MEC-E1060 - Machine Design

FEM Report - Meat Chopper

Members
Min Hein Htike (100530360)
Elias Puolakka (789981)
San Vo (100480423)

Submitted on October 9, 2022

Table of contents

MEC-E1060 - Machine Design **1**

FEM Report - Meat Chopper 1

1. Introduction 3

2. FEM Analysis 3

3. Updated CAD models for the mechanism 4

4. Materials and FEM models for all parts 5

5. Calculated displacements/stresses of all parts 10

6. Different shape testing 12

7. Optimization results for one part 15

8. FEM Analysis on New Parts 16

9. Simulation Validation 19

10. Discussion 20

11. References and Sources 22

Appendix 23

Different Shape Testing for Triangle Part 44

1. Introduction

In our previous report on MBS simulations we built a three dimensional model of our mechanism. We also built an animation of our mechanism. Then we proceeded to use our simulations to calculate the displacements, velocities, forces and torques of the mechanism. We also optimized the size, power and cycle times to suit the mechanism criterion we developed.

We started working on the report by agreeing on tasks and deciding the parts to optimize on our mechanism. Furthermore we participated in most of the assignment sessions to produce the best possible mechanical design and report provided of it.

This week we are first introducing the concept of FEM-modeling in chapter 2. After this the report will look into improving the three dimensional animated mechanism by ensuring the mechanism link members are unable to collide with one another. This is covered in chapter 3. Furthermore, we will reduce the friction and size of the mechanism further as what we have is a continuously improving project. This week we are also determining the materials used in our mechanism based on our desired functionality and the requirements set by ISO-standards as discussed in chapter 4.

Furthermore, in chapter 4 we're creating FEM-models for all the main parts based on our previous week's force and torque results and the materials selected in this report. These can then be considered with the charts that show the movement direction and thus we can produce accurate FEM-models. In this report we are calculating the displacements and stresses of the parts based on the mentioned simulations. This is covered in chapter 5.

In chapter 6 we are testing various shapes to further optimize our part and in chapter 7 we conduct a geometry optimization which will enable us to have a more optimal usage of materials with better mechanism characteristics. Near the end of the report in chapter 8 we have reconducted our FEM-analysis for a part based on the updated shapes and optimized part models. Lastly in chapters 9 and 10 we discuss the validation of simulation results and our conclusions.

2. FEM Analysis

A common technique for numerically resolving differential equations that appear in engineering and mathematical modeling is the finite element method (FEM). The conventional topics of structural analysis, heat transfer, fluid flow, mass transport, and electromagnetic potential are typical issue areas of interest.

Using two or three spatial variables, the FEM is a general numerical method for solving partial differential equations (i.e., some boundary value problems). The FEM breaks down a complex system into smaller, more manageable pieces known as finite elements in order to solve an issue. The numerical domain for the solution, which has a finite number of points, is implemented by creating a mesh of the object using a specific space discretization in the space dimensions. In the end, a set of algebraic equations emerges from the formulation of a boundary value problem using the finite element approach. The technique makes domain-wide approximations of the unknown function. [1] The small system of equations that describes these finite elements is then combined with other equations to model the full issue. The calculus of variations is used by the FEM to minimize an associated error function and then approximate a solution.

3. Updated CAD models for the mechanism

The design from our previous report is presented in image 1. The problem with the shown mechanism is that the mechanism bars F-G and B-E are able to collide with each other thus preventing the mechanism from achieving the full potential range of movement of the point E which connects with the knife used in the mechanism. Thus this mechanism could not achieve the full desired cutting movement without colliding. Furthermore, in this mechanism the bars are in constant touch causing friction which converts energy to heat and causes increased power consumption. Lastly, the mechanism is not optimized in terms of size. In the mechanism the connection areas around the pins have the same thickness as the bars themselves even though we could reduce the footprint of the mechanism by making the bars slimmer around the connection points as the main deflection will be around the mid portion of each given bar.

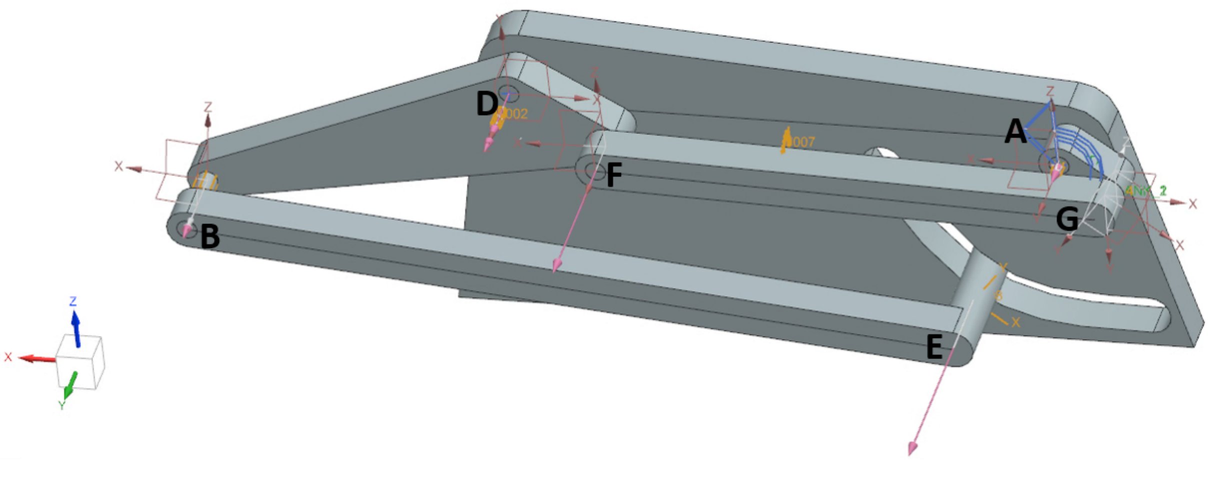


Image 1: Original mechanism design structure.

Below we present images of the updated CAD-models. We changed the mechanism so that the rod titled B-E is located at the other side of the mechanism; this prevents the rods from colliding with each other when the mechanism is fully extended to its maximum size. Furthermore we see that because the rods now have a small space between them we have greatly reduced friction. The updated mechanism design is presented in Images 2 and 3. On image 2 you see one rod missing and on image 3 you see the rod that was previously on the other side.

Furthermore, we have reduced the rod width around the pin points further making the mechanism more compact. These changes make the mechanism horizontally thinner in the y-direction of the reference Images 2 and 3.

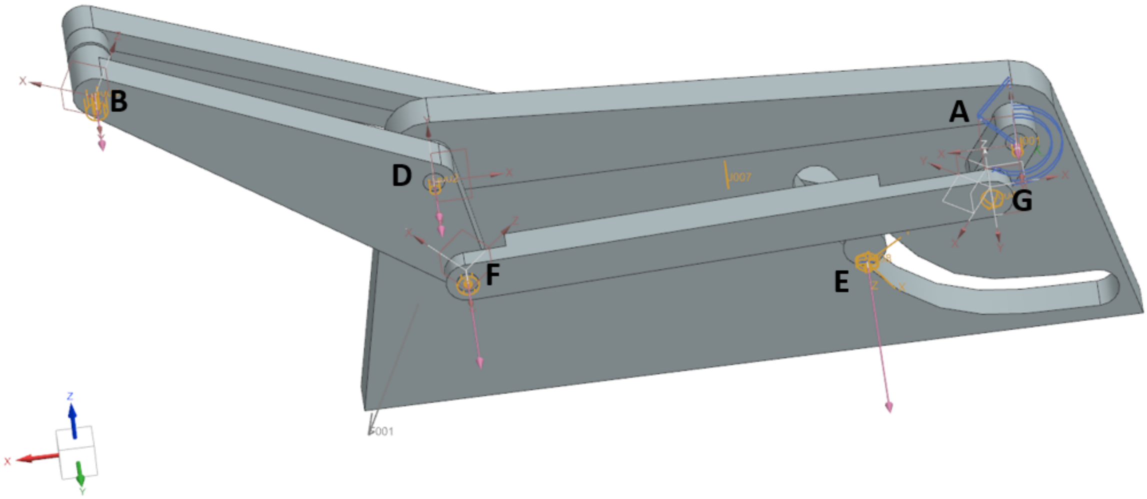


Image 2: Updated mechanism design structure front-side.

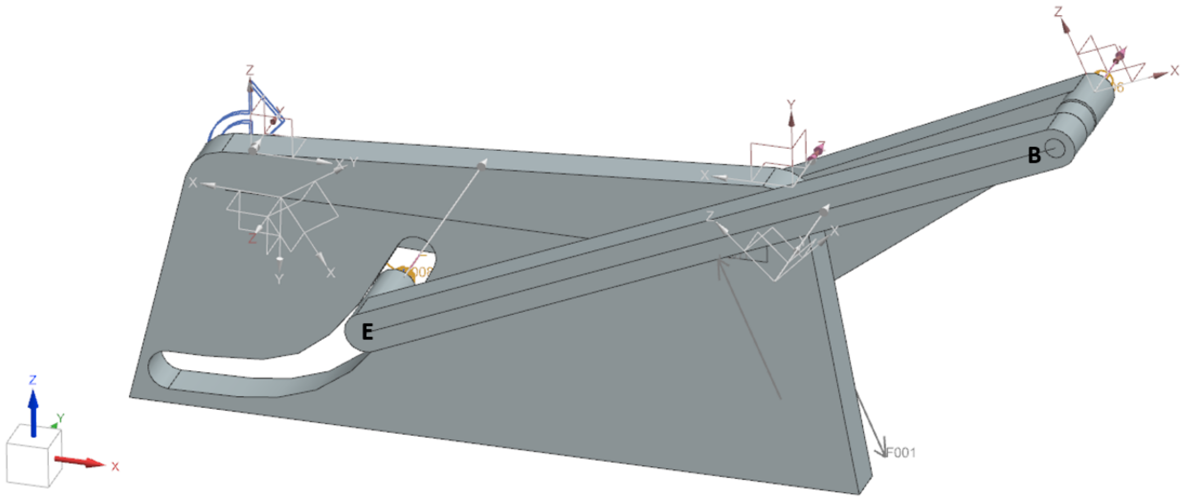


Image 3: Updated mechanism design structure back-side.

4. Materials and FEM models for all parts

Kitchenware needs to be durable and must sustain continuous exposure to water. The material we selected for the parts is Stainless steel 316. This is the second most common stainless steel in the world after grade 304. While 316 is more expensive the key advantage it has is that it contains between 2 and 3 percent molybdenum which makes it more corrosion resistant especially against industrial solvents like chlorides. This grade of stainless steel is commonly used in industrial and marine equipment as well as food production and commercial kitchens. Picture of 316 stainless steel bars is presented in Image 4.

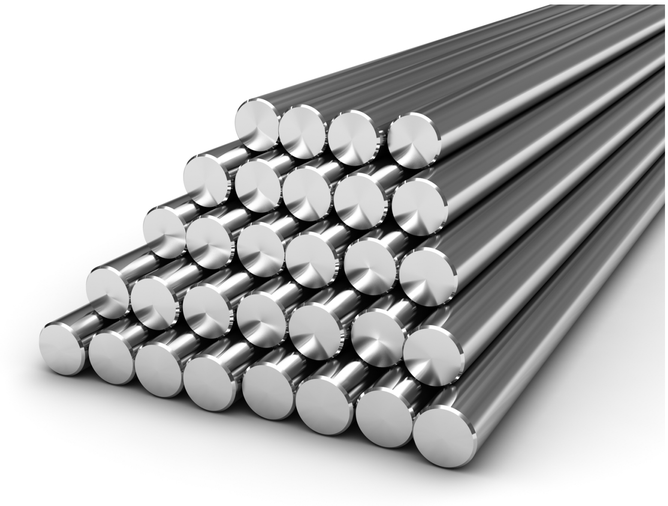


Image 4: 316 Stainless steel bars.

Our material selection is based on our list of desired features specified in table 1, and the recommendations and requirements of ISO-standards. As stated in standard SFS-EN 631-2:en [1] for kitchenware, our device must be made so that the construction does not make possible any permanent distortion of the mechanism under normal use. The 316 stainless steel is a very corrosion resistant and strong material unlike plastic. So, this requirement is clearly met. The mentioned standard also states that the appliance should be easy to clean and must not have grease, oil or other substances which could be poisonous or carcinogenic. This requirement is also met because 316 Stainless Steel is perfectly safe and non carcinogenic for humans.

Table 1 Requirements list for the mechanism

Req	Level	Measure
Cutting force ≥ 30 N	Demand	MBS Software
Cutting Cycle < 2 s	Demand	MBS Software
Weight of product < 10 kg	Non-Demand	FEM Software
Maximum Length < 30 cm	Demand	FEM Software

Standard of SFS-EN 1672-2:2020 [2] also states that material cannot be porous and must be smooth enough so we don't have food staying in small crevices; such a requirement eliminates many foams and materials with bad surface quality. Stainless steel having the high density common to metals and the possibility of surface grinding mean this standard is satisfied.

After updating the mechanism geometries, removing any overlapping connections and deciding on the materials, we look at the desired constraints. To simulate displacement we want to have one pinpoint always free to deflect under loads. In our FEM-simulations one such point is always free and the highest occurring loading is applied to it according to the force and torque results of our MBS report. For example a model with two pinpoints will receive two simulations and one with three pin points will receive three simulations. Meaning one simulation for each pin point. The other point or points in the case of the triangle member will be locked in place using the fixed constraint option of Siemens NX. Below images 5-9 present the FEM-meshes we developed for our parts. Special emphasis was placed on ensuring uniform meshes so that we receive realistic simulation results. In the pictures the meshed side represents the attached point of the mechanism and the arrows on the other end display loading that will produce the desired displacement.

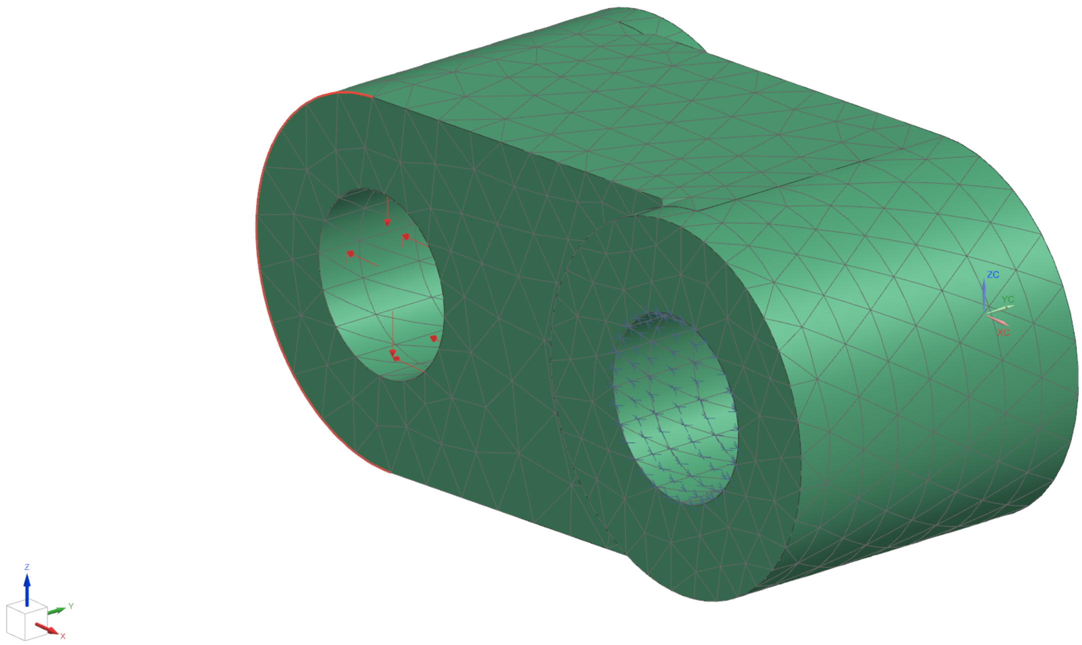


Image 5 Short Crank FEM Model

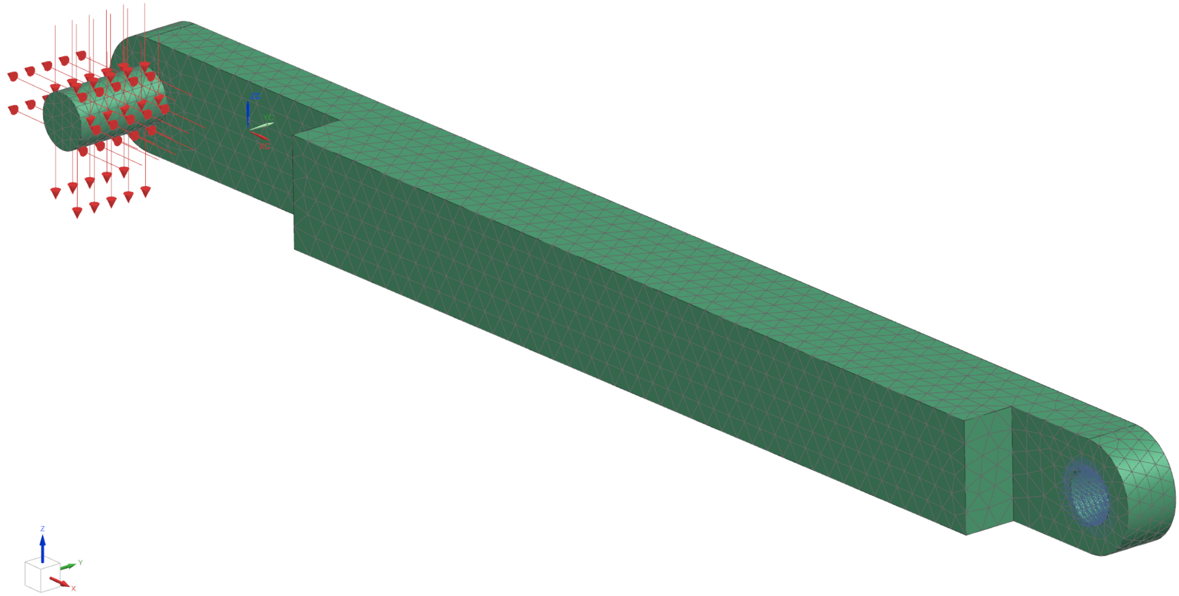


Image 6 Long Crank FEM Model

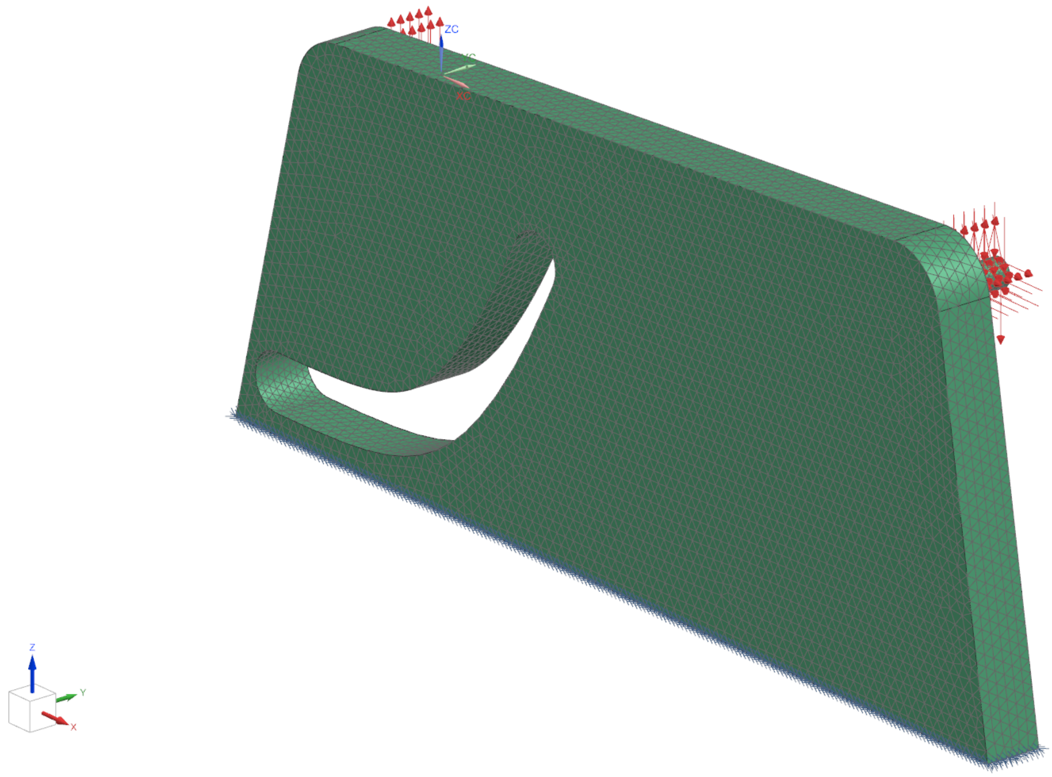


Image 7 Ground body FEM Model

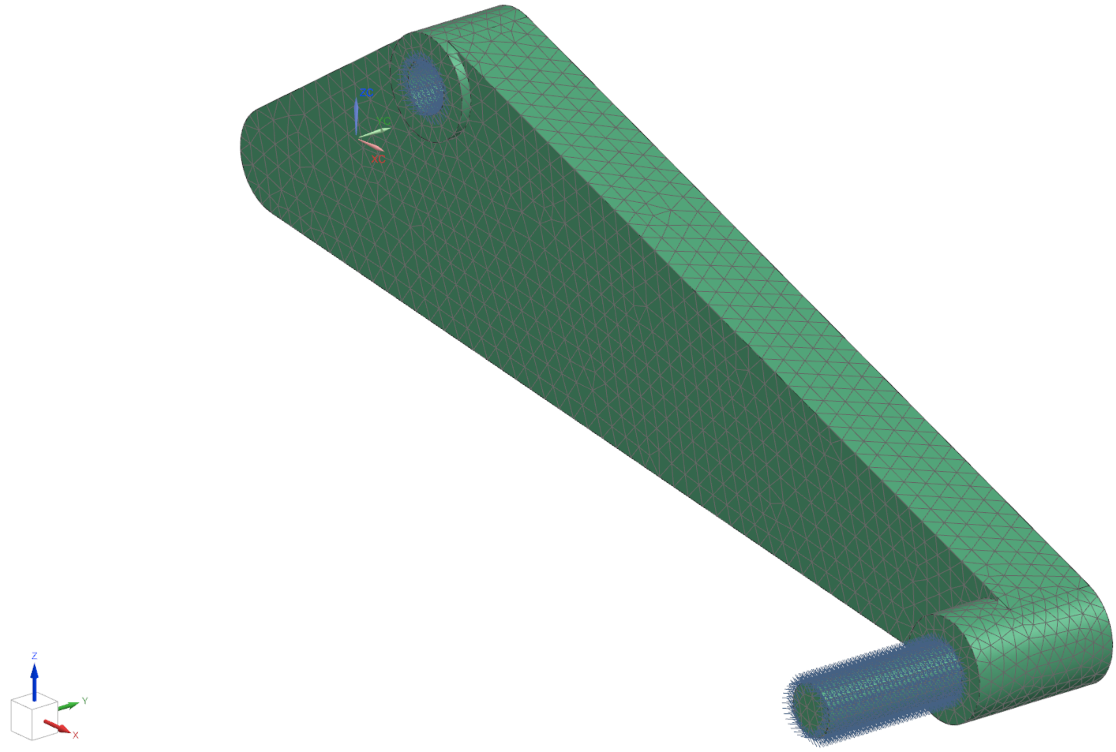


Image 8 Triangle Part FEM Model

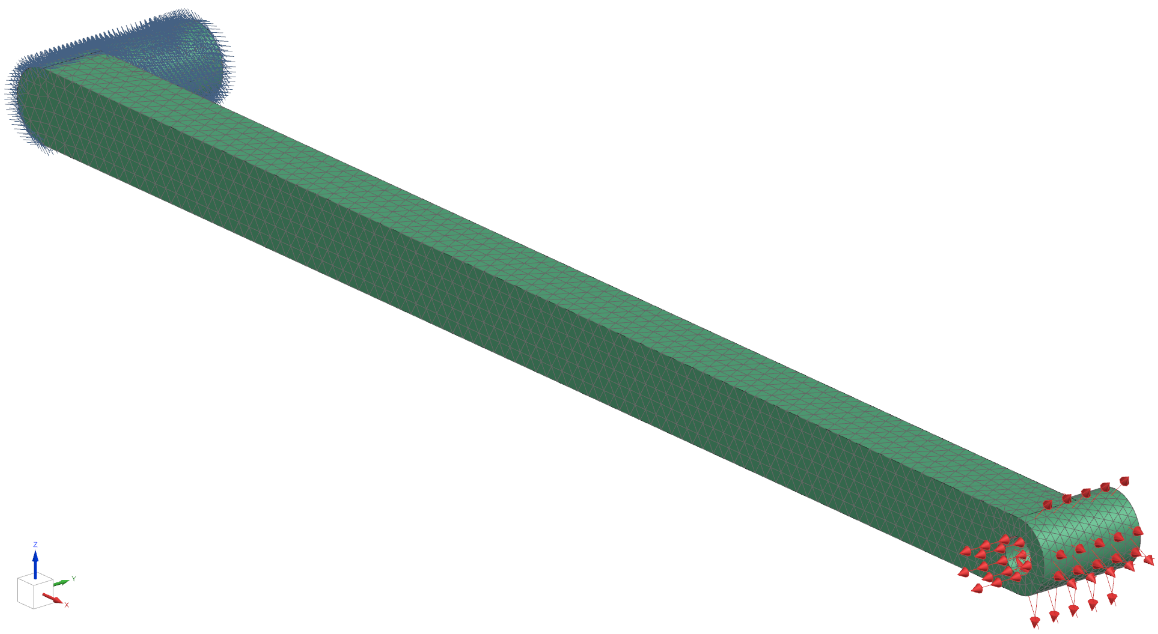


Image 9 Knife Part FEM Model

The table 2 presents the mesh sizes used in the parts. The reason we are utilizing the size of 2 on all parts is to maximize simulation accuracy while minimizing the overall simulation times to a reasonable total time. The average simulation time was under a minute.

