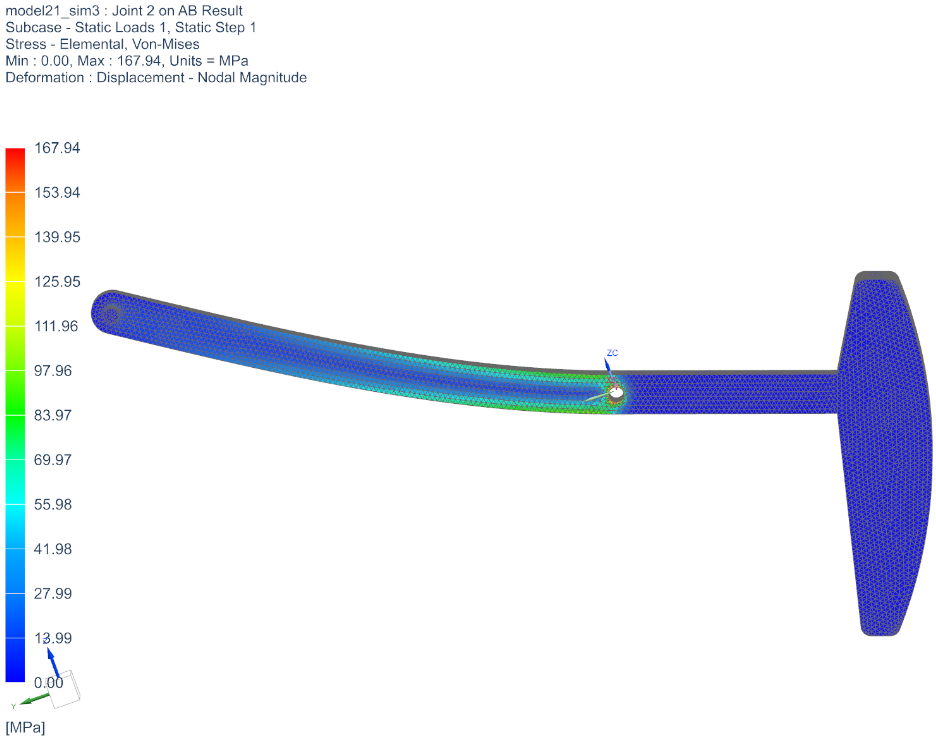


Figure 23: Stress on joint 2 on linkage AB

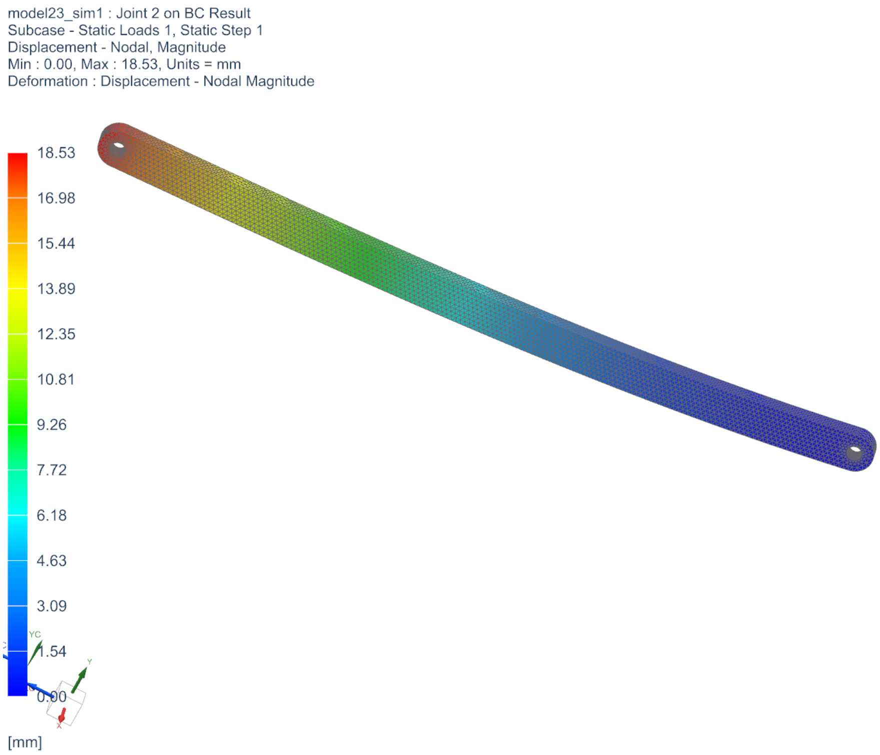


Figure 24: Displacement of joint 2 on linkage BC

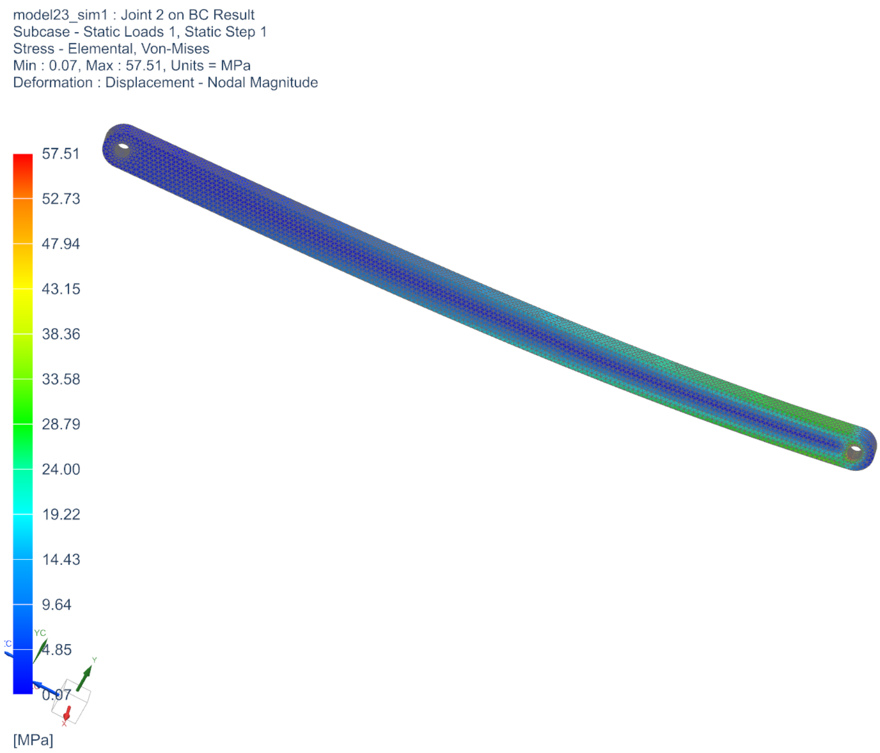


Figure 25: Stress on joint 2 on linkage BC

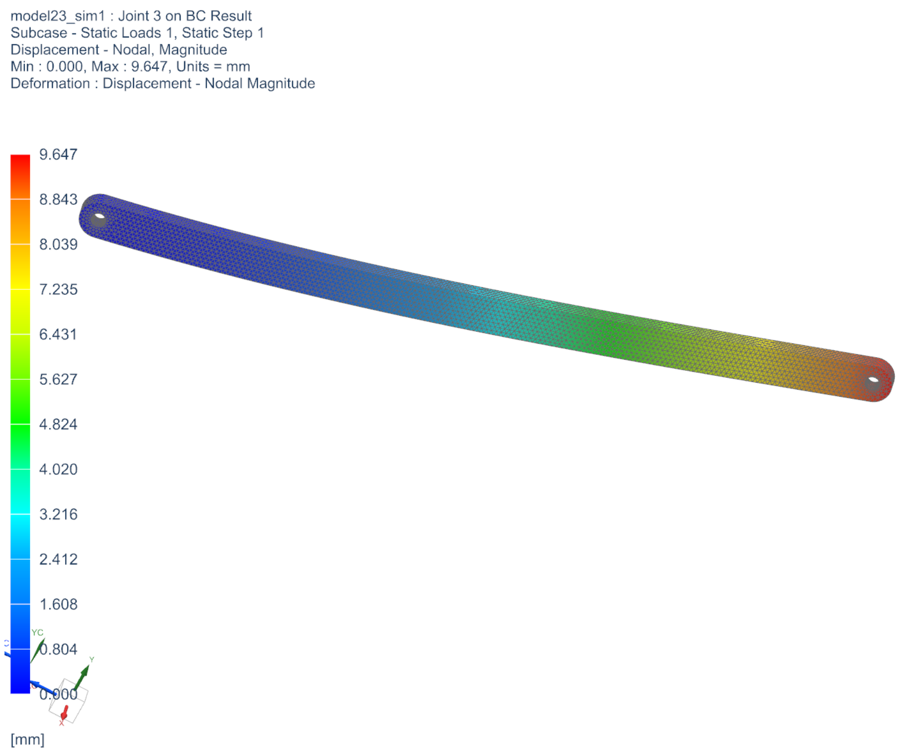


Figure 26: Displacement of joint 3 on linkage BC

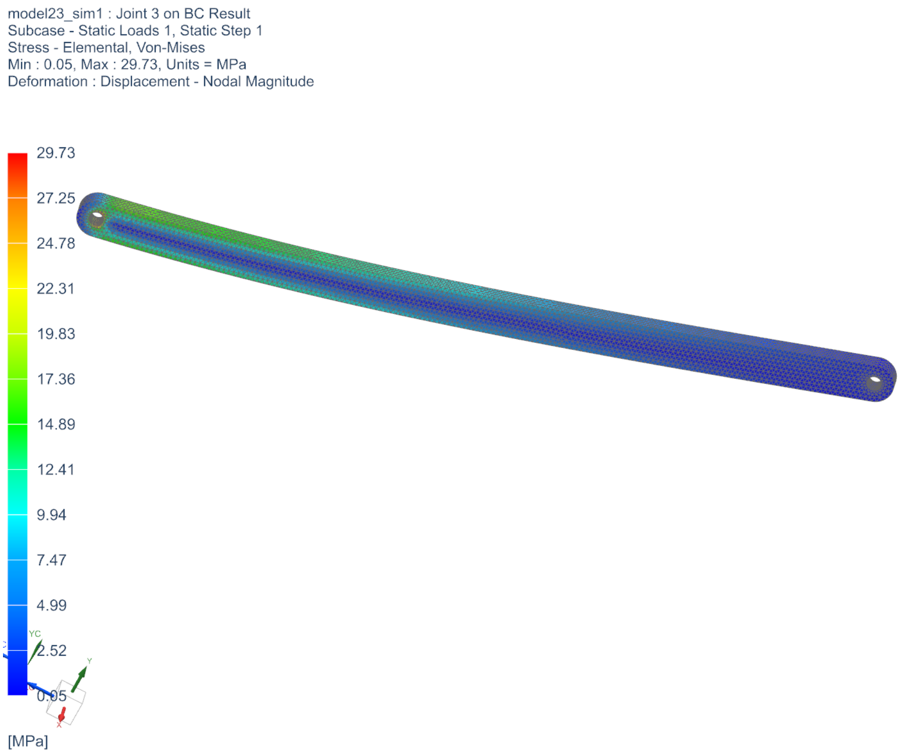


Figure 27: Stress on joint 3 on linkage BC

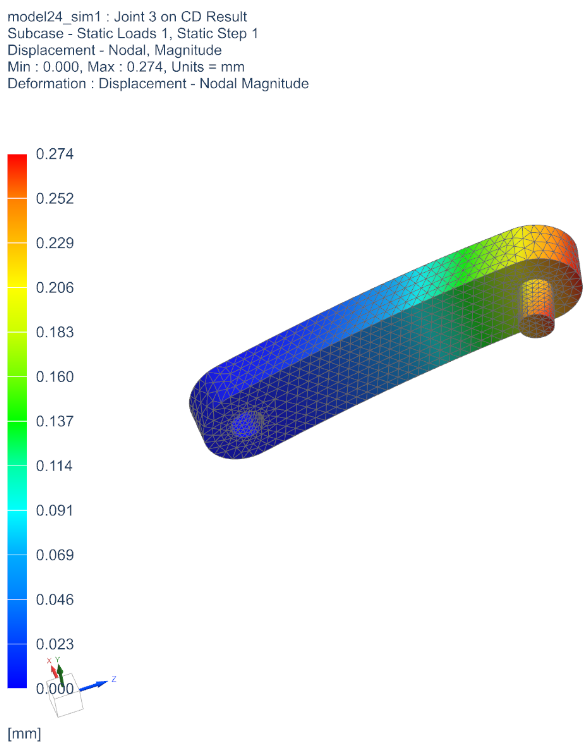


Figure 28: Displacement of joint 3 on linkage CD

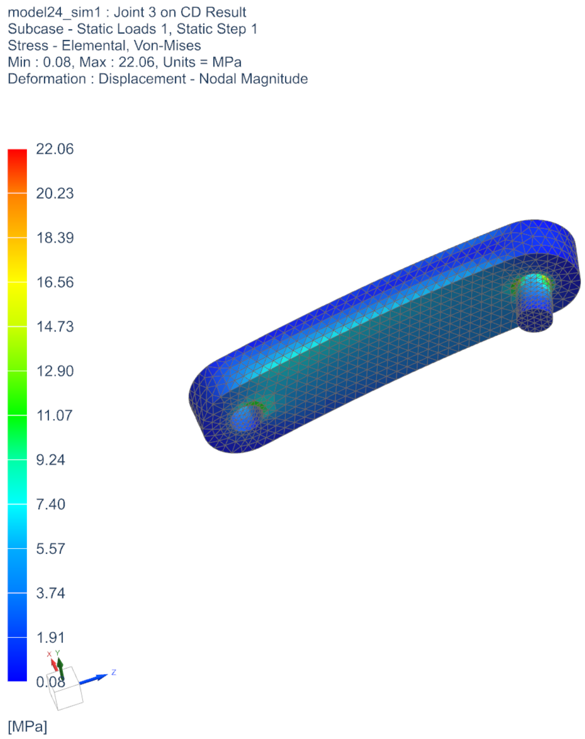


Figure 29: Stress on joint 3 on linkage CD

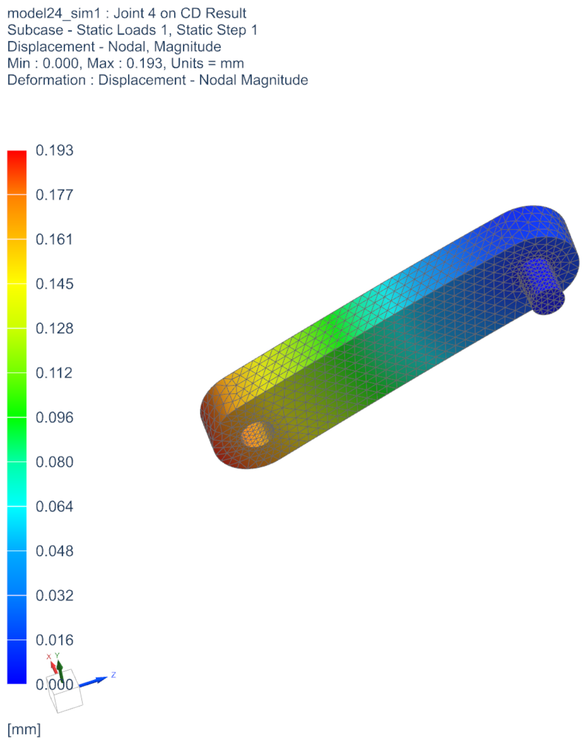


Figure 30: Displacement of joint 4 on linkage CD

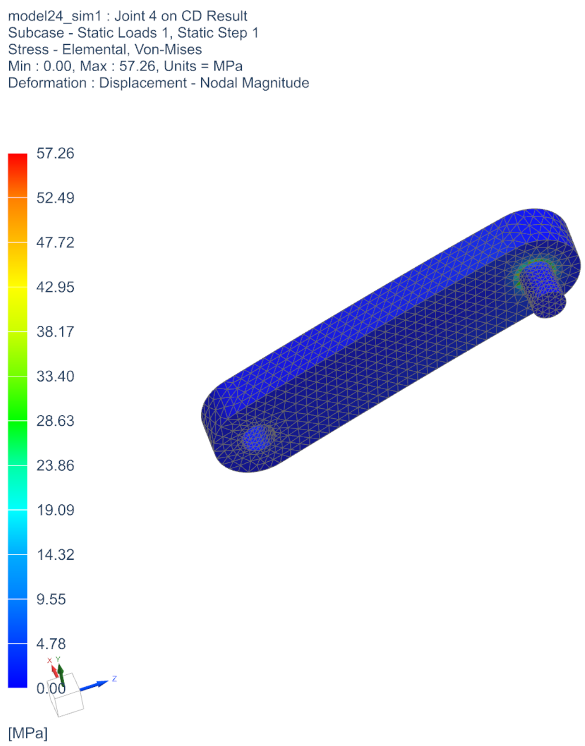


Figure 31: Stress on joint 4 on linkage CD

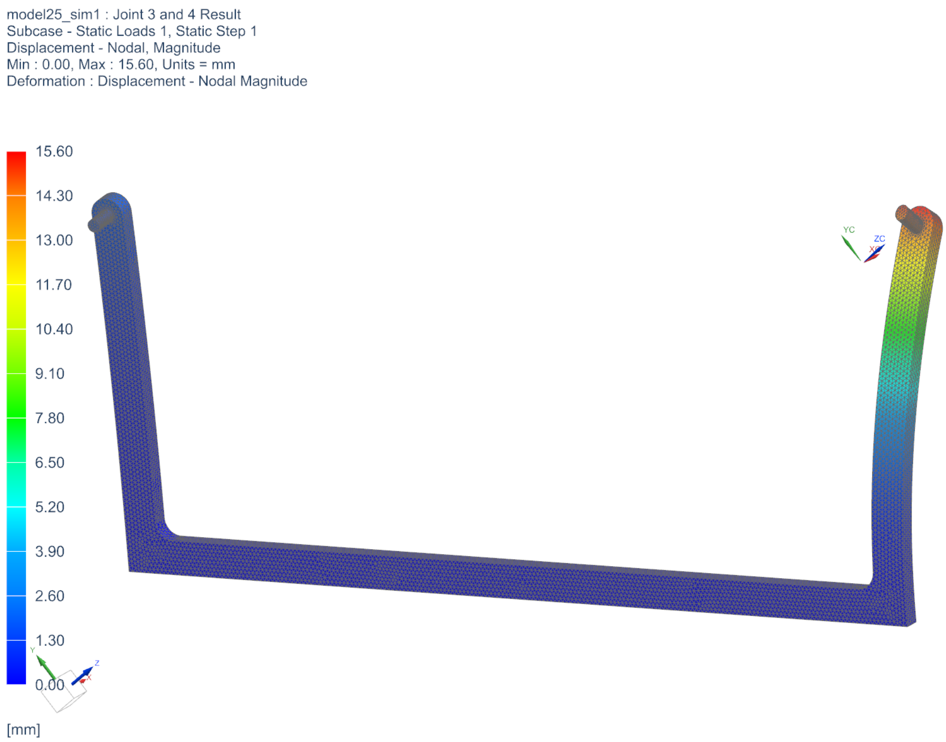


Figure 32: Displacement of joint 3 and 4 on linkage DE

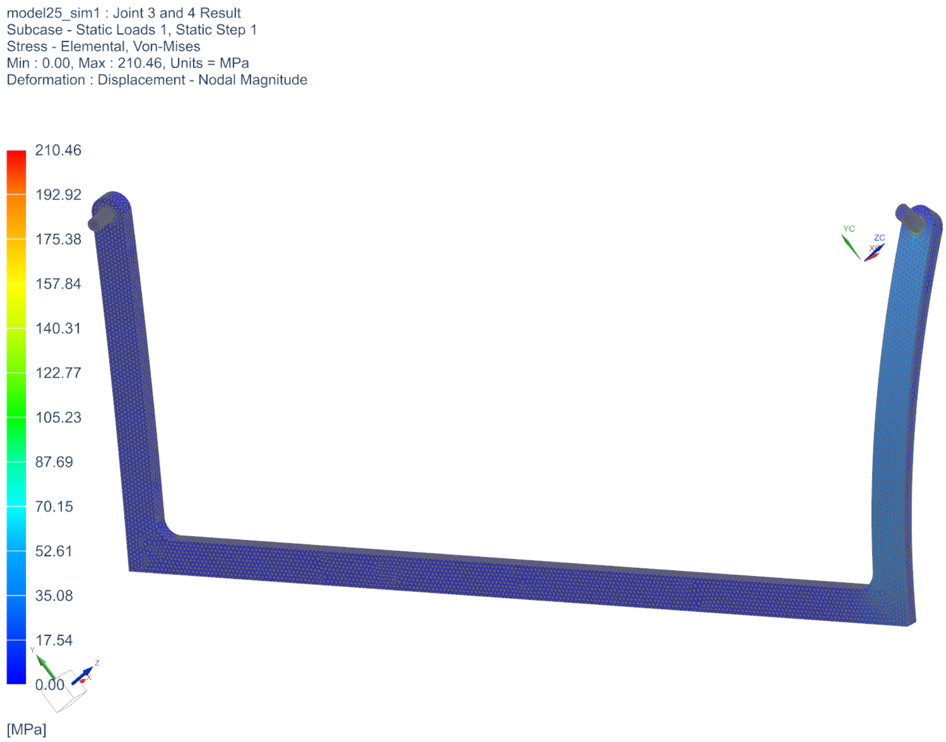


Figure 33: Stress on joint 3 and 4 on linkage DE

Table 2: Pumpjack maximum displacement and stress

joint	max. displacement (mm)	max. stress (MPa)	saftey factor
Joint 1 linkage AB	66.5	1771.41	Fail
Joint 2 linkage AB	23.45	167.94	3
Joint 2 linkage BC	18.53	57.51	9
Joint 3 linkage BC	9.64	29.73	17.5
Joint 3 linkage CD	0.27	22.06	23.5
Joint 4 linkage CD	0.19	57.26	9
Joint 3 and 4 linkage DE	15.6	210.46	2.47

Joint 1 of Linkage AB has about four times more stress than the material's ultimate tensile strength. The rest of the joints are well within the ultimate tensile strength limit and have safety factor values of over two. However, as the linkage AB fails, it must be redesigned, and further analysis must be done to ensure that the stress created on this linkage is within the material's ultimate tensile strength limit.

6 Testing various shapes for mechanism parts

The pumpjack model was further modified to increase the strength of linkage AB. Although the rest of the linkages were well within the ultimate tensile strength limit, they were also modified to reduce the weight of the overall machinery. The modified parts are shown in figures 34, 35, 36, 37, 38, and 39, and a comparison of the weight of components before and after design modification is made in table 3. Linkage DE is the ground linkage of the pumpjack mechanism, and it must ensure that the whole pumpjack mechanism can stand on it. Hence, the design of linkage DE was hugely modified, and the new model is considerably heavier than the old one. The new parts were analyzed under the exact meshing, load, and boundary conditions as the previous model. The results of the FEM analysis for each joint are shown in figures 40, 41, 42, 43, 44, 45, 46, 47, 48, 49, 50, 51, 52, 53, 54, 55, 56, and 57.

