

Analysis and Design of Torque Arm

Cuesta Bret, Cristina

Section 12158 12/05/2025

Abstract—Finite Element Analysis (FEA) is a numerical method for solving differential equations that describe many engineering problems. The objective of this report is to analyze the response of a torque-arm when subjected to horizontal and vertical loads by performing a preliminary structural analysis, a convergence study, element comparison study, and mass minimization study on the torque-arm. The software used to model the torque-arm in this study was Abaqus. The preliminary analysis results obtained assuming the torque-arm behaved like a rectangular cantilever beam were 87.33 MPa for Von Mises stress and 0.0437 cm for vertical displacement. For the convergence study, the results obtained were 148.4 MPa and 0.0932 cm for the smallest element size of 0.62 and largest number of nodes. The values for the bigger element sizes were smaller than the 0.62 results. The converged displacement obtained from Richardson Extrapolation was 0.0945 cm. For the element comparison study, it was found that the elements with the highest node numbers (8-node quadrilateral and 6-node triangle) provided stress values closer to the one obtained in the convergence study with these values being 148.5 MPa and 147.6 MPa respectively. For the mass reduction study, it was found that the values x_3 , x_2 , and x_1 of 32, 3, and 9 cm gave the best results for vertical displacement (0.135 cm) and maximum Von Mises stress (223.5 MPa) with a 14.9% mass reduction.

Index Terms—Abaqus, Displacement, FEA, Force, Stress

I. INTRODUCTION

Finite Element Analysis (FEA) is an essential method in the engineering world to analyze how a component will react under different stress conditions. FEA uses basic Mechanics of Materials (MoM) principles for stress and displacement. The objective of this report was to analyze the behavior of a torque-arm when subjected to horizontal and vertical forces about a reference point. A structural analysis, convergence study, element comparison study, and mass minimization were completed using Abaqus to analyze said behavior.

For the structural analysis of any component, two key aspects are taken into account, stress and deflection. Classical beam theory from MoM can be used to provide accurate estimates to compare to the FEA results. The Von Mises principle of stress (1) was used as the basis for stress calculations in the study, which evaluates failure under different loads by providing a single stress value. The Von Mises principle uses the three principal stresses σ_1 , σ_2 , and σ_3 to obtain the final Von Mises stress, σ_{VM} . The stress caused by the x-axis force was calculated using the equation relating the force in the x direction, F_x , and the cross-sectional area of the beam, A_c . It can be seen in equation (2).

$$\sigma_{VM} = \sqrt{\frac{1}{2} [(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2]} \quad (1)$$

$$\sigma_x = \frac{F_x}{A_c} \quad (2)$$

The torque-arm used in this study can be seen in Figure 1. The model was constrained and fixed at the center of the hole at the left and forces were applied at the center of the right hole.

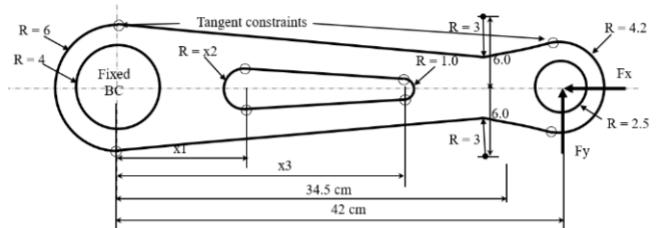


Figure 1. Geometry and dimensions of a torque-arm at the initial design (unit cm).

For a rectangular cantilever beam under bending, the principal normal stress (3), σ_1 , can be defined using the bending moment, M , the distance to the neutral axis, y , and the moment of inertia, I . The moment of inertia (4) can be defined using the base, b , and height, h of the cantilever beam.

$$\sigma_1 = \frac{My}{I} \quad (3)$$

$$I = \frac{bh^3}{12} \quad (4)$$

Deflection of a beam (5) is also a critical characteristic that is usually considered when evaluating beams under loading. Deflection, δ , can be calculated by using the applied load, P , the beam length, L , the beam's Young's Modulus, E , and the moment of inertia, I .

$$\delta = \frac{PL^3}{3EI} \quad (5)$$

To further analyze the behavior of the torque arm, a convergence study was conducted for different global sizes of the part with the objective of showing that the FEA solution will converge to a value the more elements are used. The objective here was to obtain the convergence rate, α , and converged displacement, u_0 . These values were calculated using

Richardson Extrapolation (6-7) which incorporates the element size, h , a constant, g , and the displacement or stress result for element size.

$$u_h = u_0 + gh^\alpha \quad (6)$$

$$u_0 = \frac{u_{h1}p^\alpha - u_{h2}}{p^\alpha - 1} \quad (7)$$

An element comparison study was also done for the torque-arm. To do this, four different element types were used on the mesh, 3-node triangle, 6-node triangle, 4-node quadrilateral, and 8-node quadrilateral elements. Once all elements were applied to the model, a result comparison was conducted.