Table 2 Mesh Sizes Used for the Parts

Part	Mesh Size (mm)
Short Crank	2
Long crank	2
Triangle	2
Knife Part	2
Ground Part	2

5. Calculated displacements/stresses of all parts

Nonlinear FEA simulates the shear, tension, and bending stresses using complicated equations and empirical data. A set of vector equations are used in the software computations to describe how a component would behave under stress. The overall stress response is then created by combining the directional and deformation responses.

The table 3 shows all the maximum displacements of the parts under the maximum loads we gathered from the simulations in the MBS report. These displacement results were gathered from the excel file NX produces. In the table the Case column signifies the location of the free joint that can deflect under our loads. This means two cases for most parts and three cases for our triangle rod which has three pin locations. The contour plots for the maximum displacement and stress can be seen in Appendix.

Table 3 Maximum Displacement and Stress in Parts for Different Cases

Part	Case	Maximum Displacement (mm)	Maximum Stress (MPa)
Short Crank	A free	1.726e-05	16.59
	G free	4.539e-06	7.544
Long crank	G free	7.241e-03	331.37
	F free	3.532e-04	71.29
Triangle	F free	5.715e-05	105.51
	D free	1.230e-04	41.42
	B free	9.744e-05	88.18
Knife Part	B free	1.171e-03	38.91
	E free	1.997e-03	28.35

Ground Part	A free	1.172e-04	126.20

The twin model of the mechanism is also created in another software, Solidworks, to compare the result. The model in Solidworks went through every stage that the NX model took, including MBS, dynamic simulation, and FEA. Finally, the FEM results are shown in the below table.

Table 4 Simulation result comparison

Part	Criteria	NX	Solidworks
Short Crank (G free)	Maximum Displacement (mm)	4.539e-06	3.507 e-4
	Maximum Stress (MPa)	7.544	5.83
Long crank (F free)	Maximum Displacement (mm)	3.532e-04	2.216 e-2
	Maximum Stress (MPa)	71.29	20.03
Triangle (B free)	Maximum Displacement (mm)	9.744e-05	8.383 e-3
	Maximum Stress (MPa)	88.18	11.90
Knife Part (E free)	Maximum Displacement (mm)	1.997e-03	9.847 e-2
	Maximum Stress (MPa)	28.35	14.35
Ground Part (A free)	Maximum Displacement (mm)	1.172e-04	2.908 e-2
	Maximum Stress (MPa)	126.20	162.0.

In general, from table 4, we could see that there is a big difference between the simulation results of NX and Solidworks in all criteira of all parts. This may be caused by:

- The sliding joint in NX is modeled as a point-on-curve geometry constraint, whereas Solidworks requires the model to have a Solid Body Contact in order to function.
- Although we did the best job to match the material characteristic, there are still some mismatches of the material input as one requires the property that the other doesn’t.
- The mismatching in parameters also occurs in meshing and FEM solvers.

From the difference of the results of two softwares, the team now learns that every software has its own approach and method to simulate a model, which can cause a huge impact on the conclusion of the simulation process.

6. Different shape testing

The results obtained from FEM analysis can vary with the different shape analyzed by the Siemens NX, with different displacement and stress results obtained from analysis. So it is paramount to develop a few different shapes to see if the material properties would be enhanced. Right now, we plan to use two different shapes for the triangle part since it has the most impact on the overall mechanism, having the highest stress values.

Our team tested various rod shapes to determine the best possible approach from the standpoint of loading characteristics. Below images 10-13 present the different types of loaded rods.

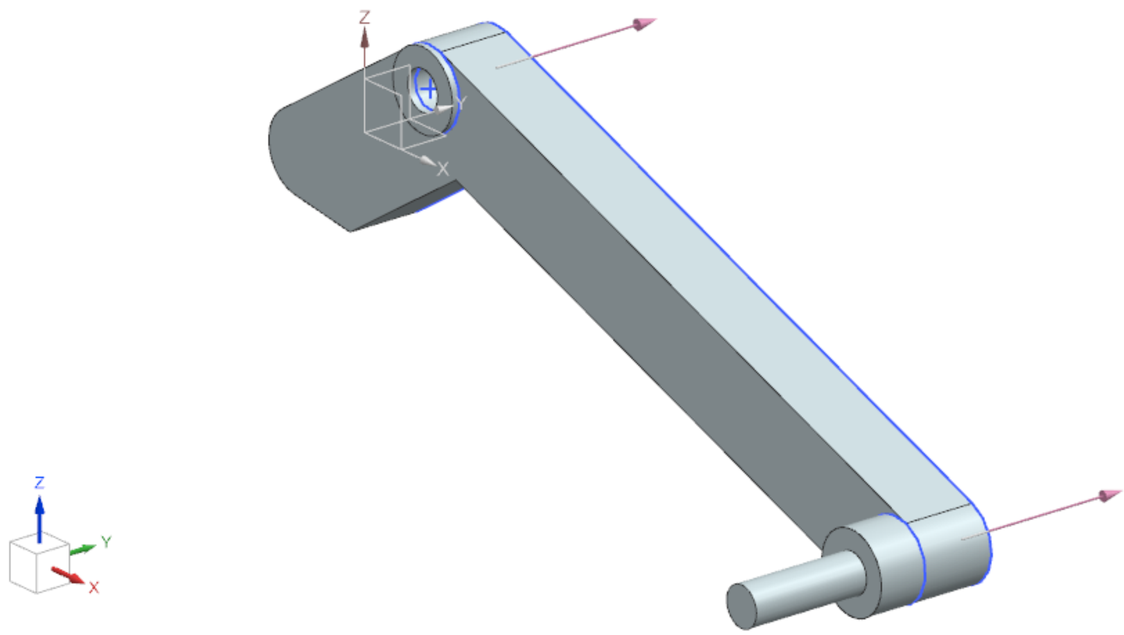


Image 10: New Design for Triangle Model Part (1)

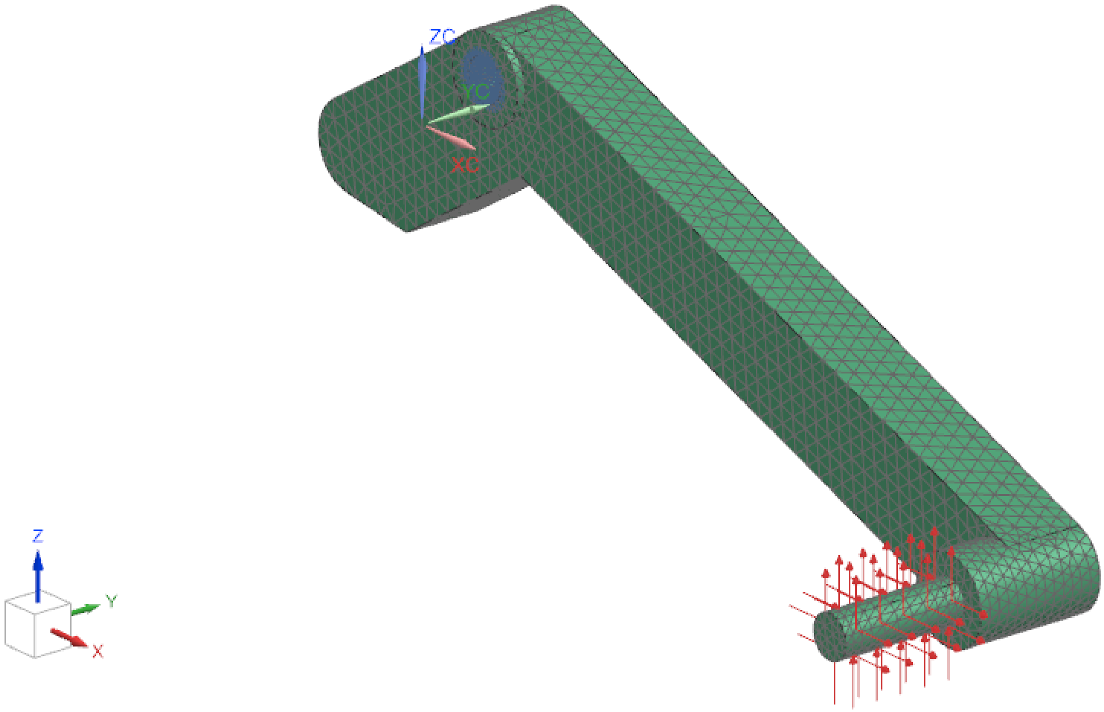


Image 11: FEM Mesh model for Triangle Model Part (1)

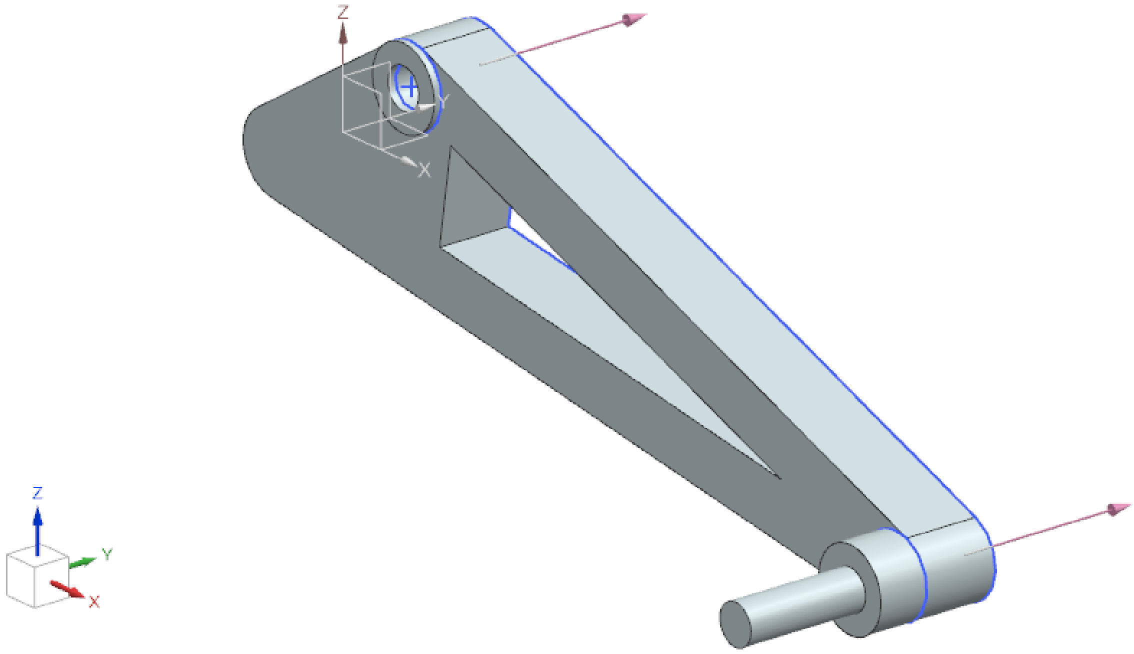


Image 12: New Design for Triangle Model Part (2)

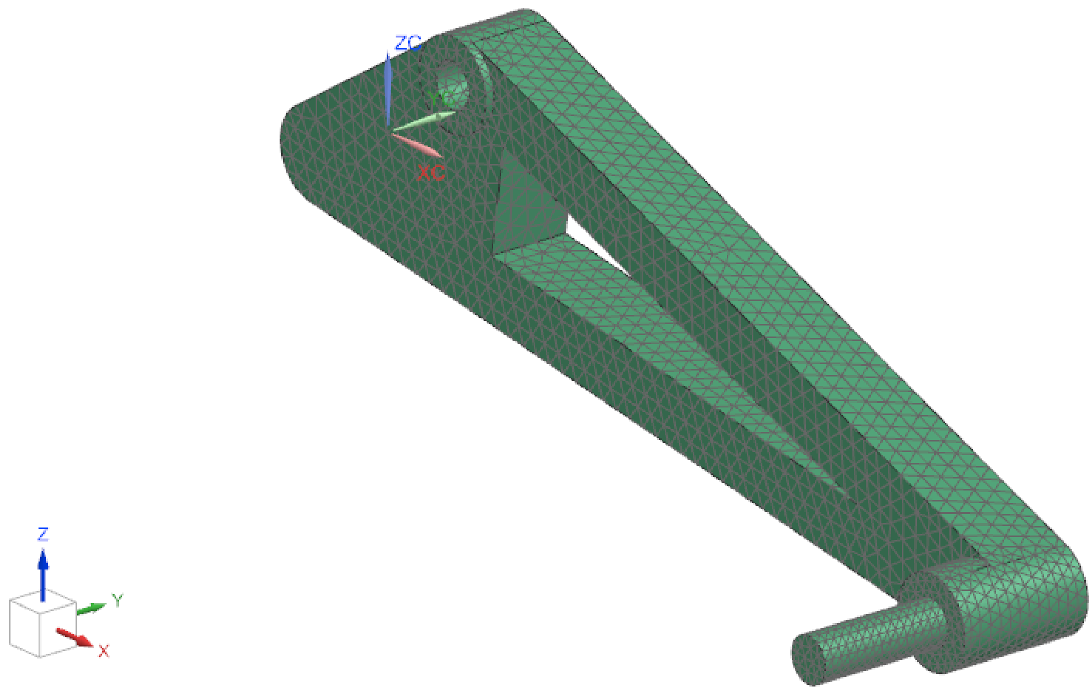


Image 13: FEM Mesh model for Triangle Model Part (2)

Table 5 Maximum Displacement and Stress in Parts with new shapes

Constraint	Criteria	Shapes		
		Original Design	New model 1	New model 2
F free	Maximum Displacement (mm)	5.715e-05	6.365e-03	4.309e-03
	Maximum Stress (MPa)	105.51	54.91	55.84
D free	Maximum Displacement (mm)	1.230e-04	2.878e-03	2.573e-03
	Maximum Stress (MPa)	41.42	55.50	55.06
B free	Maximum Displacement (mm)	9.744e-05	0.0181	1.107e-02
	Maximum Stress (MPa)	88.18	16.66	16.51

From the following observations in Table 4, we can see that the original design has the lowest displacement out of every configuration for every single case. However, the triangle part experiences higher stress for F and B free configurations but lower maximum stress for D free configuration. Thus, it can be concluded that the new parts have better stress conditions and the product designer should consider the following shapes for further analysis and development.

7. Optimization results for one part

With some restrictions, the dynamic and thermal behavior of a machine design can also be calculated using the finite element method by the machine designer. This enables him to determine, for instance, the overall machine's static stiffness at the tool center point (TCP) or the values of the undamped natural frequencies before the first physical prototype of the machine is produced. This gives him an idea of the predicted machine features. Typically, Finite Element Analysis (FEA) is used to pinpoint design flaws so they can be fixed in a later step of the design process.

The part we chose for the optimization was the triangle bar of our mechanism. The reason it was the best option is that it has the most connection links and thus we consider it the most important part of the mechanism. Our goal was to conduct the geometry optimization for the reduction of stress.

Optimization History															
Based on Optimizer															
Design Objective Function Results															
Minimum Weight [N]	0	1	2	3	4	5	6	7	8	9	10	11	12	13	14
	1.19E+09	9.54E+08	8.42E+08	1.46E+08	6.5E+08	5.69E+08	5.19E+08	4.56E+08	4.17E+08	3.9E+08	3.69E+08	3.53E+08	3.4E+08	3.29E+08	3.22E+08
Design Variable Results															
Name	0	1	2	3	4	5	6	7	8	9	10	11	12	13	14
"model8"::p20=-1	-1	-2.7	-3.50421	-8.4999	-4.88609	-5.46767	-5.81992	-6.27694	-6.55162	-6.75217	-6.89766	-7.01414	-7.1068	-7.18374	-7.23967
Design Constraint Results															
	0	1	2	3	4	5	6	7	8	9	10	11	12	13	14
Result Measure															
Upper Limit = 25.000000 [MPa]	22.477	24.243	20.114	66.174	21.287	20.926	14.823	16.923	17.954	19.423	20.21	20.944	21.494	22.389	22.848
Small change in design, run converged.															

Image 14 Optimization Process Example

Table 6 Design Objective Function Results

	0	1	2	…	12	13	14
Minimum Weight [N]	1.19E+09	9.54E+08	8.42E+08	-	3.4E+08	3.29E+08	3.22E+08
	Design Variable Results						
"model8"::p20=-1	-1	-2.7	-3.50421	-	-7.1068	-7.18374	-7.23967
	Design Constraint Results						
Upper Limit = 25.000000 [MPa]	22.477	24.243	20.114	-	21.494	22.389	22.848

For the triangle part of the mechanism, it can be seen that the design study 6 will give the best result with the lowest maximum stress acting on the body, with only 14.823 MPa. For further improvement on the mechanism, it is advised to perform further analysis in the future study.

8. FEM Analysis on New Parts

In this section, the FEM simulation results from design cycle 6 are shown and the values compared with the previous configuration. The simulation files and the mesh files are given in Image 15-17.

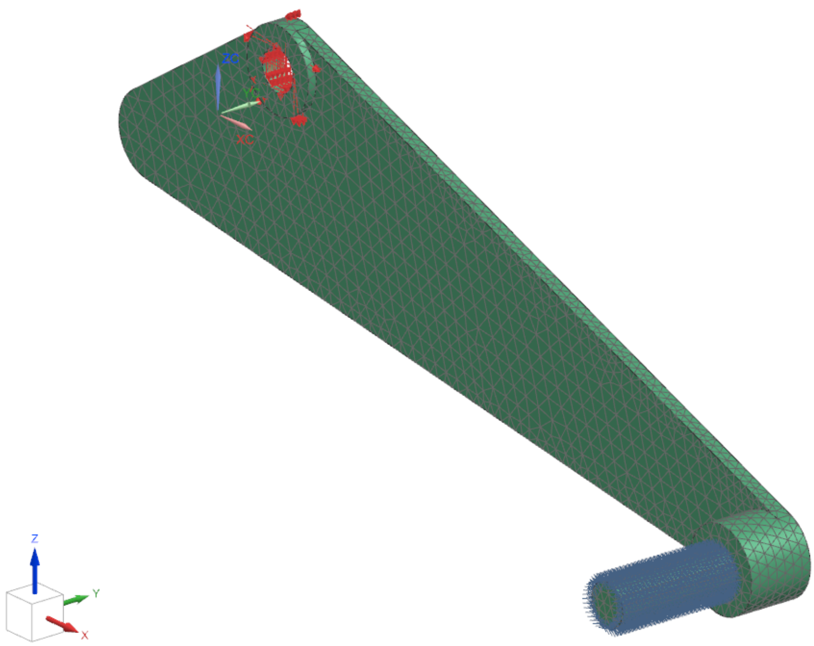


Image 15: Part file for the new FEM model

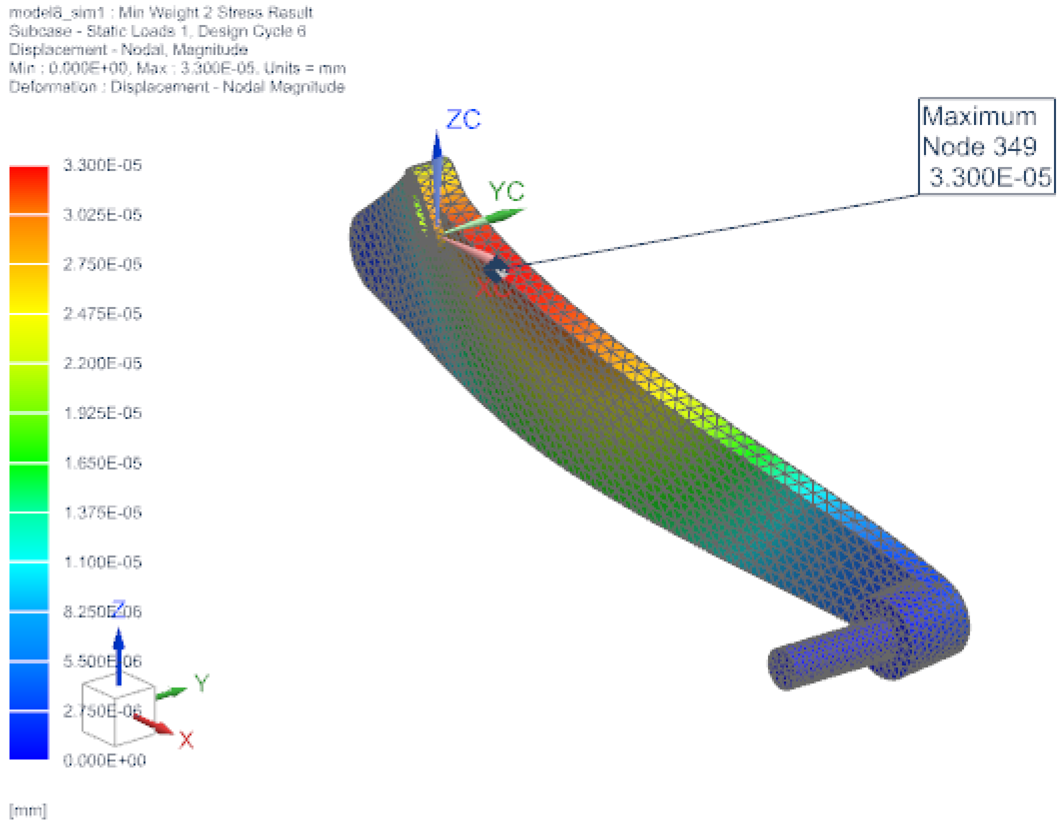


Image 16: Maximum Displacement for Design Cycle 6

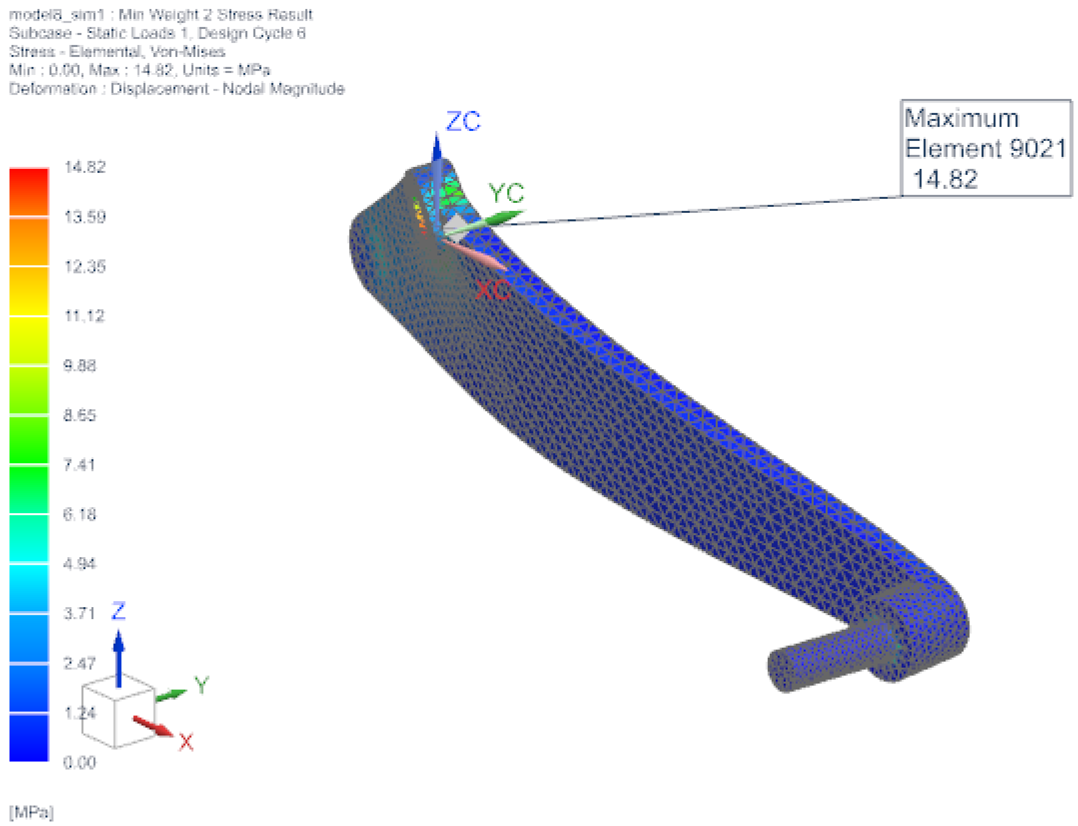


Image 17: Maximum Stress for Design Cycle 6

Table 7: Maximum Stress acting on optimized part

Case	Max Stress (MPa)
Original Configuration	41.42
Design Cycle 6	14.82
Difference	26.6
Percentage Difference	64%

From the design optimization of the triangle part in Table 5, there is 64% improvement in maximum stress acting on the mechanism, so the device will deform less and will last longer.