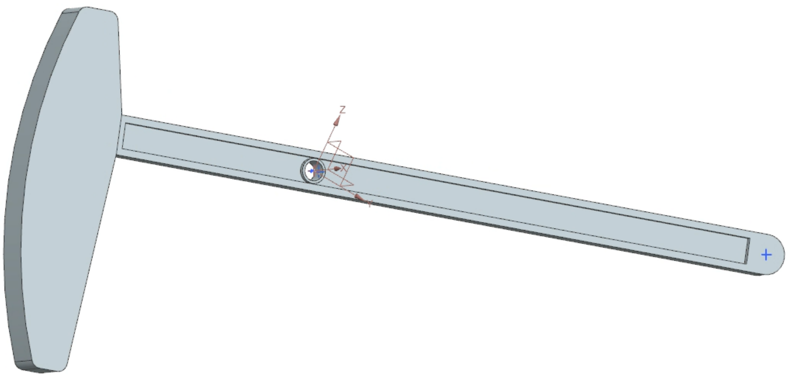


Figure 34: Modified linkage AB (front)

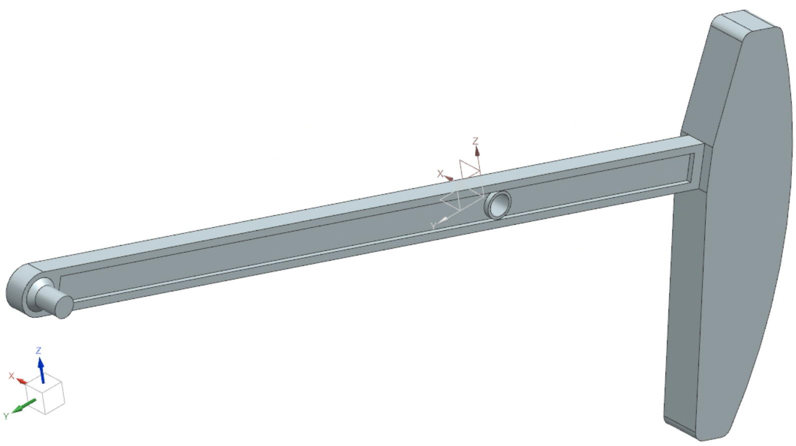


Figure 35: Modified linkage AB (back

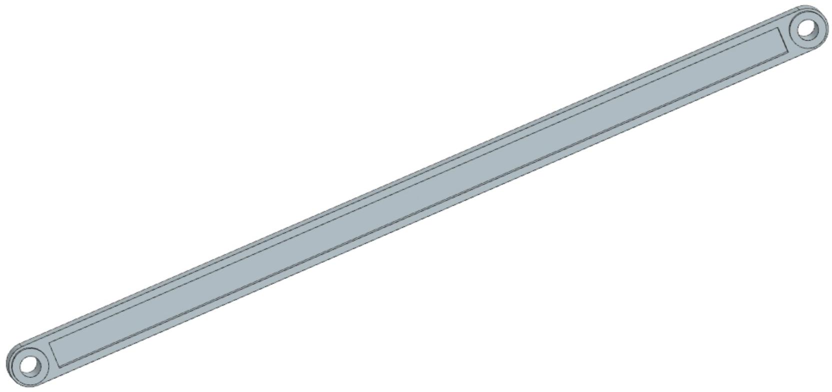


Figure 36: Modified linkage BC

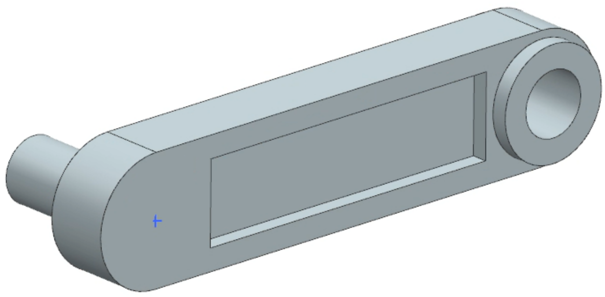


Figure 37: Modified linkage CD (front)

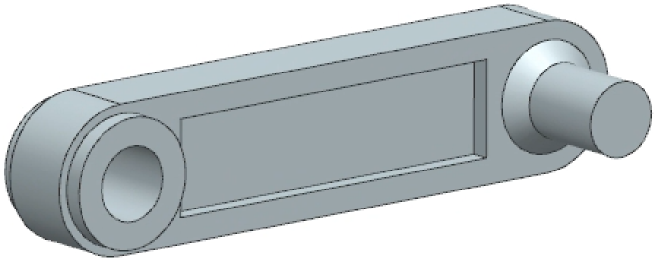


Figure 38: Modified linkage CD (back

Figure 39: Modified linkage DE

Table 3: Pumpjack parts weight comparison

Linkage name	Before weight (kg)	After weight (kg)
AB	12661.9	11343.9
BC	5089.3	2884.9
CD	1060.5	929.4
DE	11296.7	19197.6