The final method of analysis was performing a mass minimization study. The initial design was made with a margin of safety for maximum allowable stress. This analysis was made by reducing the variables in the initial model, as seen in Figure 1, as much as possible while keeping the Von Mises stress lower than the maximum allowable stress. The mass was reduced following prior knowledge of MoM topics and the differences between the initial and final results were compared.

II. PROCEDURE

Preliminary Analysis

The first step was hand-calculating an approximation of the maximum Von Mises stress and the vertical displacement at the load application point. These values were calculated using Mechanics of Materials equations (1-5). The geometry provided in Figure 1 along with the assumption that the torque-arm would behave like a rectangular cantilever beam were used to calculate σ_{VM} and the vertical deflection, δ . The largest height and length were used. To calculate the Von Mises stress, it was assumed that the second and third principal stresses were equal to zero since there was no force in the z direction, furthermore, since the forces create a bending motion on the beam, there is no normal stress in the transverse or y direction either. There was also no shear stress in the beam since all the loading was bending. The stress from the horizontal force was calculated and added to the Von Mises stress to find the maximum stress on the beam.

Constrained Geometry Modelling

Once the hand calculations were made, a model of the torque-arm was created on Abaqus. A part was created in the workspace and then a 2D mesh sketch was made. The starting point to do this, was the fixed hole and the reference point where the loads would be applied [1]. Once those circles were drawn and the one to the right was fixed, then two more were made with the 3cm radius next to the circle on the right. The circles were then connected, and the excess lines were trimmed down. All the dimensions were then added into the sketch. Then the geometry in the center of the torque arm was created following a similar procedure. To get it fully constrained, all the dimensions had to be input as well as tangencies between connecting lines and circles. The sketch was finalized when the lines were green, symbolizing that the sketch was fully constrained (Figure 2).

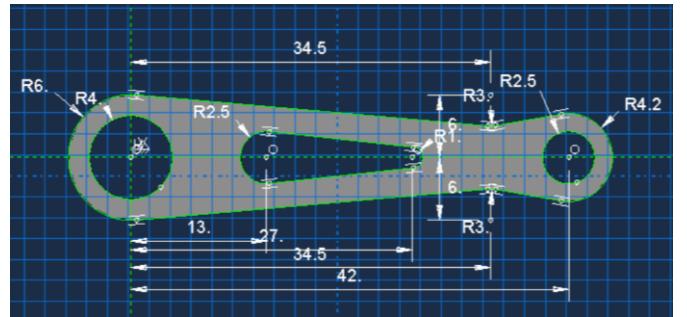


Figure 2. Fully constrained sketch for initial design of torque-arm.

After finalizing the sketch, the material properties were added, including the density, Young's Modulus, and Poisson's ratio. Then a solid, homogeneous section was made and assigned to the mesh. A linear perturbation step was created to assign the loading to. Two constraints were made to define the reference points at the center of the fixed circle (Reference Point, RP1) and the loading point (Reference Point, RP2). Then the loads were applied to RP2 using -4600 N in the x direction and 10,200 N in the y direction. Finally, a boundary condition was added to RP1 so it was fixed in both x and y directions.

Convergence Study

The third analysis method was a convergence study. This study consisted of creating a mesh using 4-node quadrilateral elements and changing the global element sizes in the seed part. Three different test cases were conducted for three different element sizes of 0.62, 1.0, and 1.61. For each size, a job was created and submitted. Using the Query tool and the probe, the vertical displacement at RP2 and the maximum stress values at the critical points were obtained for each size. The values were then input into a table on Excel to analyze and create two graphs using the data collected. The graphs plotted were displacement versus node numbers and maximum stress versus node numbers. Additionally, the convergence rate, α , and the converged displacement, u_0 , were calculated using the data obtained (6-7) [2]. An additional equation was derived to find the convergence rate on excel (10).

$$R = \frac{|U_{h3} - U_{h2}|}{|U_{h2} - U_{h1}|} \quad (8)$$

$$p = \frac{h_3}{h_2} = \frac{h_2}{h_1} < 1 \quad (9)$$

$$\alpha = \frac{\ln R}{\ln p} \quad (10)$$

Element Comparison Study

For the element comparison study, four different element types were analyzed, 3-node triangle, 6-node triangle, 4-node quadrilateral, and 8-node quadrilateral. The global size was changed to 1.5 for all element types in the seed menu. Then, the element type was changed in the mesh menu each time. For the 3 and 4 node element types, the linear order was selected, whereas for 6 and 8 nodes, the quadratic order was chosen. A separate job was created for each of them and the displacement

and Von Mises stress results were compared. The query tool was used to obtain displacement and maximum stress values.