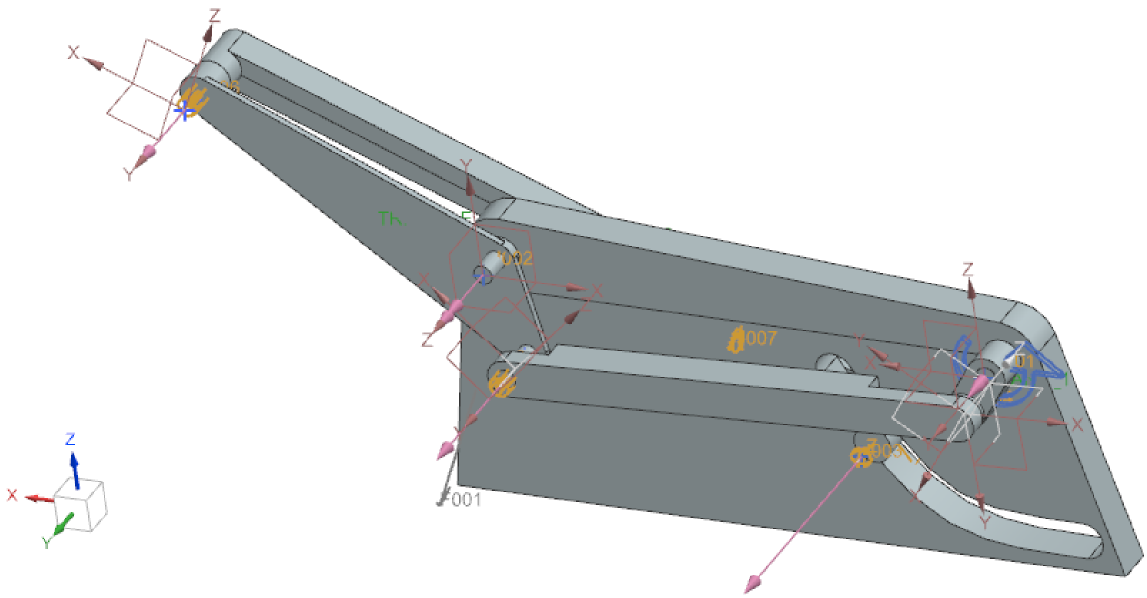


Image 18: New Mechanism Model

For the new FEM simulation with the new model shown in Image 18, we will perform the MBS simulation again to find the new power output for the mechanism. Power is a recurring theme in conversations about applications in physics and engineering. Power for rotational motion is equally as important as power for linear motion and can be calculated similarly when the force is constant. The power applied to a system rotating about a fixed axis is defined as the torque times the angular velocity, or $P = \tau\omega$. In order to convert these into an Excel spreadsheet, we first extract the graph objects representing the Angular Velocity and the Torque operating on Joint A. The power value required by the mechanism is then obtained by multiplying the two terms.

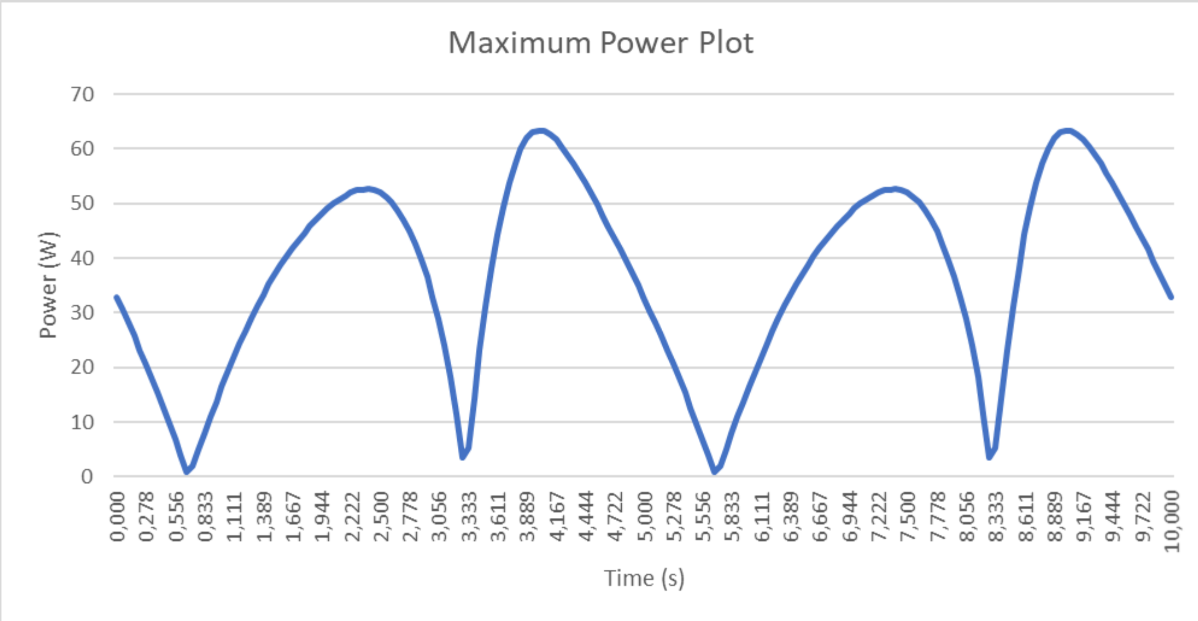


Image 19: Plot of needed power for the mechanism

From this graph in Image 19, the minimum power needed to run the mechanism is about 63.38W or **63 W**. So despite the mechanism change in design, the minimum power needed to turn the mechanism remains similar.

9. Simulation Validation

All our results are collected from Siemens NX FEM analysis. However, we used SolidWorks to produce secondary models which we used to verify our simulation results. The comparison of these results is presented in section 5. and the data is organized to table 4. We believe the extra effort has produced us a much more detailed and accurate representation of loads, stresses and strains the mechanism would experience in real life situations even though this took a lot of extra effort.

In the real life environment we would verify the simulations by producing accurate simulation loads to materials, individual mechanism members or simulating the entire system in operation; this would naturally be more expensive than just running a virtual simulation. We could for example attach a part to a siegmund table from one end. Then we could attach weights on the pin located on the other end of the rod and observe deformation of the mechanism.

A less expensive way to identify results would be to use solid mechanics calculation approaches for the deflection of the beam. We could for example calculate the deflection using beam deflection tables. Such an option would provide a fair compromise and balance between the abstract computer calculation and absolute physical model.

From the standpoint of computational modeling it is important to consider the difference between verification and validation. When we are looking at verification we are focusing on the software and mathematical models. Validation on the other hand aims to look at the accuracy of the model. All models are approximations and while such approximations can simulate reality accurately. Models cannot reach flawless accuracy capturing of the real world results. In FEM-modeling validation accuracy may be estimated using a validation pyramid as seen in picture 20. [3]

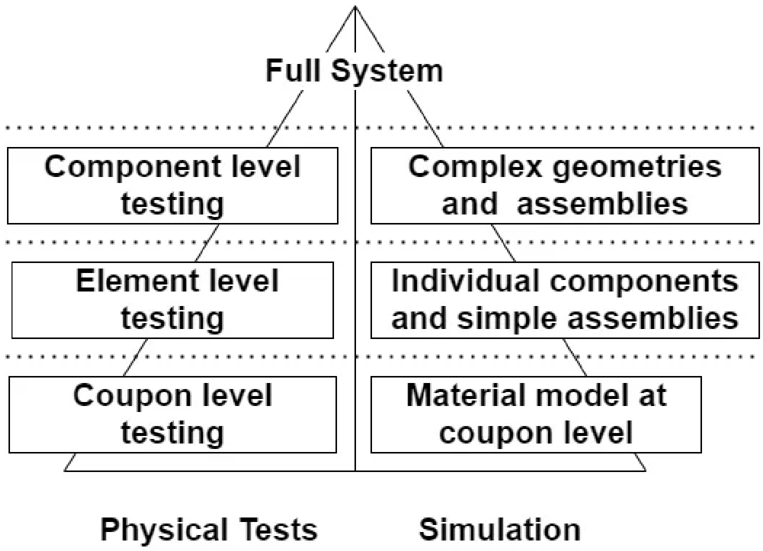


Image 20: FEM-model Validation pyramid [3]

Now, we may begin by looking at the real world behavior of our mechanism material and by validating the accuracy of the material behavior in the simulation. In the pyramid we will always proceed towards a more detailed dimension of the modeling and should real world results ever diverge from the behavior of our simulation we may pronounce the model inaccurate and estimate how much such inaccuracy may affect our desired results.

10. Discussion

This week we built FEM-models for all of the main parts and verified how the model beams deflect under loads and calculate the maximum displacement and stresses for all parts. Furthermore, we optimized the major part of the design to reduce the stress acting on the mechanism which will cause major damage to the device.

In this third stage of the project, we learnt how to use the Siemens NX software to perform FEM analysis in order to observe how much displacement and stress are acting on the parts and see if it is possible to optimize the device further to improve the longevity of the device. The necessary charts and tables showing the displacement and stress are also shown in the report. We also learnt about the relative and absolute coordinate systems and how important they are in obtaining the required results from FEM analysis.

In the first stage, we utilized Linkage to see how the mechanism would function and voted on the machine to work on, which is the knife mechanism. In the second stage, we performed MBS simulation using Siemens NX to ensure the mechanism works as expected and to plot the required displacement, force and torque plots which are required for further analysis.

This week, we learnt the basics of using the FEM simulation to observe how the mechanism functions under the forces. In order to facilitate the workload, we discussed among ourselves who would be the main input for each report, even though we would help each other on tasks which proved to be too difficult to be done.

The week began with computer sessions and tutorials uploaded to MyCourses that taught us how to utilize the software. Compared to the previous tasks, this report needs more time since the setup for correct FEM analysis was difficult and the software was unfamiliar to us. The coordinate system was also difficult to understand initially. However, we were able to successfully convert the simple diagram from the previous stages into the full model of the mechanism with complete FEM models and the potential optimisation. Just like in the first report, we will have a rotating role of main secretary for the report, who will be responsible for the report's layout. This week our report's main secretary was San Vo.

Because we finished all the descriptions, simulations, and prerequisite lists for this project and put up a significant deal of additional high-quality work, including learning how to use the program to draw and simulate results for the plots, we anticipate achieving a commendable grade of 5.

11. References and Sources

1. *SFS Online; SFS-EN 631-2:en - Materials and articles in contact with foodstuffs. Catering containers. Part 2: Dimensions of accessories and supports. (n.d.).* Retrieved October 9, 2022, from: https://sales.sfs.fi/en/index/tuotteet/SFS/CEN/ID2/6/10594.html.stx
2. *SFS Online; SFS-EN 1672-2:2020:en - Food processing machinery. Basic concepts. Part 2: Hygiene and cleanability requirements*. (n.d.). Retrieved October 9, 2022, from: https://sales.sfs.fi/en/index/tuotteet/SFS/CEN/ID2/1/957147.html.stx
3. Control automation. (n.d.). *Validation and Verification in Finite Element Analysis (FEA).* Retrieved October 9, 2022, from: https://control.com/technical-articles/validation-and-verification-in-finite-element-analysis-fea/

Appendix

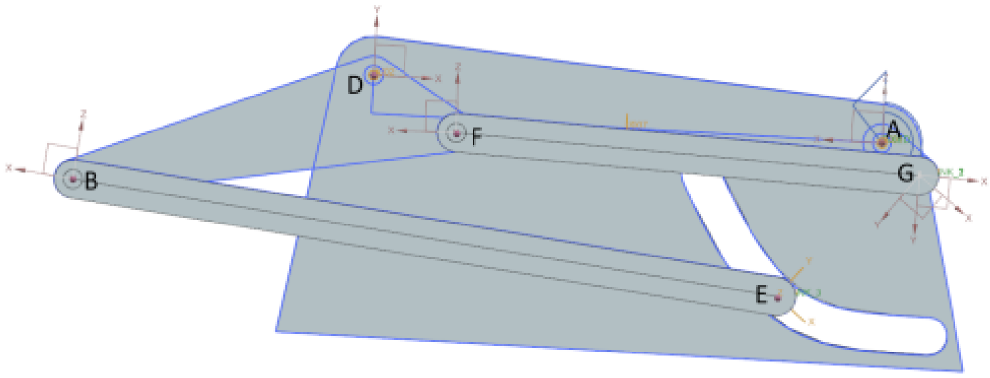
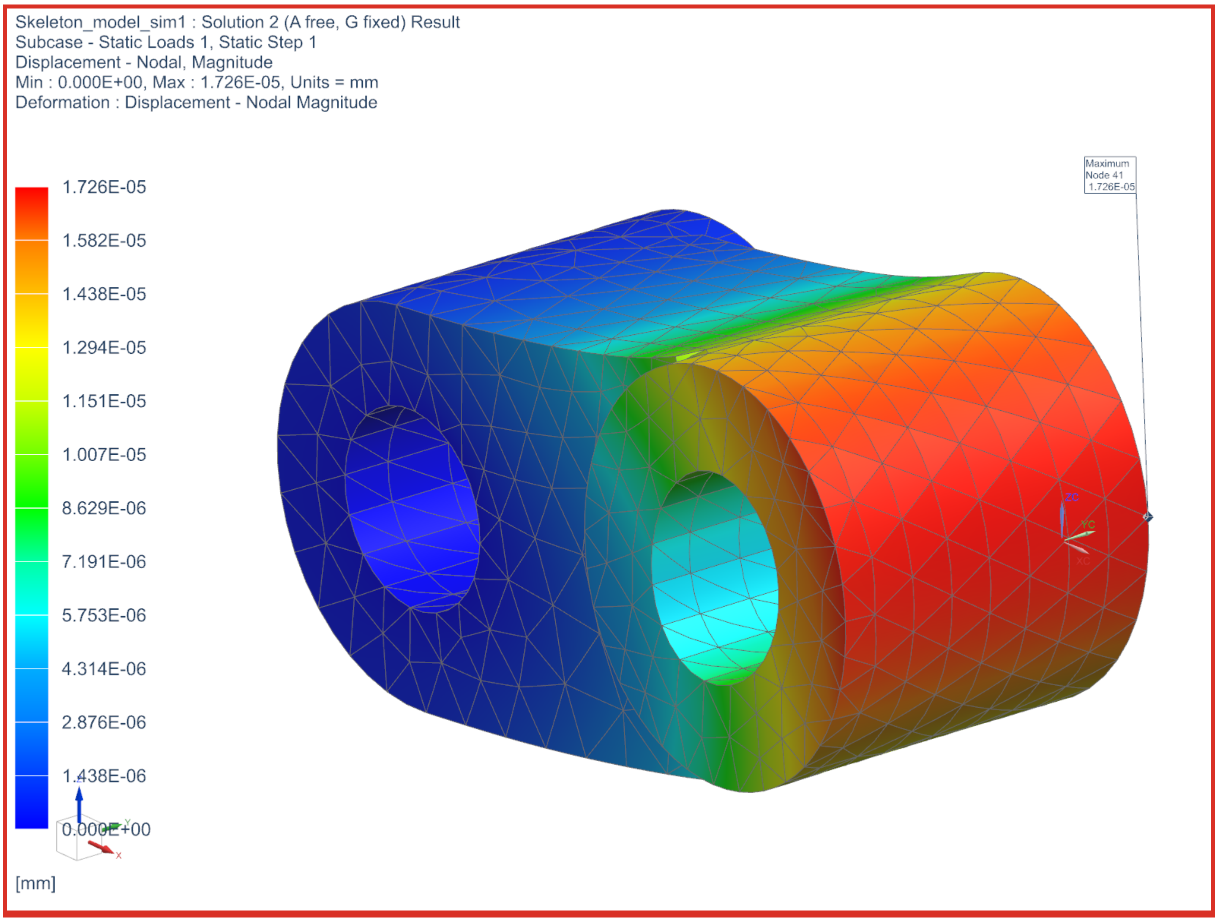


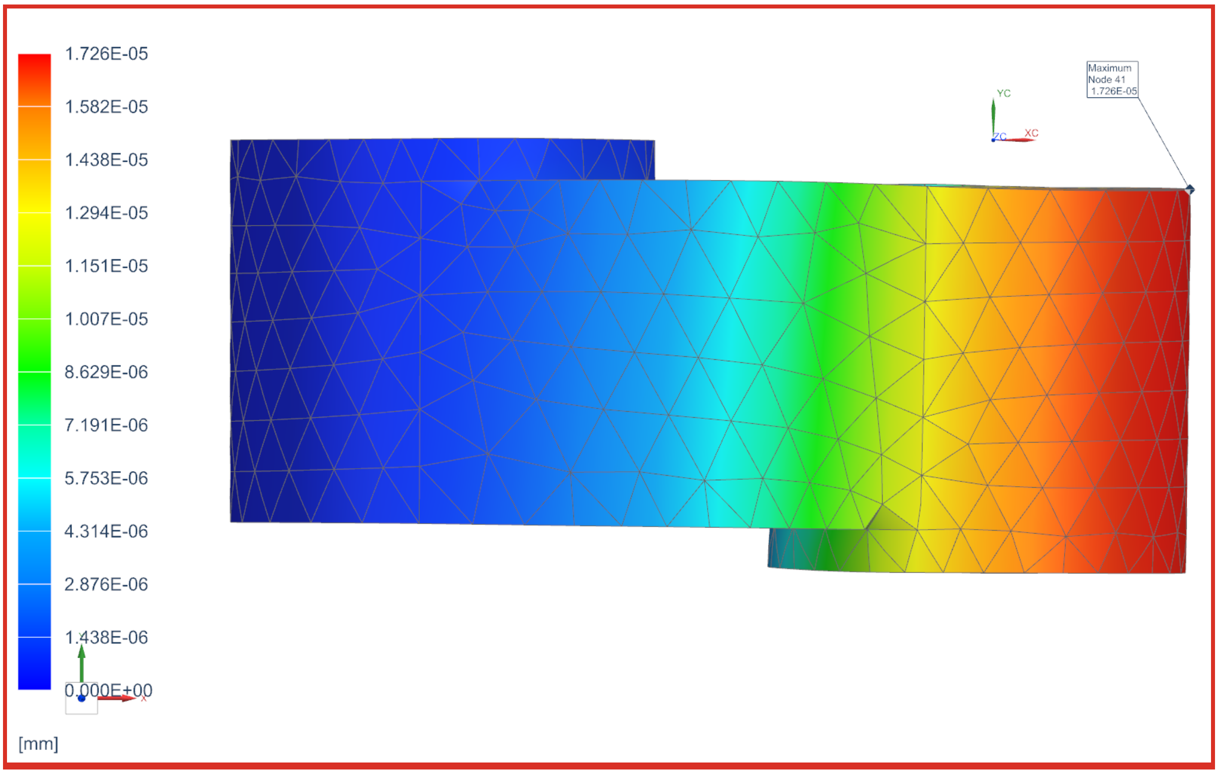
Image 6 The mechanism and its link positions.

First Part

G fixed, A free

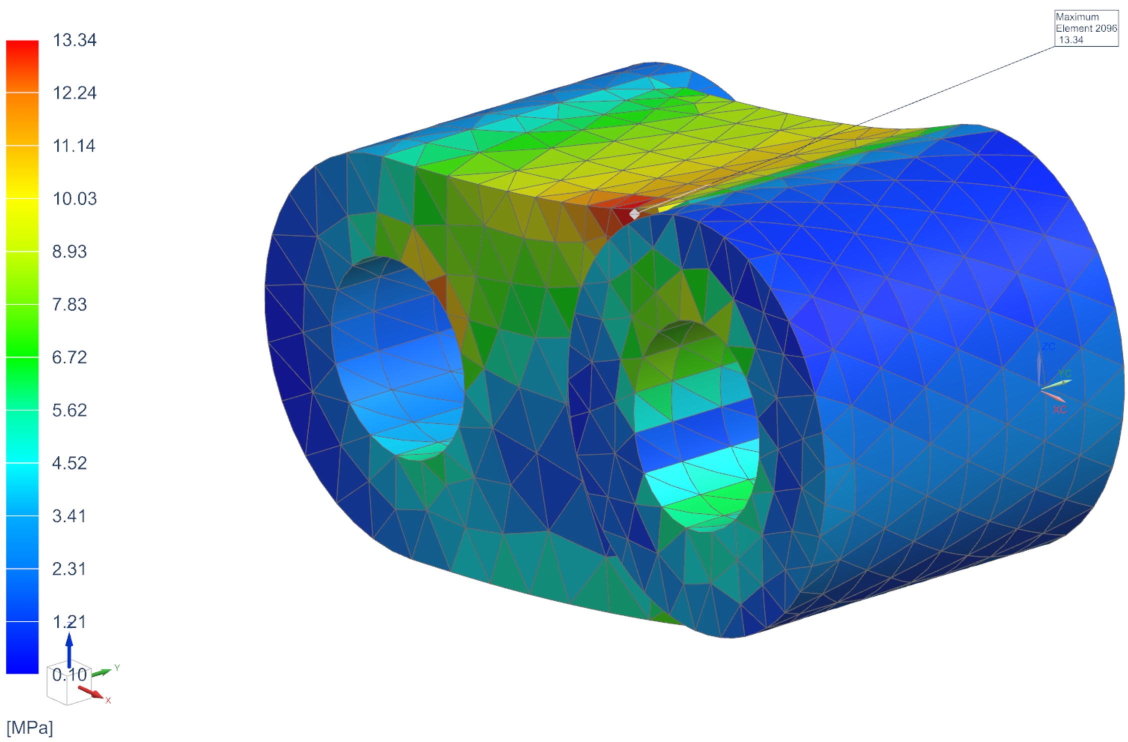
Displacement

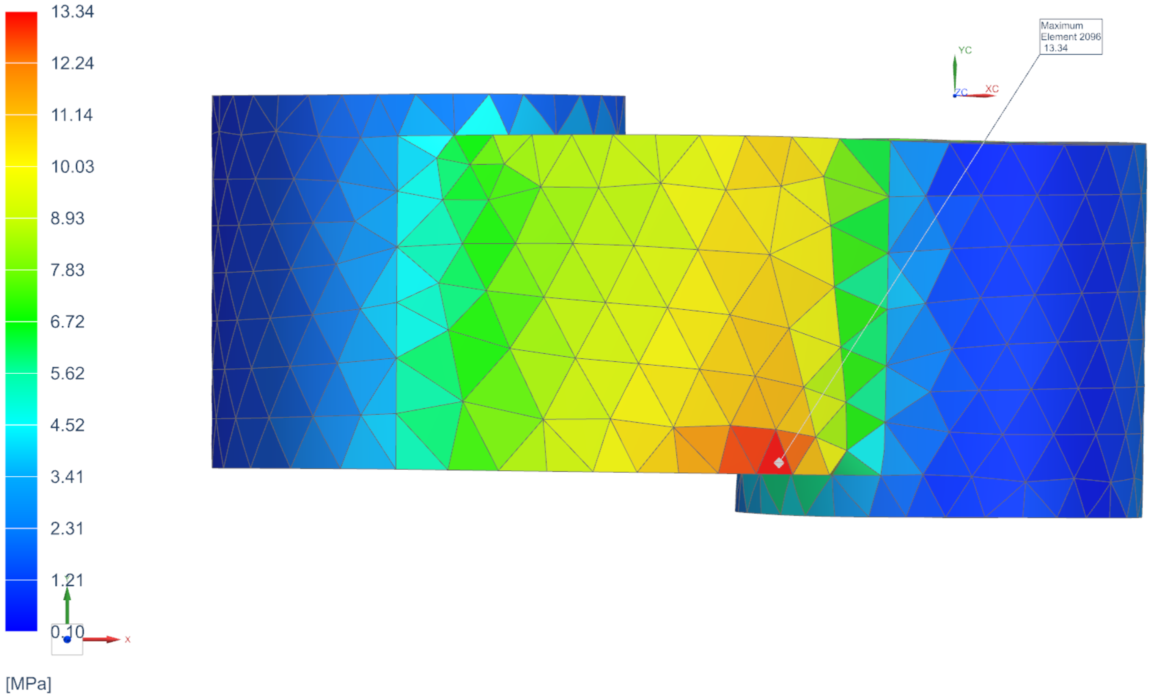




Stress

Skeleton_model_sim1 : Solution 2 (A free, G fixed) Result
Subcase - Static Loads 1, Static Step 1
Stress - Elemental, Von-Mises
Min : 0.10, Max : 13.34, Units = MPa
Deformation : Displacement - Nodal Magnitude

