493A: Finite Element Analysis

Project 2

Wrench Analysis

Jacob Bridenbecker and Andrea Mallouk
3-12-2021

Bridenbecker, Mallouk
3/12/2021

Abstract

A wrench is desired to be given with a product. Its intended purpose is to be used as a tool by the customer when assembling said product. Two wrench designs have been selected, one with and one without a cutout. This cutout would reduce weight and cost of material to manufacture; however, the decrease in cross sectional area could affect its usefulness.

In this report, we will investigate the differences between both designs. We specifically will be examining the maximum load, locations of developed stress and deformation using FEA methods to help recommend which design to use. Along with this, we will investigate our results and how they reflect our mesh selections within ANSYS.

Through our investigation, we calculated a theoretical maximum load of 74.57 N. Using ANSYS, we found a max load of P = 83 N for the simple wrench and P = 76 N for the cutout wrench. Industrial grade wrenches are rated with a max load of 50 lbs, which roughly equates to 222 N. This affirms that both situations are possible. Overall, we recommend the cutout wrench. The difference between each design's max force is 1.6 lb. This difference is so miniscule, if you were to deform the cutout wrench you would have also deformed the simple wrench. The strengths for both designs are comparable enough where the decrease in weight and cost are worth it.

Introduction

This report models 2 different wrench designs. The first design, referred to as the regular or simple wrench, has a thickness of 5 mm and follows the design below.

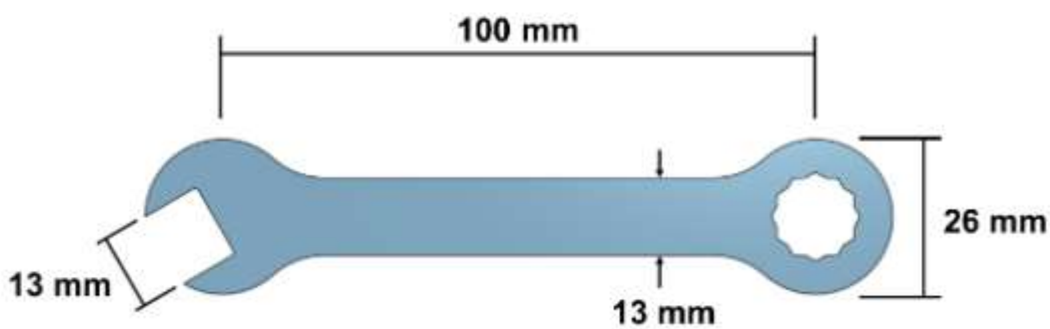


Figure 1: Simple Wrench Design, Thickness of 5 mm

The second design, referred to as the cutout wrench, has a thickness of 2.5 within the cutout and 5 mm everywhere else. It follows the design below.

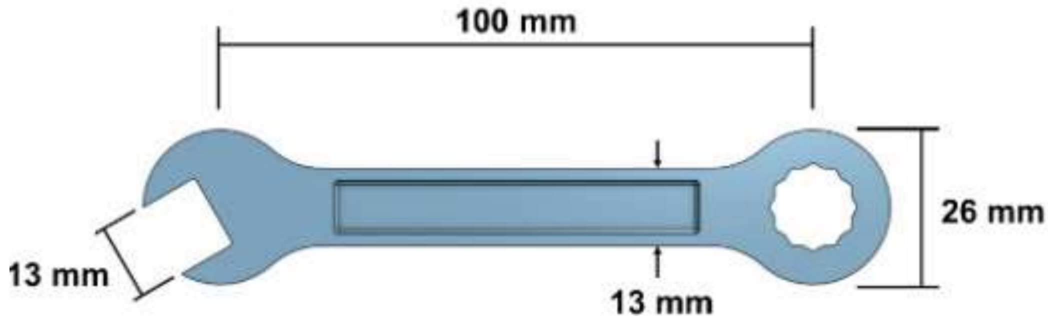


Figure 2: Internal Cutout Wrench Design, Thickness of 5 mm and 2.5 mm

Both wrenches will be made out of the same material, Aluminum, with a Modulus of Elasticity of 68 GPa, Poisson's ratio of 0.33, and Yield Strength of 50 MPa. Both wrenches will be used to tighten a bolt with the open end and an object will be used to pull down with force P at an angle of 15° on the opposite end. The end attached to the bolt will be considered fixed as it will not move or rotate.

Figure 3: Loading P and Boundary Conditions for Both Wrenches

With this, we will observe and compare the differences between both designs for the maximum load P, deformation, maximum stresses and their location, along with the effects of mesh types on our simulated results. We will then determine if the decrease in weight for the cutout design is still suitable for its intended purpose, or if the simple, but heavier, design is better suited.

Analysis Procedure

First we can start with a simple hand calculation to get a rough estimate of what we can expect oru maximum load to be. If we simplify our wrench to a cantilever beam with an unknown angled load at the end we can solve for max stress. The setup for the hand calculation appears below in figure 4.