Minimizing Mass

The last analysis method was reducing the mass by changing the variables in the initial design (Figure 1) while keeping inside the bounds provided (Table 1).

Table 1. Upper and lower bounds of design variables (cm)

Design variable	Lower bound	Initial Value	Upper bounds
x_1	8.0	13.0	15.0
x_2	0.5	2.5	3.0
x_3	20.0	27.0	37.0

The procedure followed to do this was to increase the area of the center hole as much as possible while keeping the maximum Von Mises stress under the maximum allowable stress value of 250 MPa. The procedure followed the idea that increasing x_3 reduced material between holes, which meant that there would be higher stresses. First the value of x_3 was increased to the upper bound of 37 cm. The stress turned out to be higher than the allowable, so the variable was reduced to 30 cm. This value seemed to work, so then x_2 was increased to 3 cm as well. When these two values gave good results, the x_1 value was decreased to 10 cm. This procedure was used until reasonable results were obtained.

III. RESULTS

After creating the torque-arm model on Abaqus and performing all the analysis methods, the results were put together to characterize the different elements and global element sizes. The initial design had the following values input into Abaqus as represented in Table II.

TABLE II
INITIAL DESIGN VALUES

Symbol	Quantity	Value
E	Young's Modulus	$2 \times 10^7 \text{ N/cm}^2$
v	Poisson's Ratio	0.35
t	Thickness	2 cm
ρ	Density	0.00785 kg/cm ³
x_1	Length	13 cm
x_2	Length	2.5 cm
x_3	Length	27 cm

* Units were kept in N-cm system consistently on Abaqus

The values obtained for the preliminary analysis can be seen in Table III. The equations used are (1-5). The maximum Von Mises stress was found adding the axial and von mises stresses.

TABLE III
PRELIMINARY ANALYSIS VALUES

Symbol	Quantity	Value
F_x	Force in x-direction	-4600 N
F_y	Force in y-direction	10200 N
A_c	Cross-sectional Area	24 cm ²
σ_x	Axial stress in x-axis	-1.917 MPa
M	Bending Moment	428400 N*cm
I	Moment of Inertia	288 cm ⁴
σ_{VM}	Von Mises Stress	89.25 MPa
$\sigma_{VM,max}$	Maximum Von Mises Stress	87.33 MPa

For the convergence study was then performed using three different global element sizes. These sizes were 0.62, 1.0, and 1.61. All the sizes were applied on a 4-node quadrilateral and had a deformation as shown in Figure 3. The values obtained for the number of nodes, vertical displacement, and Von Mises stress are shown in Table IV.

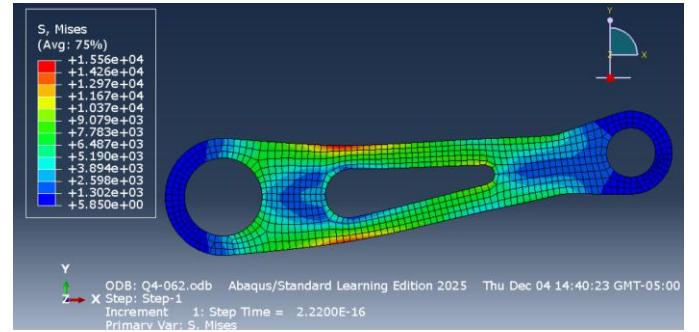


Figure 3. Torque-arm with a mesh of 4-node quadrilateral elements with element size of 0.62 and Von Mises stress deformation.

TABLE IV
CONVERGENCE STUDY RAW VALUES

Element Size	Number of Nodes	Displacement (cm)	Von Mises Stress (MPa)
0.62	939	0.09315	148.4
1.0	429	0.09237	141.2
1.61	223	0.09111	133.4

For the element comparison study, four different element types were created and the number of nodes, vertical displacement, and maximum Von Mises stress was collected for all of them. The global element size was kept at 1.5 for all element types, and data can be seen on Table V.

