ME130L – COMPUTER APPLICATIONS FOR MECHANICAL ENGINEERING

PLATE 3 (PRACTICE EXERCISE) - STATIC STRESS ANALYSIS OF KNUCKLE JOINT

2020168321

BERGONIO, JOSE LUIS ANINO.

SUBMITTED TO: **ENGR. ALEXANDER HAMITLON ATIENZA**

SUBMITTED ON: **NOVEMBER 5, 2022**

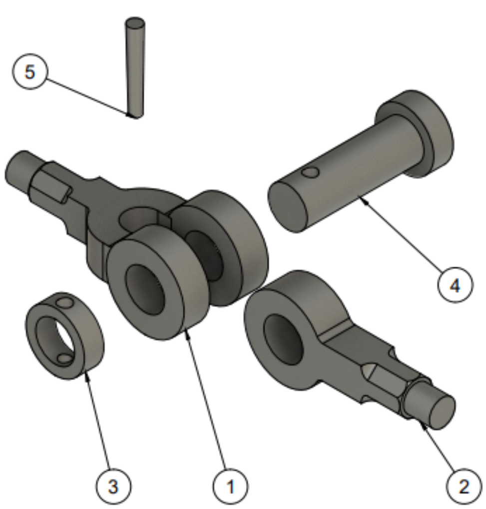
B38

MCL
Malayan Colleges Laguna
A MAPÚA SCHOOL

INTRODUCTION

The calculations for a basic stress analysis take for granted that the components are flawless, have an identical cross-section, and are free of any abnormalities. In reality, however, almost all of the components used in engineering need to have some kind of modification to its section or shape. The joints play a very important part in the mechanical and automotive fields. The joints that are utilized can either be temporary or permanent, depending on the requirements of the application. Temporary joints, such as bolted joints, cotter joints, sleeve cotter joints, universal joints, and knuckle joints, are typically used in applications involving the transmission of power or the transfer of motion (J & Vanka, 2015). The knuckle joint is a type of joint that is utilized in the steering system in between the steering rod and the pinion of the steering gear. Due to the fact that the line of the action axis of both the mechanical parts are intersecting and lying-in different planes, the knuckle joint is the only joint that can be utilized in this particular application. It is imperative that the production technology be flexible in order to achieve the highest possible levels of plant productivity.

In order to connect two rods that are going to be subjected to tensile force, a knuckle joint is utilized. This junction allows for the rods to be misaligned at an angle and has the potential to take compressive force if it is directed. In the construction of bridges, these joints are utilized for a variety of different kinds of connections, such as tie rods and tension links. In this configuration, one of the rods has an eye at the end of the rod, and the other end is forked, with eyes at both ends of the fork. A collar and a split pin are used to fasten a pin that has been placed through the rod-end and fork-end eyes (N & R, 2015). This pin is known as a knuckle pin. Because the breaking of a knuckle joint could result in an accident, it is essential that these joints be designed to remain intact even when subjected to tension. The efficient design of a mechanical device or assembly requires the predictive knowledge of its behavior in working state. It has become vital for the designer to be aware of the forces and stress that are created while the device is in operation (D & K, 2015).



Parts List		
Item	Part Number	Material
1	Fork End	Steel
2	Eye End	Steel
3	Coller	Steel
4	Pin	Steel
5	Taper Pin	Steel

Figure 1. Components of Knuckle Joint

ASSEMBLY AND DESIGN

Symbol	Element	Result
P	Tensile load acting on the rod	50 KN
D	Diameter of each rod	20 mm
D_1	Enlarged diameter of each rod	25 mm
d	Diameter of knuckle pin	30 mm
d_0	Outside diameter of eye or fork	60 mm
d_1	Diameter of pin head	45 mm
a	Thickness of eye end of rod A	20 mm
b	Thickness of eye end of rod B	25 mm
x	Distance of the center of fork radius R from the eye	10 mm
σ_t	Tensile Stress	$170\ \frac{N}{mm^2}$
σ_c	Crushing Stress	$170\ \frac{N}{mm^2}$
τ	Shear Stress	$85\ \frac{N}{mm^2}$

Table 1.1. Calculation of Dimension prior to the given forces, factor of safety, and yield stress.

$\sigma_{t\ eye} < \sigma_t$	Check the failure of maximum tensile stress in eye	$66.67\ \frac{N}{mm^2} < 170\ \frac{N}{mm^2}$
$\sigma_{c\ eye} < \sigma_c$	Check the failure of maximum crushing stress in eye	$66.67\ \frac{N}{mm^2} < 170\ \frac{N}{mm^2}$
$\tau_{\ eye} < \tau$	Check the failure of maximum shear stress in eye	$66.67\ \frac{N}{mm^2} < 85\ \frac{N}{mm^2}$
$\sigma_{t\ fork} < \sigma_t$	Check the failure of maximum tensile stress in fork	$55.56\ \frac{N}{mm^2} < 170\ \frac{N}{mm^2}$
$\sigma_{c\ fork} < \sigma_c$	Check the failure of maximum crushing stress in fork	$55.56\ \frac{N}{mm^2} < 170\ \frac{N}{mm^2}$
$\tau_{\ fork} < \tau$	Check the failure of maximum shear stress in fork	$55.56\ \frac{N}{mm^2} < 85\ \frac{N}{mm^2}$

Table 1.2. Failure Stress Analysis of the Knuckle Joint

It is necessary to design a knuckle joint that is capable of withstanding an axial tensile force of 50 KN (which is derived from the initial letter of my surname, B). The rods and pin of this knuckle joint must be fabricated from the material Steel-Cast (Syt = 500 MPa), and it must have a factor of safety of 3.

Based on the information in Table 1.2, it appears that the intended knuckle joint can be operated without risk. According to the findings, the failure analysis for the eye and fork end that was investigated found that the tensile stress, shearing, and crushing were acceptable within the criteria. This allowed the knuckle joints to obtain the safety factor that they required, which was the allowable shear, tension, and crushing stresses.

MATERIAL SELECTION

For the purpose of design analysis, it is extremely helpful to be aware of and correctly declare the value of the material property. The results are variable due to the fact that various kinds of materials have varying densities. The Young's modulus of a material is a numerical constant that shows the elasticity of a solid and its power to endure changes when subjected to strain in a particular direction. When a material has a higher Young's modulus, it is more resistant to deformation and requires more force to do so. The strength and type of how a material structure deforms in response to a certain restriction can be described using Poisson's ratio and young's modulus, respectively. The yield strength and the tensile strength of a material are two other crucial parameters that define when the material loses its elastic nature and the greatest stress that a material is capable of enduring, respectively. (Nipun, 2015).

Density	7.85E-06 kg / mm^3
Young's Modulus	210000 MPa
Poisson's Ratio	0.3
Yield Strength	207 MPa
Ultimate Tensile Strength	345 MPa
Thermal Conductivity	0.056 W / (mm C)
Thermal Expansion Coefficient	1.2E-05 / C
Specific Heat	480 J / (kg C)

Table 2. Mechanical Properties of Steel-Cast

We are able to validate that the mechanical parameters offered by the simulation report are almost identical to the mechanical qualities that stainless steel or structural steel possesses. Because of this, we are able to include steel-cast in the branch of stainless steel that is resistant to corrosion, has a high tensile strength, is highly durable, and possesses a number of other positive properties that are helpful in the building of the knuckle joint.

We are able to verify, with the assistance of the journal report of performance analysis created by (D, M, & A, 2018), that the mechanical properties of steel cast are almost identical to what is indicated in the figure. Nevertheless, there was a slight difference in some properties, most notably in ultimate tensile strength.