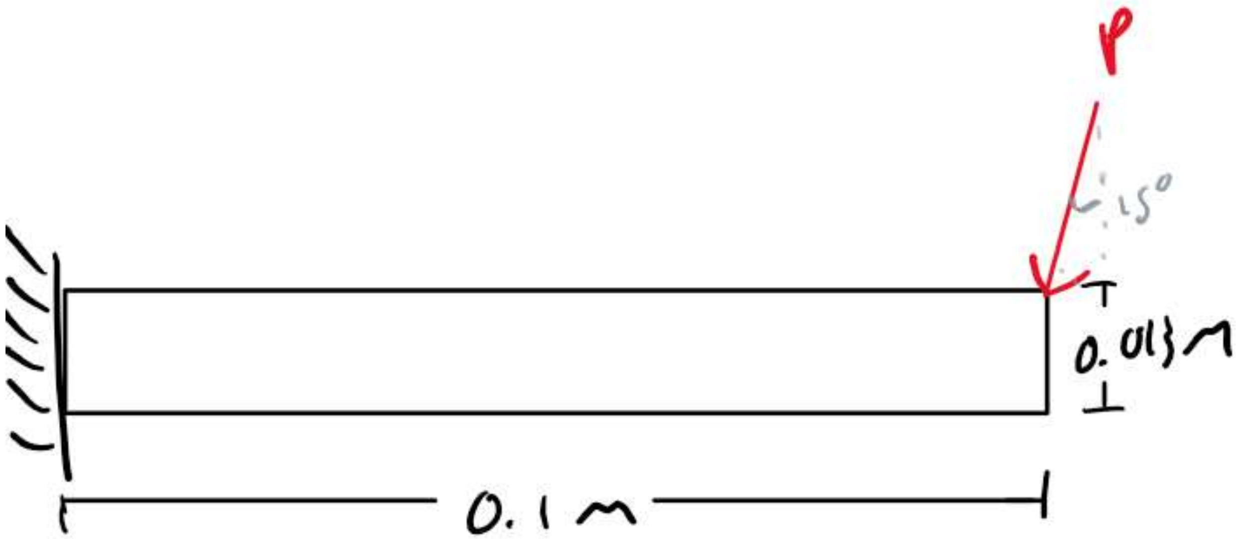
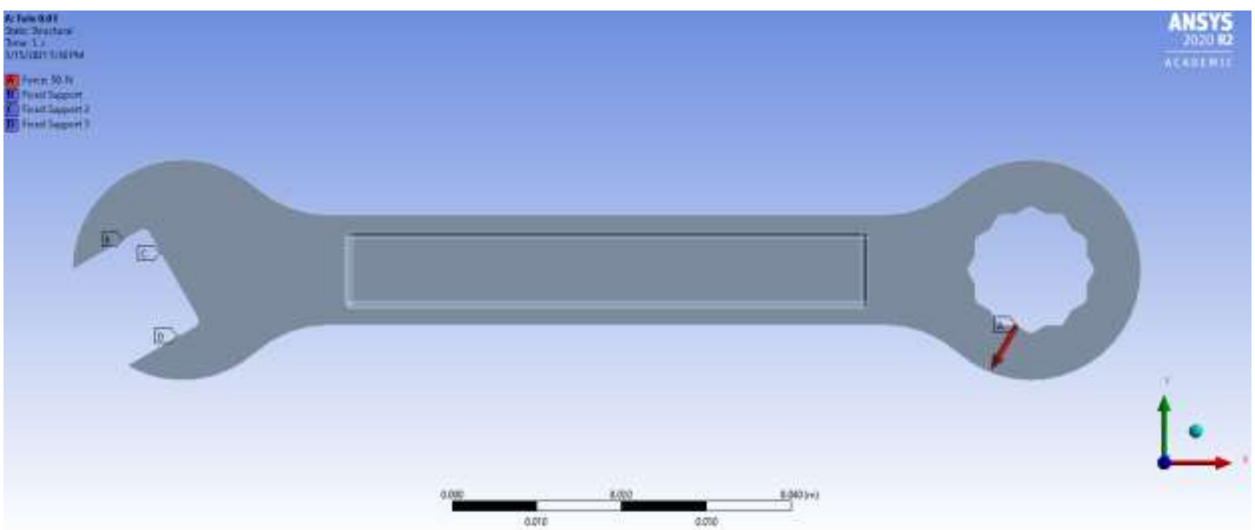


Figure 4: Simple Setup

In ANSYS we will apply various boundary conditions to the provided model in order to properly simulate the forces on the wrench. For the bolt that doesn't move we will apply fixed conditions to the three flat surfaces where the wrench contacts the bolt. On the other end where a bar is inserted and pulled on we will apply an angled force across the areas where the bar contacts as seen below in figure 5.



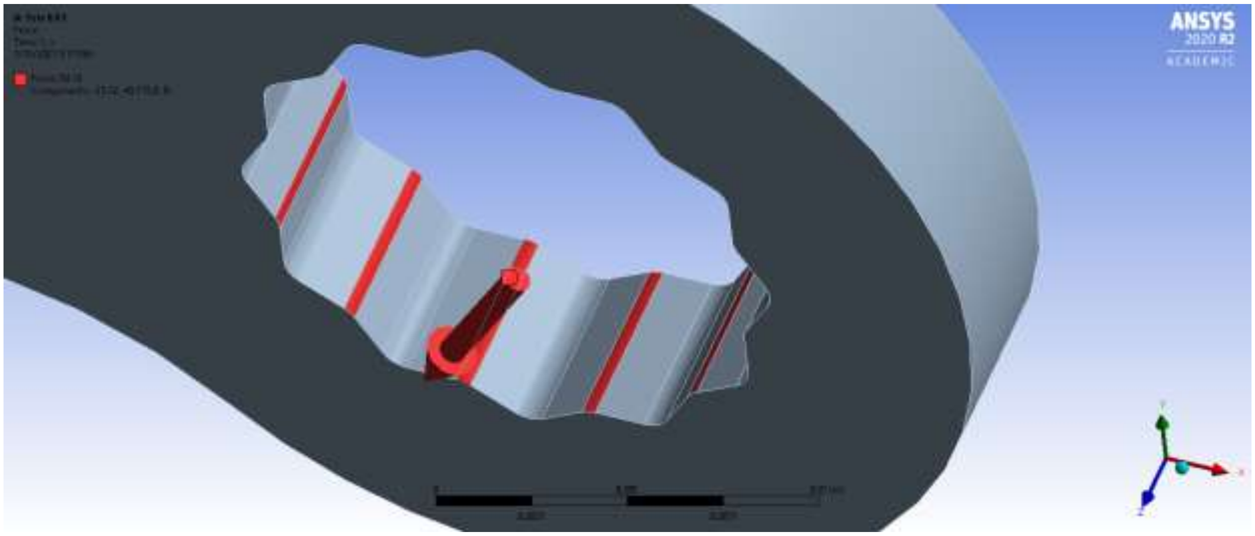


Figure 5: Boundary Conditions

The next step will be to perform a mesh analysis on our simple wrench. This will allow us to determine if we are using a refined enough mesh. This involves running multiple simulations at different mesh sizes and plotting how the maximum stress is affected as the number of elements increase. In theory the maximum stress should plateau as we increase the number of elements. At this point we can use that mesh to run our simulations. We will utilize localized mesh sizing to maximize the number of elements where our forces are the highest.