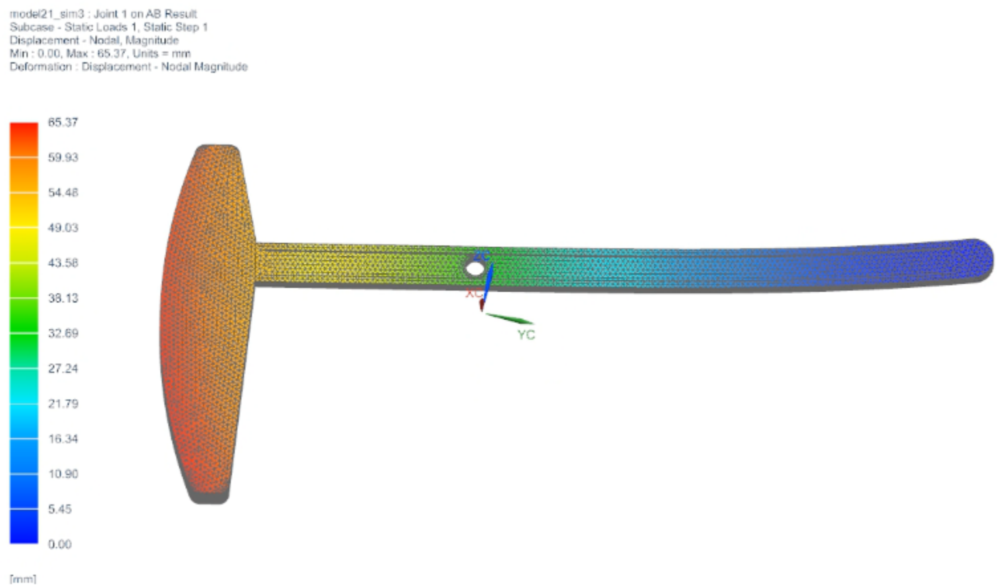


Figure 40: Displacement of joint 1 on modified linkage AB

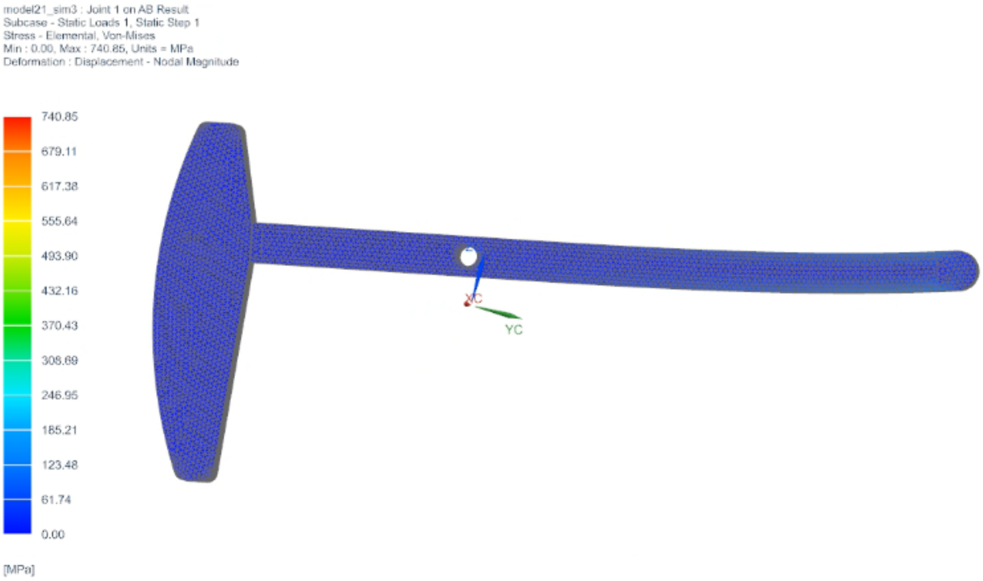


Figure 41: Stress on joint 1 of modified linkage AB

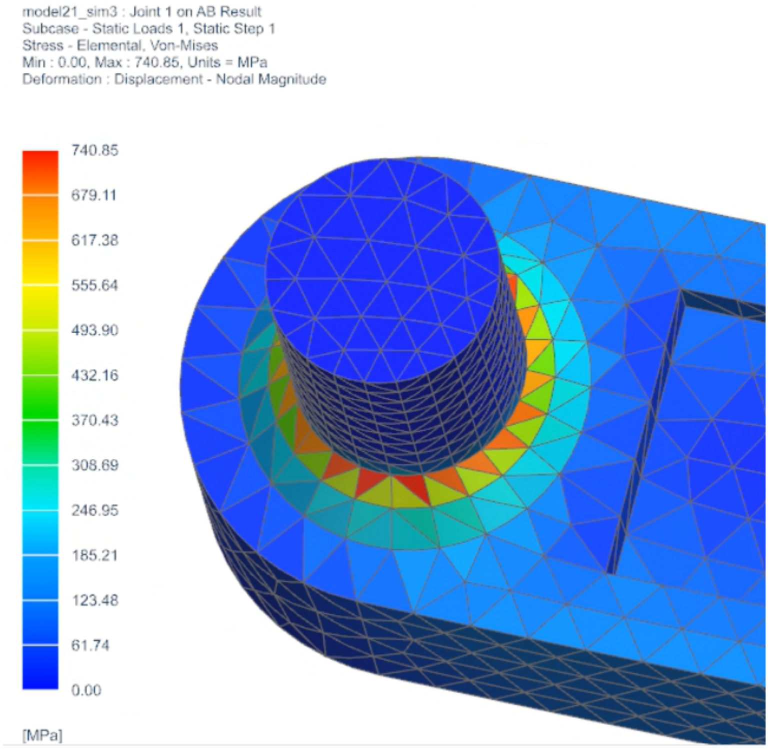


Figure 42: Close-up view of maximum stress on joint 1 of modified linkage AB

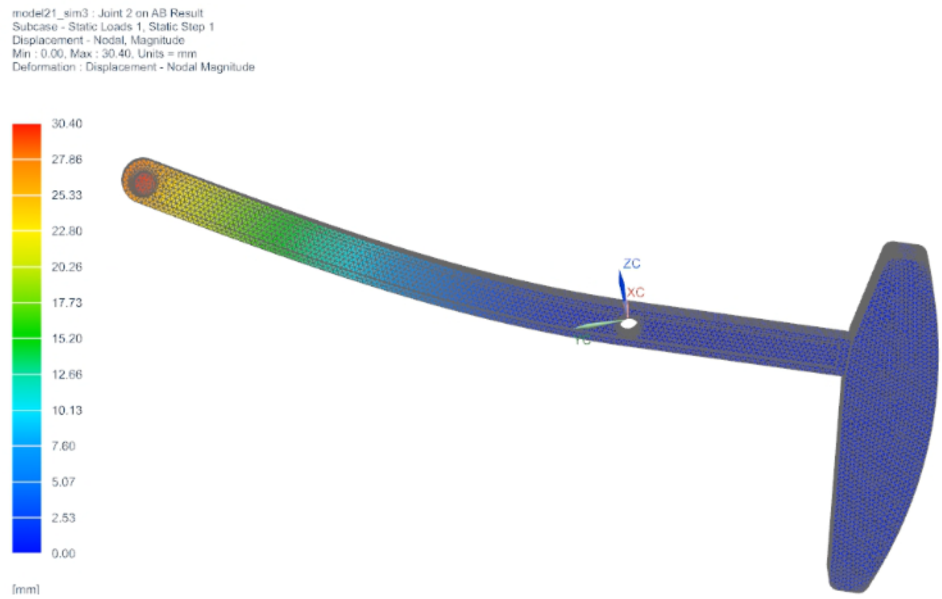


Figure 43: Displacement of joint 2 on modified linkage AB

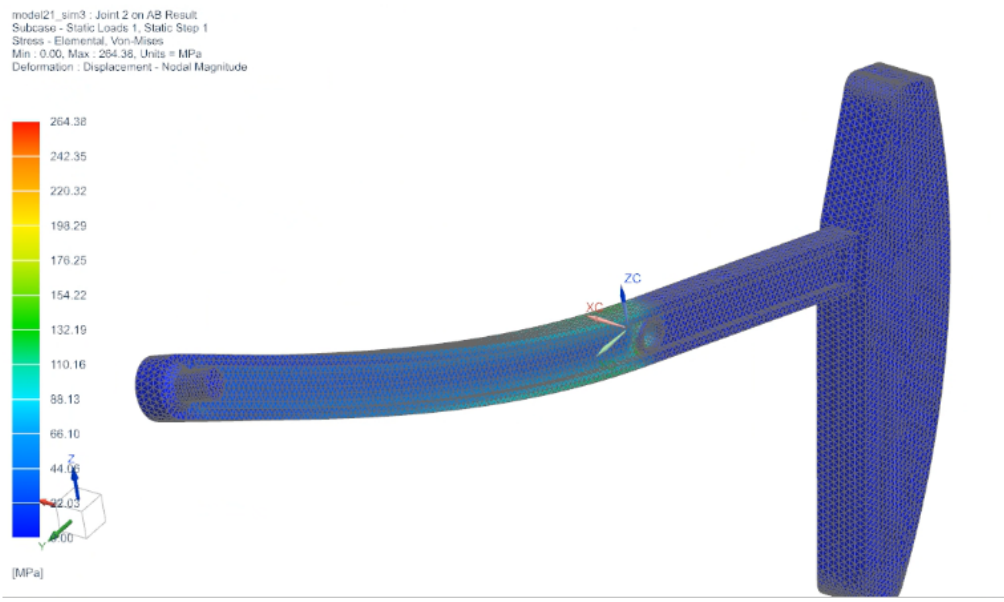


Figure 44: Stress on joint 2 of modified linkage AB

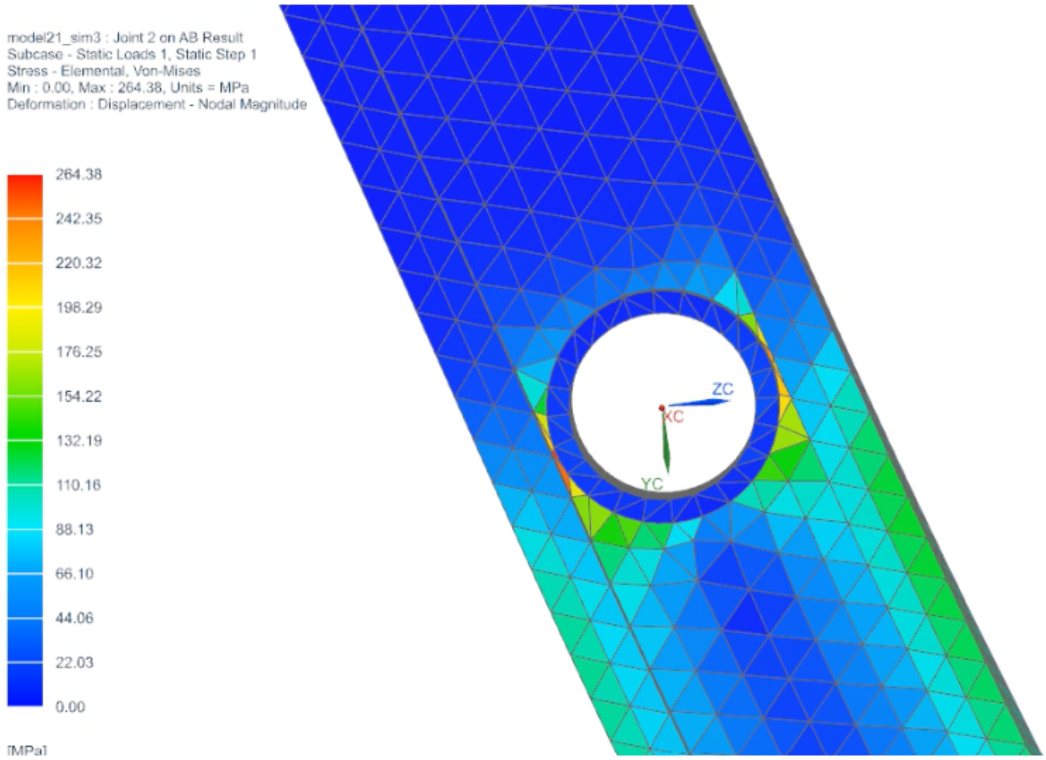


Figure 45: Close-up view of maximum stress on joint 2 of modified linkage AB

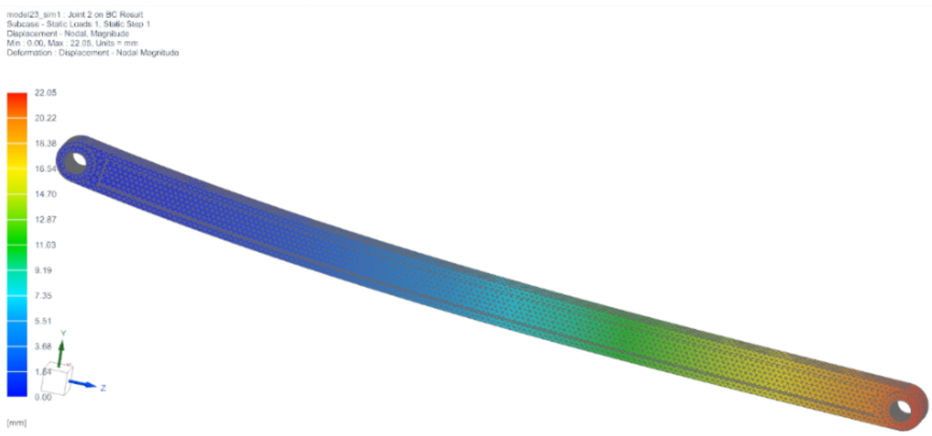


Figure 46: Displacement of joint 2 on modified linkage BC

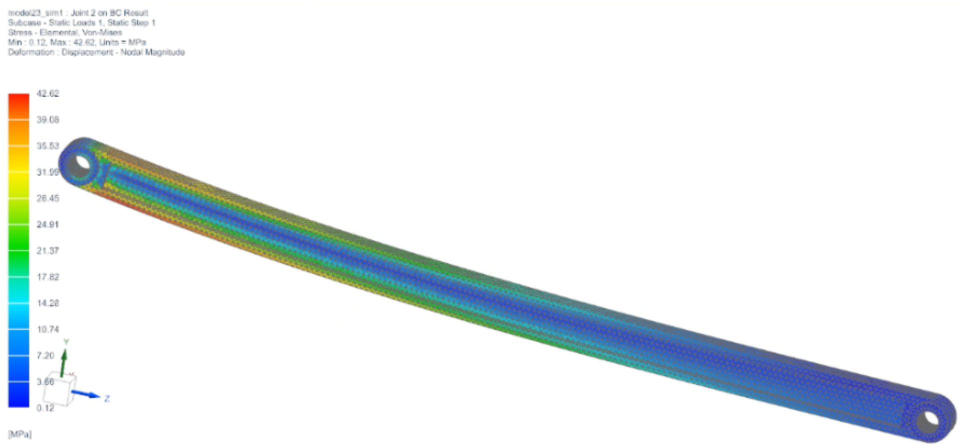


Figure 47: Stress on joint 2 of modified linkage BC

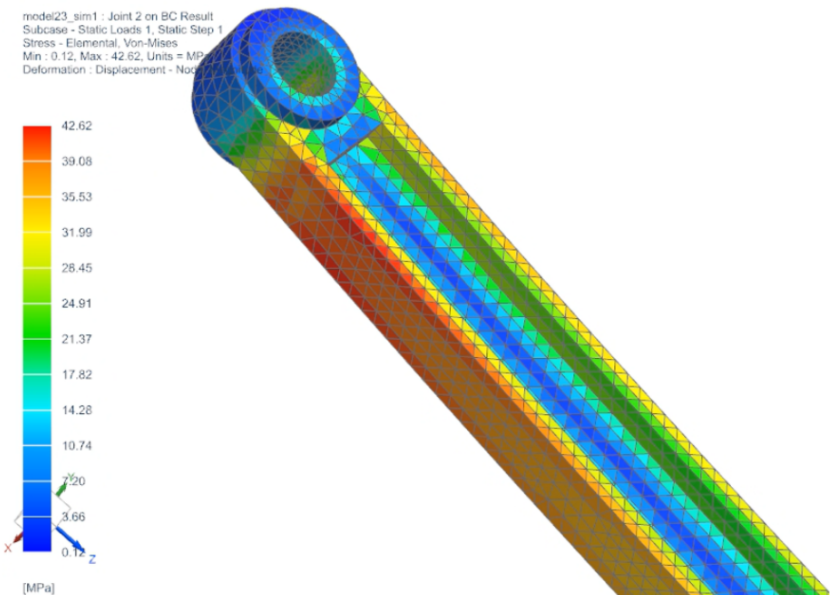


Figure 48: Close-up view of maximum stress on joint 2 of modified linkage BC

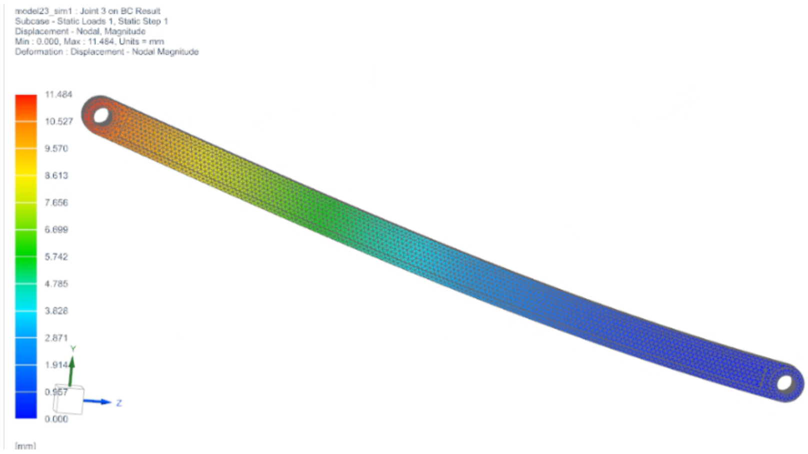


Figure 49: Displacement of joint 3 on modified linkage BC

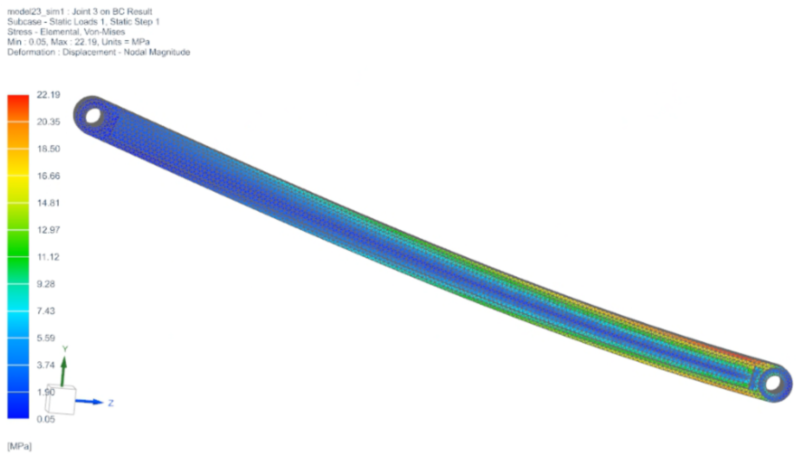


Figure 50: Stress joint 3 of modified linkage BC

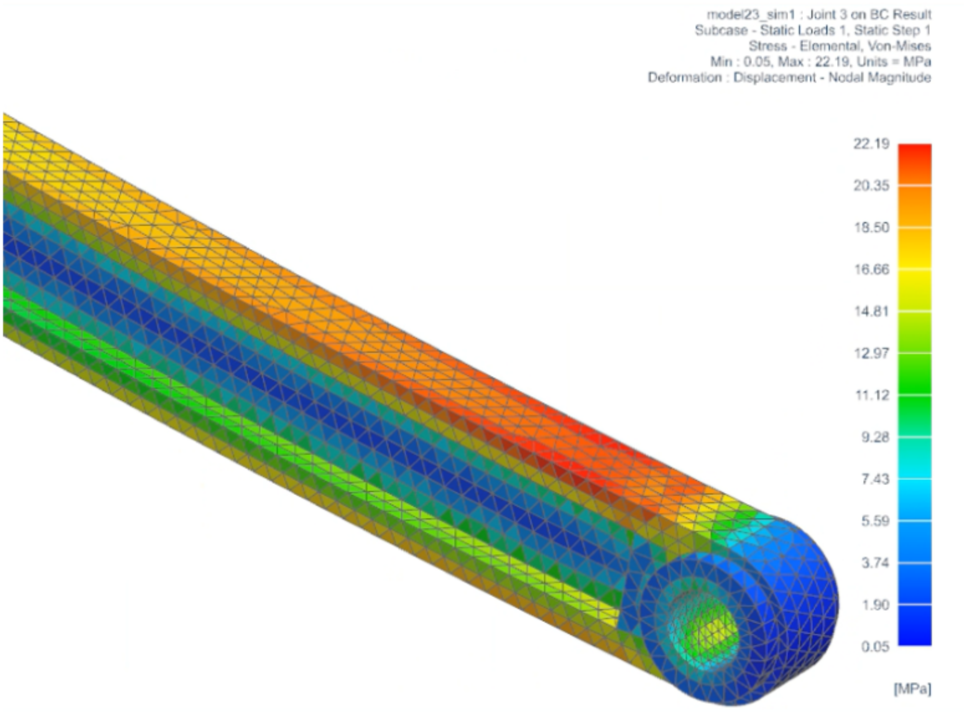


Figure 51: Close-up view of maximum stress on joint 3 of modified linkage BC

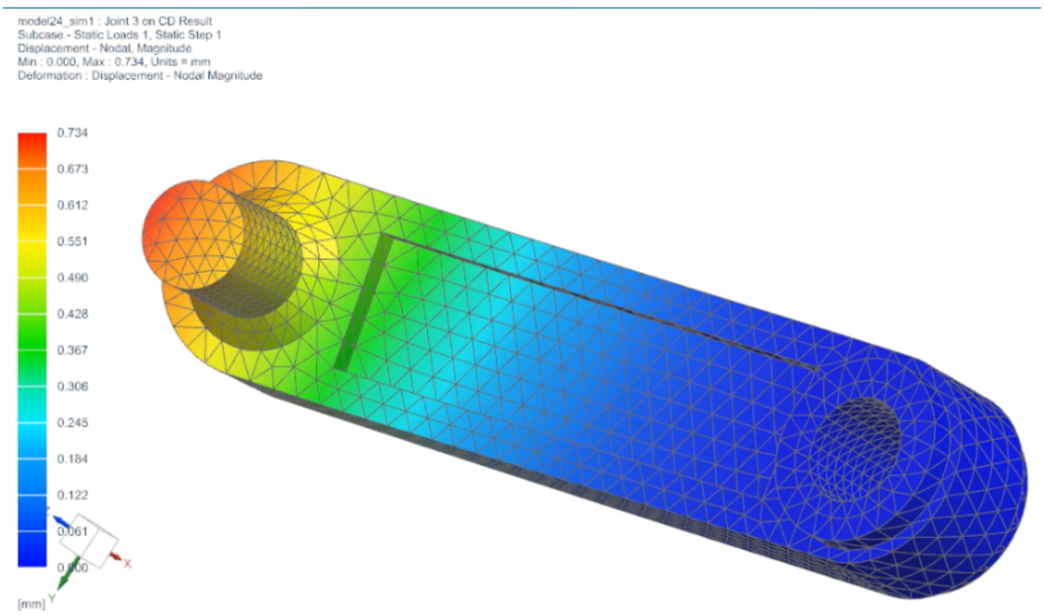


Figure 52: Displacement of joint 3 on linkage CD

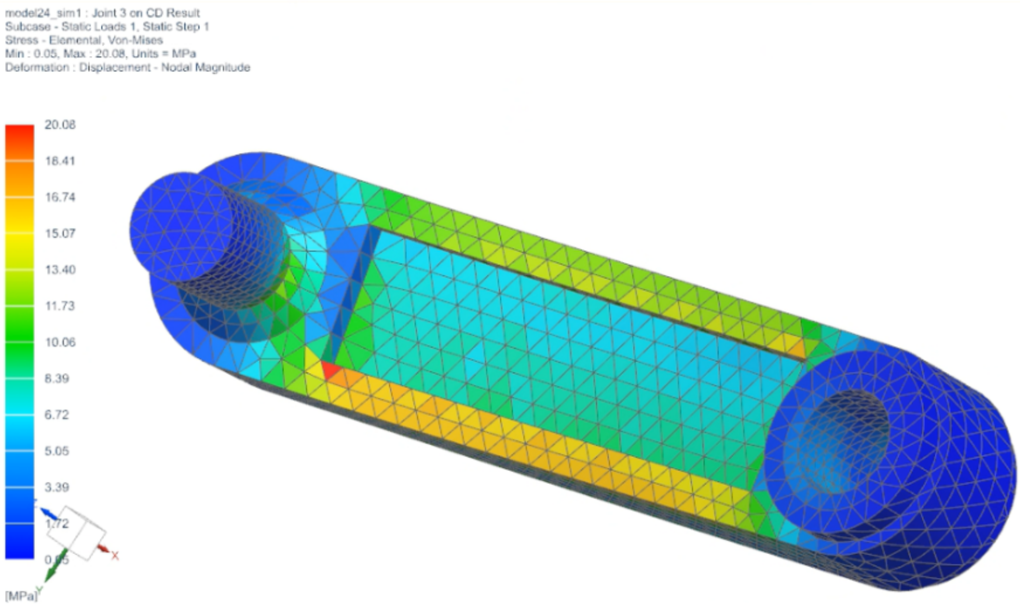