Mechanical property	Stainless Steel	Cast Iron
Density (Kg/m^3)	7850	7200
Coefficient of Thermal Expansion (1/°C)	1.7e-005	1.7e-005
Specific Heat (J/kg/°C)	480	490
Thermal Conductivity (W/m°C)	15.1	53.3
Compressive Yield Strength (MPa)	207	970
Tensile Yield Strength (MPa)	207	190
Ultimate Tensile Strength (MPa)	586	276
Reference Temperature (° C)	22	22
Young's Modulus (Pa)	1.93e+011	1e+006
Poisson's Ratio	0.31	0.23
Bulk Modulus (Pa)	1.693e+011	6.1728e+005
Shear Modulus (Pa)	7.366e+010	4.065e+005

Figure 2. Mechanical Properties of Stainless Steel

MESH GENERATION

Following the creation of the automatic contacts between all of the pieces at a distance of 0.10 mm, we are able to compute the mesh for the simulation model depending on the settings that have been provided. The number of nodes and elements that make up the created mesh are listed in the table that follows. Within the mesh, there are a total of ***17,399 nodes and 9799 elements.*** R&D World states that the most fundamental and accurate method for determining the quality of a mesh is to refine the mesh until a critical result, such as the maximum stress in a particular region, converges. This means that the result does not vary considerably while the mesh is refined (R&D Editors, 2015). The accuracy of the overall model depends heavily on the quality of the mesh, which can ultimately determine whether or not it is possible to accurately anticipate whether or not a design will be unsuccessful. The selection of such a large number of elements was made with the intention of increasing the complexity of our component, which in turn enables us to obtain more genuine findings based on a more advanced technique of fatigue life calculation

Average Element Size (% of model size)	
Solids	10
Scale Mesh Size Per Part	No
Average Element Size (absolute value)	-
Element Order	Parabolic
Create Curved Mesh Elements	Yes
Max. Turn Angle on Curves (Deg.)	60
Max. Adjacent Mesh Size Ratio	1.5
Max. Aspect Ratio	10
Minimum Element Size (% of average size)	20

Table 3. Mesh Properties

Number of Refinement Steps	0
Results Convergence Tolerance (%)	20
Portion of Elements to Refine (%)	10
Results for Baseline Accuracy	Von Mises Stress

Table 4. Adaptive Mesh Refinement

Type	Nodes	Elements
Solids	**17399**	**9799**

Table 5. Values of Nodes and Elements of the mesh

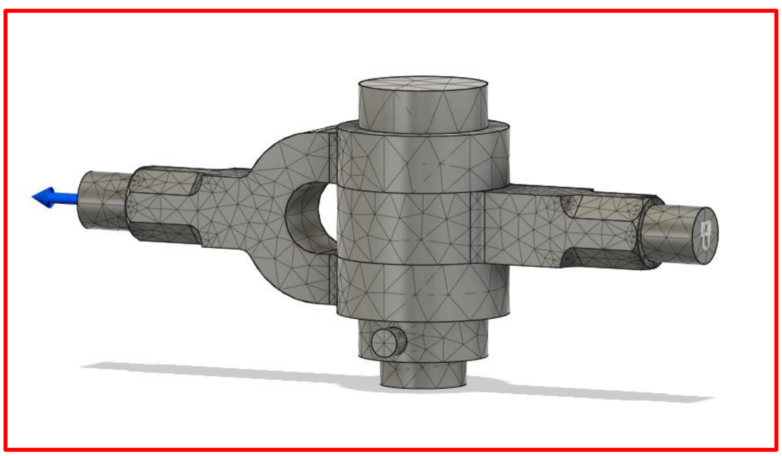


Figure 3. Mesh Contour of Knuckle Joint with Boundary Conditions

CONTACTS

When describing the connections between different assembly components in simulation research, you can either use Automatic contacts or Manual contacts. The interaction between pieces in an assembly can be described using the term "simulation contacts." You have the option of either automatically or manually specifying contact conditions. When a static stress or thermal stress analysis is performed, automatic contacts are generated in accordance with the default contact type and contact tolerance set for the analysis.

When the material that will travel through simulations has a body of two or more, contacts are created with it. Because the knuckle joint in question has the five sections that have been discussed, it is essential to have automatic contacts in order to satiate the safety factor. The list is broken down into the following categories according to contact set and contact type in the following table:

Name
[S] Bonded1 [Pin:1||Taper Pin:1]
[S] Bonded2 [Coller:1||Pin:1]
[S] Bonded3 [Coller:1||Taper Pin:1]
[S] Bonded4 [Coller:1||Taper Pin:1]
[S] Bonded5 [Eye End:1||Pin:1]
[S] Bonded6 [Fork End:1||Eye End:1]
[S] Bonded7 [Fork End:1||Pin:1]
[S] Bonded8 [Fork End:1||Eye End:1]
[S] Bonded9 [Fork End:1||Eye End:1]
[S] Bonded10 [Fork End:1||Coller:1]
[S] Bonded11 [Fork End:1||Pin:1]
[S] Bonded12 [Fork End:1||Pin:1]

Table 4. Automatics Contacts Summarization

We need to make sure that any problems that could arise with the study are resolved so that we can successfully set up the research project. One of the problems I came across was that the model had five fully unconstrained groups in the load case. This refers to the contact tolerance of the five body parts of the model, and because the software reads the model as a single unit, the model had to be assembled before the simulation study could begin. I established a contact detection tolerance of around 0.10 millimeters for solids and did not eliminate any stiff body modes.

Contact Tolerance	0.1 mm
Remove Rigid Body Modes	No

Table 5. General Optimization of Contact Tolerance

BOUNDARY CONDITIONS

Boundary Condition 1:

The knuckle joint constraint is fixed at one end on the fork side. After the meshing is complete, constraints like fixed support and forces are imposed to the model; this is a highly significant stage that serves as a major requirement for the analysis. (Talikoti et al., 2016).

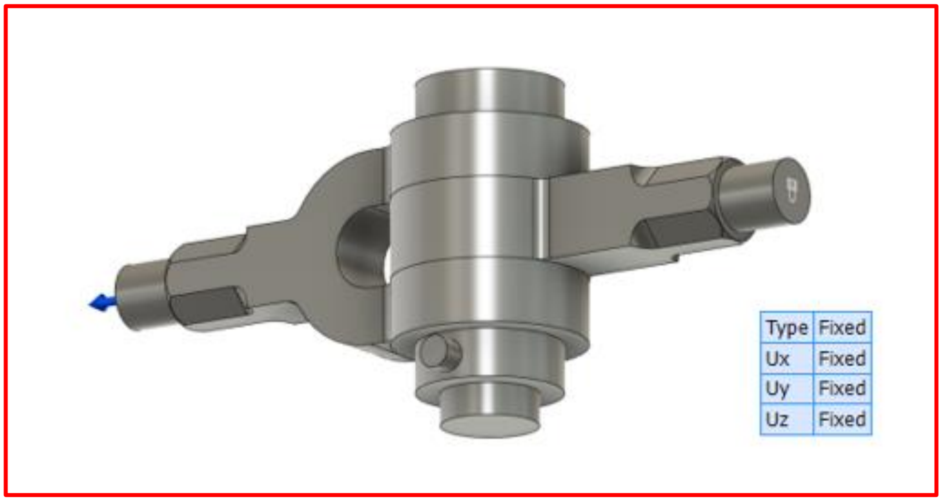


Figure 4. Boundary Condition 1 with constraints and selected entities

Boundary Condition 2:

The knuckle joint axial tensile force of 15000 N is applied on the other end.

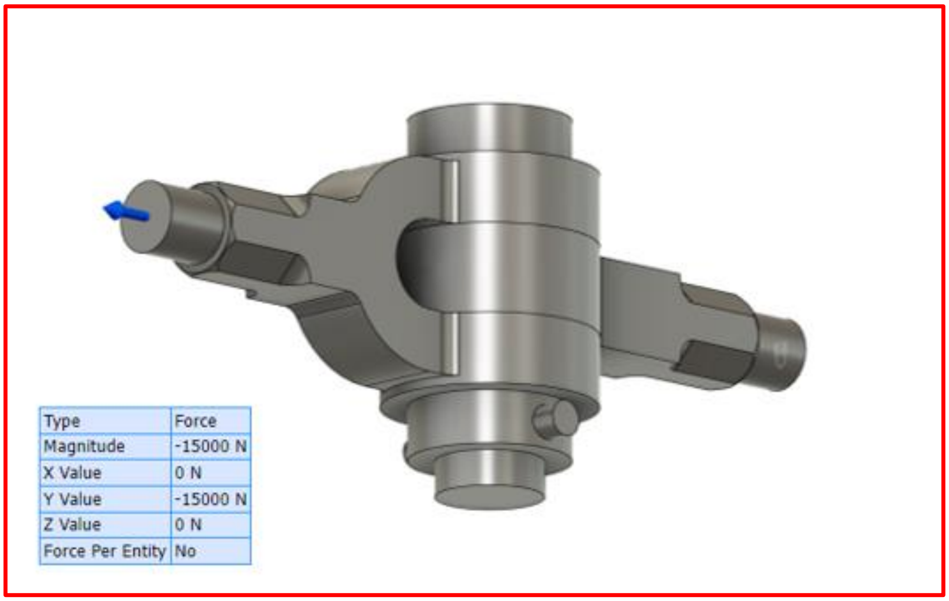


Figure 5. Boundary Condition 2 with force and selected entities

STATIC STRESS ANALYSIS

In static structural analysis, the stresses, displacements, strains, and forces in structures induced by a load that does not induce significant inertia and damping effects are calculated using a load that does not move the structure much. It is anticipated that the loading and response conditions will remain constant; more specifically, it is assumed that both the loads and the structure's response would change slowly in relation to time.