Because we are using a linear elastic material, our elements will have quadratic properties as linear elements with a linear material produce bad results. Tetrahedrons tend to map better with curved surfaces, but hexahedrons tend to produce better results. We will compare how using tetrahedrons vs hexagonal mesh affects our results.

If we were using the pro version of Ansys we could parameterize the load and have the software automatically iterate until it finds a force that causes stress equal to the material yield strength. However with the student version we will have to do that manually and gradually increase the applied load until we reach the yield strength of the given material. Finally we will analyze the internal cutout wrench to see how it compares to the simple wrench.

Results

Show the figures and numerical results of your analysis here. Describe any important features from the results in text. This should be a purely descriptive section – do not include your interpretation of the results here. All results figures should have a reference in the text. Make sure to include results for all the questions asked on the project assignment.

Part 1: Hand Calculations

$$\sigma_{max} = \sigma_{bend} + \sigma_{axial} = \frac{M * c}{I}$$

Where M is the moment applied to the beam, c is the radius of the cross section, and I is the moment of inertia.

$$I = \frac{1}{12} * b * h^3$$

Thus

$$5 * 10^7 Pa = \frac{F * \sin(75) * 0.1m * \frac{0.013m}{2}}{\frac{1}{12} * 0.05m * (0.013m)^3} + \frac{F * \cos(75)}{0.005m * 0.013m}$$

so

$$F = 74.57N$$

Part 2: Mesh Convergence Study

Regular

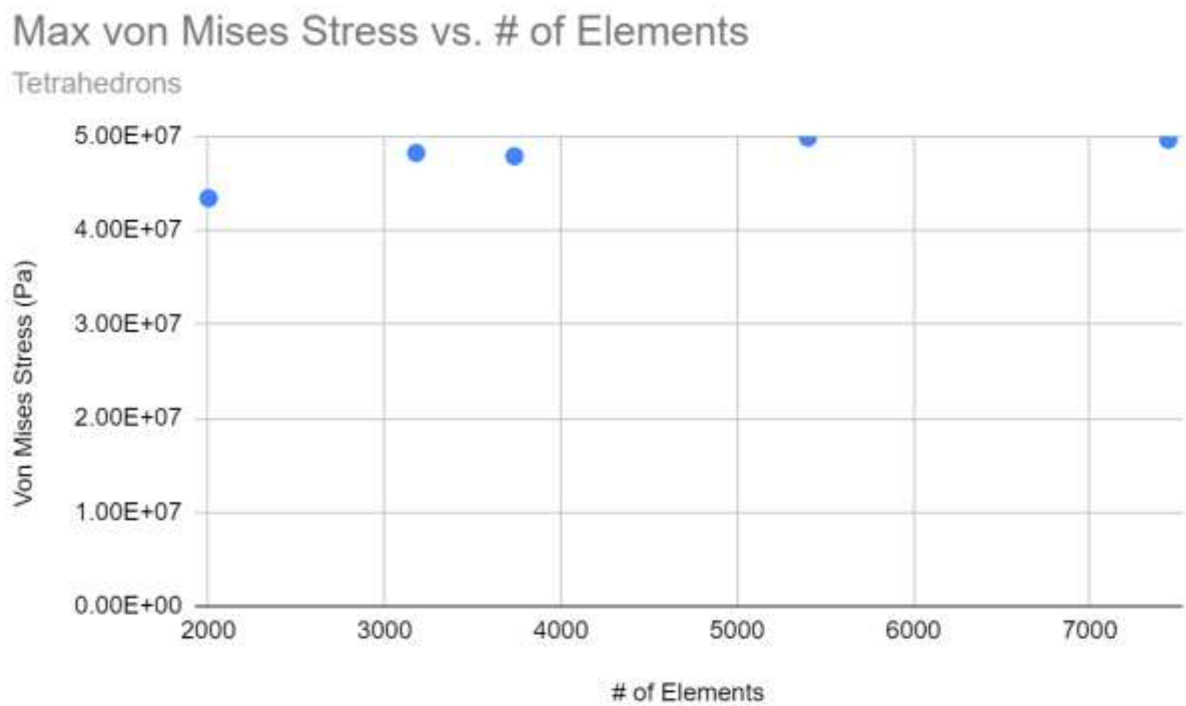


Figure 6: Convergence with P = 83 using Tetrahedrons of Simple Wrench

At element size 2E-03 m, we had 2001 total elements with a max stress of 4.34E+07 Pa. At element size 1.00E-03 m, we had 3179 total elements with a max stress of 4.82E+07 Pa. At element size 0.0008 m, we had 3736 total elements with a max stress of 4.79E+07 Pa. At element size 0.00065 m we had 5401 total elements with a max stress of 4.99E+07 Pa. At element size 0.0005 m, we had 7445 total elements with a max stress of 4.97E+07 Pa.