Figure 53: Stress on joint 3 of modified linkage CD

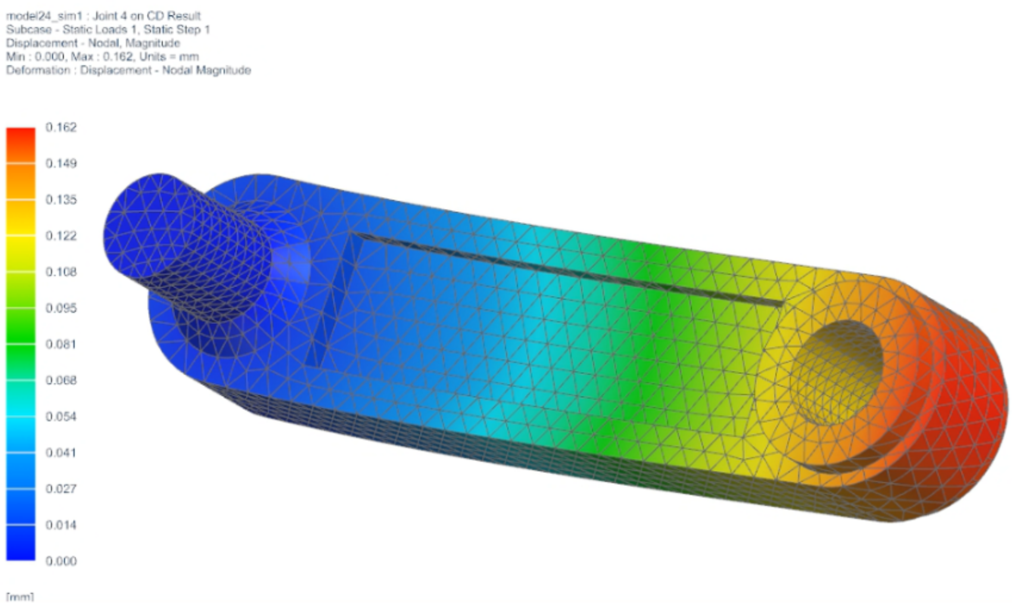


Figure 54: Displacement of joint 4 on modified linkage CD

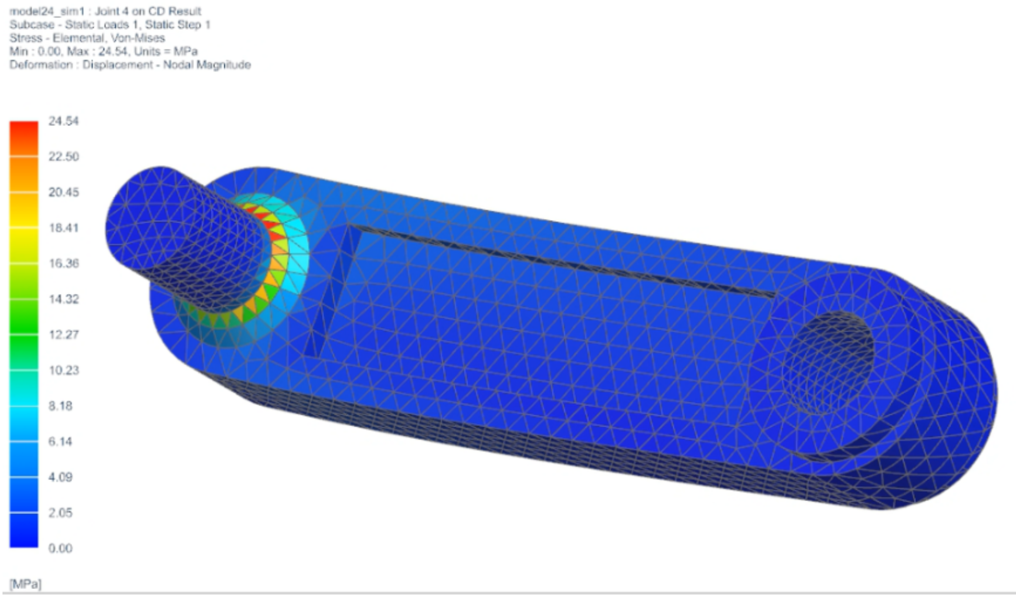


Figure 55: Stress on joint 4 of modified linkage CD

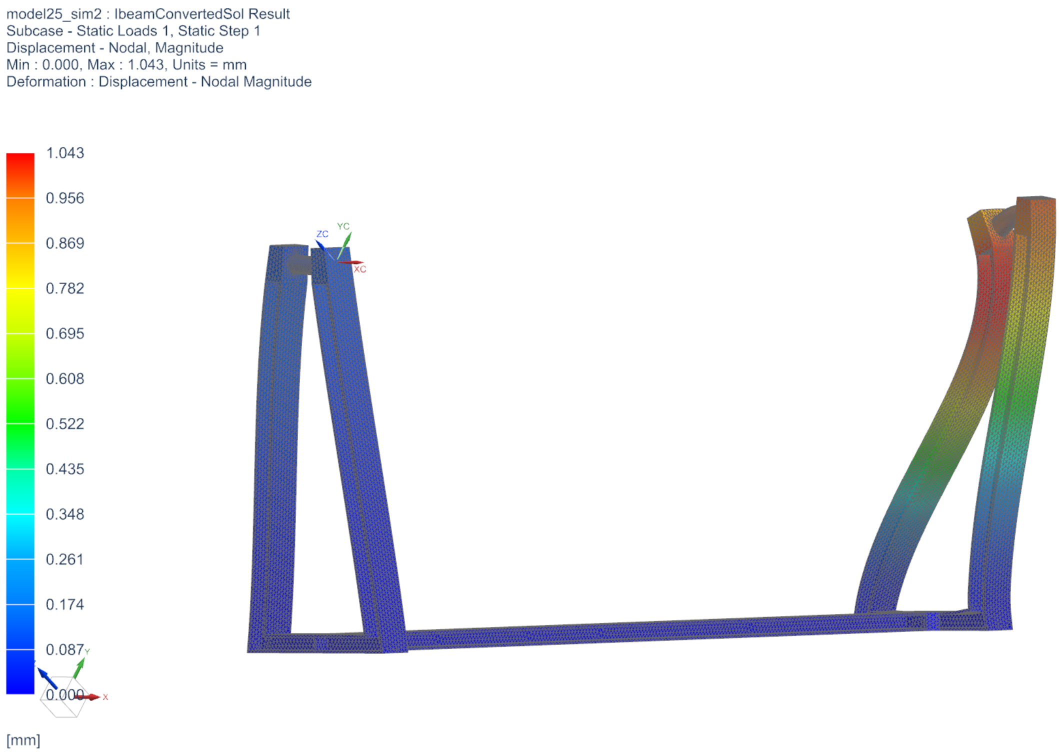


Figure 56: Displacement of joint 3 and 4 on modified linkage DE

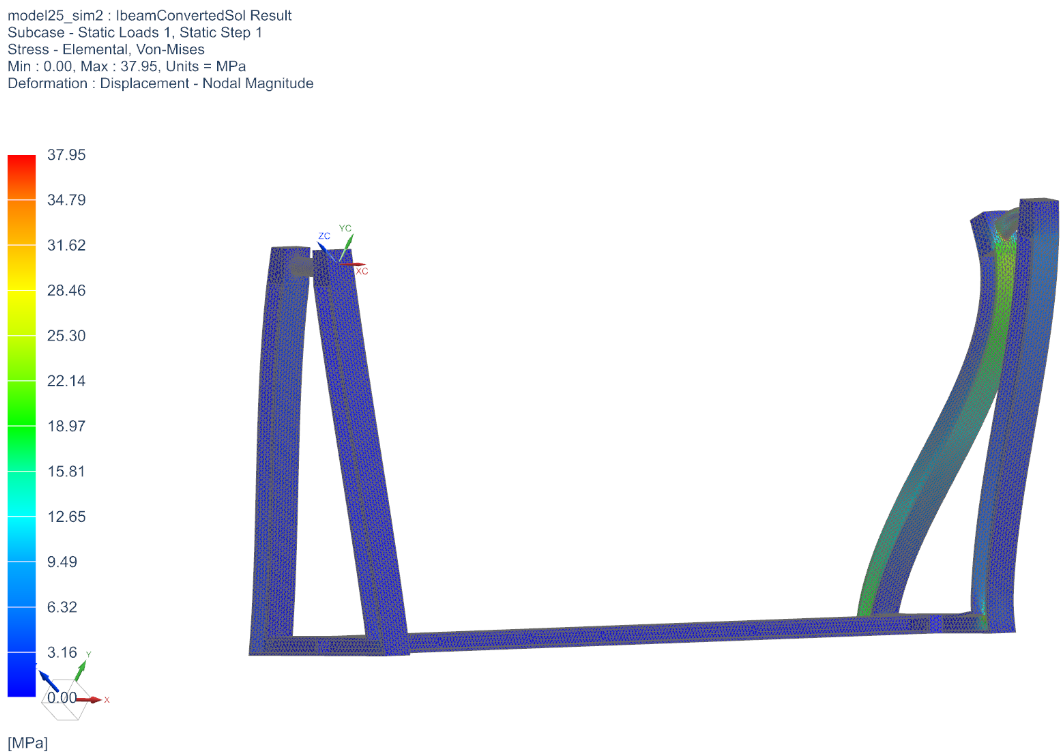


Figure 57: Stress on joint 3 and 4 on modified linkage DE

The maximum stress on join 1 of linkage AB is still higher than the material's ultimate tensile strength. However, as shown in the closed-up view figure 42, the high-stress values are concentrated at the sharp corner between the camphor and the pin. Such corners cause the creation of artificially high stress. The value of such stresses increases with mesh refinement.[2] Linkage AB was further analyzed with a mesh size of 20 mm, and the stress results are shown in Figure 58. As shown in the close-up view in Figure 59, the stress values at the sharp corner go up drastically as the mesh size increases. This result confirms that these stress values are artificially high stress and should be ignored. The stable stress values between the analyses done with mesh sizes of 50 mm and 20 mm will be considered as the maximum stress on the body. The same results were observed for joint 2 of linkage AB. This joint was also analyzed with the mesh element size of 20 mm, and the results are shown in Figures 60 and 73 . The maximum stress, displacement, and safety factor values of each modified model are summarized in table 4.