In this simulation, a force of -15000 N (because the fork end is facing the negative side of the x-axis) pushes to the left, while a force of -15000 N (because the eye end is facing the positive side of the x-axis) pulls against it from below. Such phenomena are observable in the actual world, and can be seen in the context of a pull contact with a stationary object.

The minimum or maximum parameters designate the locations on the part that exhibit the highest and lowest result values with the actual value. These parameters identify the areas on the component that exhibit the highest and lowest result values. In order to make advantage of the maximum and lowest annotations, you first need to obtain simulation results that are legitimate. In this column, the position of the maximum and minimum parameter values is displayed in contour map form for each of the figures that follow:

Name	Minimum	Maximum
Safety Factor		
Safety Factor (Per Body)	3.144	15
Stress		
Von Mises	0.03233 MPa	65.84 MPa
1st Principal	-6.065 MPa	76.42 MPa
3rd Principal	-32.3 MPa	19.07 MPa
Normal XX	-11.19 MPa	30.18 MPa
Normal YY	-13.6 MPa	65.38 MPa
Normal ZZ	-27.33 MPa	30.64 MPa
Shear XY	-20.02 MPa	22.33 MPa
Shear YZ	-29.61 MPa	29.43 MPa
Shear ZX	-8.16 MPa	7.784 MPa
Displacement		
Total	0 mm	0.03078 mm
X	-0.001046 mm	6.471E-04 mm
Y	-0.0307 mm	0 mm
Z	-0.002484 mm	6.604E-04 mm
Reaction Force		
Total	0 N	1040 N
X	-207.5 N	195.4 N
Y	-53.96 N	1030 N
Z	-175.8 N	197.4 N
Strain		
Equivalent	1.919E-07	4.612E-04
1st Principal	9.662E-08	4.895E-04
3rd Principal	-2.739E-04	-2.142E-07
Normal XX	-8.711E-05	7.588E-05
Normal YY	-3.016E-05	2.802E-04
Normal ZZ	-1.327E-04	8.536E-05
Shear XY	-2.478E-04	2.765E-04
Shear YZ	-3.666E-04	3.644E-04
Shear ZX	-1.01E-04	9.637E-05
Contact Pressure		
Total	0 MPa	30.25 MPa
X	-2.583 MPa	2.987 MPa
Y	-13.17 MPa	14.88 MPa
Z	-26.33 MPa	19.71 MPa
Contact Force		
Total	0 N	820.7 N
X	-116 N	76.97 N
Y	-817.9 N	518.2 N
Z	-406.1 N	339 N

Table 6. The summary of the results obtained from the simulation

SAFETY OF FACTOR

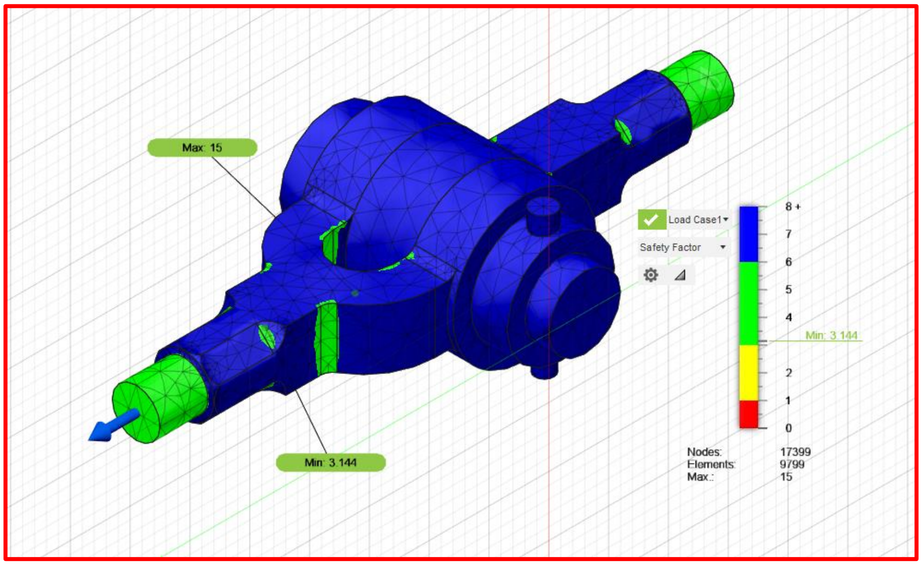


Figure 6. Safety of Factor with Min/Max Parameters

The maximum amount of stress that may be placed on any object used in building is determined by the type of material that was used to make it. If the safety factor of the model does not exceed the specified safety of 3, the process of simulation cannot be continued. Fortunately, the design was able to pass the mandatory safety factor with a score of 3.144, which was determined to occur in the joint of the fork-end portion. According to this, ***the model can withstand fifteen times the load*** in the blue part of the model; nevertheless, the green sections dictate that it is most likely to encounter the yielding part. This is the case because the green parts of the model.

The majority of the model's components are represented by the color blue in the Safety Factor, and the model's overall safety measure is 15, which indicates that the model is ***robust***. The Safety Factor is a useful metric that may be used to measure how well a model matches the function for which it was designed. A scale ranging from 0 to 15 has been assigned to the graph legend for the results of the Safety Factor that can be found in the lower right-hand corner of figure 6. ***The model was required to have a minimum safety factor of between three and six, but this range varied depending on the requirements***. For the purpose of determining conformance to the design safety factor, the most prominent anticipated loading situation is considered (Autocad, n.d).

To summarize, considering that there is a force of 15000 Newtons acting on one side of the model, it is ***still stable***. When a model is subjected to the pressures that are being applied, the Safety Factor reveals whether the model is able to withstand those forces without breaking, bending, or collapsing.

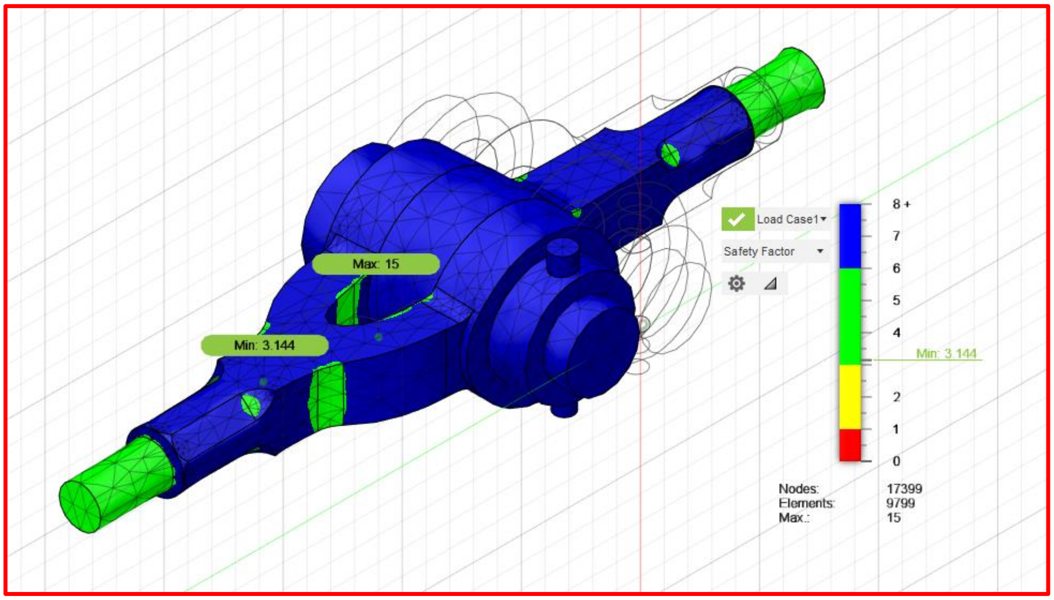


Figure 7. Total Deformation Results

The Fusion 360 software contains a one-of-a-kind module that can measure the maximum deformation as a percentage of the model size, such as 2.5% all the way up to 25%, to display the model's actual deformation, also known as a change in the length of the joints. Figure 7 presents a typical graphic that illustrates the deformation that occurs in each individual component of the knuckle joint. As can be seen, the greatest amount of deformation takes place at the two ends of the joints, whereas the pin section saw the least amount of elongation.