TABLE V
ELEMENT COMPARISON STUDY VALUES

Element Type	Number of Nodes	Displacement (cm)	Von Mises Stress (MPa)
T3	245	0.08916	133.7
T6	846	0.09350	147.6
Q4	225	0.09077	132.3
Q8	610	0.09378	148.5

* T= Triangle; Q= Quadrilateral

For the mass minimization study several steps were taken until the lowest possible mass was obtained. A total of 8 changes were made. The changes made can be seen in Table VI and the results can be seen in Table VII.

TABLE VI
MASS MINIMIZATION ITERATION CHANGES

Iteration	x_1	x_2	x_3
1	13 cm	2.5 cm	27 cm
2	13 cm	2.5 cm	37 cm
3	13 cm	3 cm	30 cm
4	13 cm	3 cm	30 cm
5	10 cm	3 cm	30 cm
6	9 cm	3 cm	32 cm
7	8 cm	3 cm	33 cm
8	8 cm	2.5 cm	33 cm
9	8 cm	3 cm	32 cm

TABLE VII
MASS MINIMIZATION VALUES

Iteration	Element Size	Mass (kg)	Displacement (cm)	Von Mises (MPa)
1	1	5.009	0.09237	141.2
2	1	4.461	0.16916	287.7
3	0.62	4.642	0.10741	179.5
4	0.62	4.642	0.10741	189.8
5	0.62	4.451	0.11534	186.7
6	0.62	4.263	0.13498	223.5
7	0.62	4.138	0.15042	263.6
8	0.62	4.405	0.13959	263.3
9	0.62	4.201	0.13816	229.6

IV. DISCUSSION

The objective of this report was to analyze the behavior of a torque-arm using Abaqus FEA and to compare the data obtained with hand calculations made assuming the arm would behave as a rectangular cantilever beam. At first glance, the vertical displacement calculated by hand is of the same order of magnitude as all the displacements found using Abaqus, which makes sense since the forces applied don't change. The value obtained is not exactly the same because the moment of inertia calculated by hand was for a rectangular beam, therefore the Abaqus displacements would be more accurate.

The maximum Von Mises stress values differ by a factor of 1.6 using the initial design values from Table IV. This means that the initial model sketched on Abaqus is stiffer than the assumed rectangular cantilever beam calculated. The hand calculations assume a fully rectangular beam without discontinuities and no critical stress points. In the FEA model, geometry causes stress concentrations as seen in Figure 3. Additionally, the original geometry has fillets and holes that can cause stress higher than the ones predicted by a rectangular cantilever beam.

The convergence study clearly shows that increasing the number of active nodes makes the values more accurate. It is observed that the values would converge to a certain value if the number of nodes were to be increased more. The smaller the element size, the more elements and thus more active nodes there will be. Based on this, the maximum Von Mises stress for the initial design was around 148.4 MPa and the displacement 0.093153 cm.

Two graphs were plotted using the values obtained, one for the number of nodes vs Von Mises and another one for the number of nodes vs displacement.

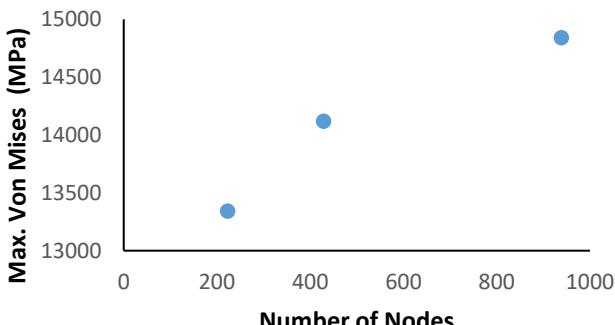


Figure 4. Max. Von Mises stress (MPa) vs number of nodes.

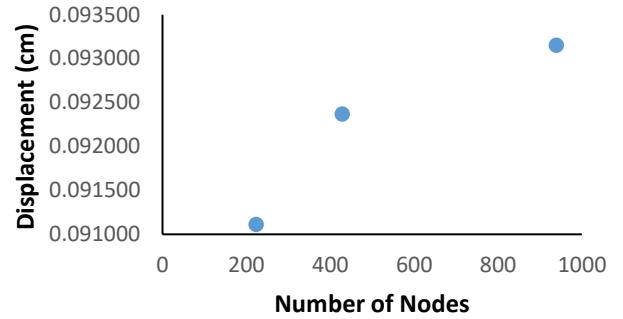


Figure 5. Displacement (cm) vs number of nodes.