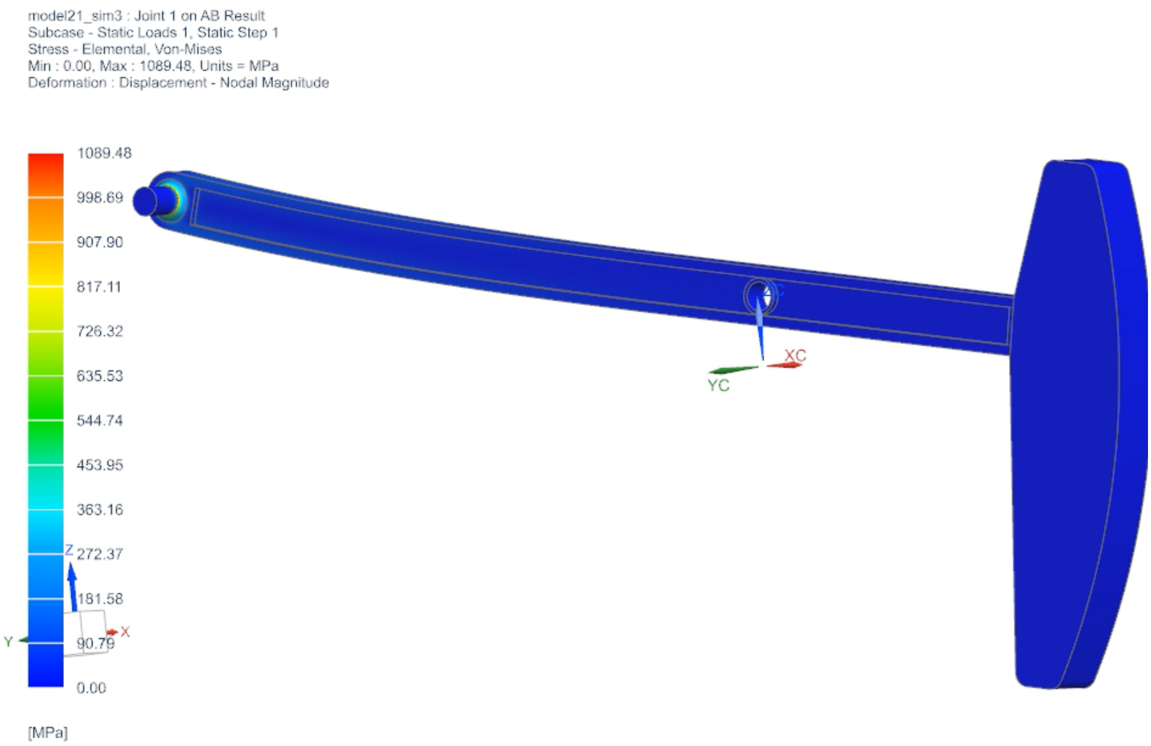


Figure 58: Stress on joint 1 of linkage AB at mesh size 20 mm

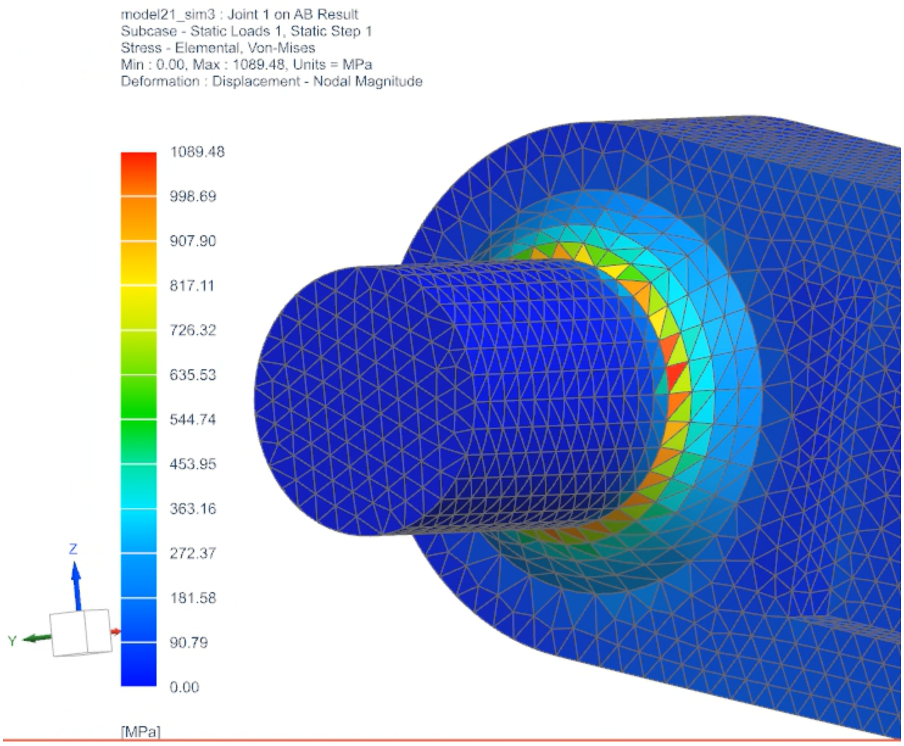


Figure 59: Close-up view of stress on joint 1 of linkage AB at mesh size 20 mm

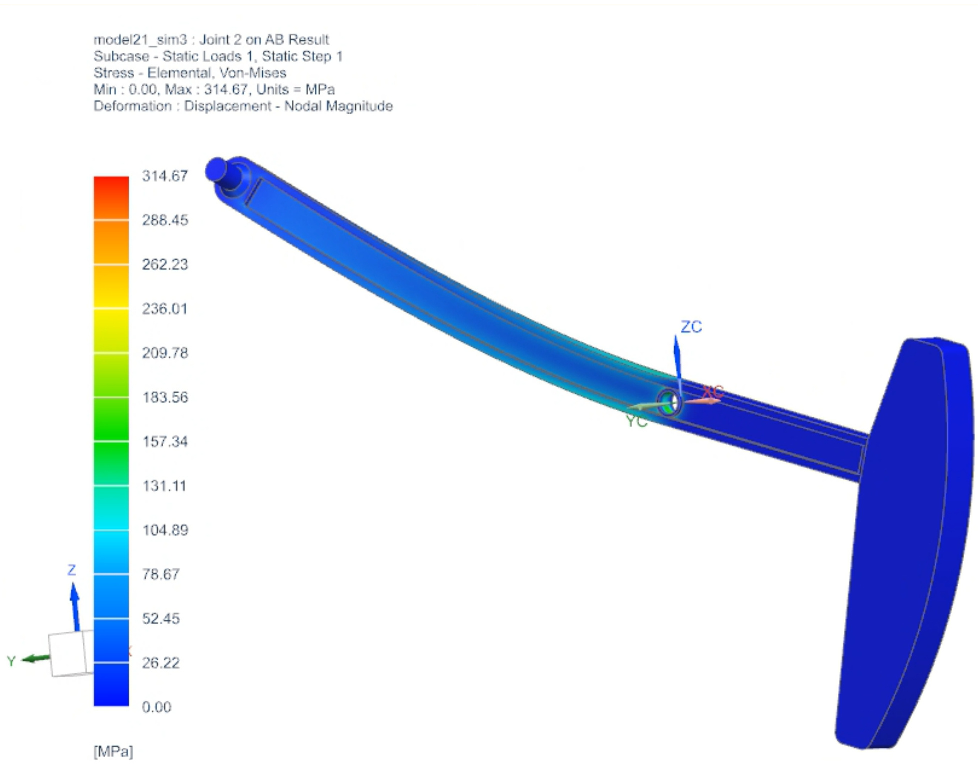


Figure 60: Stress on joint 2 of linkage AB at mesh size 20 mm

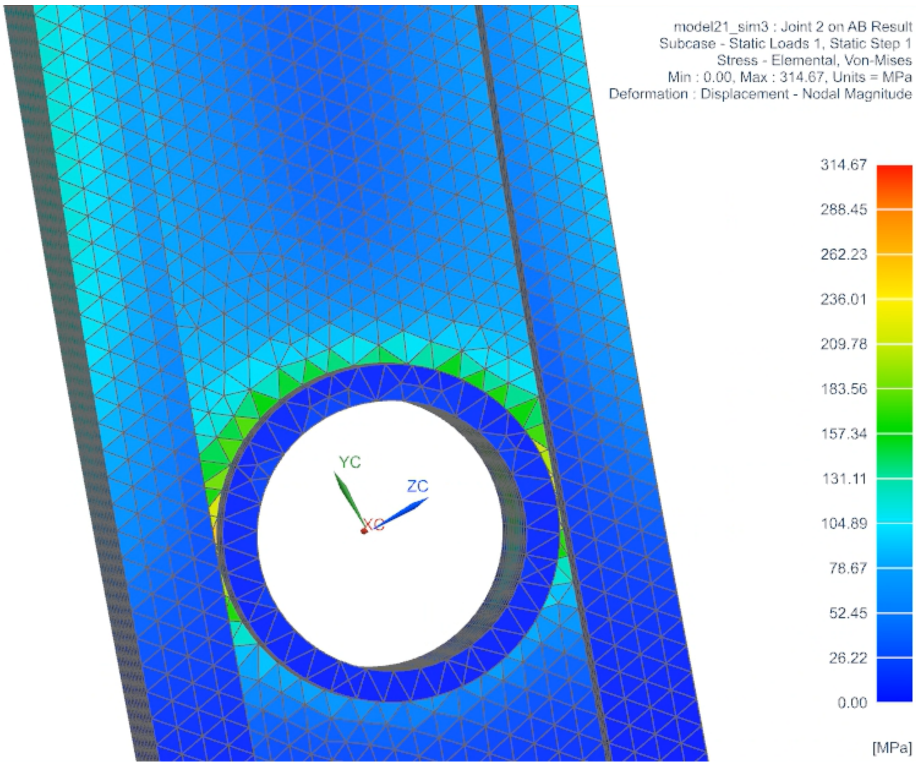


Figure 61: Close-up view of stress on joint 2 of linkage AB at mesh size 20 mm

Table 4: Modified Pumpjack model maximum displacement and stress

joint	max. displacement (mm)	max. stress (MPa)	safety factor
Joint 1 linkage AB	65.37	246.95	2
Joint 2 linkage AB	30.4	264.38	1.9
Joint 2 linkage BC	22.05	42.62	12.2
Joint 3 linkage BC	11.48	22.19	23.4
Joint 3 linkage CD	0.73	20.08	25.89
Joint 4 linkage CD	0.162	24.54.26	21.1
Joint 3 and 4 linkage DE	1.4	37.96	13.69

7 Optimization results for one part

Since Linkage AB is the heaviest one and has the highest von-Mises stress, it is good idea to optimize the geometry of this linkage alone. The aspect we want to minimize is the maximum stress that this linkage is subject to. This can be achieved by going to New Solution Process -> Geometry Optimization of the model in the simulation file in NX.