Cutout wrench

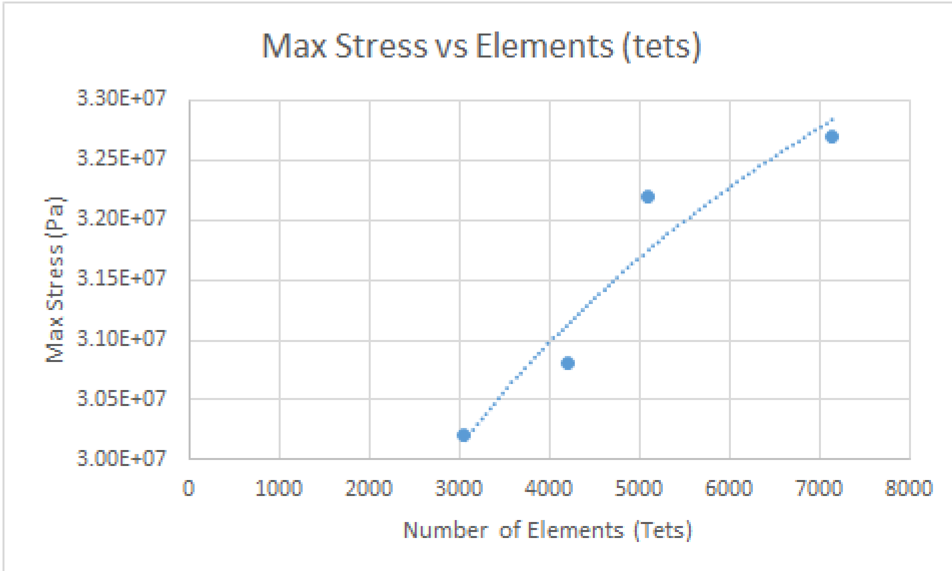


Figure 7: Mesh Convergence for Cutout Wrench at P=50N Using Tetrahedrons

The mesh does not converge on the cutout model before the student version runs into the limit.

Part 3: Deformation and stresses

Regular

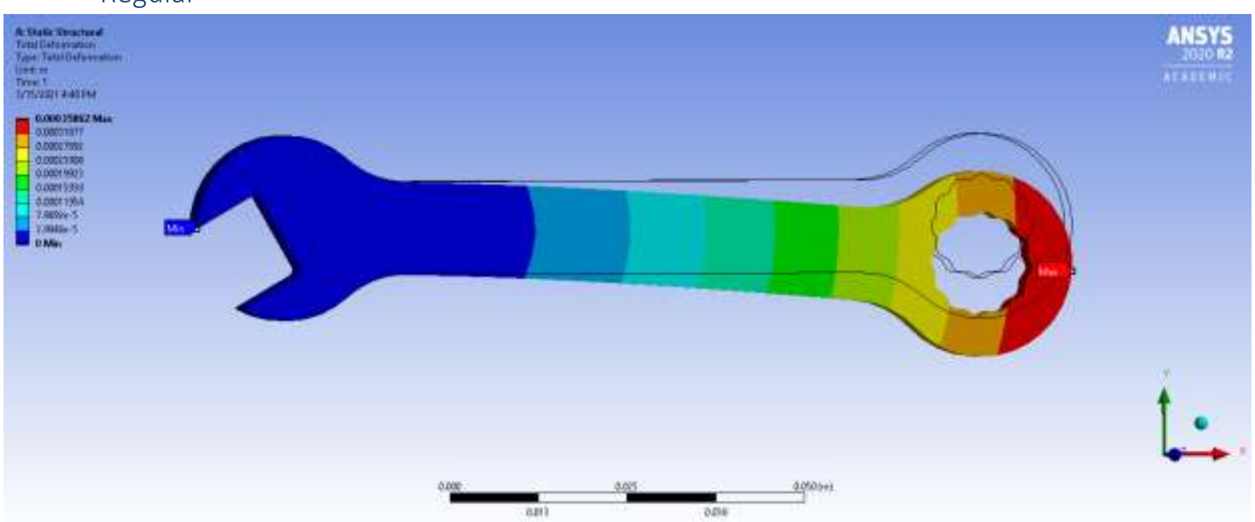


Figure 8: Deformation with Load of 83 N for Simple Wrench

The wrench experiences a max deformation of 0.3562 mm at the farthest point of the handle. The wrench has a minimum deformation of 0 at the location of the bolt.

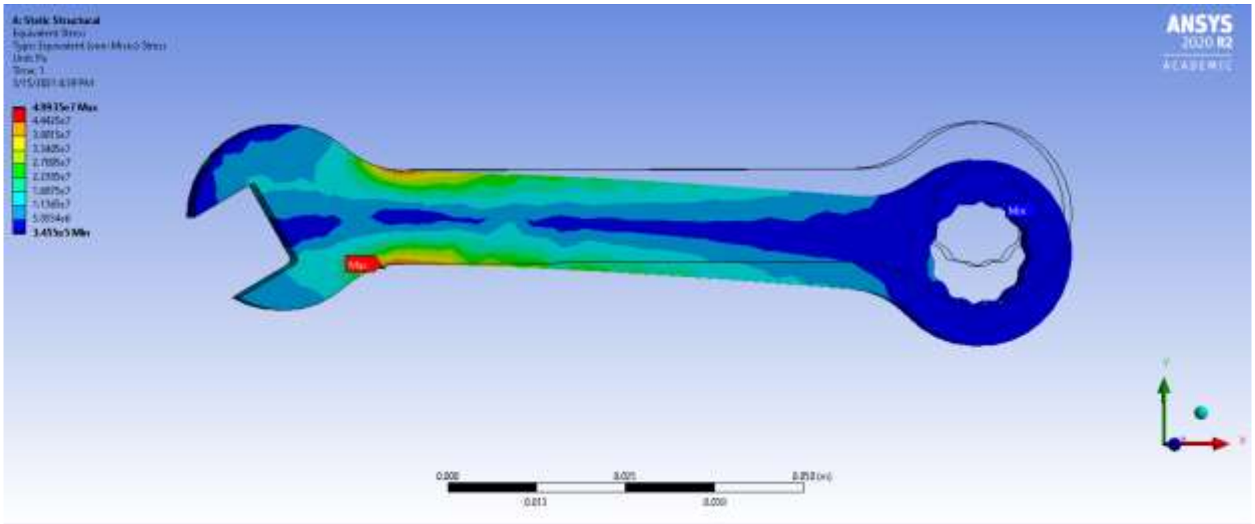


Figure 9: Von Mises with Load of 83 N for Simple Wrench

The wrench experiences a maximum stress of 4.9935E7 Pa at the apex of the bottom curve. The minimum is 3.455E5 Pa near the top right corner inside the handle.