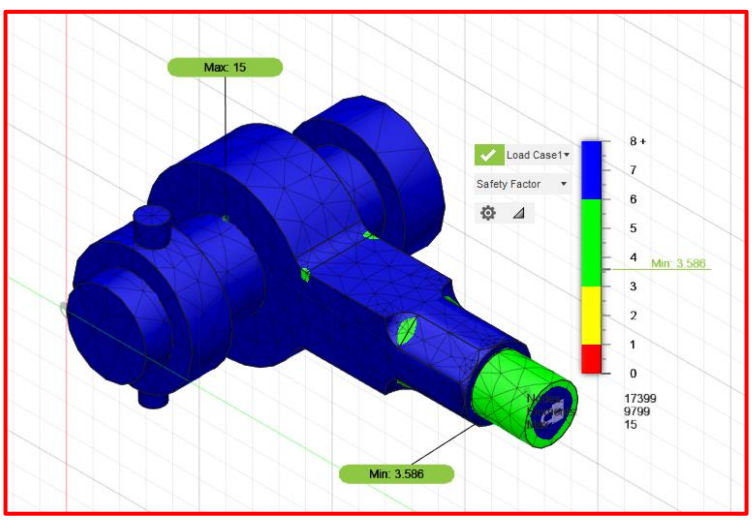


Figure 8. Safety of Factor (Eye-End View)

In figure 8, the fork-end is purposefully left out so that it can be demonstrated that the highest possible safety factor happens at the pin, while the lowest possible safety factor occurs in the joint of the eye-pin. This shows that the location of the greatest parameter is ***most likely not to be distorted when elongation occurs in the central area of the model*** in which there is not a single shade of green that can be observed. This is because there is not a single shade of green seen in this part of the model. However, if we look at figures 7 and 8, we can see that there is a region around the pin portion at each end that has the lowest possible safety factor. This region is surrounded by a shade of green, ***which indicates that part is most likely to experience the yielding.***

STRESS (VON MISES)

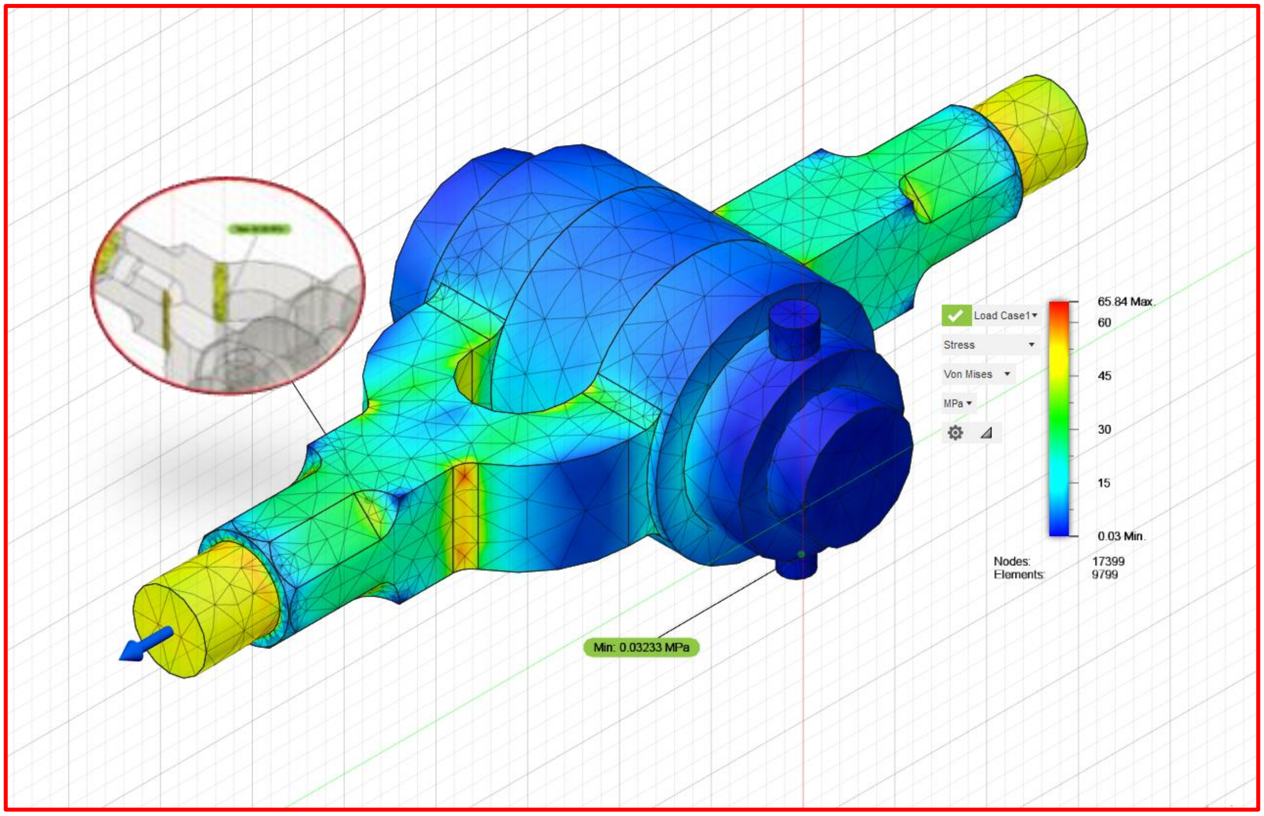


Figure 9. Equivalent Von-Misses Stresses

The Von Mises stress is a value that is utilized in the process of determining whether or not a certain material will yield or fracture. The majority of the time, it is applied to ductile materials like metals (SIMSCALE, 2022). According to the information presented in the figure that is found above, the point at which the fork end's midspan experiences the greatest amount of bending moment is also the one at which the coller experiences the least amount of bending stress. To determine the maximum (Von-Mises) stress and deformation of the knuckle joint. The constrains needs to be carefully applied based on the prevailing conditions (Manesh P. Sharma, 2014).

According to Midas Bridge, the failure can be checked by Von-Mises Stress by applying Von-Misses yield criterion in order to determine the yield. This is how the yield is determined. It is verified that the Von Misses stresses are well within the elastic yield limit by performing checks on them. When we look at table 1 (Material Properties of the Steel), we see that the knuckle joint has a Poisson's Ratio of 0.3 and a Yield Strength of 207 MPa. This allows us to conclude that the knuckle joint is quite durable. Because the maximum bending stress parameter in the design, ***which is calculated to be 65.84 MPa, is lower than the yield strength value, which is calculated to be 207 MPa***, we are able to validate that the design is **SAFE**.

Therefore, it is probably a good idea to mention that the minimal bending stress is also noted as a SAFE. This is because it is most likely true. The stress diagram makes it abundantly evident that the support is the location where the most significant strains are generated.

STRESS (1^{ST} AND 3^{RD} PRINCIPAL)

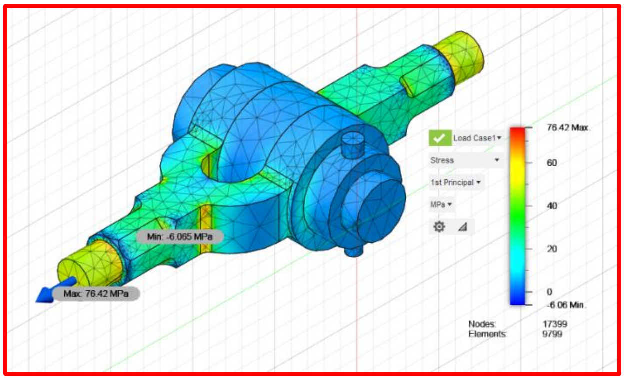


Figure 9. Maximum and Minimum 1^{st} Principal Stress

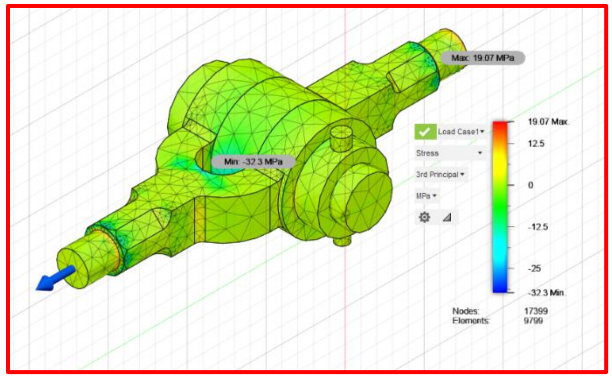


Figure 10. Maximum and Minimum 3^{rd} Principal Stress

The theory of maximum main stress is helpful for analyzing brittle materials (Ghosh, 2009). The simulation also provides additional information to see results of stresses via on its first and third principal. It is stated that these stresses can also validate if the design is safe if von-misses stresses is not provided. It is possible to determine stresses along axes other than the standard X, Y, and Z. By shifting the frame of reference, we may cancel out the shear stresses and be left with only the normal stresses.