Additionally, the convergence rate, α , and converged displacement, u_0 were found using (6-10). The results are shown in Table VIII. The values used to find p were the largest size (h_1) over the second largest size (h_2). The converged displacement value obtained was 0.09445 cm based on Richardson Extrapolation.

TABLE VIII
CONVERGENCE STUDY DERIVED VALUES

Element size	Symbol	Quantity	Value
$h_1 = 1.61$	p	Element size ratio	0.62
$h_2 = 1.0$	R	Displacement ratio	0.62
$h_3 = 0.62$	α	Convergence rate	0.993
--	u_0	Converged displacement	0.09445 cm

The element comparison study compared four different types of elements: 3-node triangle, 6-node triangle, 4-node quadrilateral, and 8-node quadrilateral. As shown in Table V, the quadratic element types (6-node and 8-node) show more accurate results for both the displacement and the maximum stress. This can be mostly due to them having a higher number of nodes. Additionally, differentiating between element shapes, the quadratic should have more accurate results since the shape can represent curvature better than triangles, comparing Figure 3 and Figure 6. Having said this, the maximum Von Mises stress for the initial design according to this analysis was 148.5 MPa and a displacement of 0.0938 cm.

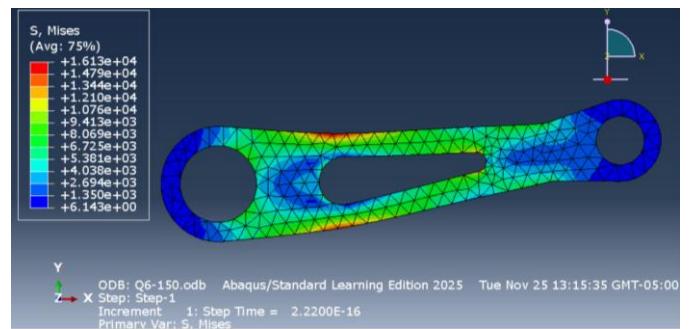


Figure 6. 6-node triangle element type with global size 1.5.

Minimizing the mass was the last study conducted to analyze the behavior of the torque-arm. A total of nine iterations were made with the objective of minimizing the mass while keeping the displacement at a reasonable value and the maximum Von

Mises stress under the allowable 250 MPa. The results shown in Tables VI and VII show that maximizing the center hole will decrease the area and therefore decrease the mass of the torque-arm. It can also be seen, on iteration 2, that increasing the value of x_3 to the maximum bound value (37 cm) will create a high stress of 287.7 MPa that goes past the allowable value of 250 MPa. This result shows that making the hole get too close to the load point increases the stress in that area. The minimum mass obtained was 4.138 kg which corresponds to iteration 7. This iteration also went past the allowable stress value, which meant the distances to the fixed and load points were too small, creating critical stress points. It was concluded that the best values obtained were for iterations 6 and 9. Iteration 9 gave the lowest value for the mass, 16% reduction, had an increase of displacement by 49.6% and showed an increase in maximum stress of 62.6%. Iteration 6 had a 14.9% mass reduction, with increases in displacement of 46.2% and stress of 58.2%. Both iterations are under the allowable stress value of 250 MPa. Taking this into consideration, iteration 6 would be the best option to choose since it achieves a mass reduction difference of 1.1% compared to iteration 9 and has better results for displacement and maximum Von Mises stresses than iteration 9.

V. CONCLUSION

To conclude, the smaller the size of the elements, the more active nodes, which results in a more accurate representation of both the maximum Von Mises stress and the displacement on the torque-arm (Figure 4 and Figure 5). The preliminary analysis provided a good estimate of the vertical displacement but proved to be incorrect regarding the maximum Von Mises stress. This was due to the assumptions made when making hand-calculations about the torque-arm and that hand-calculations don't take into account geometric behavior and bending, whereas FEA does. The element study showed that the quadratic element types were more accurate due to the higher number of active nodes, with the quadratic element being the most representative of the displacement and stress. Finally, minimizing the mass can be very useful but must be done with precaution. Reducing the mass too much can result in high stresses and unwanted deformation at critical points.

REFERENCES

- [1] Ting Dong. "Abaqus Tutorial Torque Arm." *YouTube*, 7 Nov. 2020, www.youtube.com/watch?v=d7UyLqJhURk.
- [2] Kim, Nam H, et al. *Introduction to Finite Element Analysis and Design*. Chichester, Wiley Blackwell, 2018.