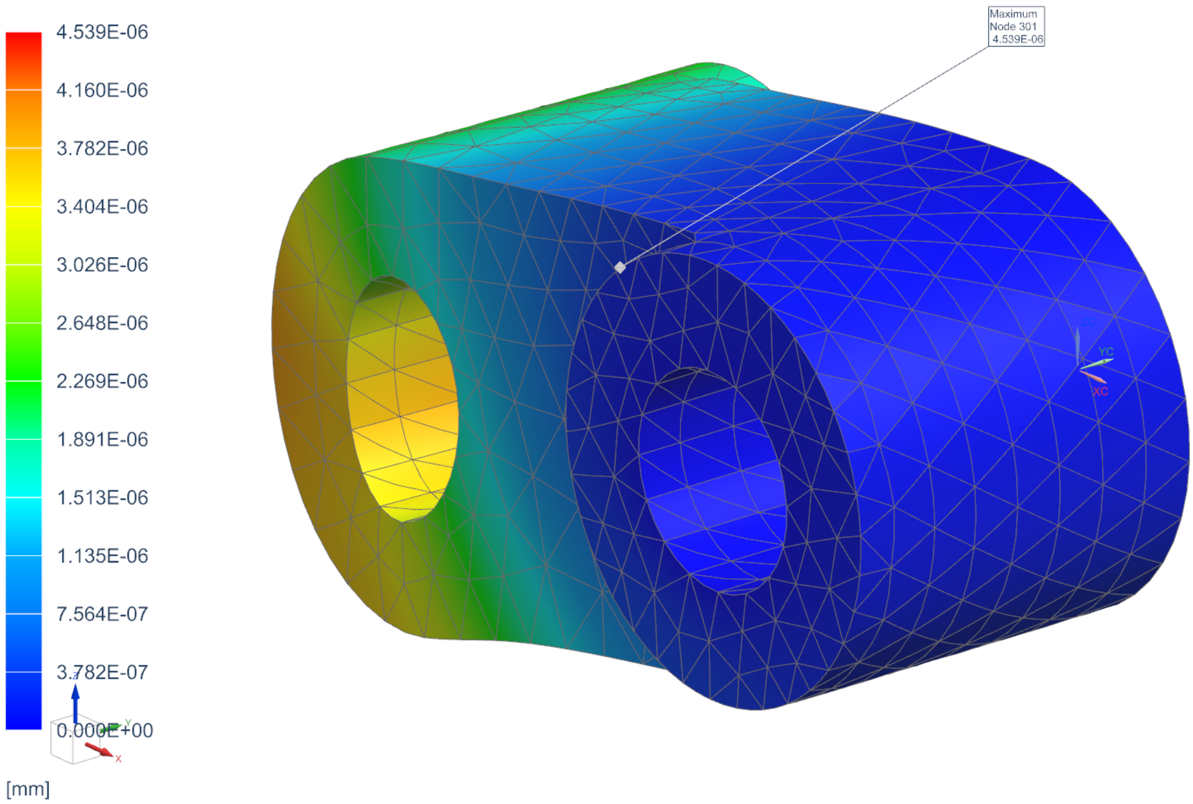


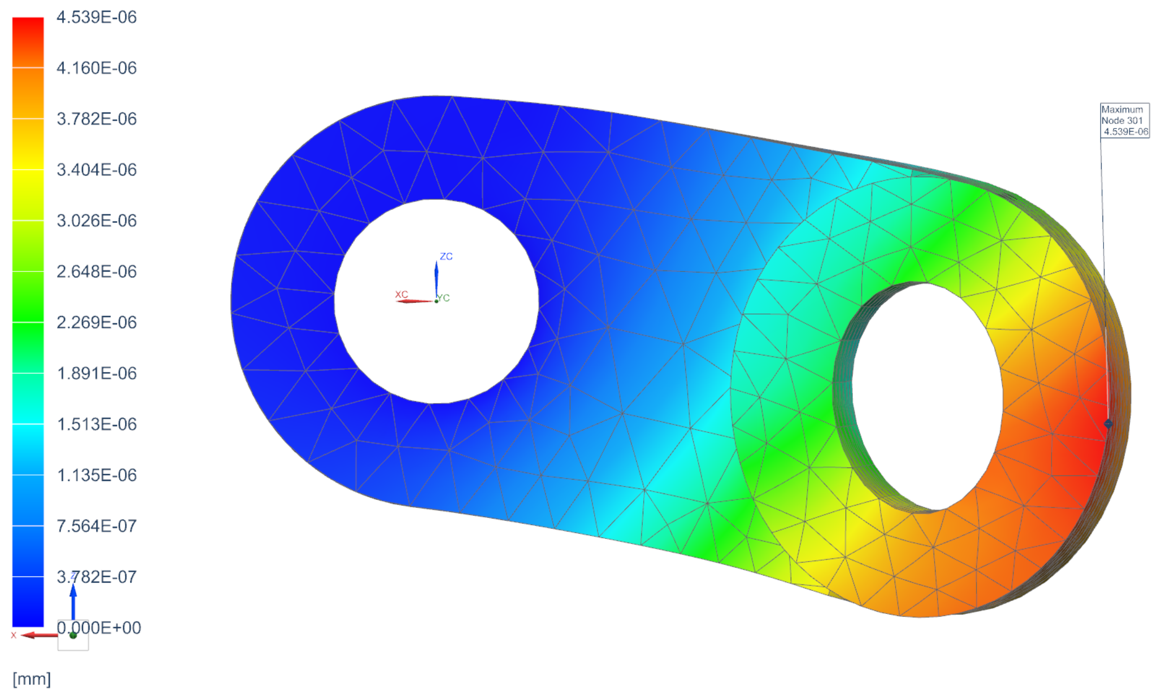


A fixed, G free

Displacement

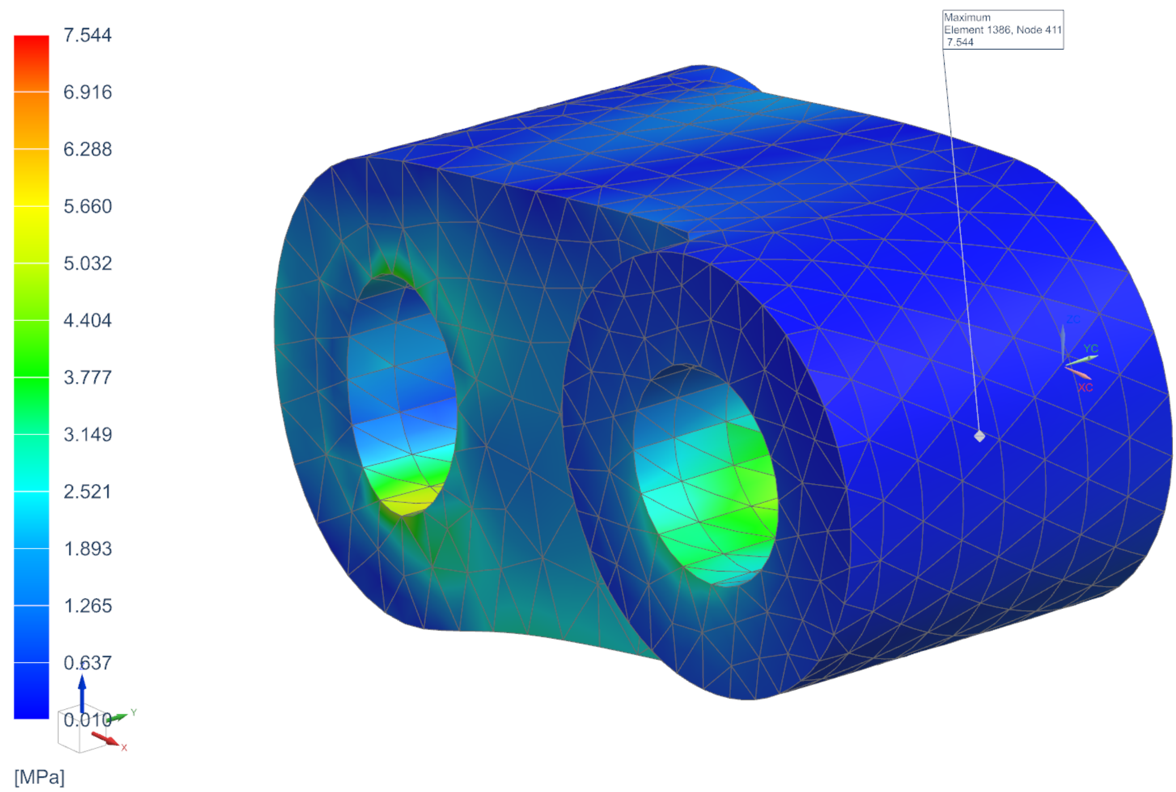
Skeleton_model_sim1 : Solution 1 (G free, A fixed) Result
Subcase - Static Loads 1, Static Step 1
Displacement - Nodal, Magnitude
Min : 0.000E+00, Max : 4.539E-06, Units = mm
Deformation : Displacement - Nodal Magnitude

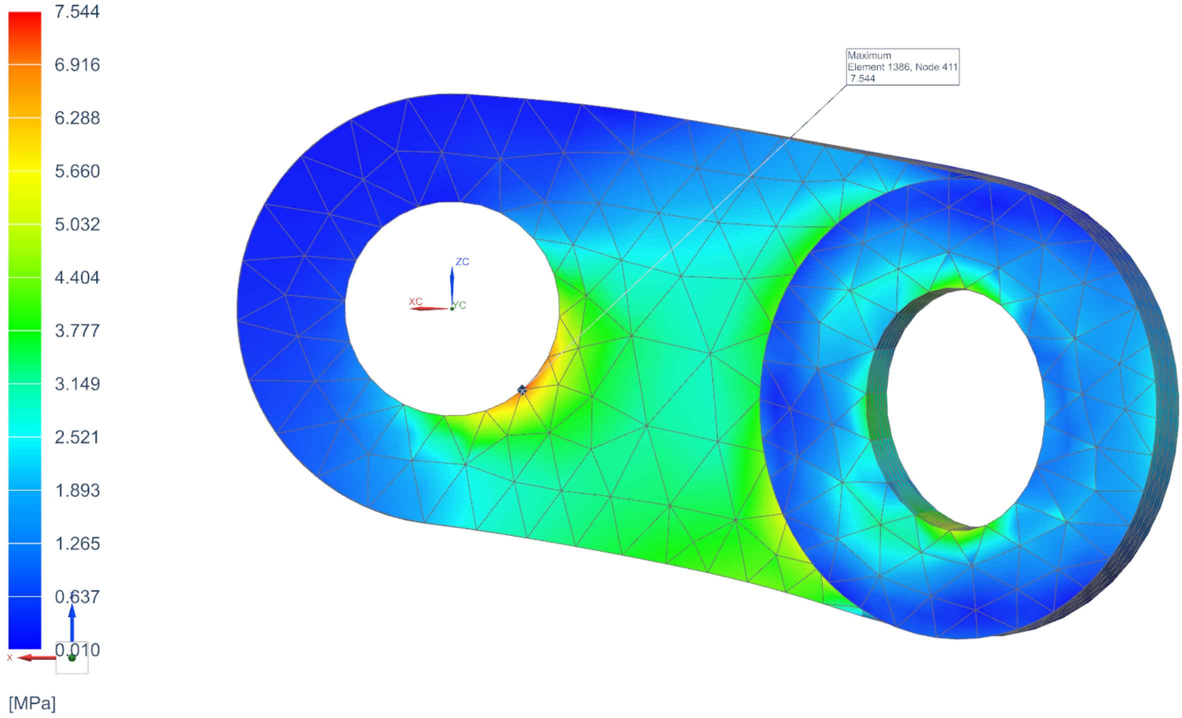




Stress

Skeleton_model_sim1 : Solution 1 (G free, A fixed) Result
Subcase - Static Loads 1, Static Step 1
Stress - Element-Nodal, Unaveraged, Von-Mises
Min : 0.010, Max : 7.544, Units = MPa
Deformation : Displacement - Nodal Magnitude



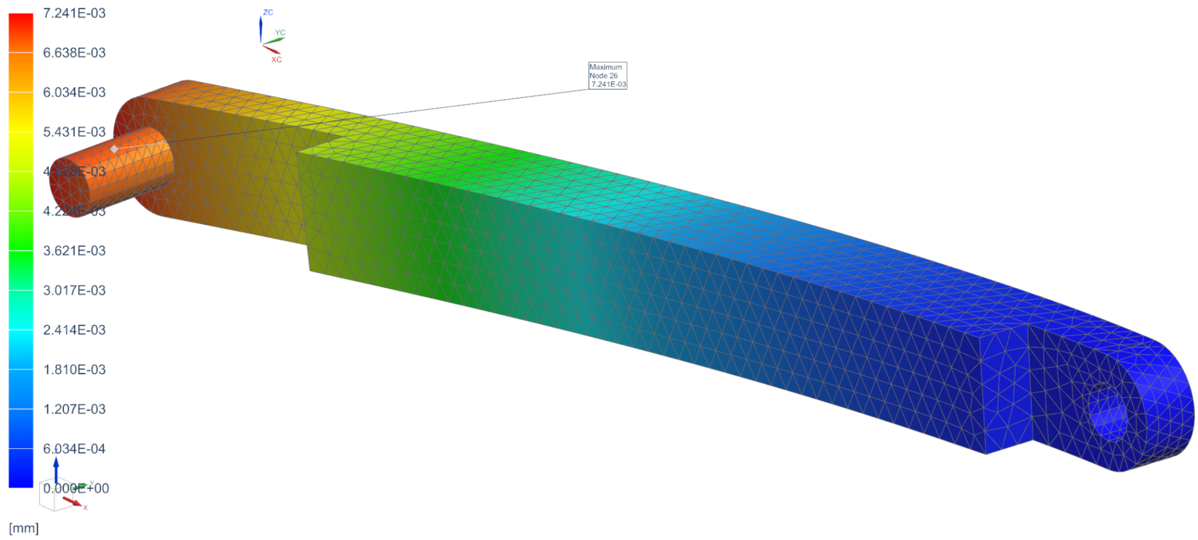

7.544
6.916
6.288
5.660
5.032
4.404
3.777
3.149
2.521
1.893
1.265
0.637
0.010
[MPa]
Maximum
Element 1386, Node 411
7.544
ZC
XC
YC

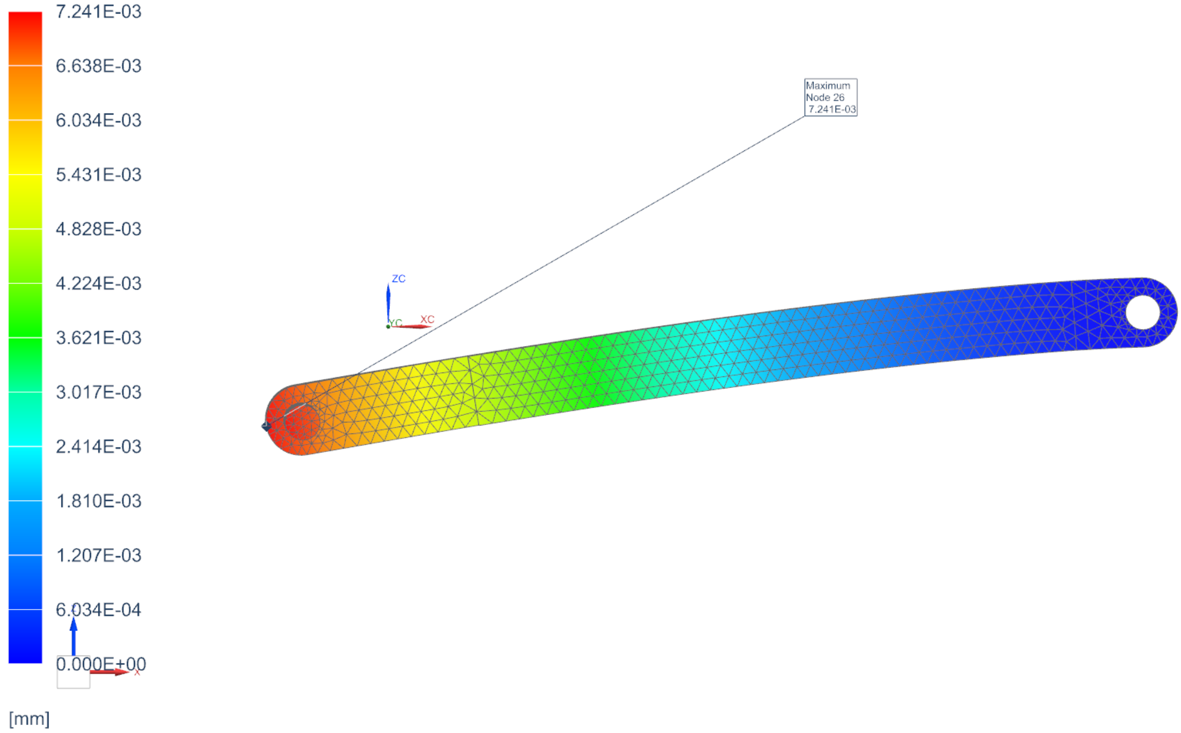
Second Part

F fixed, G free

Displacement

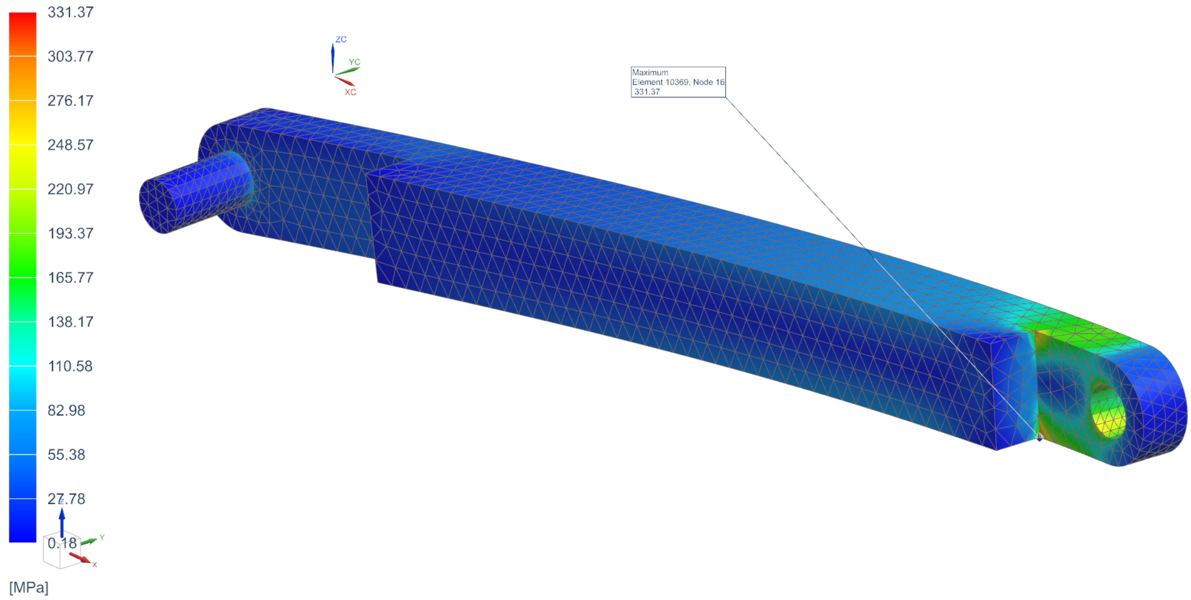
Skeleton_model_sim2 : Solution (F fixed, G free) Result
Subcase - Static Loads 1, Static Step 1
Displacement - Nodal, Magnitude
Min : 0.000E+00, Max : 7.241E-03, Units = mm
Deformation : Displacement - Nodal Magnitude

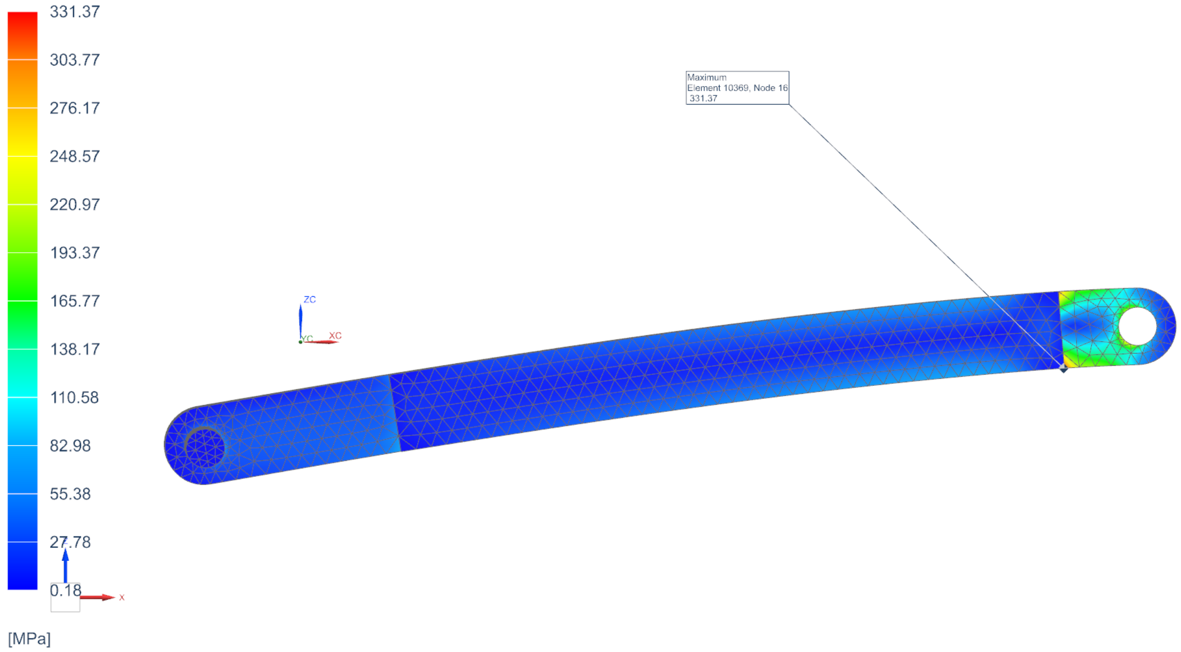




Stress

Skeleton_model_sim2 : Solution (F fixed, G free) Result
Subcase - Static Loads 1, Static Step 1
Stress - Element-Nodal, Unaveraged, Von-Mises
Min : 0.18, Max : 331.37, Units = MPa
Deformation : Displacement - Nodal Magnitude