1^{st} Principal Stress	**3^{rd} Principal Stress**
It exhibits the greatest amount of tensile stress and is perpendicular to the surface of the plane.	This line is perpendicular to the surface of the plane and reveals the least amount of compressive stress.
It provides the highest possible level of stress on the system.	It provides the system with the least amount of stress possible.

Figure 9 illustrates the greatest primary stress, which indicates the value of the stress that is normal to the plane. This is due to the fact that the shear stress is zero in this plane. It has a stress value of 76.42 MPa, which indicates that it is predominantly tensile. When compared to the value of Von-Mises, which is 65.84, we are able to determine that the value of the first principal is, in fact, slightly greater. This leads one to believe that the state of pure tension exists anytime the most stressful condition is present (MDME, n.d.). Since shear failure is possible, maximum main stress theory is not suited for ensuring the safe design of machine components made of ductile material, as stated by Bansal (2019). Because steel-cast is a material that is both hard and brittle, ***the design of the knuckle joint is regarded as being appropriate and safe in light of the fact that the material that it is formed of is hard.***

If we look at figure 10, we can see that the surface that is exposed to the greatest amount of external force has a primary stress that is around -32.3 MPa (a negative sign indicating that the force is compressive). In this scenario, the major stress that has the most negative value also has the biggest magnitude; as a result, ***the design is safe when viewed from the surface that is exposed to the environment.***

For the stress on normal XX, the entire contoured area of the fork-end indicates a ***significant amount of strain that leans toward yellow hues*** at a value of 30.18 MPa. The strains are still at their greatest, and the peaks on this graph went from a shade of blue to a shade of blue, ***suggesting a very low amount of stress of -11.19 MPa.***

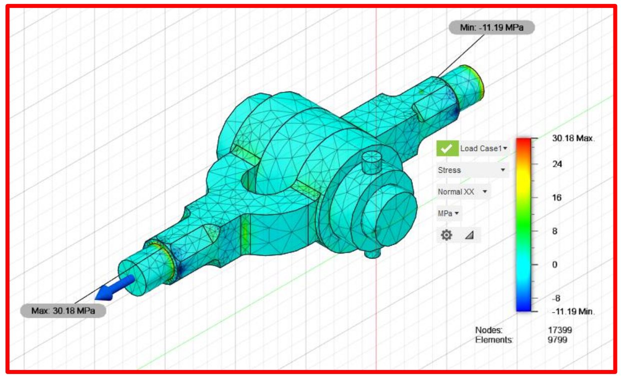


Figure 11. Maximum and Minimum Normal XX Stress

Regarding the stress on the normal YY, we are able to determine that the value of -13.6 MPa indicates that it has ***a minimum stress in the blue colors*** that occupy the center regions of the model. This can be deduced from the fact that the model has blue hues. On the other hand, the point along the y-axis ***where there is a load equal to 15000 Newtons causes the stress to peak at 65.83 MPa.***

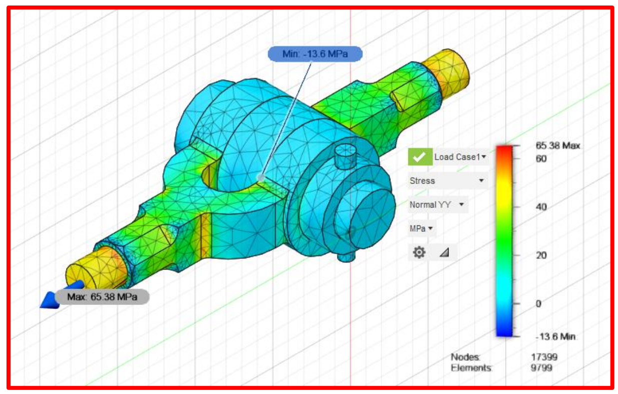


Figure 12. Maximum and Minimum Normal YY Stress

As the progression of the normal xx, yy, zz continues, we are able to identify that the shades of the color range from the xx, which primarily consists of blue hues, to the zz, which occupies the majority of its parts with a shade of green that tends to yellow hues. In between these two extremes is the yy, which primarily consists of yellow hues. The normal ZZ stresses have a minimum that is around -27.33 MPa and a high that is approximately 30.64 MPa. This would imply that ***the model is occupying a greater amount of stress which is fairly uniform until it gets***

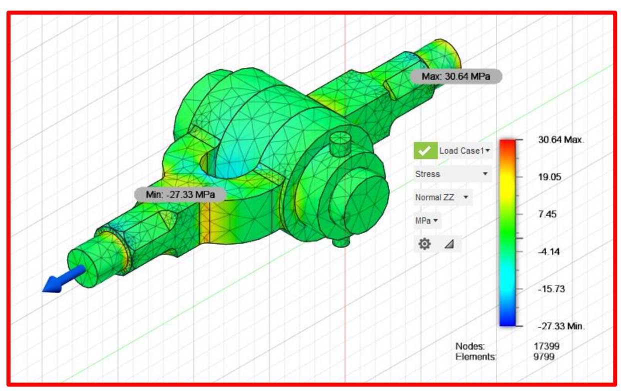


Figure 13. Maximum and Minimum Normal ZZ Stress

close to the restraint in this particular stress section in comparison to the other stress sections that have been reviewed.

The coloration of the graphic shows that the total stress for shear XY is a shade that is somewhere between yellow and green. This would ***imply that the stress on the X plane that is caused by a force acting in the Y direction occupies stress on most of the X plane's*** components with a range of -20.02 MPa to 22.33 MPa.

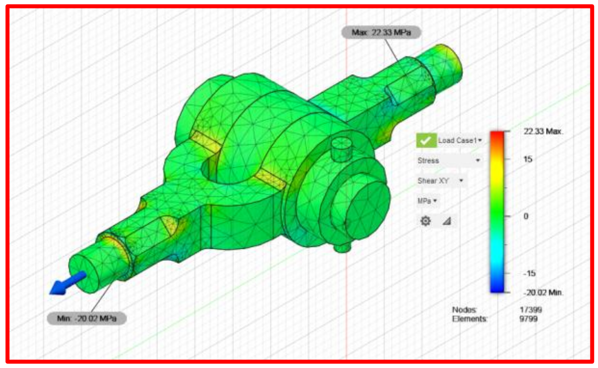


Figure 14. Maximum and Minimum in Shear Stress XY

When viewed from the perspective of the shear stress XY, it appears to be the same because the stress is located on the opposite side of the eye in the XY. In this particular instance, it is situated on the fork, which is the region where the load resides, and it has a stress range of around -29.61 MPa to 29.43 MPa. This would imply that ***the Z-plane plays a significant role in determining where the stresses are located.***

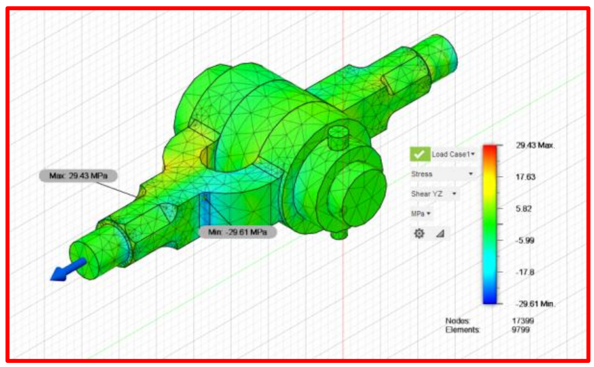


Figure 15. Maximum and Minimum in Shear Stress YZ

The stresses are located on the contour part of the fork-end with a range of 7.784 MPa to -8.16 MPa. In situations analogous to those seen on other coupled planes, ***the range stresses are essentially identical to the absolute value of one another***. In this shear stress ZX, the stresses are located in the load where it resides.