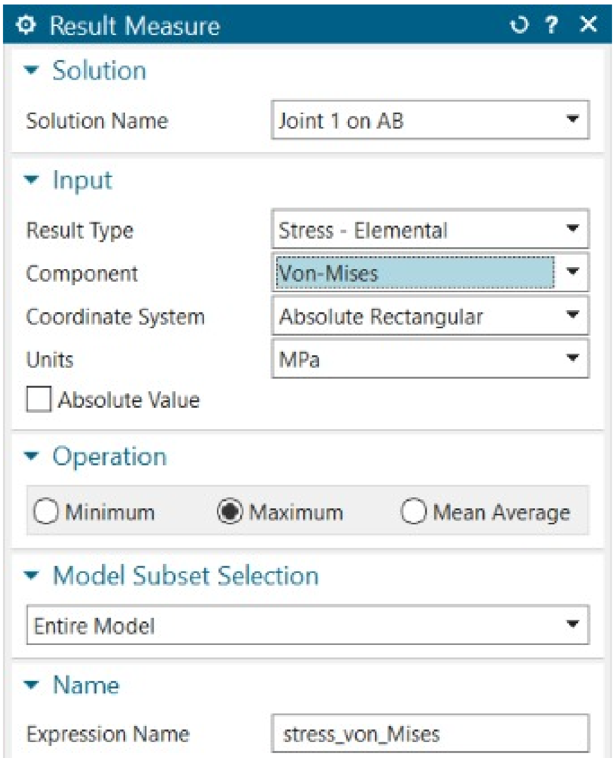


Figure 62: Geometry objective: minimizing the maximum von Mises stress

The constraints are relaxed: the volume or mass can freely change on their magnitude range with a relaxed lower bound.

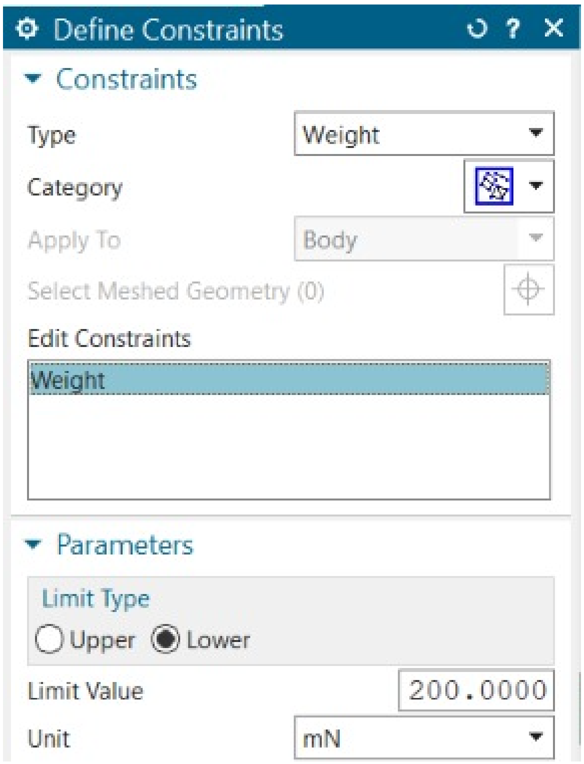


Figure 63: Geometry constraint: relaxed lower bound of weight

After that, we need to choose the variable dimensions for optimization. Two dimensions (p30 and p31) are selected in the Design Variable stage, where Nastran solver will try to solve the optimization problem to find the optimal value of those two dimensions. They are located in the sketch as follows:

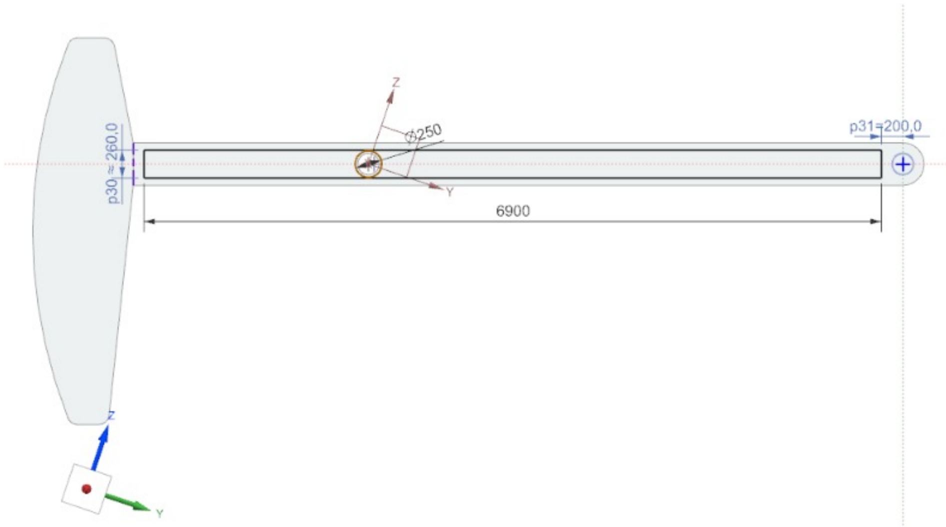


Figure 64: Dimension variables for optimization

Define Design Variables

Name	Initial Value	Upper Limit	Lower Limit
"model21"::p30=259.9...	246.673319	286.000000	234.000000
"model21"::p31=200	210.153760	220.000000	180.000000

Figure 65: Geometry design variables: Two dimensions for optimization

After everything is set up, the Setup is solved to find the optimal dimensions.

Optimization History								
Based on Optimizer								
Design Objective Function Results								
Minimum Result Measure [MPa]	0	1	2	3	4	5	6	7
	592,75	610,06	575,14	597,53	624,89	588,96	574,49	587,31
Design Variable Results								
Name	0	1	2	3	4	5	6	7
"model21"::p30=259.999999998	260	270,4	260	234,018	256,693	234,0382	247,2914	246,6733
"model21"::p31=200	200	200	208	219,9772	219,9772	207,382	209,7718	210,1538
Design Constraint Results								
	0	1	2	3	4	5	6	7
Weight								
Lower Limit = 200.000000 [mN]	1,11E+08	1,11E+08	1,11E+08	1,14E+08	1,11E+08	1,14E+08	1,14E+08	1,14E+08

Figure 66: Geometry optimization results

We can observe that the best design with lowest von-Mises stress is at the 3rd case, where $\sigma_{vM} = 574.49MPa$ with $p30 = 247.29mm$ and $p31 = 209.77mm$. The weight stays constant after optimization due to the relaxed lower bound.

8 Run MBS simulation with updated parts

After we have obtain the optimized dimensions, the sketch of linkage AB is updated according to the two new dimensions. After update, the volume of the hammer becomes a little smaller because both dimensions are shrunk.

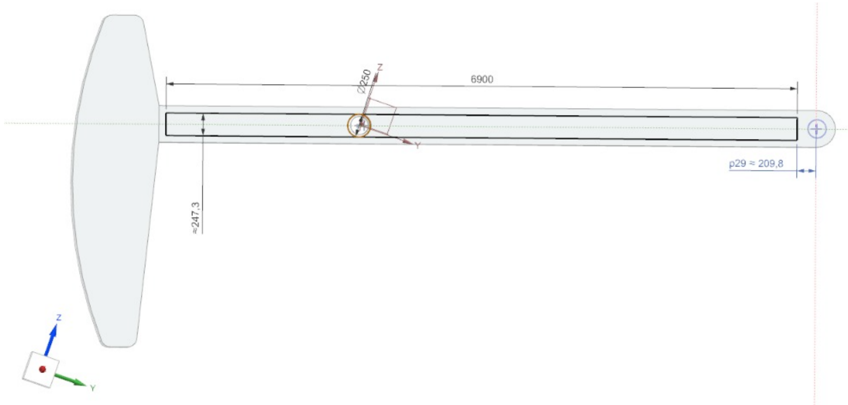


Figure 67: Linkage AB updated with new dimensions

The MBS simulation is built up again with the Dynamic solution on the updated geometry. The same settings is used as the ones in the previous second report of MBS analysis. The Power Driver is located at Joint J004.

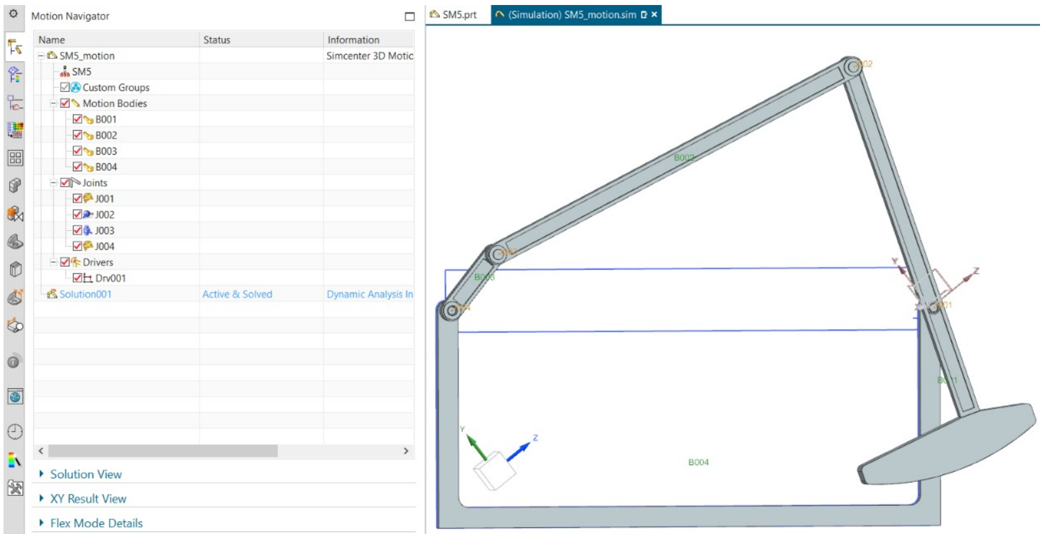


Figure 68: MBS simulation setup for the updated geometry

The Driver has an angular velocity of 60 degrees per second, which means that the pumpjack makes one full rotation in 6 seconds. The plots below are the absolute force of each joint plotted against time (12 seconds)

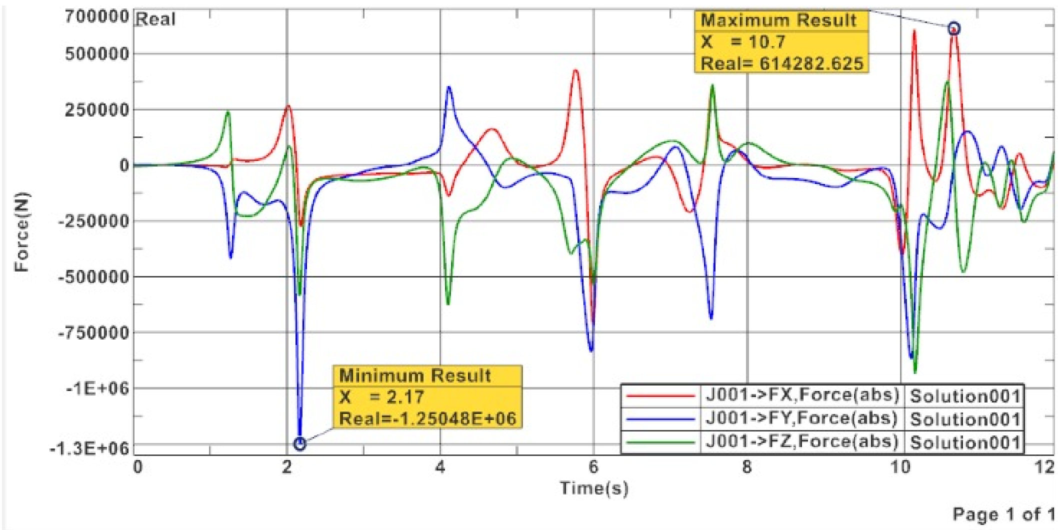


Figure 69: Force plot of J001

We can see that compared to the previous report, the Force Plot becomes much more turbulent due to the complex geometries of the new MBS simulation. Additionally, none of Forces are zero, since the linkages are in 3-dimensions.