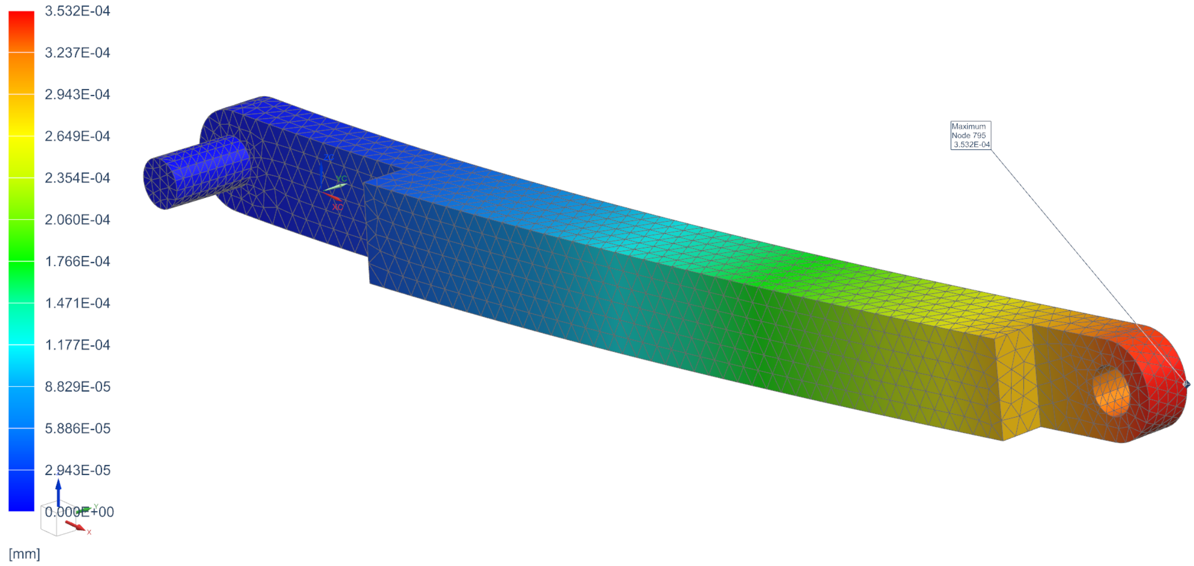


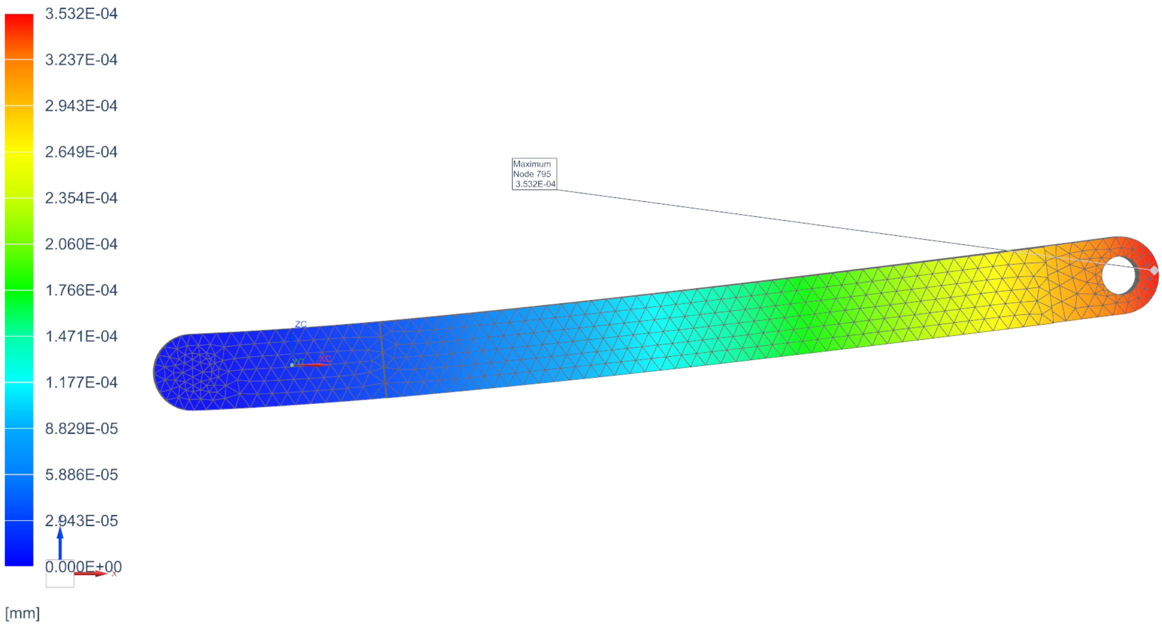


G fixed, F free

Displacement

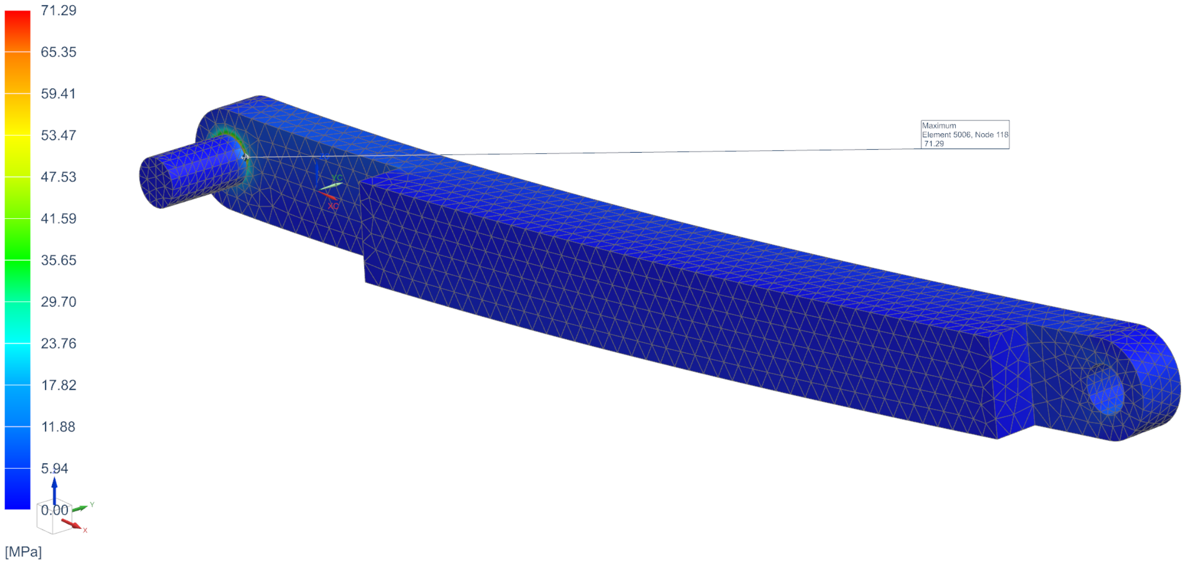
Skeleton_model_sim2 : Solution (G fixed, F free) Result
Subcase - Static Loads 1, Static Step 1
Displacement - Nodal, Magnitude
Min : 0.000E+00, Max : 3.532E-04, Units = mm
Deformation : Displacement - Nodal Magnitude

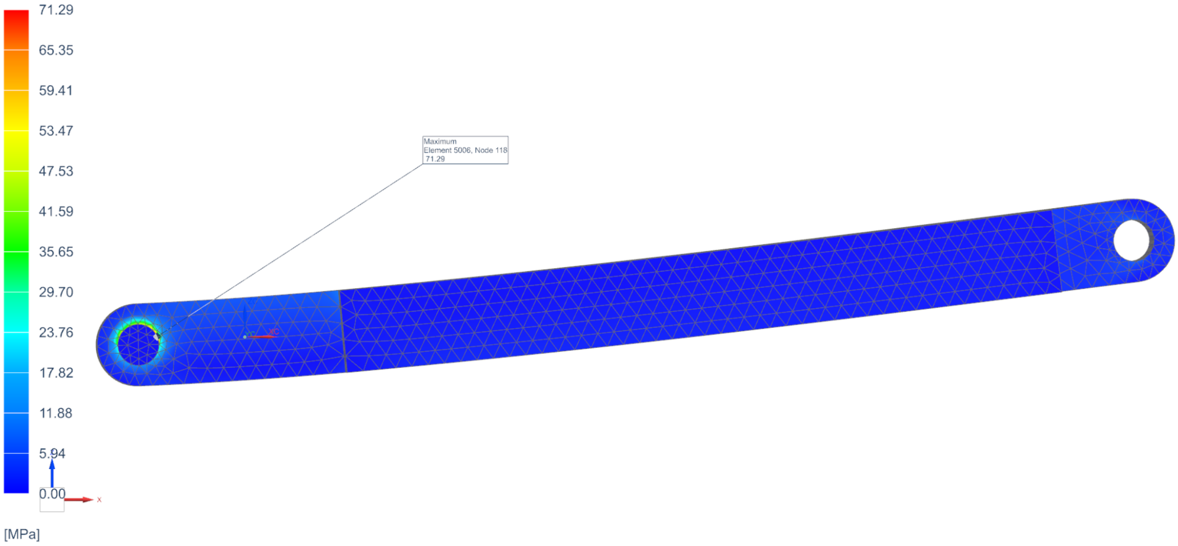




Stress

Skeleton_model_sim2 : Solution (G fixed, F free) Result
Subcase - Static Loads 1, Static Step 1
Stress - Element-Nodal, Unaveraged, Von-Mises
Min : 0.00, Max : 71.29, Units = MPa
Deformation : Displacement - Nodal Magnitude



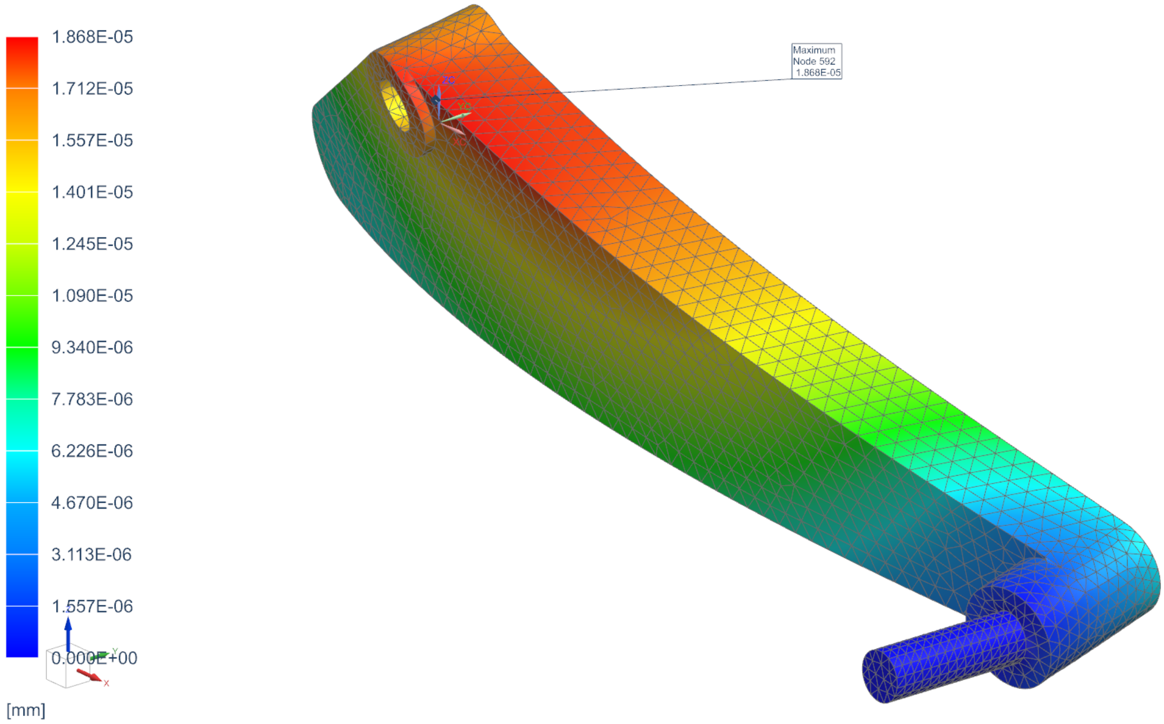


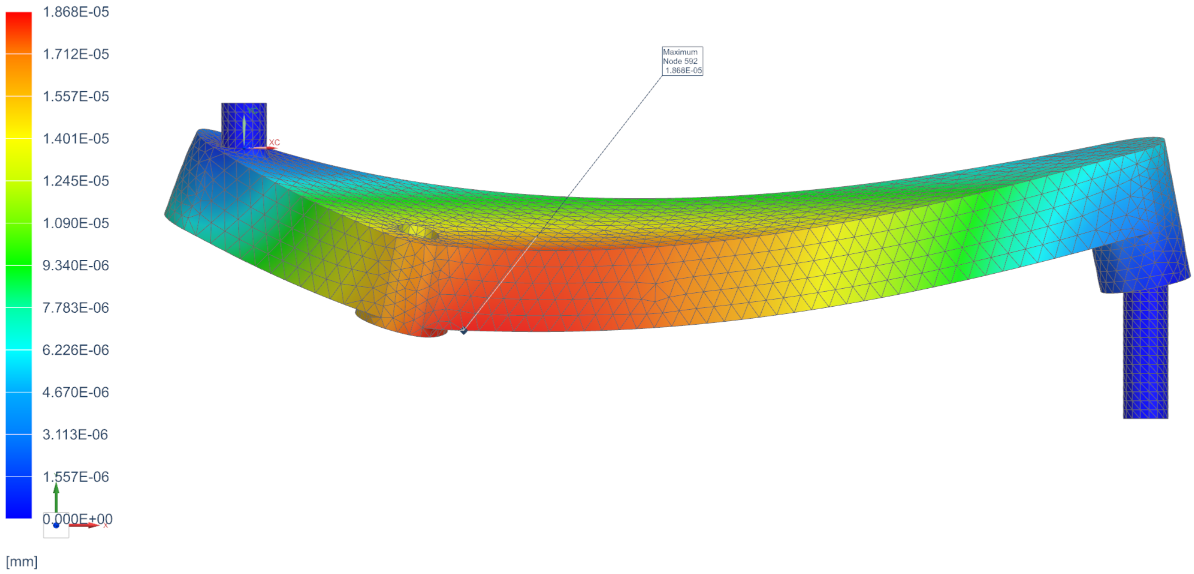
Third Part (Triangle Part)

D free

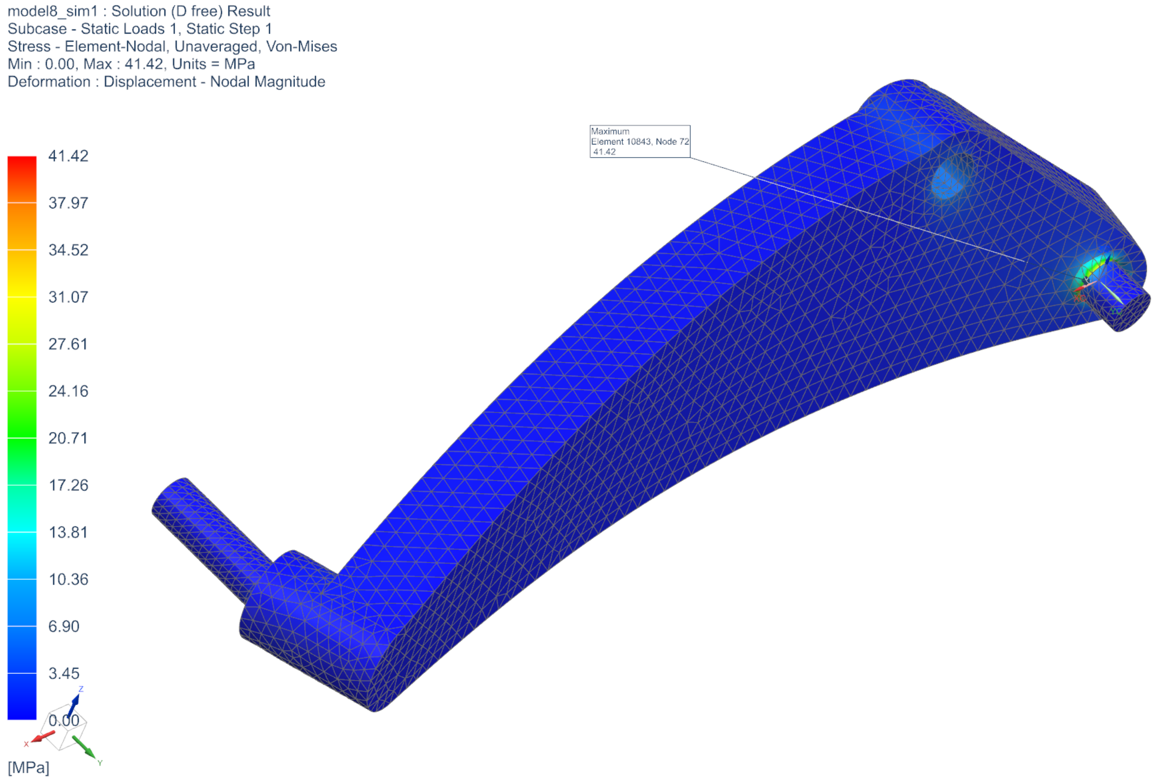
Displacement

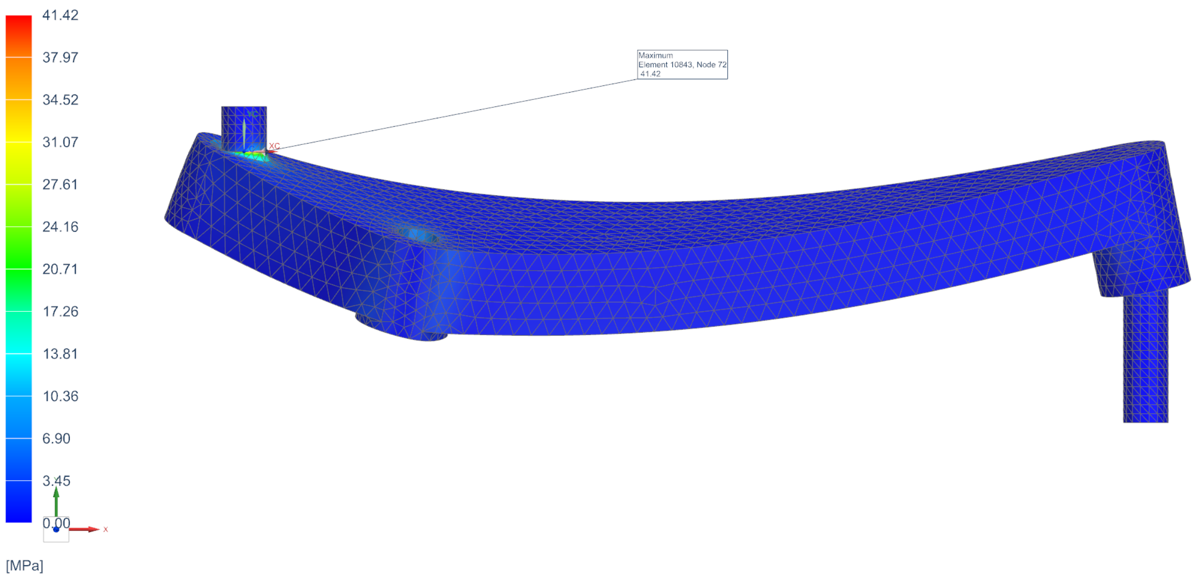
model8_sim1 : Solution (D free) Result
Subcase - Static Loads 1, Static Step 1
Displacement - Nodal, Magnitude
Min : 0.000E+00, Max : 1.868E-05, Units = mm
Deformation : Displacement - Nodal Magnitude





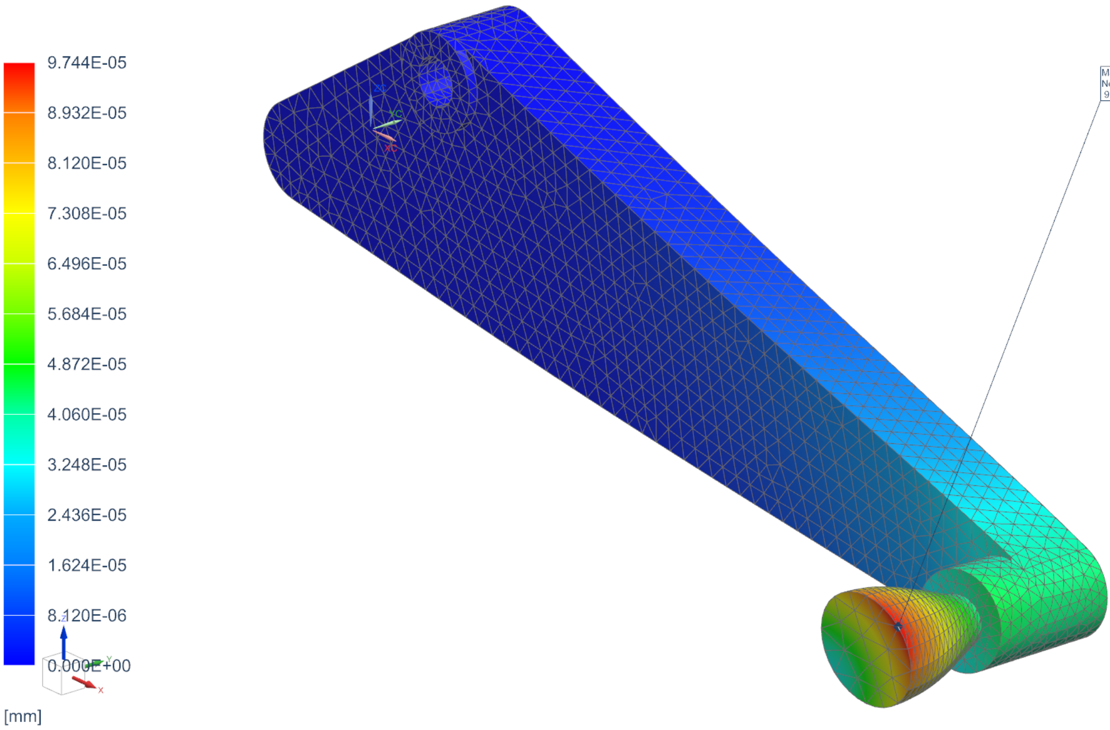
Stress

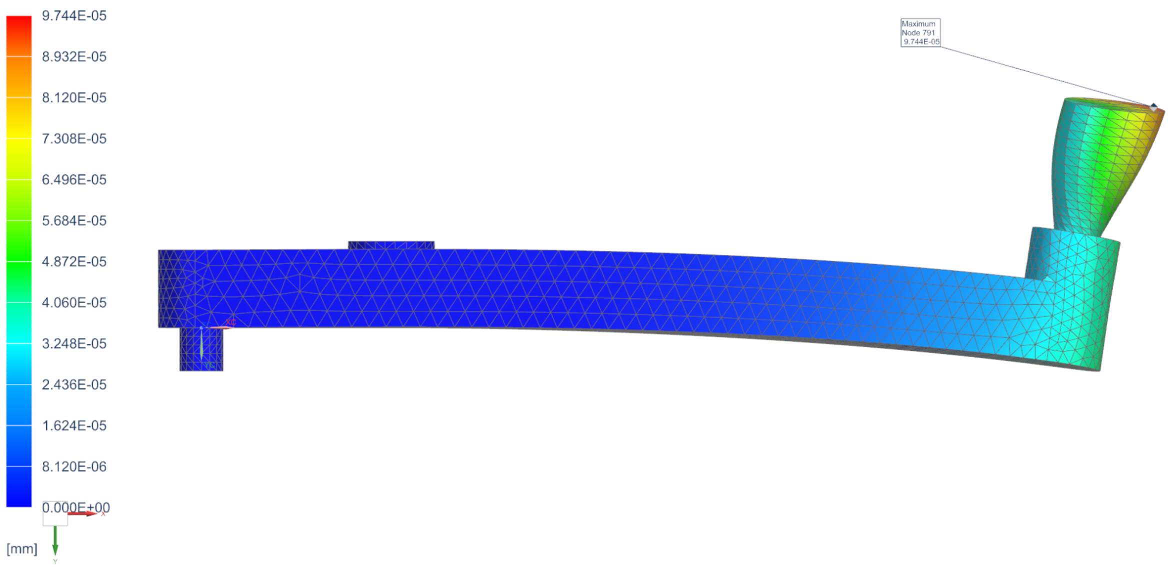

model8_sim1 : Solution (D free) Result
Subcase - Static Loads 1, Static Step 1
Stress - Element-Nodal, Unaveraged, Von-Mises
Min : 0.00, Max : 41.42, Units = MPa
Deformation : Displacement - Nodal Magnitude
41.42
37.97
34.52
31.07
27.61
24.16
20.71
17.26
13.81
10.36
6.90
3.45
0.00
[MPa]
Maximum
Element 10843, Node 72
41.42


41.42
37.97
34.52
31.07
27.61
24.16
20.71
17.26
13.81
10.36
6.90
3.45
0.00
[MPa]
Maximum
Element 10843, Node 72
41.42

B free
Displacement

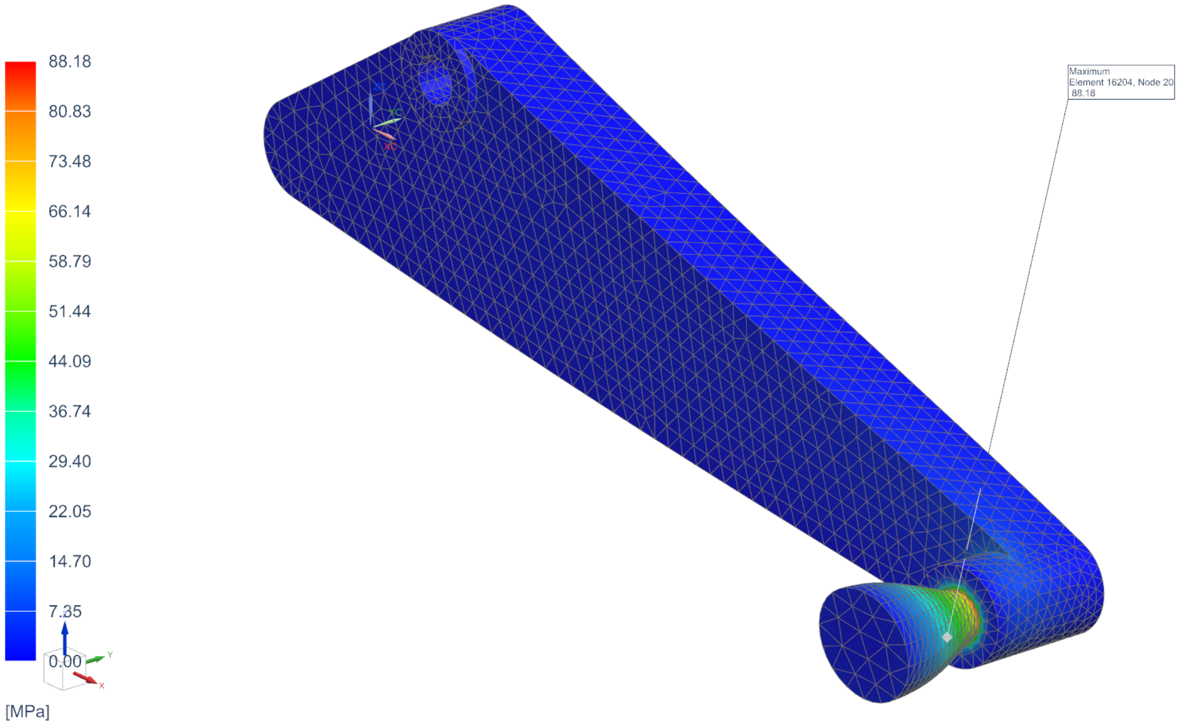
model8_sim1 : Solution (B free) Result
Subcase - Static Loads 1, Static Step 1
Displacement - Nodal, Magnitude
Min : 0.000E+00, Max : 9.744E-05, Units = mm
Deformation : Displacement - Nodal Magnitude