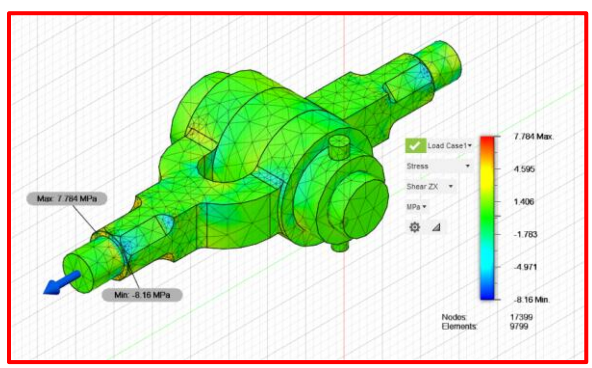


Figure 16. Maximum and Minimum in Shear Stress ZX

DISPLACEMENT

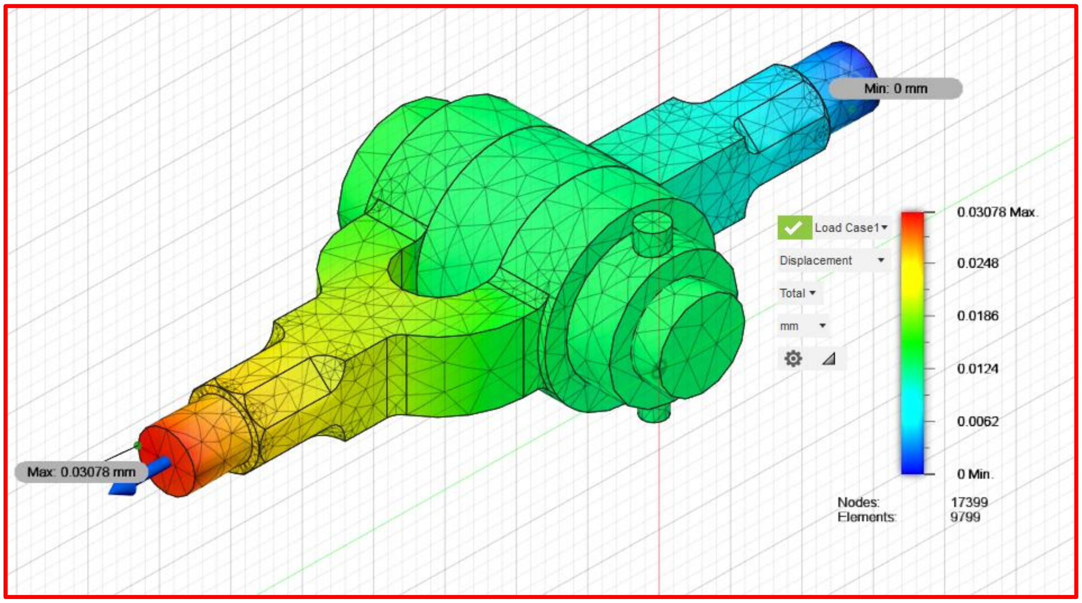


Figure 17. Equivalent Displacement of Knuckle Joint

A vector quantity known as displacement denotes "how far out of place an object is"; it is the object's overall change in position (including translation, rotation, and scale) (The Physics Classroom, 2018). In the meantime, the simulation has produced a color contour map for the displacement result, which illustrates how far the model has traveled in comparison to where it was initially located (AutoDesk, 2017). The phenomenon of displacement takes place whenever a load or force, such as compression, is applied to an item. Compression takes place as a result of the interaction between the force of 15,000 Newtons coming from the fork end of the model and the response force coming from the eye end.

The three-dimensional model has a maximum displacement of 0.03078 millimeters from the fork-end of the model and a minimum displacement of 0 millimeters from the eye-end of the model, as shown in the image. We are able to reach the conclusion that there is no deformation occurring in the eye-end. The greatest amount of motion occurs in the direction of where the force is being applied directly. Because the structure was designed with fewer holes, the numbers show that the deformation is less likely to occur because there is more space to spread and withstand greater compressional force

We can infer from these numbers that there will be no deformation at the eye end (0 mm) because it is not subjected to the load, it receives a lesser amount of force. Minimal distortion at the force location (0.03078 mm) because it is sturdy and capable of supporting a greater load.

REACTION FORCE

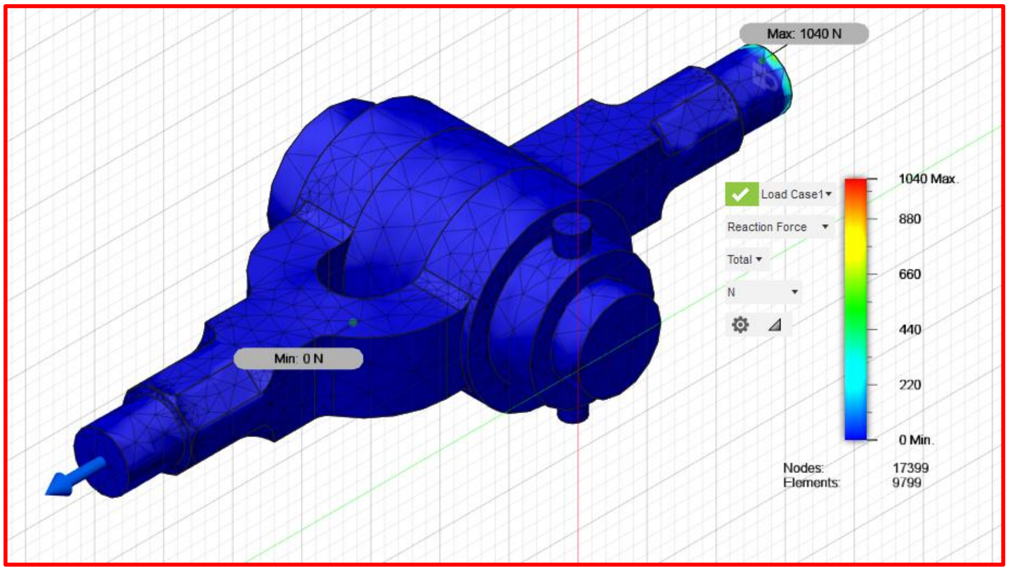


Figure 18. Total Reaction Force Simulation Results

The response force is the amount of force that Fusion 360 determines is necessary to prevent the restricted object from moving. All unrestricted vertices, edges, and faces yield a value of zero (Autodesk, n.d.). Another way to describe the effect of an applied force is as response force. I saw those unrestricted vertices, edges, and faces all yielded a value of zero in the reaction force result. Reaction moments are calculated using the distribution and orientations of the reaction forces (Autodesk, n.d.). Since structural failure occurs when response forces exceed action loads, resulting to model distortion or destruction, this modeling and analysis is crucial.

As what we can see in the figure 12, the reaction force is the opposite of the action force, and their direction is perpendicular to each other. Stated in the boundary section, the action force is the -15000 load on the leftmost of the design, and the expected reaction force can be found at the rightmost of the design thus they are perpendicular to each other. The maximum reaction force is 1040 Newtons which is located to the opposite side where the load which have a minimum of 0 Newtons. The findings here are that eyeballing the process or inferring the results are not reliable in engineering, how important simulation programs to accurately visualize the results.

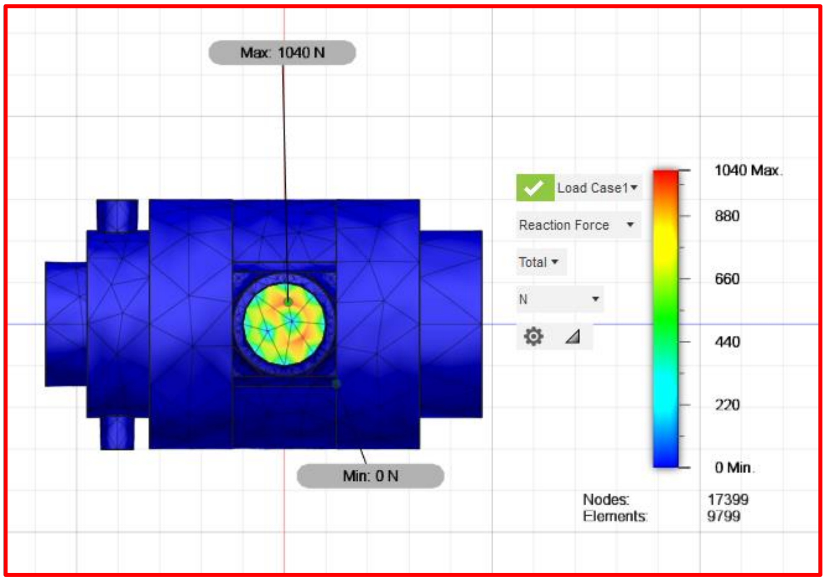


Figure 19. Reaction Force Results (Side View)

The home view of the knuckle joint does not visually capture where the real location of the maximal reaction force is; rather, this area is on the eye-end of the joint. Because it is the component of the eye-end that is responsible for maintaining the force that was applied by the fork-end of the model, the lower portion of the eye-end was subjected to the greatest amount of response force. Therefore, the component of the model that forces against the forces applied from the fork-end will be the bottom area of the eye-end. This will be the part of the model that maintains the object's position even when a force is applied. I've observed that the location of a limitation is the best place to look for the response force.