Cutout wrench

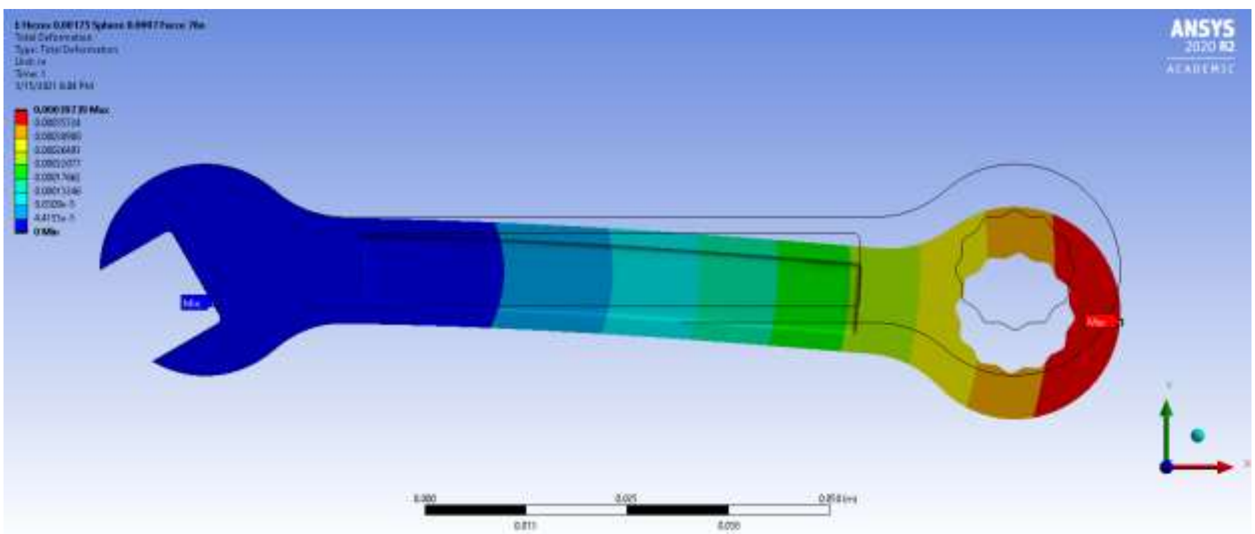


Figure 10: Total Deformation of Cutout Wrench

The cutout wrench Deforms to a maximum of 0.00039739 meters at the end.

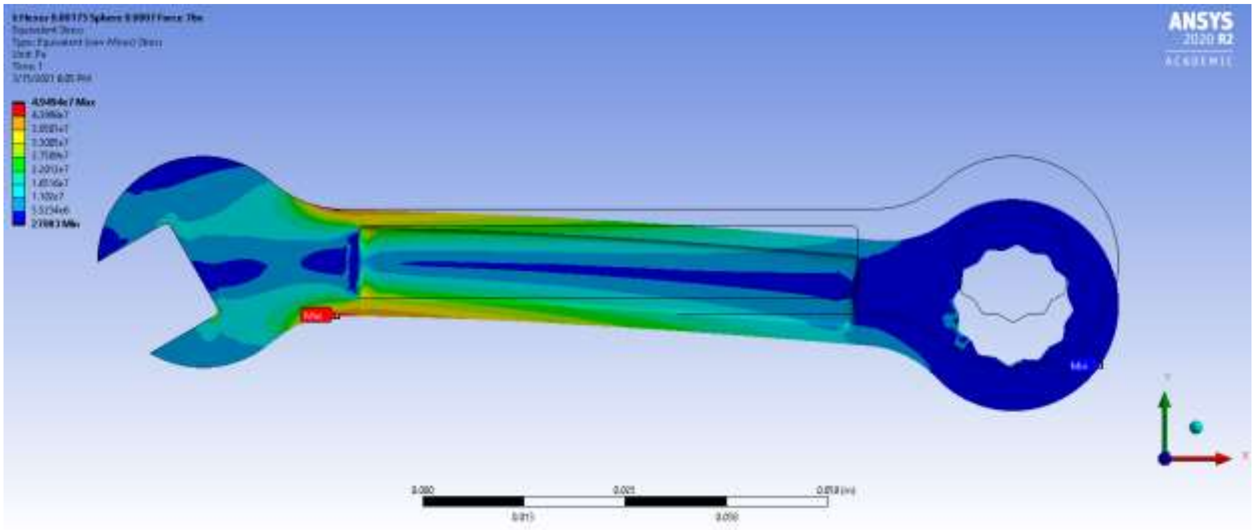


Figure 11: Maximum Von Mises Stress of Cutout Wrench

The cutout wrench experiences a maximum of 4.9494e7 Pa at the underside neck.