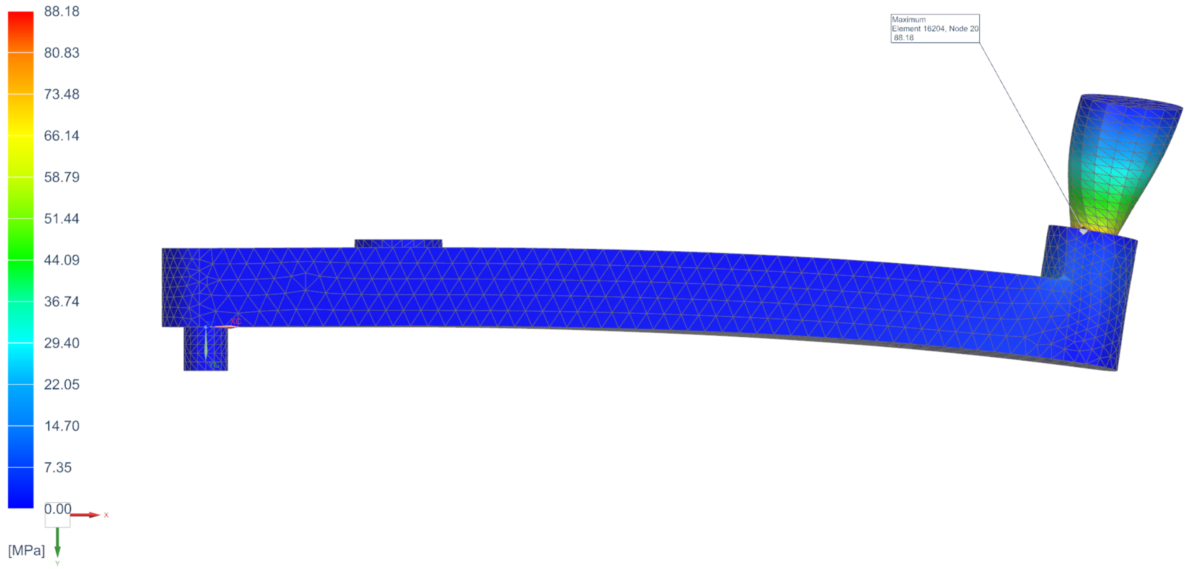




Stress

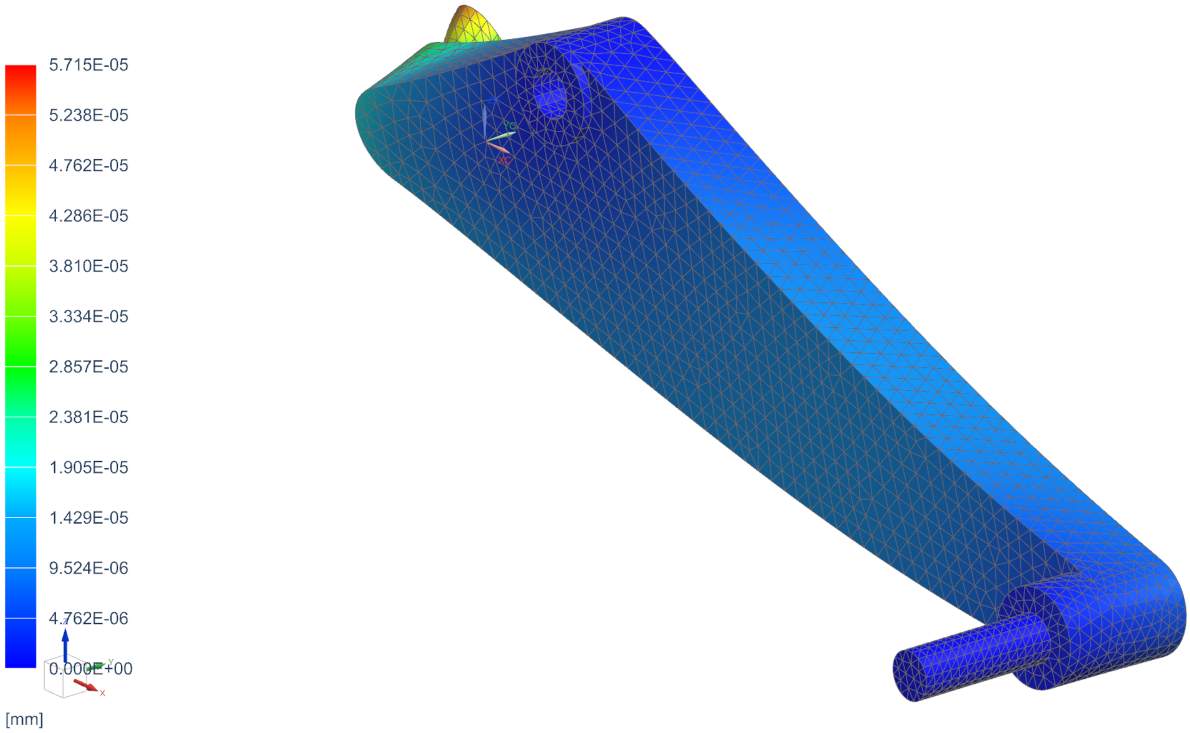
model8_sim1 : Solution (B free) Result
Subcase - Static Loads 1, Static Step 1
Stress - Element-Nodal, Unaveraged, Von-Mises
Min : 0.00, Max : 88.18, Units = MPa
Deformation : Displacement - Nodal Magnitude

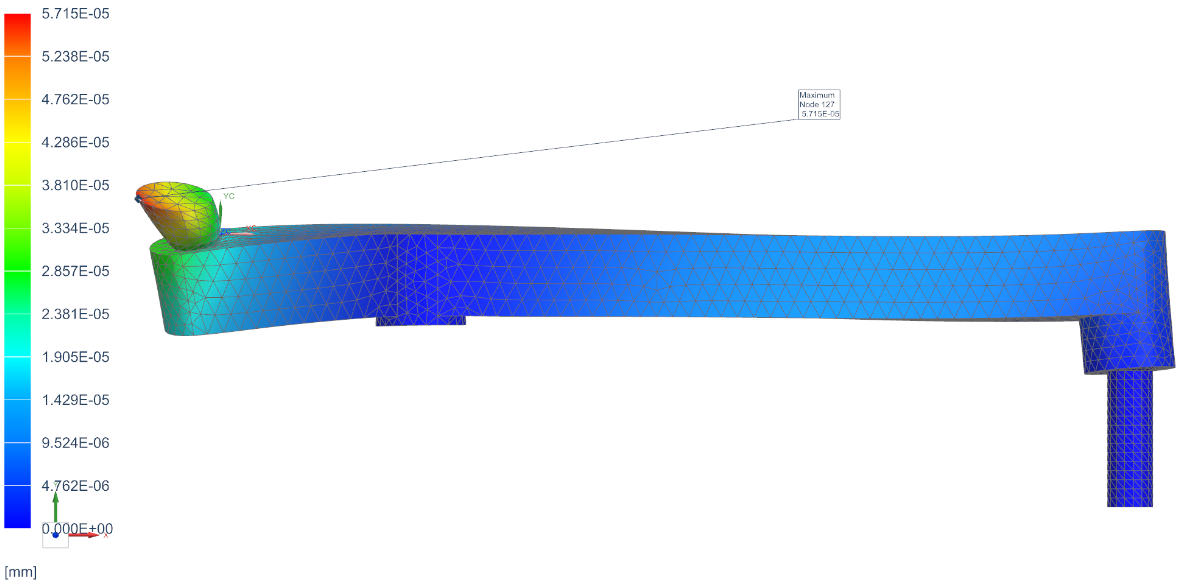




F free
Displacement

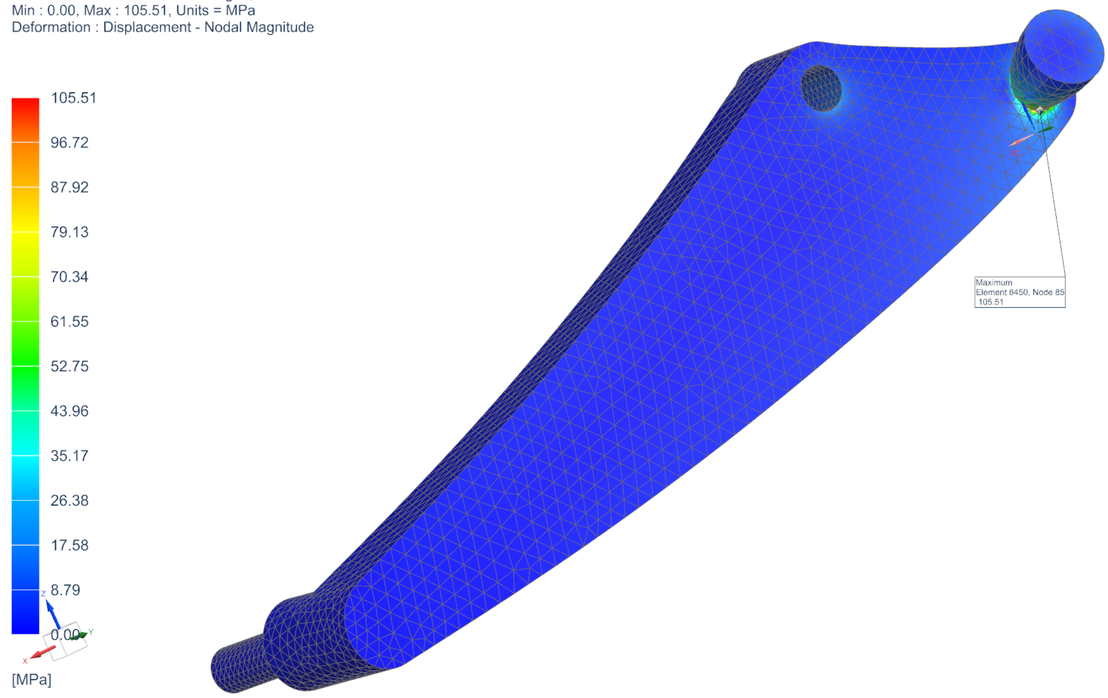
model8_sim1 : Solution (F free) Result
Subcase - Static Loads 1, Static Step 1
Displacement - Nodal, Magnitude
Min : 0.000E+00, Max : 5.715E-05, Units = mm
Deformation : Displacement - Nodal Magnitude

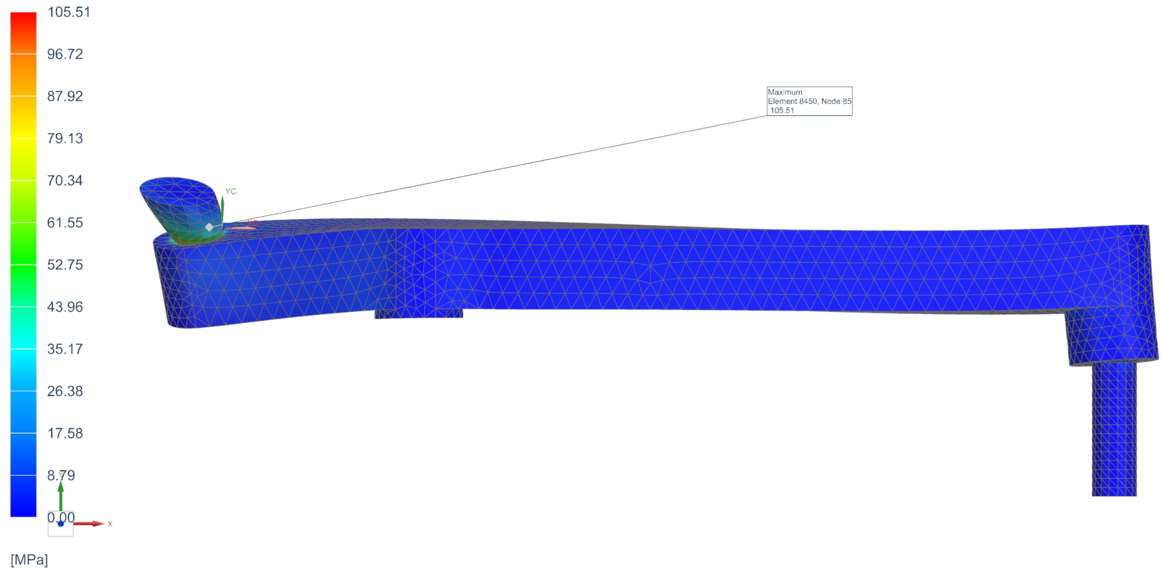




Stress

model8_sim1 : Solution (F free) Result
Subcase - Static Loads 1, Static Step 1
Stress - Element-Nodal, Unaveraged, Von-Mises
Min : 0.00, Max : 105.51, Units = MPa
Deformation : Displacement - Nodal Magnitude



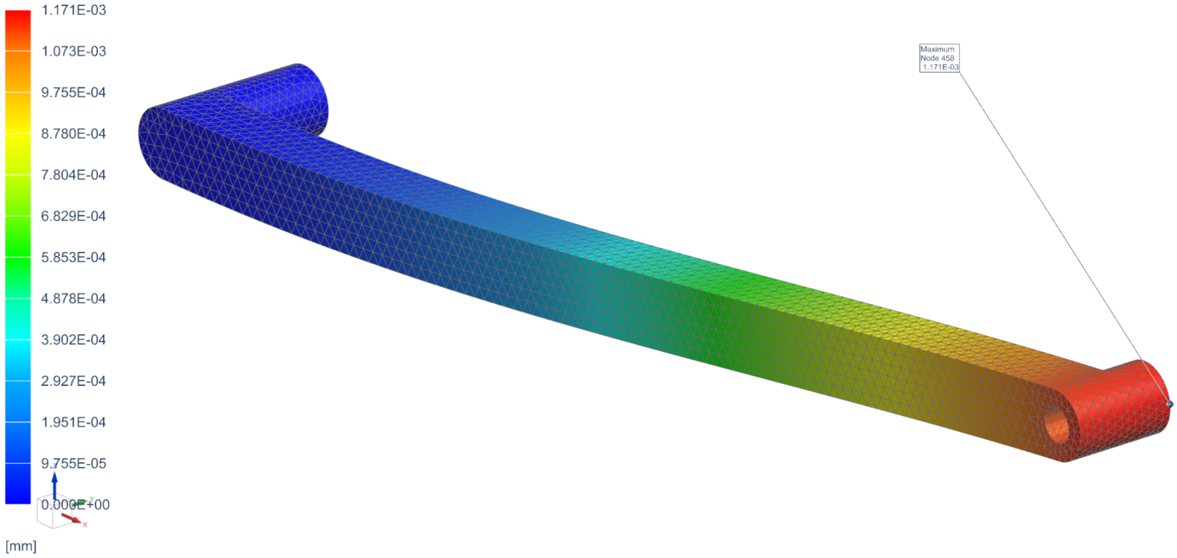


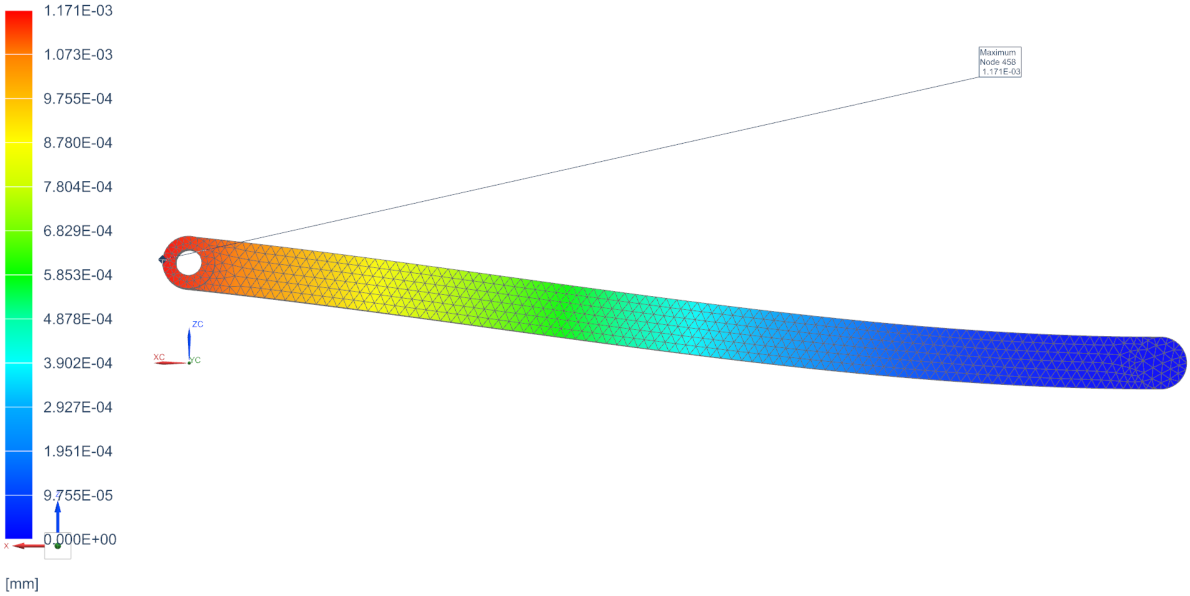
Final Part

B free

Displacement

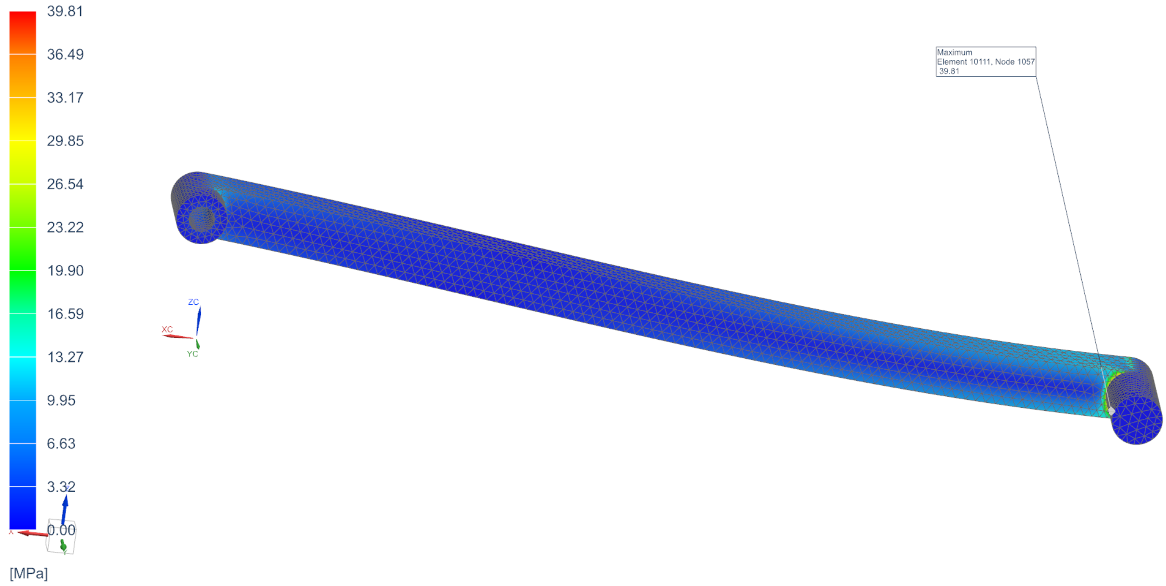
model9_sim1 : Solution (B free) Result
Subcase - Static Loads 1, Static Step 1
Displacement - Nodal, Magnitude
Min : 0.000E+00, Max : 1.171E-03, Units = mm
Deformation : Displacement - Nodal Magnitude

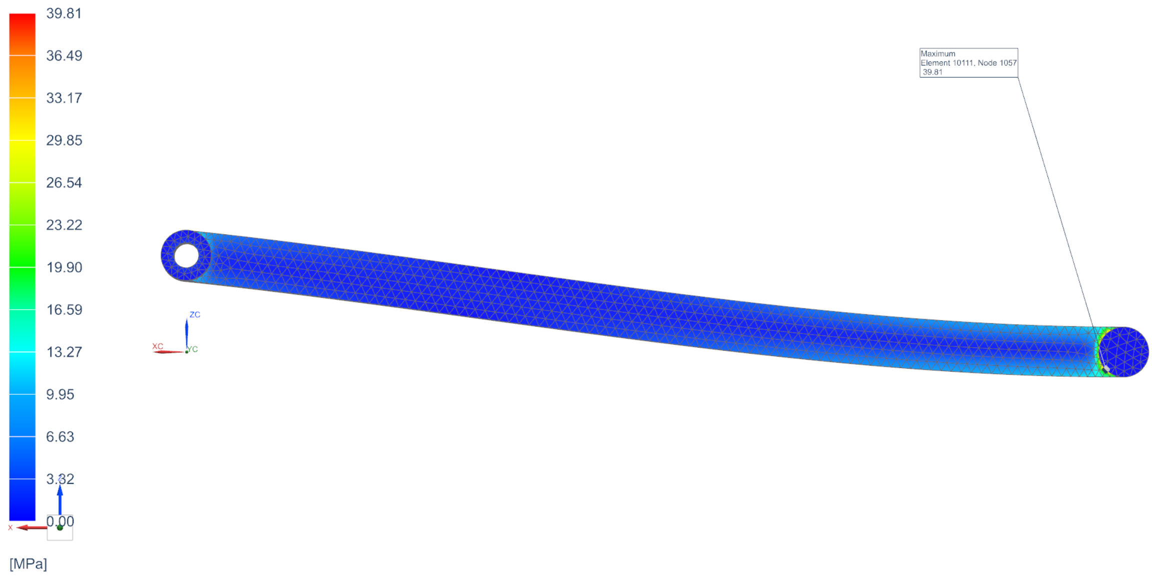




Stress

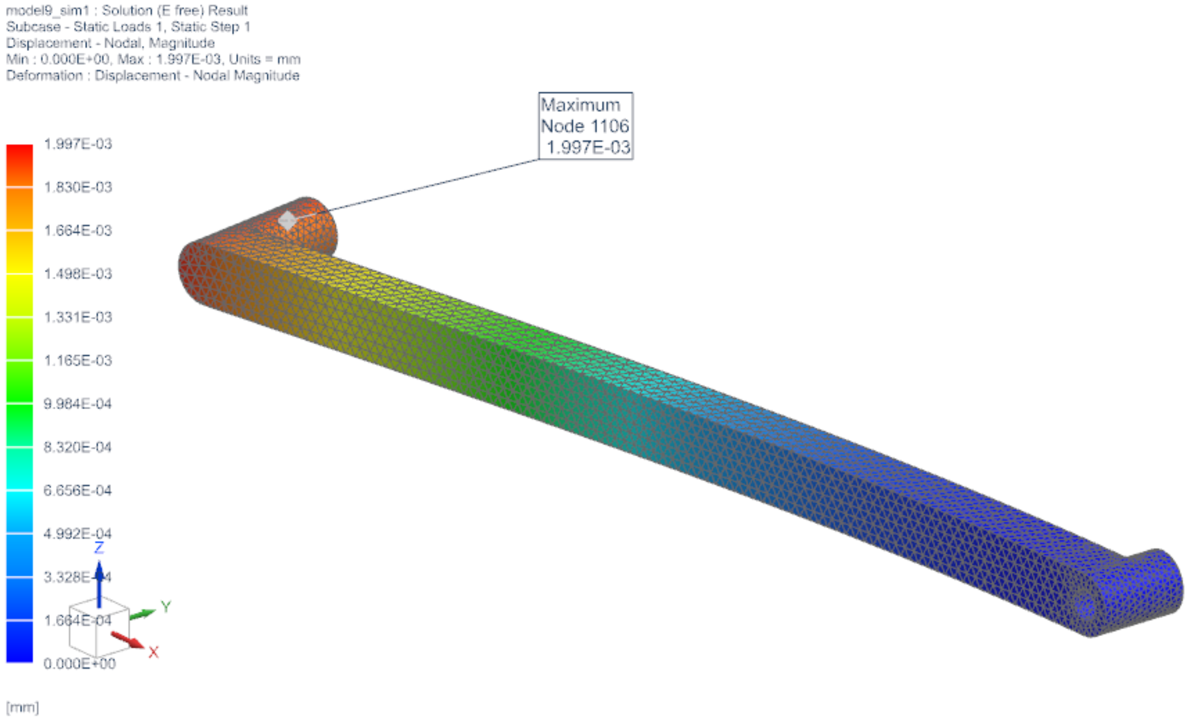
model9_sim1 : Solution (B free) Result
Subcase - Static Loads 1, Static Step 1
Stress - Element-Nodal, Unaveraged, Von-Mises
Min : 0.00, Max : 39.81, Units = MPa
Deformation : Displacement - Nodal Magnitude