As we can see in figure 19, the reaction force consumes the base of the bottom half of the eye-end in a consistent manner. This is something that we can determine for ourselves. The region of the semicircle ring with varying intensities of green, yellow, and blue shade was the spot where the largest reaction force was measured. This ring had a color that was somewhere between orange and red.

STRAIN

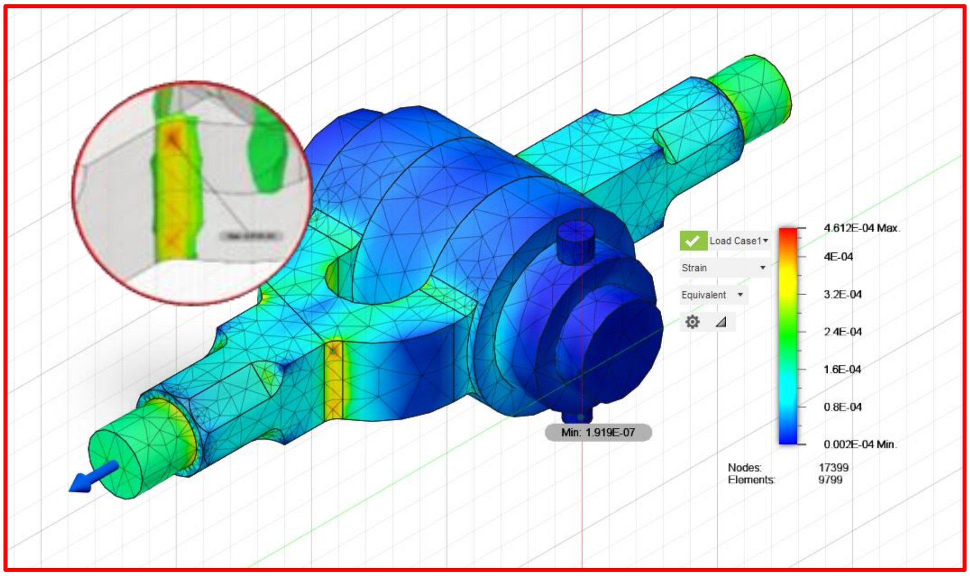


Figure 20. Equivalent Maximum and Minimum Strain

Strain is a non-unitary measurement that indicates how much an object expands or contracts as a result of an applied load. When an object elongates in response to the normal force acting on it, this phenomenon is known as normal strain (Holmes, n.d.). We can observe from the simulation that the greatest amount of strain takes place on the fillet side of the fork-end, with a value of 4.612E-04, and that the pin part of the knuckle joint experiences the least amount of strain, with 1.919E-07. This particular pin placement exhibits the least amount of strain since it has a ***substantially bigger surface to distribute the force that is being applied***. The edge reduced the surface area, which led to the material becoming less resistant to stress. Meanwhile, the fillet side is where the largest strain is placed. This is because ***the edge reduced the surface area***.

It's possible that the strain and stress figures, when seen in different locations***, look visually similar to one another***. Despite the fact that stress and strain are two separate variables, figure 9 shows that the stress is located on the opposite side of the fillet side of the fork-end from where the strain is. We are able to provide evidence in support of the hypothesis that a normal strain takes place whenever an object elongates in reaction to a normal stress (i.e., ***perpendicular to a surface***).

A strain that is positive indicates a tensile strain, whereas a value that is negative indicates a compressive strain. Because of this piece of knowledge, the simulation analysis makes use of an equivalent for the strain section. This is because the load that is applied is the total strain intensity that is created by the conjunction of the six components that make up the strain matrix, and this value is consistently positive.

CONTACT PRESSURE

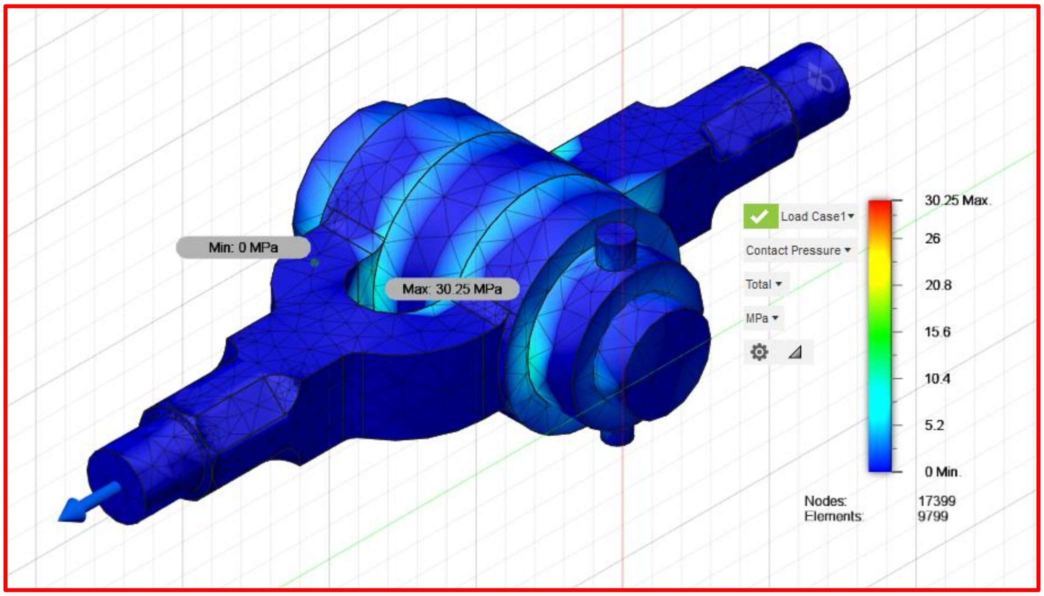


Figure 21. Contact Pressure of Knuckle Joint

The normal load divided by the total contact area (the sum of the front and back areas) gives you the contact pressure. This number is expressed as a ratio. Only in the event of plastic contact is it appropriate to refer to it as the scratch hardness (C & R, 2006). When a separation contact is formed amongst two neighboring portions of a model, results for contact pressure are generated (AutoDesk, 2016). One force is exerted at each node, and this total is then divided by the total surface area around each node.

According to what is presented in the figure, the lowest possible contact pressure is 0 MPa, while the highest possible contact pressure is 30.25 MPa. According to John Holtz, who works at AutoDesk, the contact pressure is calculated and reported at the "centroid," which means that there is only one result per element face (Holts, 2022). In this particular instance, just one set of contact pressure results was acquired from this design, and it was positioned in close proximity to the centroid.

It suggests that the pressure that is operating on one part is the same magnitude but acts in the opposite direction compared to the contact pressure that is acting on the part that is adjacent to it. According to Stirickland and Taylor, it is common knowledge that some contact under excessive contact pressures over the yield limit can result in excess damage, which will exacerbate material failure. This was noted in their paper (M & M, 2014). ***The design has considered safe to use since the maximum contact force of 30.25 MPa is not beyond the yield limit of 207 MPa*** (from table 2).