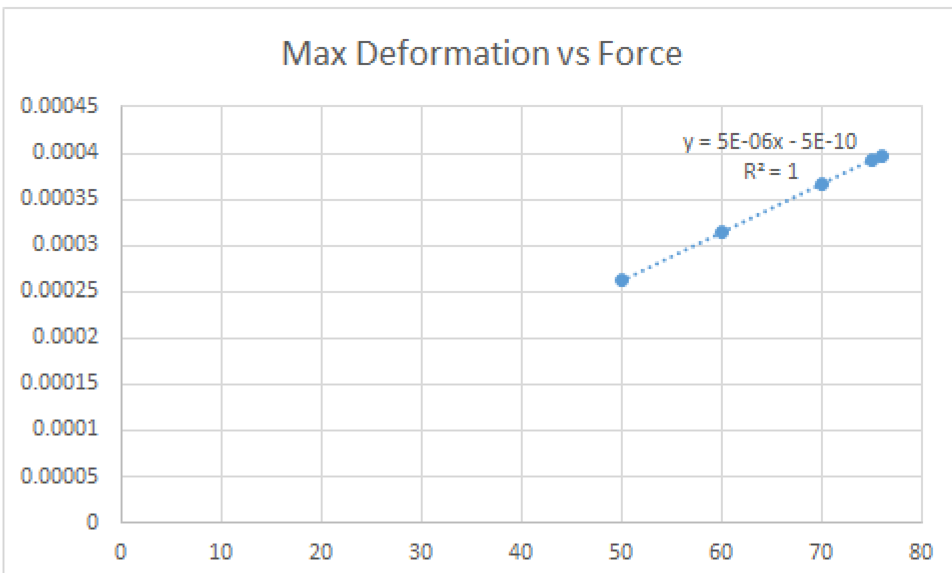


Figure 12: Max Deformation vs Force for Cutout Wrench

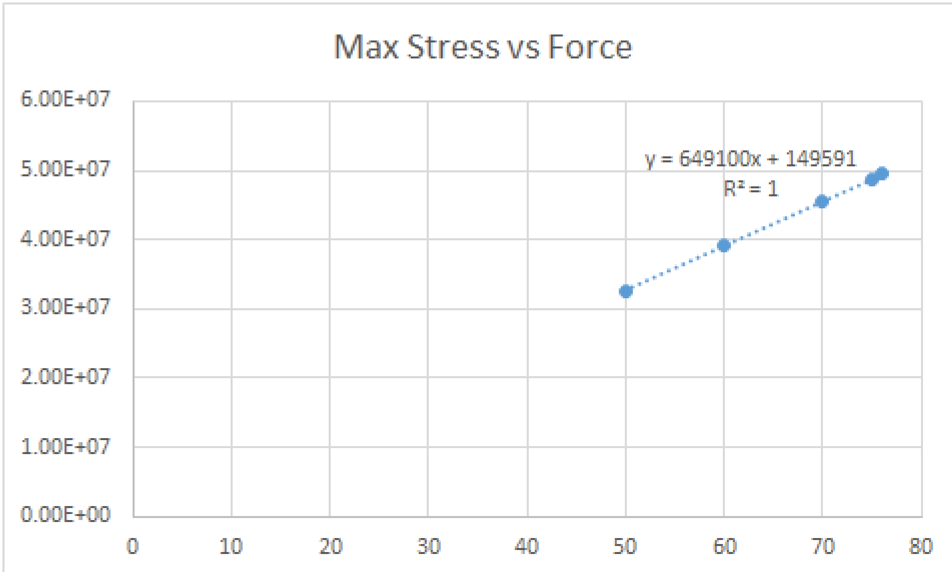


Figure 13: Max Stress vs Force for Cutout Wrench

Part 3: Mesh Effects

Regular

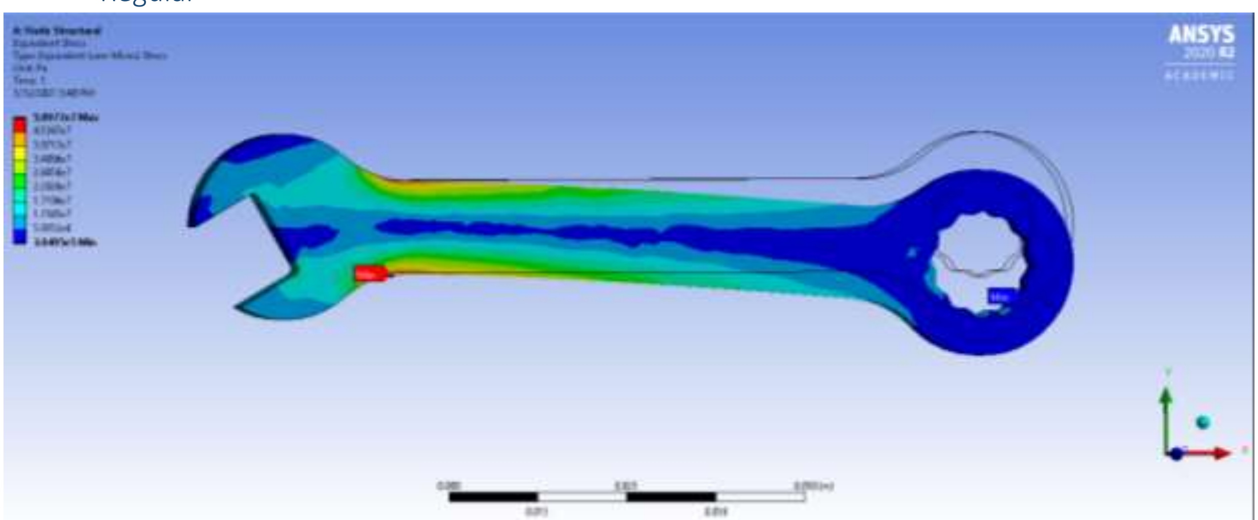


Figure 14: Von Mises Hex Mesh with Load 83 N for Simple Wrench

The wrench experiences a maximum stress of 5.0977E7 Pa at the apex of the bottom curve by the bolt.