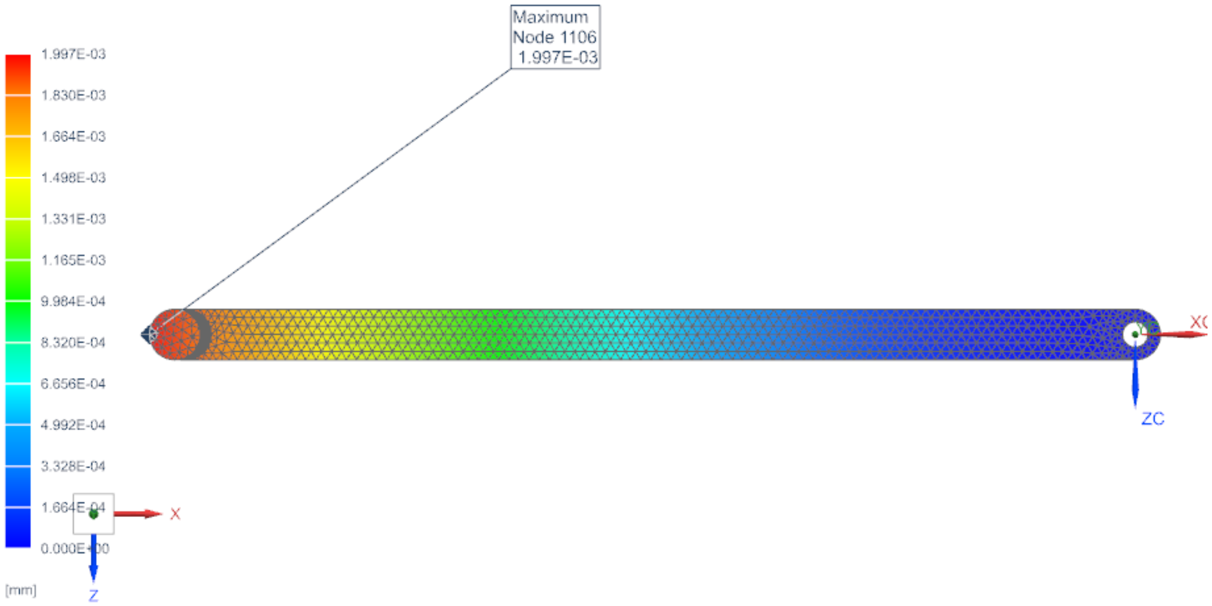




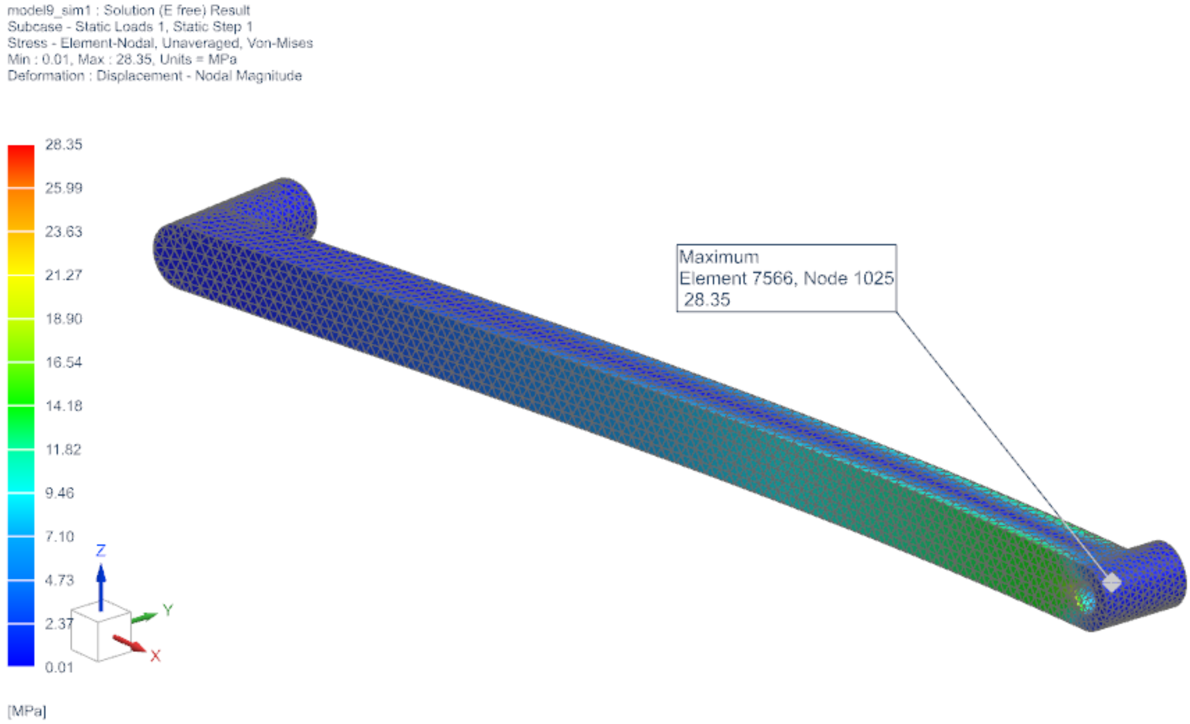
E free

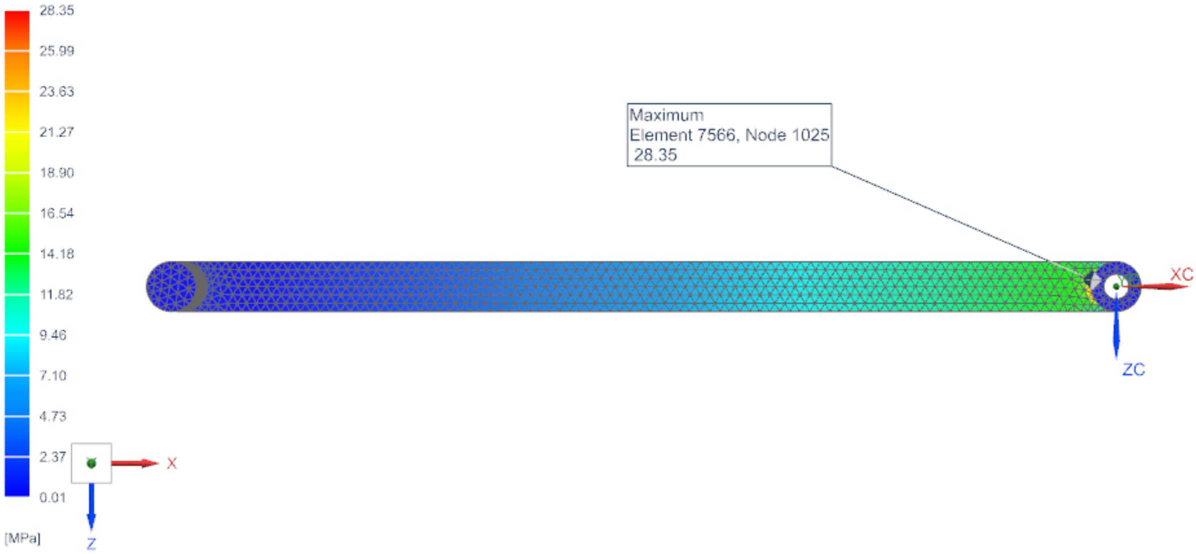
Displacement





Stress

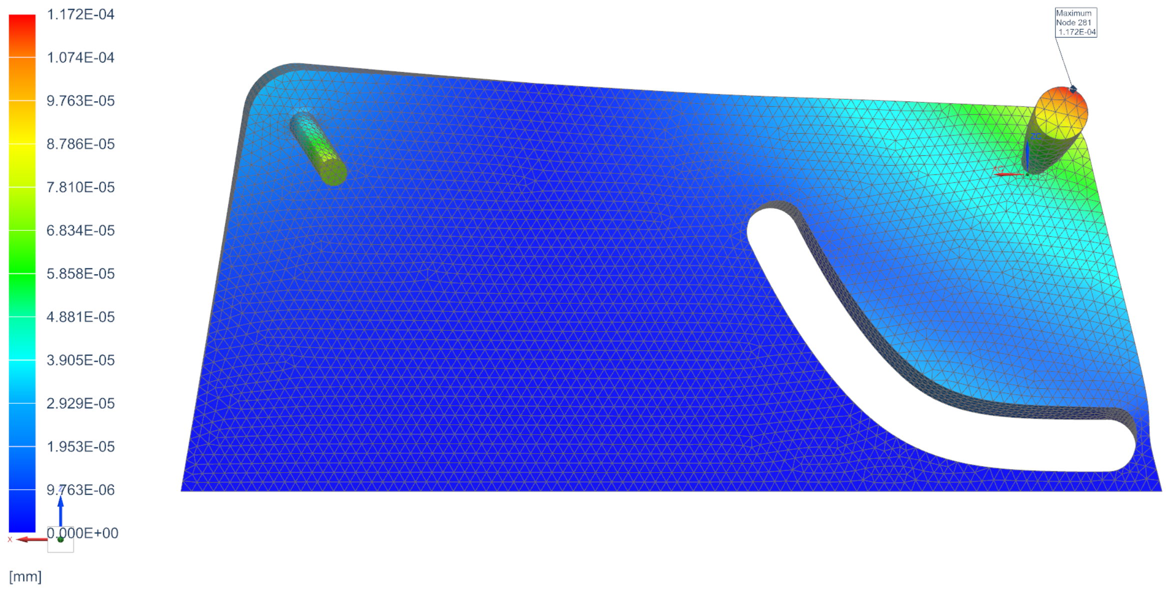

model9_sim1 : Solution (E free) Result
Subcase - Static Loads 1, Static Step 1
Stress - Element-Nodal, Unaveraged, Von-Mises
Min : 0.01, Max : 28.35, Units = MPa
Deformation : Displacement - Nodal Magnitude
28.35
25.99
23.63
21.27
18.90
16.54
14.18
11.82
9.46
7.10
4.73
2.37
0.01
[MPa]
Maximum
Element 7566, Node 1025
28.35
Z
Y
X

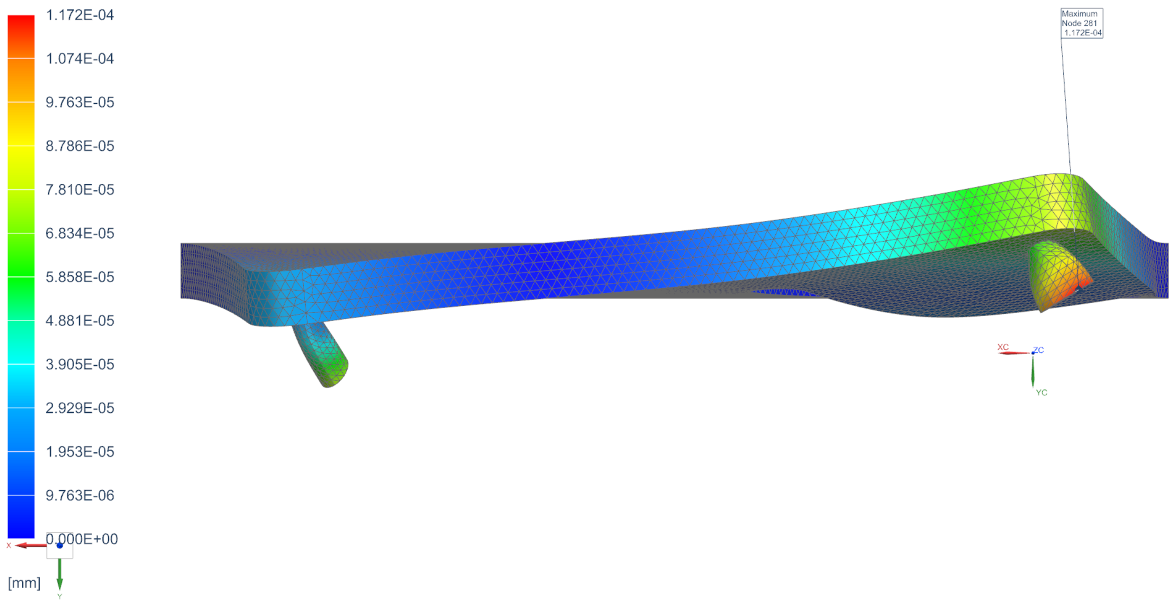

28.35
25.99
23.63
21.27
18.90
16.54
14.18
11.82
9.46
7.10
4.73
2.37
0.01
[MPa]
Maximum
Element 7566, Node 1025
28.35
XC
ZC
X
Z

Body Part

Displacement

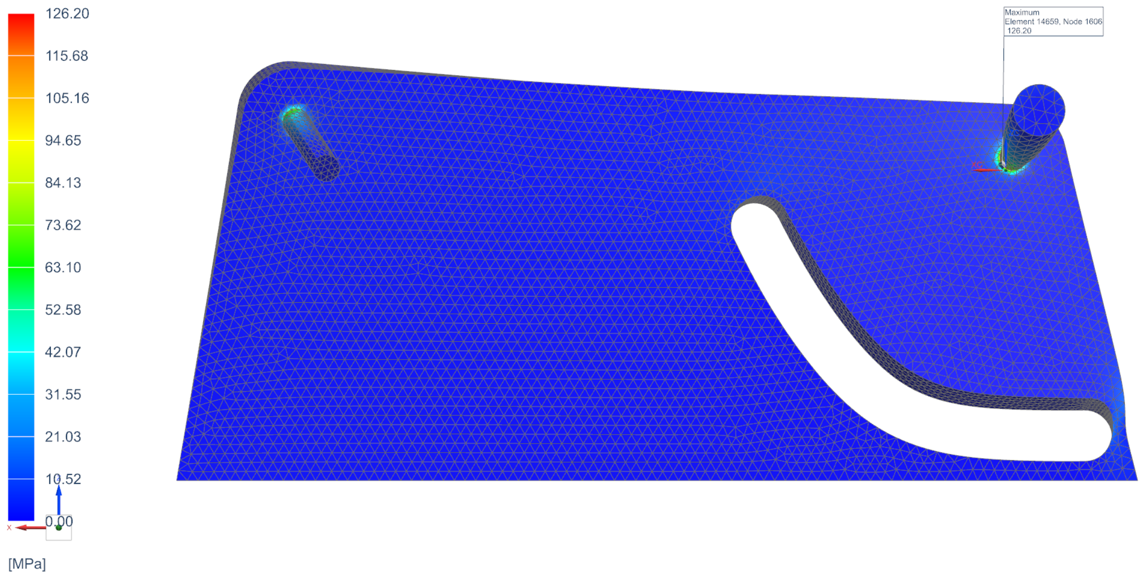
model4_sim2 : Solution (D free) Result
Subcase - Static Loads 1, Static Step 1
Displacement - Nodal, Magnitude
Min : 0.000E+00, Max : 1.172E-04, Units = mm
Deformation : Displacement - Nodal Magnitude

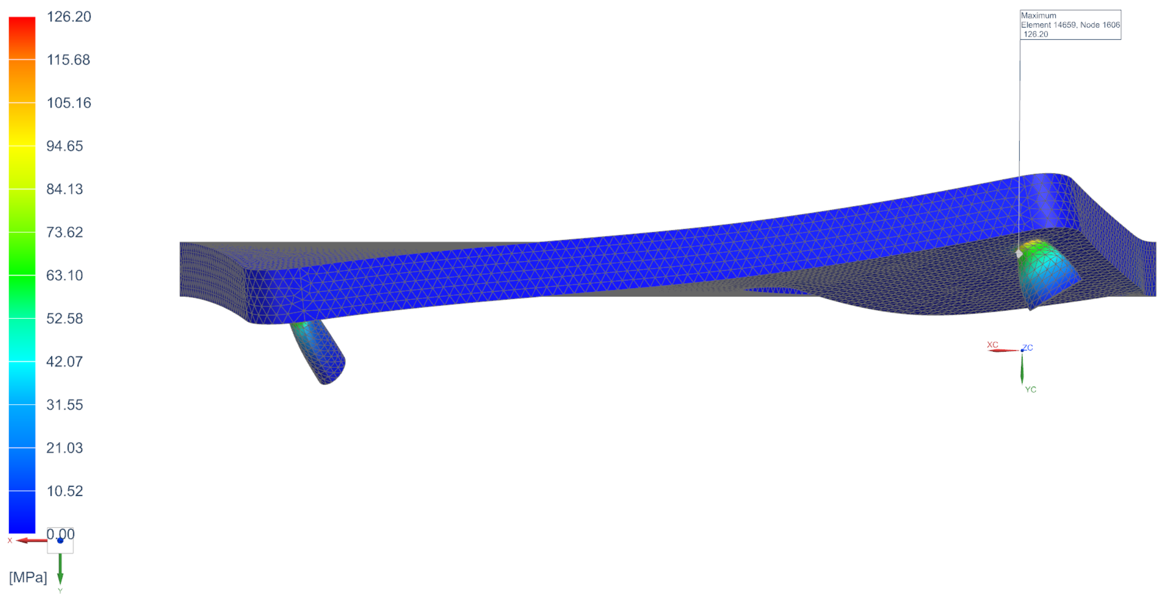




Stress

model4_sim2 : Solution (D free) Result
Subcase - Static Loads 1, Static Step 1
Stress - Element-Nodal, Unaveraged, Von-Mises
Min : 0.00, Max : 126.20, Units = MPa
Deformation : Displacement - Nodal Magnitude