CONTACT FORCE

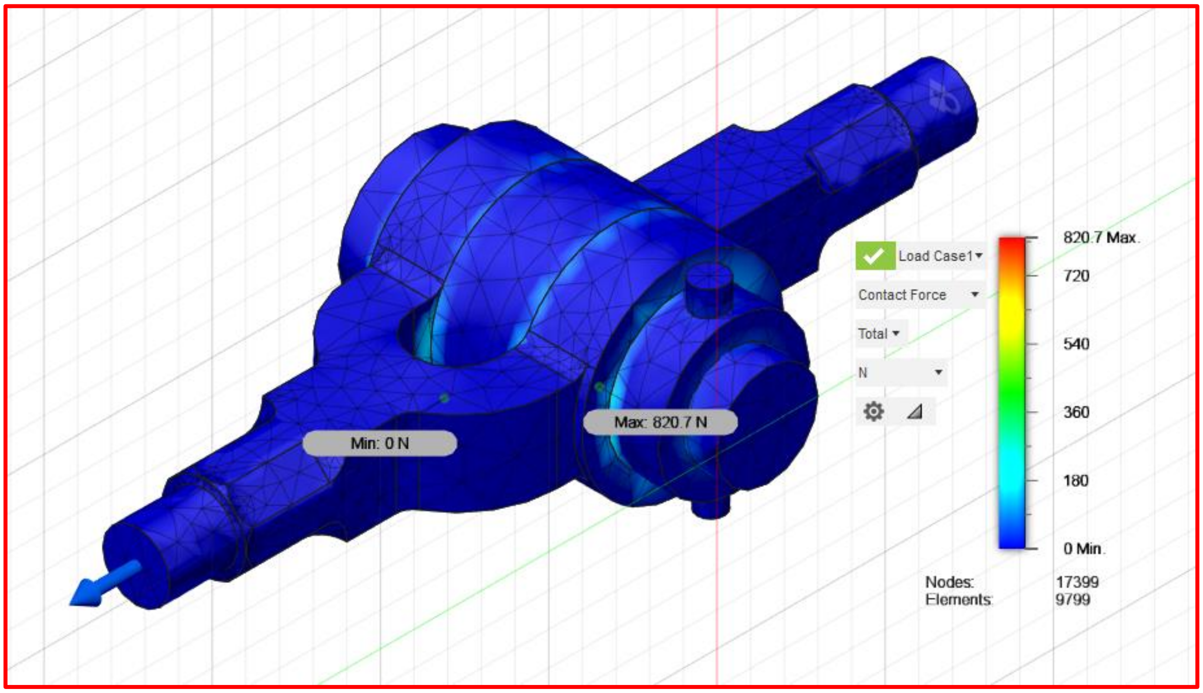


Figure 22. Contact Force of Knuckle Joint

Contact-impact phenomena are widely found in various mechanical systems, and accurate modeling of the contact-impact will directly affect the accuracy of modeling and simulation of related mechanical systems (Zhang, Fang, Zhao, & Zhao, 2022). The contact force is defined as the force which is exerted by bodies in contact with one another. Newton's Laws have a direct impact on the contact forces. The contact force results in Fusion 360 are described as the force applied to operate along the contact surface (Fusion 360, n.d.).

The contour map shows that the greatest contact force is 820.7 Newtons, while the minimum contact force is 0 Newtons. You can see this information displayed in the figure. From what I can tell, the results of the contact pressure are almost exactly the same. The one and only difference is that the area occupied by the minimum contact force of 0 Newtons occupies both the fork-end and the eye-end area, leaving the entire thing in a blue hie, which indicates that the object is at rest. The weight of 15,000 pounds coming from the fork end is equivalent to the force acting laterally (normal force). Because of this, ***the two forces were able to nullify one another***, and as a result, it became zero.

On the other hand, in the maximum contact force, it only has an in the centroid portion of the design, which indicates that there is a contact force in that area which is not all that terrible. This is the case since it only has a occurs in the centroid part of the design.

CONCLUSION

Fusion 360, which is software developed by AutoCAD, is used to perform all of the necessary calculations and analysis for the simulation. Steel-cast makes up the majority of the material that goes into making this knuckle joint. However, based on our research into the mechanical properties of various steels, we are able to classify steel-cast alongside structural steel due to the fact that both types of steel have identical properties. The knuckle joint is a versatile component that finds widespread use in a variety of applications, including those involving automobiles and other fields. Therefore, it should have sufficient strength to withstand the varying amounts of load that are applied to the system; otherwise, there is the potential that it will fail. Due to the existence of a knuckle split-pin, the design of a knuckle joint makes it easy to manufacture, assemble, and dismantle the joint. This is because the split-pin can assist in disconnecting other joint components.

In the eye, as well as in the fork-end and eye-end of the knuckle joint, the checking tensile, crushing, and shear stresses were all lower than the permissible stress. Following the completion of all of the processes involved in the study, the article comes to the conclusion that the steel-cast knuckle joint has a maximum stress limit of 170 MPa, while the greatest stress that can be generated by simulation is 65.84 MPa. ***As a result, we are able to draw the conclusion that the design has been securely organized and examined, and that it is suited for its uses.***

REFERENCES

AutoDesk. (2016). *Contact Pressure Result*. Retrieved from AutoDesk: https://help.autodesk.com/view/fusion360/ENU/?guid=SIM-CONTACT-PRESSURE-VIEW-CONCEPT

AutoDesk. (2017). *Displacement Results*. Retrieved from AUTODESK: https://help.autodesk.com/view/fusion360/ENU/?guid=SIM-DISPLACEMENT-VIEW-CONCEPT

Autodesk. (n.d.). *Reactions Results*. Retrieved from AUTODESK: https://help.autodesk.com/view/fusion360/ENU/?guid=SIM-REACTION-FORCE-VIEW-CONCEPT

Bansal, R. (2018). *What is Maximum Principal Stress Theory*. Retrieved from hkdivdei: https://www.hkdivedi.com/2017/08/what-is-maximum-principal-stress-theory.html

C, G., & R, S. (2006, October 11). *Chapter 1 - Mechanical Analysis of the Scratching Properties of Coated Polymers*. Retrieved from ScienceDirect: https://doi.org/10.1016/S0167-8922(06)80037-4

D, B., M, N., & A, C. R. (2018, March 6). *PERFORMANCE ANALYSIS OF KNUCKLE JOINT MADE OF METAL AND POLYMER*. Retrieved from IJCRT: https://www.ijcrt.org/papers/IJCRT1803225.pdf

D, S., & K, K. (2015). FE Analysis of Knuckle Joint Pin used in Tractor Trailer. *Asian Research Publishing Network, Vol.10,*, 2227-2232.

Ghosh, S. (2009, January 7). *Theories of failure for brittle materials (part-3): Maximum principal stress theory*. Retrieved from mechGuru: https://mechguru.com/machine-design/theories-of-failure-for-brittle-materials-part-3-maximum-principal-stress-theory/

Holmes, D. O. (n.d.). *Mechanics of Materials: Strain*. Retrieved from Mechanics of Slender Structures: https://www.bu.edu/moss/mechanics-of-materials-strain/

Holts, J. (2022, October 24). *Contact Pressure Vs. Normal Stress*. Retrieved from AutoDesk: https://forums.autodesk.com/t5/inventor-nastran-forum/contact-pressure-vs-normal-stress/td-p/11483567?us_oa=forums-us&us_si=94d0a3d9-781d-4534-936c-7ec4830733d2&us_st=contact%20pressure

J, S., & Vanka, H. (2015). Modeling and Analysis of Knuckle Joint. *International Journal & Magazine of Engieering, Technology, Management and Research, vol.2, issue 11*, 292-298.

M, S., & M, T. (2014). Joint wear simulation. *Computational Modelling of Biomechanics and Biotribology in the Musculoskeletal System*, 39-89.

Manesh P. Sharma, D. S. (2014). Static Analysis of Steering Knuckle and Its Shape Optimization. *IOSR Journal of Mechanical and Civil Engineering*, 34-38.

MDME. (n.d.). *Introduction to FEA* . Retrieved from MDME: Manfacturing, Design, Mechanical Engineering: http://www.mem50212.com/MDME/MEMmods/MEM09155A-CAE/010-Intro-FEA/Intro-FEA.html

N, S., & R, R. (2015). Study and analysis of knuckle joint with the replacement of material by using Teflon. *Journal of Engineering Research & Application, Vol.5, Issue 3*, 67-71.

Nipun. (2015, October 14). *Difference Between Yield Strength and Tensile Strength*. Retrieved from Pediaa: https://pediaa.com/difference-between-yield-strength-and-tensile-strength/

R&D Editors. (2015, July 23). *How to Determine If a Mesh is Fine Enough for Accurate Results*. Retrieved from R&D World: https://www.rdworldonline.com/how-to-determine-if-a-mesh-is-fine-enough-for-accurate-results/

SIMSCALE. (2022, September 9). *What is von Mises Stress?* Retrieved from SimScale : https://www.simscale.com/docs/simwiki/fea-finite-element-analysis/what-is-von-mises-stress/

The Physics Classroom. (2018). *Distance and Displacement*. Retrieved from The Physics Classroom: https://www.physicsclassroom.com/class/1DKin/Lesson-1/Distance-and-Displacement

Zhang, J., Fang, M., Zhao, L., & Zhao, Q. L. (2022). A continuous contact force model for the impact analysis of hard and soft materials. *Mechanism and Machine Theory*. Retrieved from https://doi.org/10.1016/j.mechmachtheory.2022.105065