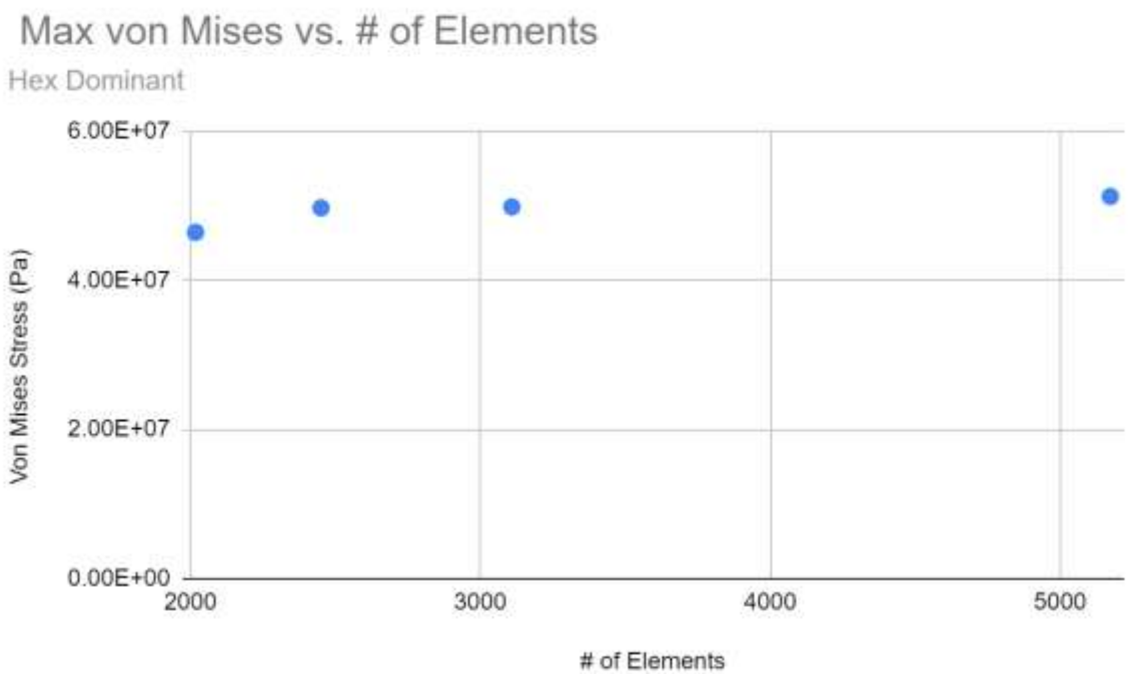


Figure 15: Convergence with P = 83 using Hex Dominant of Simple Wrench

At element size 5.00E-03 m, we had 2018 total elements with a max stress of 4.64E+07 Pa. At element size 1.00E-03 m, we had 2451 total elements with a max stress of 4.97E+07 Pa. At element size 8.00E-04 m, we had 3109 total elements with a max stress of 4.99E+07 Pa. At element size 5.00E-04 m we had 5173 total elements with a max stress of 5.13E+07 Pa.

Cutout wrench

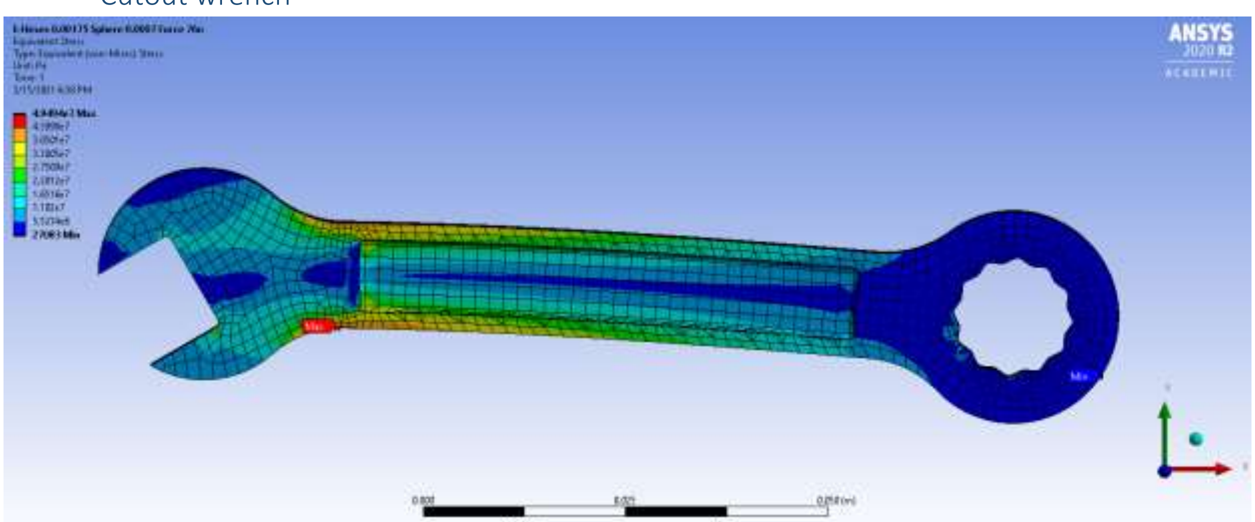


Figure 16: Cutout Wrench With Hex Mesh

Using 50N load, the max stress with the best tetrahedral model created a maximum stress of 3.27E+07 Pa, compared to 3.26E+07 Pa using the hex model. This is a 0.13% error while using fewer elements than the tetrahedral model.

Discussion

Part 1: Hand Calculations

The hand calculations produced an estimated max load of 74.6N. This is an extremely simplified calculation that doesn't take into account other geometry factors. The load could be low due to not properly taking into account how the location and distribution of the applied force affects the material.

Part 2: Mesh Convergence Study

Regular

In Figure 6, we see our data plateauing to 50 MPa. Even when increasing the total amount of elements from 5401 to 7445, there is only a 0.02 MPa difference. Unfortunately we could not increase the number of elements much higher due to constraints of using the student version. Based on our Ansys simulation, we got a maximum load of P = 83 N, as this was the closest to 50 MPa we could get with our mesh. Through our hand calculations, we expected a maximum load of P = 74.52 N. This gives us a percent error of 11.38% and is well within range of our expected P value.