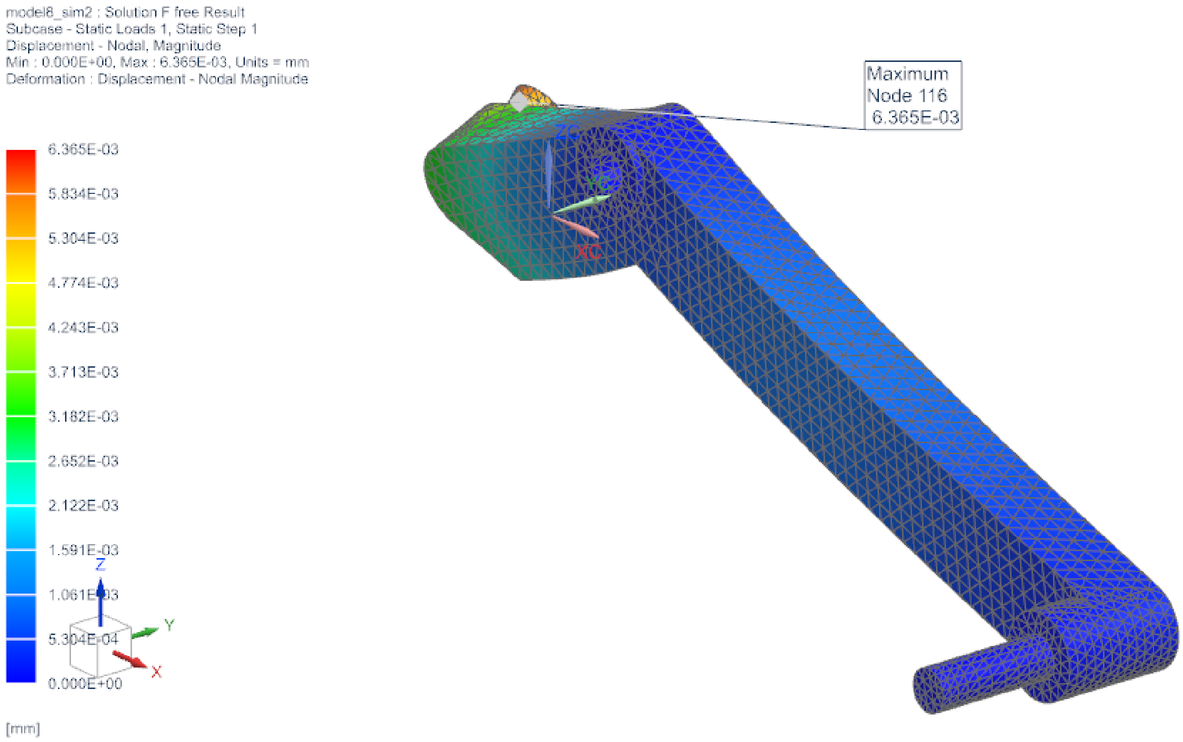


Different Shape Testing for Triangle Part

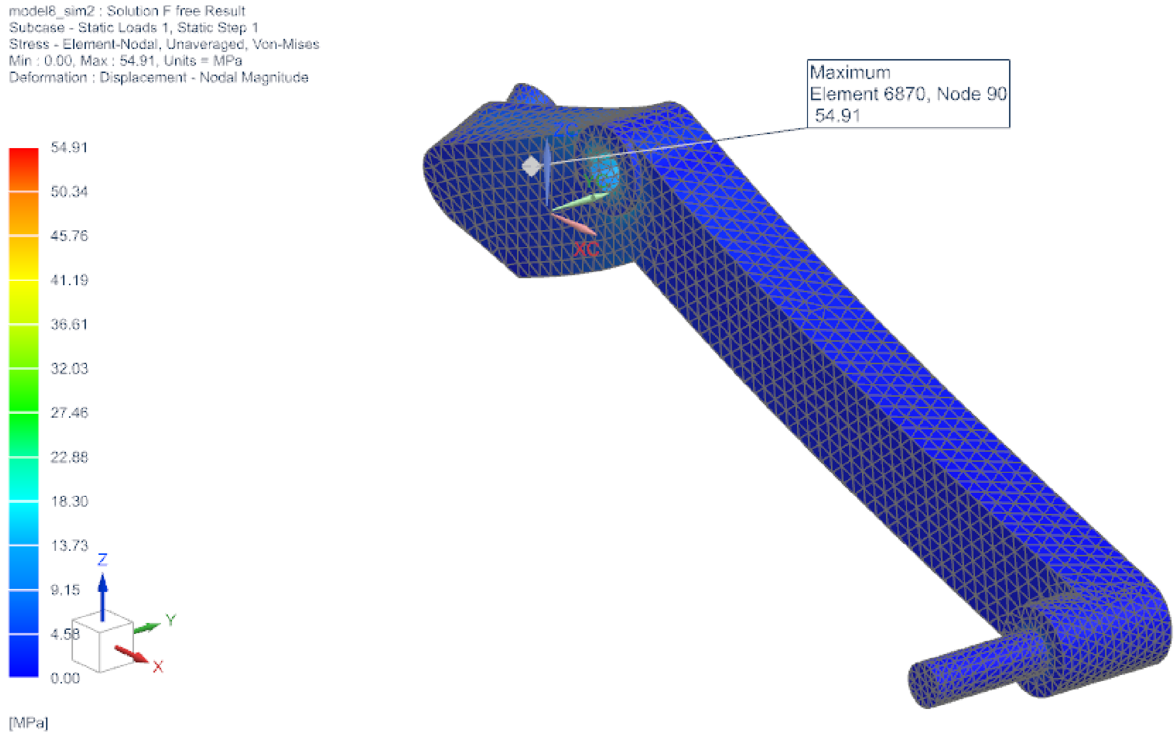
Part 1

F free

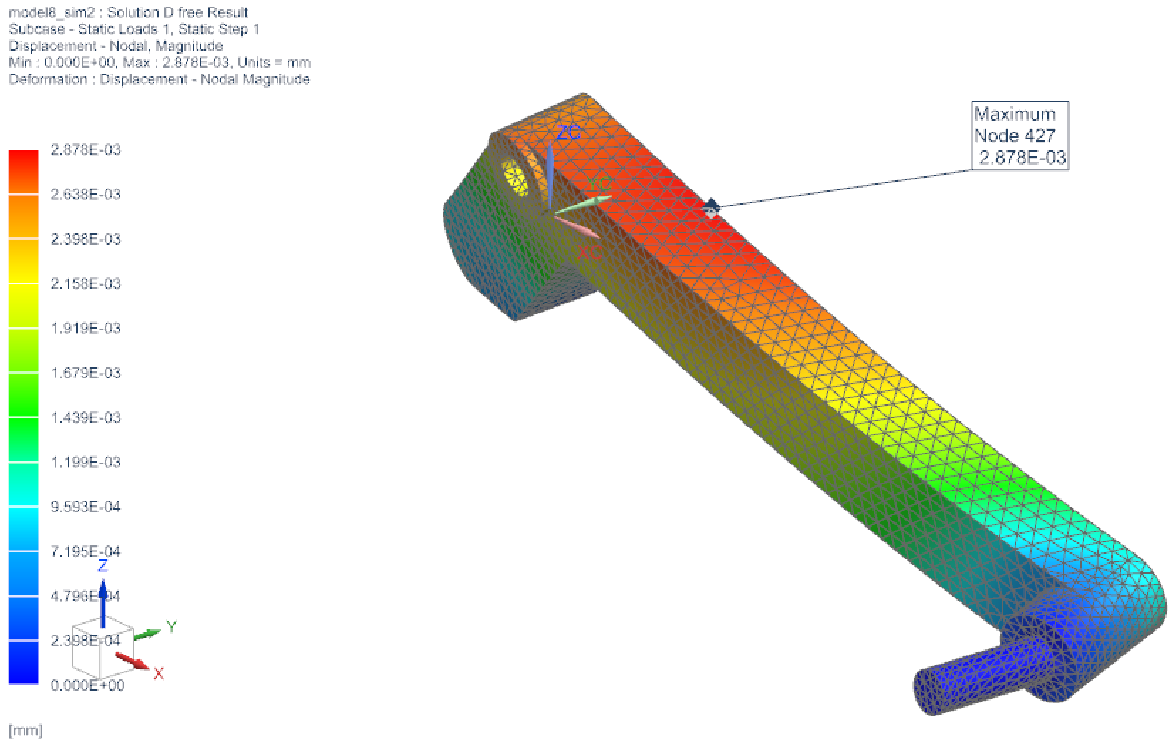
Displacement



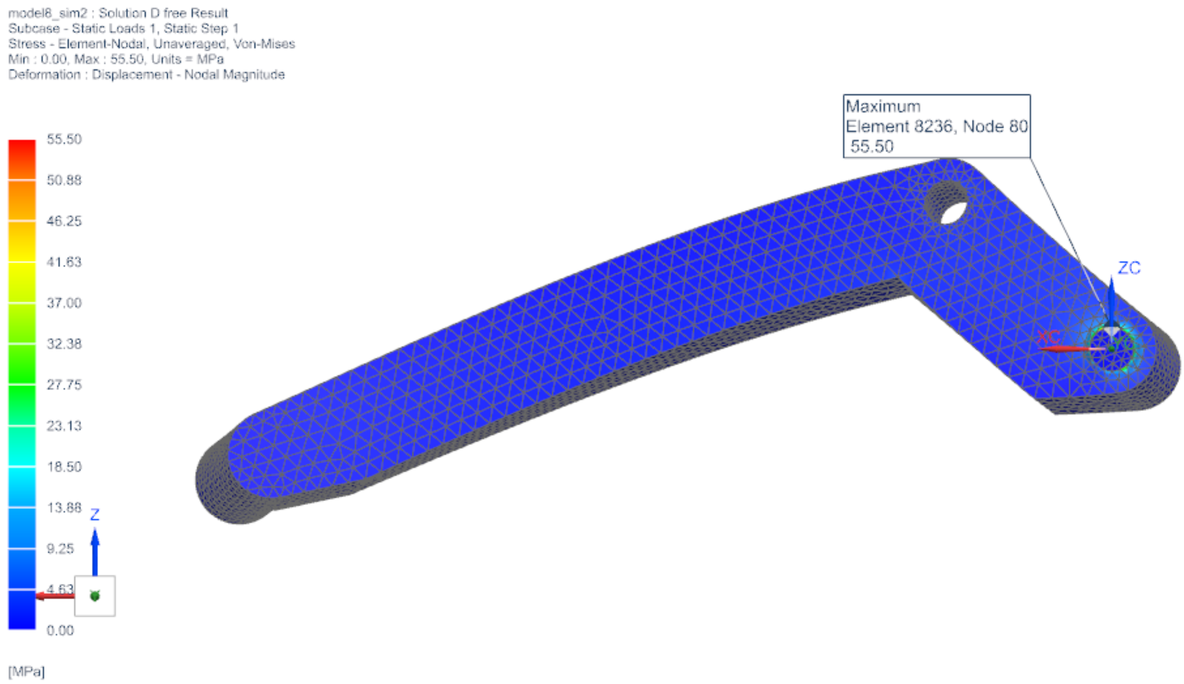
Stress



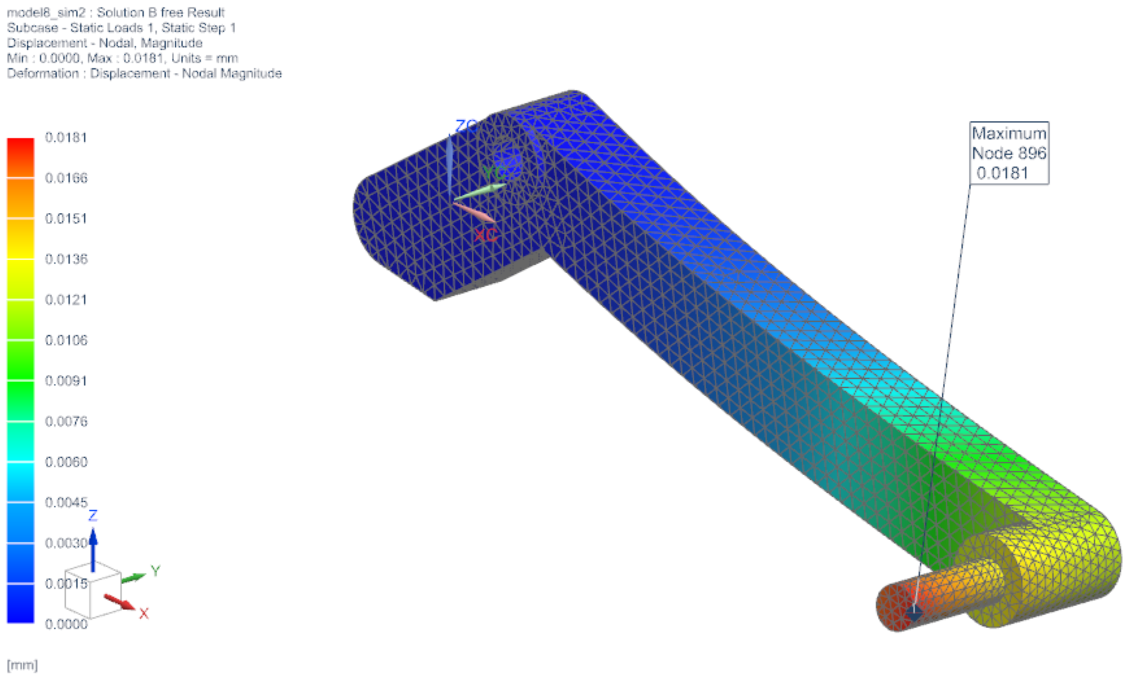
D free
Displacement



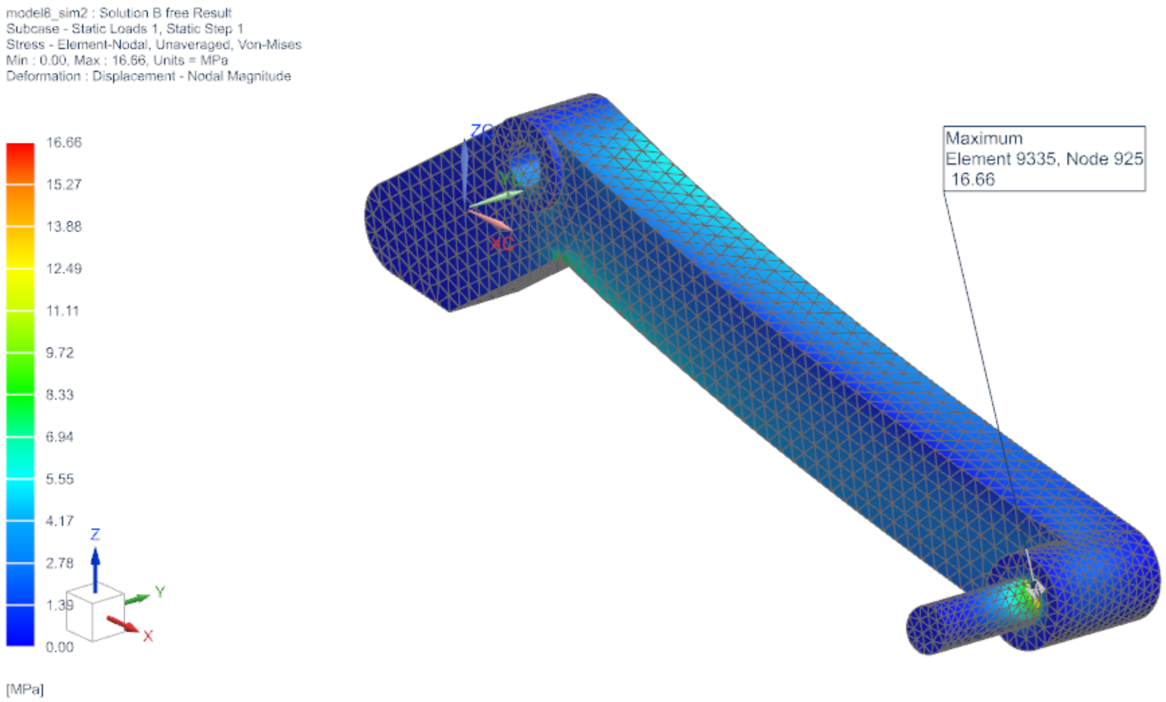
Stress



B free
Displacement



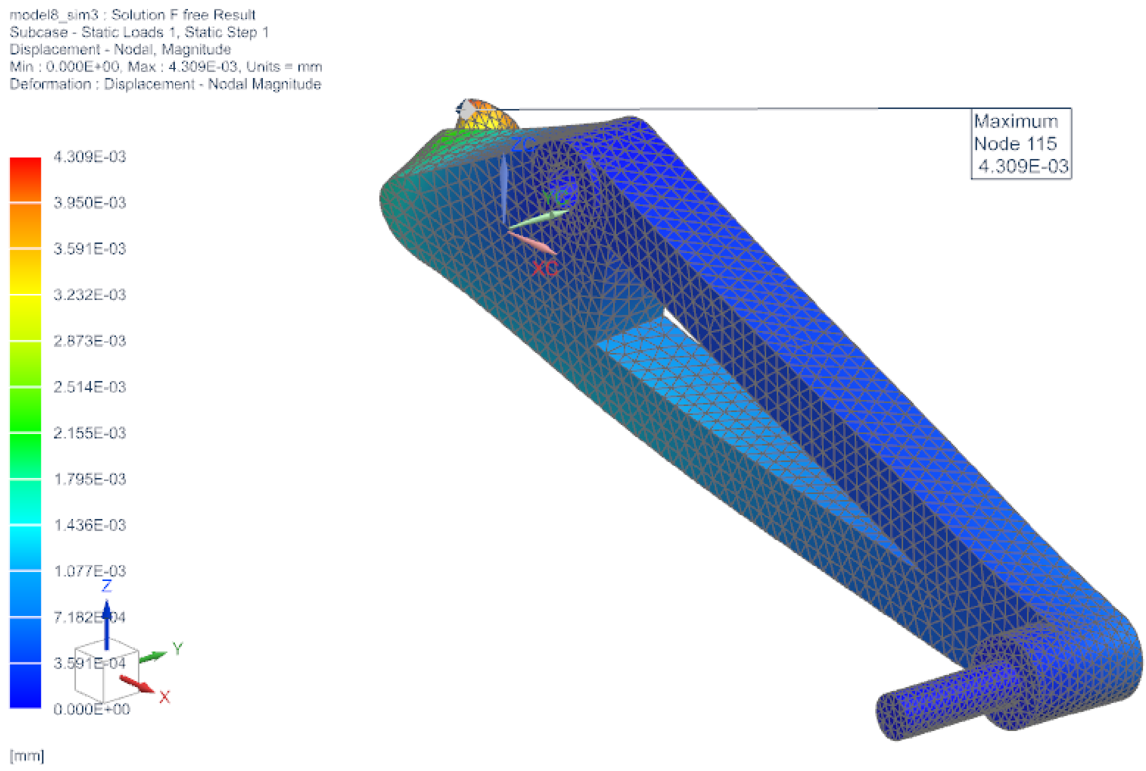
Stress



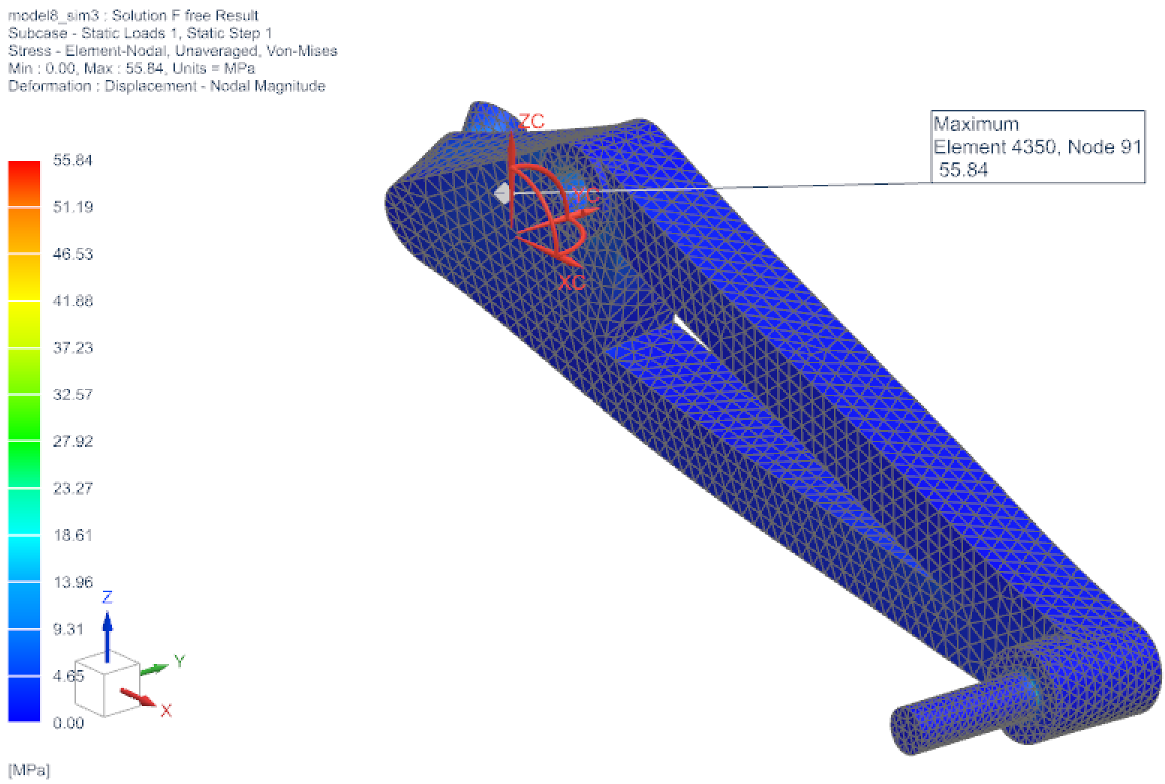
Part 2

F free

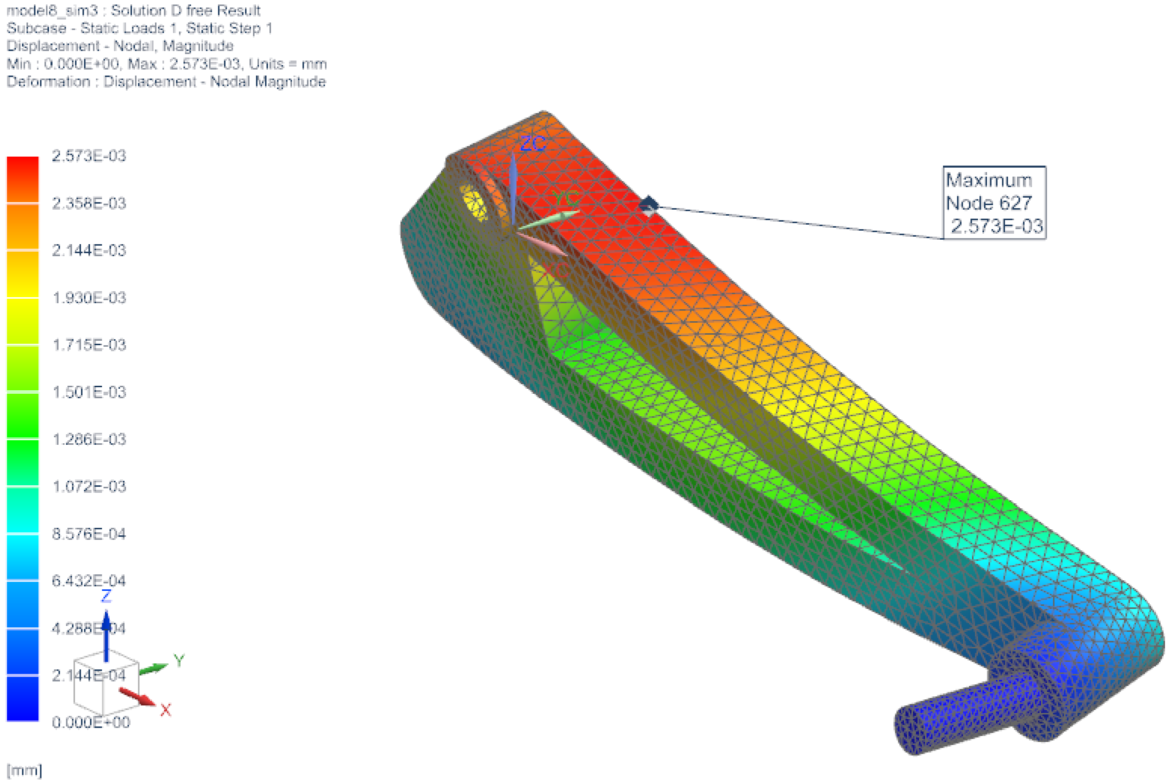
Displacement



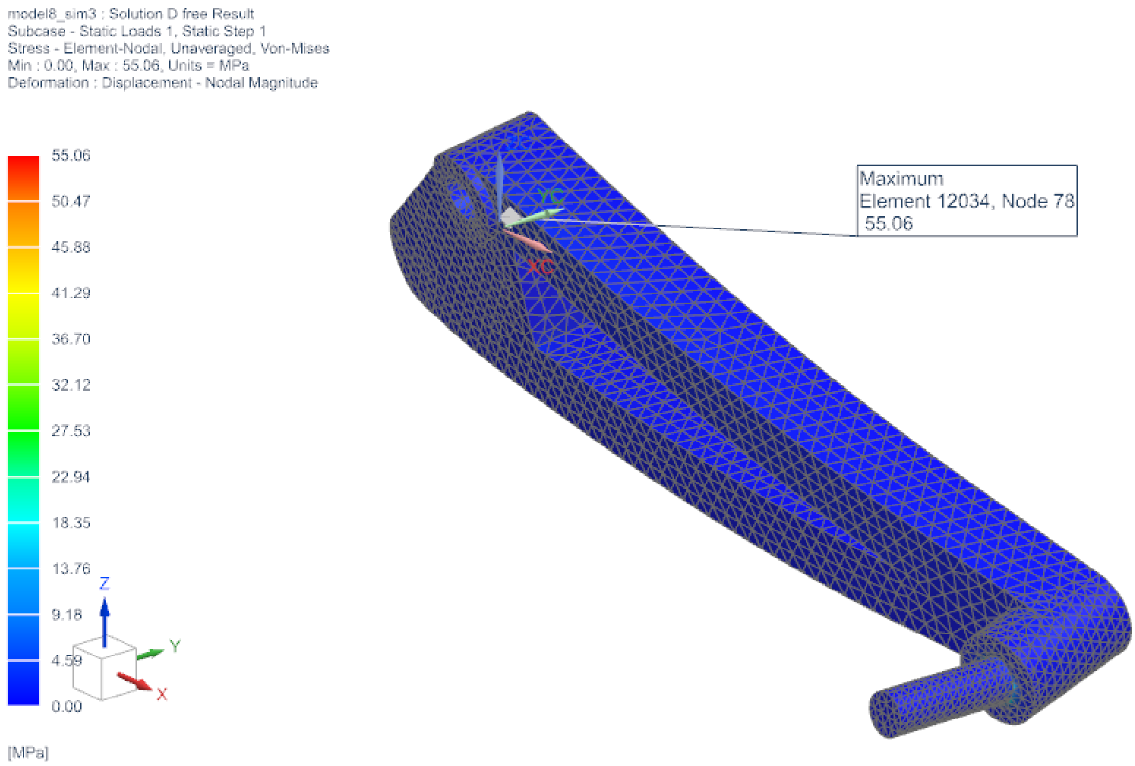
Stress



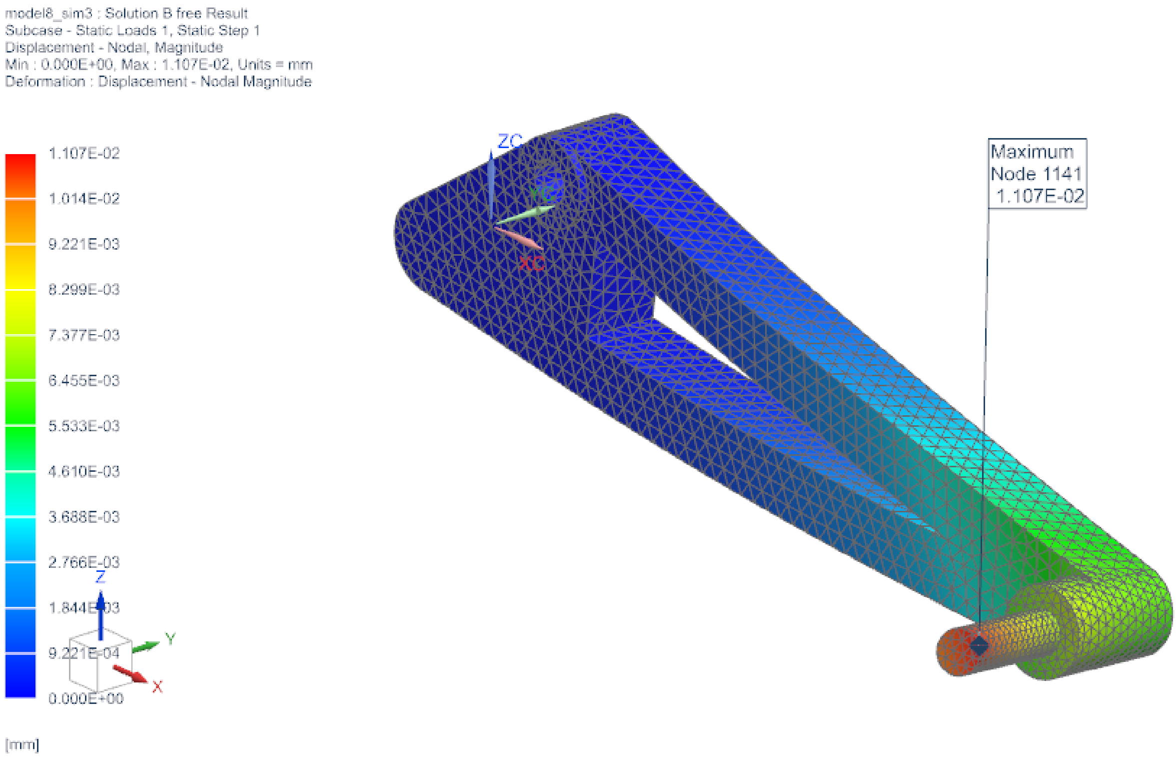
D free Displacement



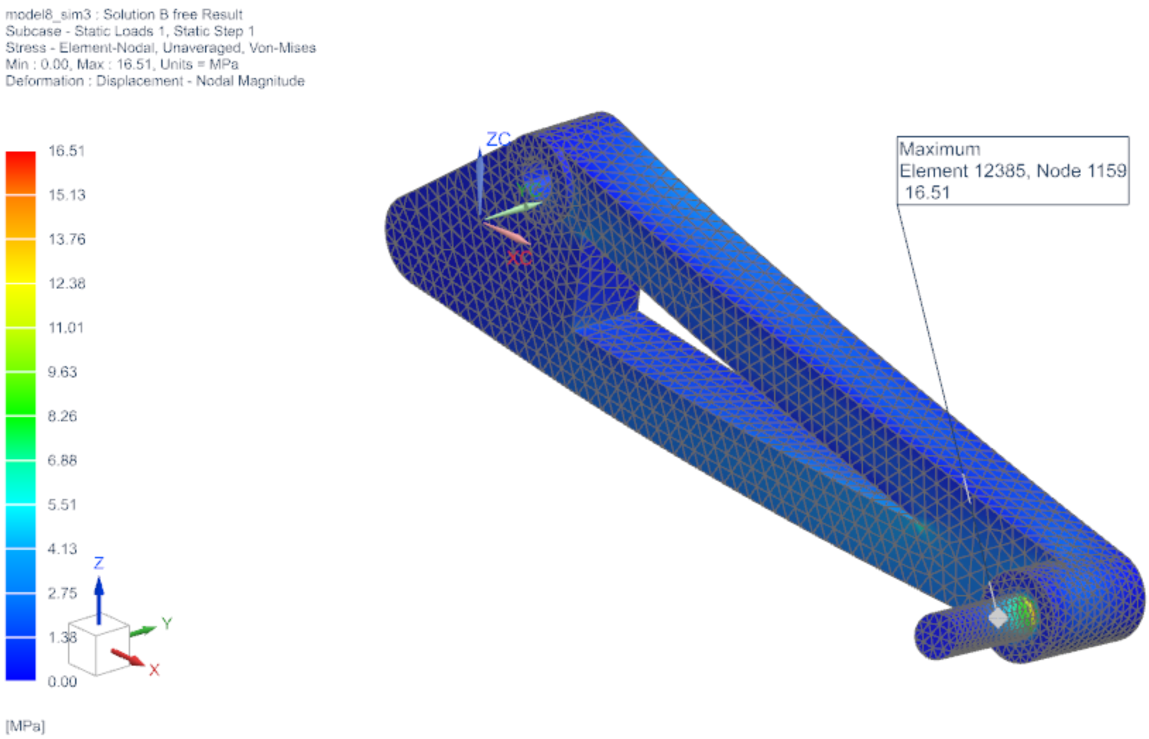
Stress



B free
Displacement


model8_sim3 : Solution B free Result
Subcase - Static Loads 1, Static Step 1
Displacement - Nodal, Magnitude
Min : 0.000E+00, Max : 1.107E-02, Units = mm
Deformation : Displacement - Nodal Magnitude
1.107E-02
1.014E-02
9.221E-03
8.299E-03
7.377E-03
6.455E-03
5.533E-03
4.610E-03
3.688E-03
2.766E-03
1.844E-03
9.221E-04
0.000E+00
[mm]
Maximum
Node 1141
1.107E-02
ZC
XC
Z
Y
X

Stress


model8_sim3 : Solution B free Result
Subcase - Static Loads 1, Static Step 1
Stress - Element-Nodal, Unaveraged, Von-Mises
Min : 0.00, Max : 16.51, Units = MPa
Deformation : Displacement - Nodal Magnitude
16.51
15.13
13.76
12.38
11.01
9.63
8.26
6.88
5.51
4.13
2.75
1.38
0.00
[MPa]
Maximum
Element 12385, Node 1159
16.51
ZC
XC
Z
Y
X