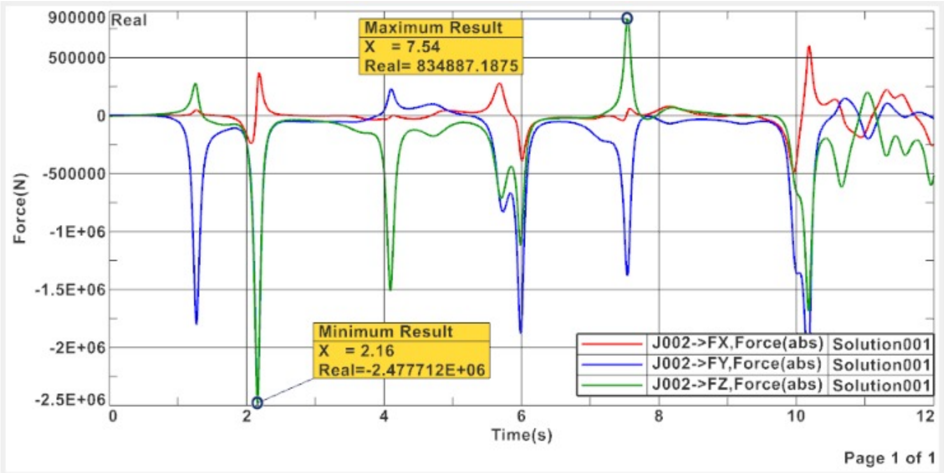


Figure 70: Force plot of J002

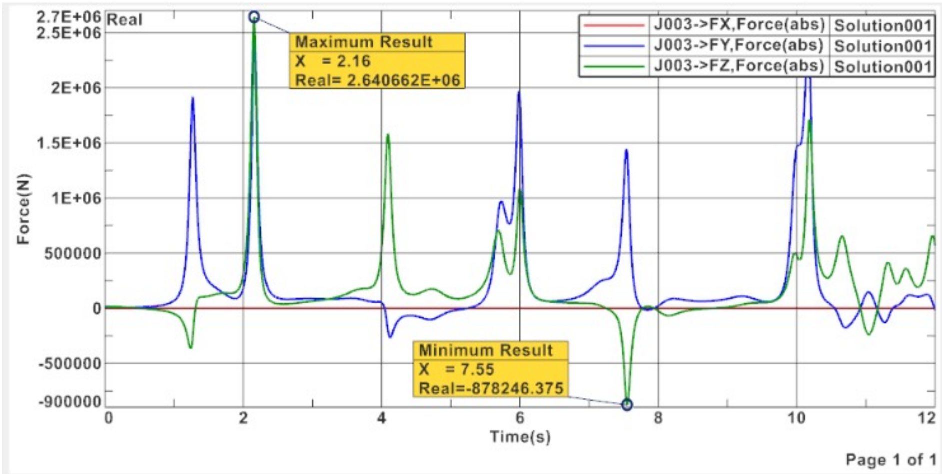


Figure 71: Force plot of J003

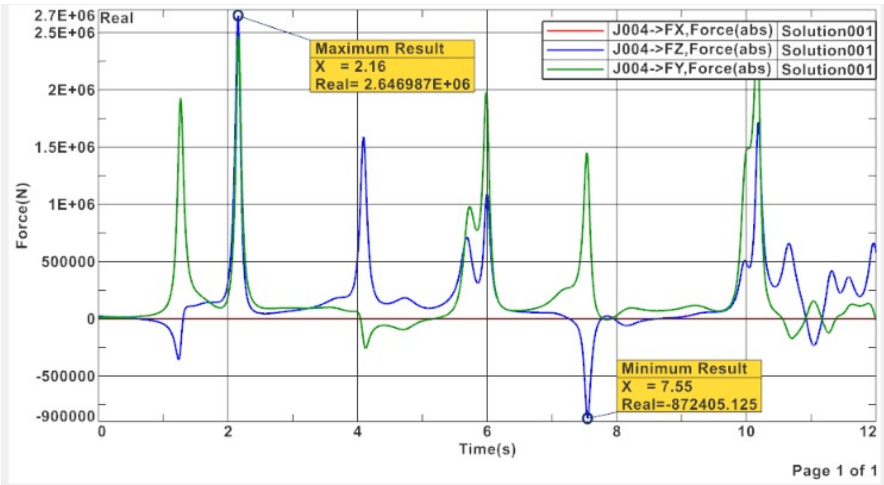


Figure 72: Force plot of J004

To derive the Power needed to run the pumpjack, we need to obtain the maximum relative Torque on the Driver, which is joint J004.

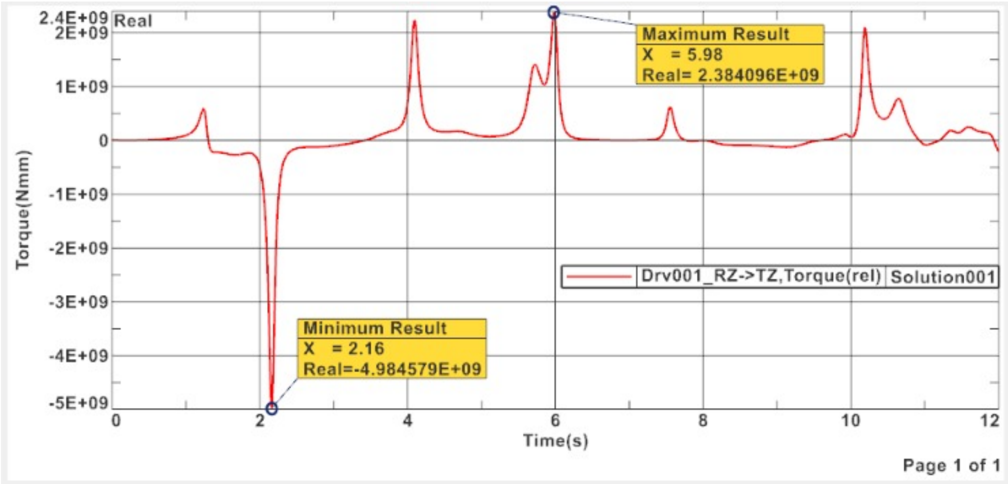


Figure 73: Torque plot of J004

From the above figure, we can see the absolute maximum torque is 4.984e9 (Nmm) or 4.984e6 (Nm). Power P can be calculated from the torque and rotational velocity according to the following equation:

$$P = T \times \omega \tag{1}$$

The angular velocity of the power source is $60\ degrees/s$ or $1/3\ radian/s$. Therefore, the updated power to run the pumpjack is:

$$P = 4.984e6 \times (1/3) = 1.661e6 \approx 1661000Watt$$

The original power calculated in the previous report is only 70310.6 Watts. The updated power is larger by magnitude of a hundred, suggesting that we have miscalculated somewhere, or we have underestimated the true power of operating the pumpjack in the previous report.

9 Validation of simulation results

As a part of validating the FEM analysis, the same analyses under the exact same loading conditions were done using Ansys. The stress analysis result for the linkage DE using Ansys is shown in Figure 74. According to the Ansys analysis results, the maximum stress on the linkage is 34.6 MPa; from NX results, it was 37.96 MPa. Although the numerical values of the results are not exactly the same, they are quite close to each other. These results confirm that the primary analysis method adopted for NX analysis was accurate.

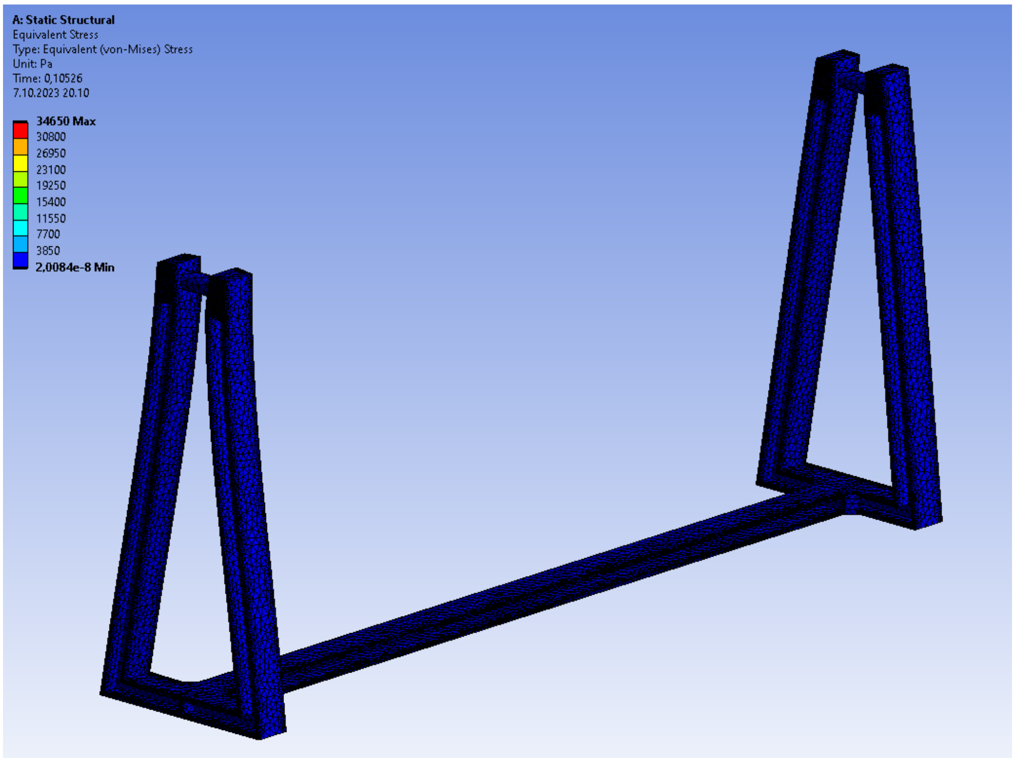


Figure 74: Stress on joint 3 and 4 of linkage DE using Ansys

Simplified hand calculations are another way of validating the simulation results, as the pumpjack structure is still relatively simple. Producing scale models and applying loads will also improve the designer's understanding of which component or joint will fail first. In real life, cylindrical and thrust bearings will be used between the joints. These bearings will help distribute the load around shafts equally. For the analysis done in this report, only the loads calculated from Multi-body simulation were used. However, as the pumpjack is heavy machinery, each component is also considerably heavy and applies significant load on other components. The weight of the component also must be considered for more accurate simulation results.

10 Learning outcomes

This week's most crucial learning outcome was the fundamentals of FEM analysis and learning how to use it. All of our team members had theoretical knowledge of the FEM analysis; however, it was our first time using it to analyze a part that we had designed. This week was most exciting as we did multiple iterations of design modifications and stress analysis. Making an assumption, designing, and then seeing the analysis results, which sometimes confirmed our assumption and sometimes gave completely different results, was a particularly stimulating part of this task. When the analysis results did not go as we had assumed, it made us question whether the settings done for the analysis were wrong, or our assumption was wrong. The same analysis was done both in Ansys and NX throughout this report to ensure our analysis settings are correct. However, one strange effect was observed: even though the stress and displacement values were close, the deformation animation in NX differed from the one in Ansys. Ansys deformation seemed more accurate and intuitive.

The next step is to modify the design with the manufacturability of pump-jack linkages in mind. As the weight of each linkage component is noteworthy, the analysis should be redone to add the load acting on each body due to the weight of other bodies. Sharp corners must also be avoided as they create artificially high stress and make the FEA analysis unreliable. Because of our combined efforts, we expect to receive a grade of 5 for this report.

References

1. *410 Stainless Steel Properties* https://www.kvastainless.com/410.html#:~:text=AISI%20Type%20410%20is%20a,from%20500%20to%201400%20MPa.. (accessed: 2023-10-07).
2. *Artificially High Stress* https://www.youtube.com/watch?v=iTqPgcTurXc&t=94s. (accessed: 2023-10-07).