Cutout wrench

Complicated geometry in the cutout wrench resulted in no convergence being found. The student version of ANSYS had greatly limited the number of elements that could be generated. Hexahedral elements don't handle curves well, so many elements were needed in order to generate a usable model. There was only one element size for hexes that produces a usable model without exceeding the student limitations.

Part 3: Deformation and Stresses

Regular

Using P = 83 N for our load, we get a max deformation of 0.3562 mm that was at the furthest point away from where the bolt would be fixed. This is what we expected since it is in the location where our force is acting. For our von Mises stress, we got a maximum value of 4.9935E7 Pa. Compared to our yield strength, our model's max stress has a percent error of 0.13%. This maximum was at the apex of the bottom curve, between the open end of the wrench and the rectangular segment. This location makes sense as it is part of the segment where the cross-sectional area is changing. If we think of the left half of the wrench as a fillet, this location would have the greatest stress concentration, meaning that it would see the most stress.

Cutout wrench

A load of P = 76 N created a deformation of 0.39739 mm and a stress of 4.95E+07 which is extremely close to the yield stress of the steel. Further refinement could be tested to find a more accurate and precise load, but would be excessive given other computing limitations. The maximum stress was located at the underside neck of the wrench, directly in the center. This location is where we expected as it is a fillet. The location is the same in both wrenches. The load on the notched wrench is lower than the simple wrench which also makes sense since there is less material and thickness in the notched wrench.

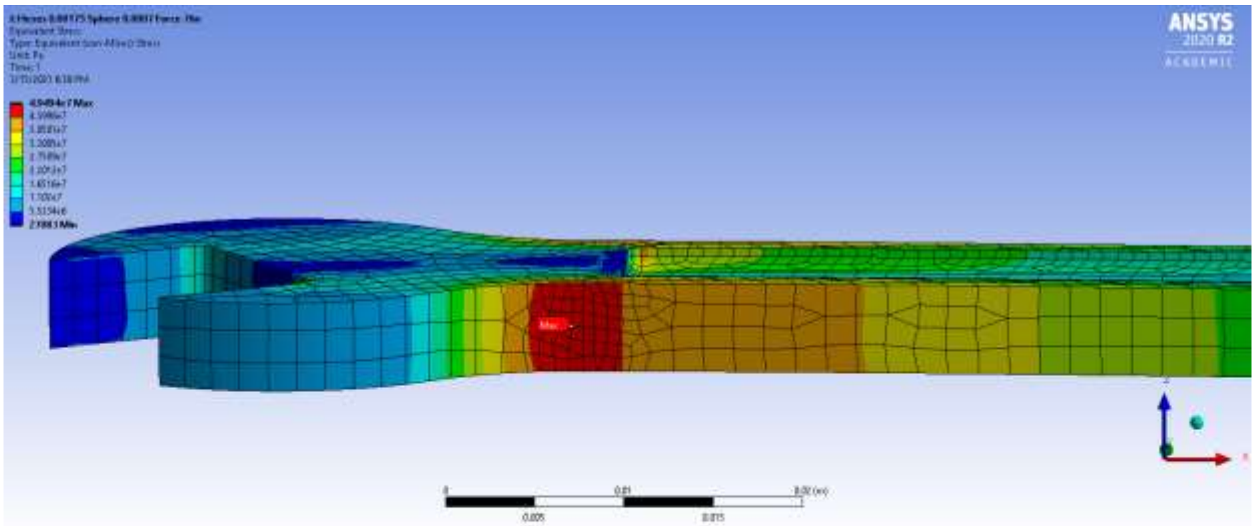


Figure 17: Location of maximum stress

Part 4: Mesh Effects

Regular

When trying to apply a hex dominant mesh with the same element sizing as our study with tets, we ran into multiple errors. This caused us to change the mesh and essentially guess at which values would the program actually allow, due to limitations of the student version. This makes our comparison between tets and hexs not totally comparable. With the closest mesh, as shown in Figure 14, we got a maximum stress of 5.0977E7 Pa. This element type is more accurate to our calculated model. With a load greater than 74 N, we should expect a stress greater than our yield. At a load of 83 N, tets resulted in a stress just below the yield while hexs resulted in a stress above the yield. When comparing the convergence graphs for tets, Figure 6, and hexs, Figure 15, we see that the hex model resulted in greater stress values overall. The hex method also appears to converge more nicely than the tet, as the tet model had more variation between the resulting values. This change in element types resulted in a change in our convergence. While the tet converges to 50 MPa, the hex model converges to somewhere between 50 MPa and 60 MPa. Unfortunately, we could not achieve higher stress or number of element values due to the limitations from the student version and failures to form hex meshes.

Cutout wrench

The cutout wrench has more complex geometry and is exceedingly limited with the hex elements. Due to hex elements having a harder time with curved geometry, we needed more of them to make a good model. Thus due to limitations we were only able to test one mesh size. However, compared to the max tetrahedral mesh this produced an error of only 0.13% less max stress. This was deemed good enough to use for all of the max load simulations.

Conclusion

The simple wrench had a good mesh analysis and produced a max deformation of 0.3562 mm and max stress of 4.9935E7 Pa at a load of 83N. This was comparable to the cutout wrench which had a maximum deformation of 0.39739 mm and a max stress of 4.95E+07 Pa at a load of 76 N. The maximum force was located at the center of the underside neck as seen in figure 5. The hex mesh produced a slightly higher load in the simple wrench than the tet mesh did. This however was not replicable in the cutout wrench due to more complex geometry. Because the cutout wrench withstood a force close to the simple wrench, we recommend the cutout wrench for its weight and cost savings.